



# 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

### ■ 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
- AGGROT, 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

- 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
- YTTITLE, 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

- 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
- MRPBUSH, 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 $\geq$ 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 gpid=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. <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. (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. <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)      NO MORE MEMORY SPACE AVAILABLE FOR EXECUTIVE TABLES.      USER ACTION: INCREASE DEFAULT NUMBER OF EXECUTIVE TABLES ENTRIES VIA SYSTEM CELL 162.  <b>SYSTEM(162) IS CURRENTLY SET TO 6000</b></p> <p>A DMAP workaround is of the form:      Just before the offending module      NP=GETSYS(NP,162) \$      NP1= value \$ value &gt; 6000      PUTSYS(NP1,162) \$      Just after offending module      PUTSYS(NP1,162) \$</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.
<a href="#">CHEXAINT (212)</a>	Specifies CHEXA element’s integration rule for p-adaptive analysis and p=2x2x2 (only).	
	0	Reduced (Default)
	1	Full
<a href="#">DISTORT (213)</a>	Element distortion fatal termination override. Applies to all p-elements and the TETRA h-elements.	
	0	Terminate run (Default)
	1	Do not terminate run.
<a href="#">T3SKEW (218)</a>	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.	
<a href="#">(219)</a>	Rank to use for complex symmetric sparse decomposition high rank update (Default = 1).	
<a href="#">(220)</a>	Rank to use for real unsymmetric sparse decomposition high rank update (Default = 1).	
<a href="#">(221)</a>	Rank to use for complex unsymmetric sparse decomposition high rank update (Default = 1).	
<a href="#">(253 - 262)</a>	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.	
<a href="#">MAXSET (263)</a>	Controls the default number of vectors in block or set for Lanczos Eigenvalue extraction. See <a href="#">EIGRL, 1924</a> . The default is 7 for most machines but it is machine dependent.	
<a href="#">QUARTICDLM (270)</a>	A value of 1 selects the new quartic formulation of the doublet lattice kernel (N5KQ), while 0 selects the original quadratic form (Default = 0).	
<a href="#">(273)</a>	A value of 1 selects the old Lanczos shift logic from Version 70 and previous systems (Default = 0).	
<a href="#">DBCFACT (274)</a>	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  Default value=1024 words The physical I/O size is IFPBUFF words. The maximum value of IFPBUFF is 65536 words.  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. 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. 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  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.  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  > 100000, work area size in word	
<a href="#">H5GM34 (751)</a>	Write GEOM3 and GEOM4 data in NH5RDB	
	-1	Use OP2GM34 setting (Default).
	1	Write GEOM3 and GEOM4 data in NH5RDB
<a href="#">UDEFGRID(753)</a>	0      Do not write GEOM3 and GEOM4 data in NH5RDB	
	Control whether having undefined grid points on ASET/ASET1 entries is allowed.  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> <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> <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 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 per 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]
```

$$\left[ \begin{array}{l} [\text{STATUS} = ] \\ \left\{ \begin{array}{l} \text{NEW} \\ \text{OLD} \\ \text{UNKNOWN} \end{array} \right\} \end{array} \right]$$

$$\left[ \begin{array}{l} [\text{FORM} = ] \\ \left\{ \begin{array}{l} \text{FORMATTED} \\ \text{UNFORMATTED|UNFORMATTED\_64} \\ \text{UNFORMATTED\_32} \\ \text{UNFORMATTED\_MIXED} \end{array} \right\} \end{array} \right]$$

```
[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. “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. “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. “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. If “NO”, the unit (if any) and logical name are reserved by MSC Nastran.
Open	If “YES”, the file is opened by default. If “NO”, the file must be explicitly opened.

Access	If “SEQ”, the file is opened for sequential access. 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>  <b>BEAMEVAL</b>  <b>DRESP3</b>  <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 (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 (Old Group Format)	Meaning
group	Group name referenced by the GROUP field on DRESP3, GMCURV, GMSURF, PBRL, 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 PBRL 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	<pre>(FWIDTH = w [.d] DWIDTH = w [.d] AWIDTH = a IWIDTH = i, LWIDTH = k COLSPACE = c VALUE = w, colname = col-width, . . . ),</pre>
--------	---

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, 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 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 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 describers.

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 IWIDTHT=5 LWIDTHT=1 ,
         FWIDTH=12.5 COLSPACE=2) SORT(KEY=A) ,
  LABEL('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      22

MODULE NAME = DBDICT , SUBDMAP SEKRRS , OSCAR RECORD NUMBER      18
                           QUALIFIERS
KEY APRCH B2GG B2PP BMETH CMETH CONFIG DEFORM DELTA DESITER DLOAD DRMM DYRD EXTRCV FMETH FREQ
   FSCOUP GUST HIGHQUAL HINDEX IC IKEBAR IMACHNO IPANEL IQ ISA ISOLAPP K2GG K2PP LOAD
   M2GG M2PP MACHINE METH MFLUID MODEL MPC MTEMP NCASE NL99 NLOAD NLOOP NOQUAL OPERALEV
OPERASYS P2G PEID PVALID SDAMP SEDWN SEID SOLAPP SOLID SPC STATSUB SUBDMAP SUBMODEL SUPORT
   TEMPPLD TFL TSTEP ZNAME ZUZR1 ZUZR2 ZUZR3

----- 335   '
          '           0                                     F  0
          '           '           0                                     0
          '           '           0           0           0
----- 336   '
          '           0                                     0
          '           '           0           0           0
          '           '           0           0           -1
-----
```

### Figure 3-3 DBDICTIONARY 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   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   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 {  $\left\{ \begin{array}{c} \text{BINARY} \\ \text{NEUTRAL} \end{array} \right\}$  } {  $\left\{ \begin{array}{c} \text{OVRWRT} \\ \text{NOOVRWRT} \end{array} \right\}$  }

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.  Modifies the values for PROJECT, VERSION, DBSET, and qualifiers selected by the where-expr. The format of convert-expr is:  PROJECT=project-expr; VERSION=version-expr; DBSET=DBset-name; quali=qual-expr[;...]  For example, CONVERT (SEID=100+SEID; SPC=102). See the beginning of this section for more information on WHERE and CONVERT clauses.  The default action for VERSION and PROJECT is to use the same version IDs and project IDs; i.e., CONVERT(PROJECT=PROJECT; 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 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 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.  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:  PROJECT=project-expr; VERSION=version-expr; DBSET=DBset-name; quali=qual-expri[;...]  For example, <code>CONVERT (SEID=100+SEID; SPC=102)</code> See the beginning of this section for more information on WHERE and CONVERT clauses.  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 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' , UNIT=12 FORMATTED

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

DBUNLOAD	DATABLK=(KAA) WHERE(PROJECT='FRONT BUMPER' ,AND 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, 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 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 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 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) 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. Following order:

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.

a. Objects created with the COMPILE statement in the current run.

b. 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. 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, 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, 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 (ACMS)	Selects the number of domains as follows: If NUMDOM = 0 or 1, then the model will not be split. Default depends on the model size and the value of PARTOPT.  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 (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 (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 (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 (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 (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 (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 (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. 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:  SUPER      Specifies the SUPER compression method. (Default) GRID        Specifies the GRID compression method. NONE        Specifies that no compression should be done.
RUN OPT	Specifies the DMP run option. The following are valid options for RUN OPT:  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.  MULTIMST    Specifies "multiple-master" DMP execution mode. All DMP processes execute the full Nastran solution. This is optimal for multiple-host DMP execution.  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.  BEND        Specifies the Extreme/BEND partitioning method MSCMLV     Specifies the MSCMLV partitioning method 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.  Default: 1000 (ACMS Version=OLD); 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:  0      Default N      Set parallel group size to "N" SMP threads. -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

Descriptor	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.”
[...] 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.
\$	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: Linear steady state heat transfer. Alternate reduction. Inertia relief.
103	SEMODES	Normal modes.
105	SEBUCKL	Buckling with options: Static analysis. Alternate reduction. 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: 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: Linear or nonlinear analysis.
159	NLTCSH	Transient structural and/or transient heat Transfer analysis with options: 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 SOL 600, ID, 153)	MSC Nastran API into MSC MARC nonlinear
700	NLTRAN ( See SOL 700, ID, 172)	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:
	<ul style="list-style-type: none"> <li>0 Store output in OESNLBR and OESNLXR (OESNLXR will be empty like SOL 106)</li> <li>1 Store output in OESNLBR</li> <li>2 Store output in OESNLBD</li> <li>3 Store output in OESNLXD</li> </ul>
<b>Note:</b>	<ol style="list-style-type: none"> <li>1. SOL 106 outputs both OESNLBR and OESNLXR but OESNLXR is empty.</li> <li>2. SOL 129 only outputs the OESNLXD datablock and it is empty.</li> <li>3. Case Control BOUTPUT is also required to obtain this type of output.</li> <li>4. The default is 0 if contact is present in the model and OUTR=op2 (or .xdb, punch and/or .f06).</li> <li>5. This option is specified like the example shown: OUTR=OP2,F06,CDB=0</li> <li>6. The datablocks have the same names and type of information whether executing SOL 600,106 or SOL 600,129.</li> </ol>

### 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, MATE, 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 or 3) 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 MRENUxxx 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.  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)  (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)  (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" The format of hostfile is as follows for the example for machine: machine1 2 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)  (Default is to use NP on the SOL 700 line. If NP and nproc are omitted, the default is 1.) NOTE: The number of requested processes must be a power of 2.
ncpus	Number of processors. (only for SMP)  (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 =  $\left( \left[ \begin{array}{l} \text{[NO]CHART} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right] \left[ \begin{array}{l} \text{nsegs} \end{array} \right] \right)$ ,  
 $\left[ \begin{array}{l} \text{[NO]TABLE} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right] \left[ \begin{array}{l} \text{nratios} \\ \text{NMAXRAT} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right] \left[ \begin{array}{l} \text{maxratio} \\ \text{MAXRAT} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right]$ ,  
 $\left[ \begin{array}{l} \text{SORT} \\ \left\{ \begin{array}{l} \text{GRID} \\ \text{VALUE} \\ \text{BOTH} \end{array} \right\} \end{array} \right]$ ,  
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). 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). 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

<p>Brackets [ ] indicate that a choice of descriptors is optional.</p>		<p>Applied Load Output Request</p>	A brief description of the command is given.
<p>Requests the form and type of applied load vector output.</p>			Descriptors in uppercase letters are keywords that must be specified as shown.
<p><b>Format:</b></p>		$OLOAD \left[ \begin{array}{l} \text{[SORT1]} \\ \text{[SORT2]} \end{array} \right] , \text{PUNCH}, \left[ \begin{array}{l} \text{[ } \text{ or } \text{]} \\ \text{PHASE} \end{array} \right] \right] = \left\{ \begin{array}{l} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$	Descriptors in lower case letters are variables.
<p><b>Examples:</b></p>		<p>OLOAD = ALL</p> <p>OLOAD(SORT1,PHASE)</p>	Braces { } indicate that a choice of descriptors is mandatory.
<p>If the descriptors are stacked vertically, then only one may be specified.</p>			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. 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).		

### Remarks:

- Both PRINT and PUNCH
- See the *MSC Nastran User's Guide* for descriptions of SORT1 and SORT2 formats and their default output.
- In a statics problem, a request for zero load output is equivalent to a request for nonzero load output.

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.

## 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> or <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> or <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

[DEFORM \(Case\)](#)

Selects the element deformation set.

[CLOAD \(Case\)](#)

Requests a CLOAD Bulk Data entry that defines a list of superelement loads and their scale factors in nonlinear static analysis only.

[LOAD \(Case\)](#)

Selects an external static loading set.

[LOADNAME \(Case\)](#)

Provides a name to be associated with a loading condition.

### 2. Dynamic Load Selection

[DLOAD \(Case\)](#)

Selects a dynamic load or an acoustic source to be applied in a transient or frequency response problem.

[LOADSET \(Case\)](#)

Selects a sequence of static load sets to be applied to the structural model. The load sets may be referenced by dynamic load commands.

[NONLINEAR \(Case\)](#)

Selects nonlinear dynamic load set for transient problems.

### 3. Constraint Selection

[AXISYMMETRIC \(Case\)](#)

Selects boundary conditions for an axisymmetric shell problem or specifies the existence of fluid harmonics for hydroelastic problems.

[AUTOSPC \(Case\)](#)

Requests that stiffness singularities and near singularities be automatically constrained via single or multipoint constraints.

[BC \(Case\)](#)

Identifies multiple boundary conditions for normal modes, buckling, and flutter analysis in SOLs 103, 105, 145, and 200.

[DSYM \(Case\)](#)

Provides for either one or two planes of overall symmetry in DIH-type cyclic symmetry problems.

[MPC \(Case\)](#)

Selects a multipoint constraint set.

[SPC \(Case\)](#)

Selects a single-point constraint set to be applied.

[STATSUB \(Case\)](#)

Selects the static solution to use in forming the differential stiffness for buckling, normal modes, complex eigenvalue, frequency response, and transient response analysis.

[SUPORT1 \(Case\)](#)

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.

**AEXREF (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).  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), VECTOR (Case), or 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 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 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. 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.	$SDAMP \Rightarrow TABDMP1$ SDAMP above subcase.
sesdamp = yes	Modal damping for each superelement using the fixed boundary CMS modes.	$SDAMP \Rightarrow TABDMP1$ SDAMP in superelement subcase.
For part superelements 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:

- 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.
- 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

```

- 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=NSTAT** 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.  value1: Value of the lower bound.
V2ORTHO	Higher frequency bound of the Craig-Bampton modes in cycles/unit time.  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 BC CONTACT=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. CTYPE can be specified by bulk data section BCAUTOP also, the value of CTYPE in BCAUTOP will be used if it is specified.
7. When BC CONTACT=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 BC CONTACT=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 BC CONTACT=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:

<i>BEGIN[BULK]</i>	<i>AFPM = afpmid</i> <i>ARBMODEL = arbmid</i> <i>AUXMODEL = auxmid</i> <i>MASSID = massid[LABEL = masslabel]</i> <i>MODULE= moduleid[APPEND][LABEL = modlabel]</i> <i>FLXBDY = flexbody</i> <i>SUPER = seid</i> <i>TRMC = trimid</i> <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.

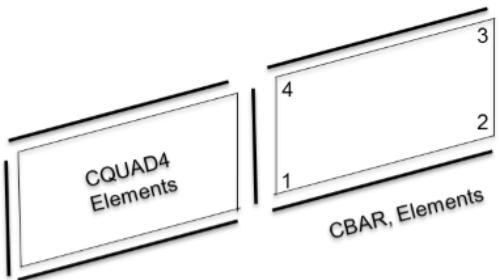
BENDL (PRINT) = ALL

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, 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, 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:**

$$\text{ELSDCON} \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix} = \begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$$

**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

$$\text{ELSENS(MASS)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r]$$

$$\text{ELSENS(STIFF)} = [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r]$$

$$\text{ELSEND(DYNAMIC)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r] + [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r]$$

$$\text{ELSENS(SQMMASS)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r] + [A_{\text{setf}}]^{*t} [M_{\text{elem}}]^* [A_r]^*$$

$$\text{ELSENS(SQSTIF)} = [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r] + [U_{\text{setf}}]^{*t} [K_{\text{elem}}]^* [U_r]^*$$

$$\text{ELSEND(DYNAMIC)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r] + [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r] + [A_{\text{setf}}]^{*t} [M_{\text{elem}}]^* [A_r]^* + [U_{\text{setf}}]^{*t} [K_{\text{elem}}]^* [U_r]^*$$

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, CTRIAx6, 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:**

$$ESE \left[ \begin{array}{c} \boxed{PRINT, PUNCH} \\ PLOT \end{array} \right] \left[ \begin{array}{c} AVERAGE \\ AMPLITUDE \\ PEAK \end{array} \right] [THRESH = p][NOPERCENT] ) = \left\{ \begin{array}{c} ALL \\ n \end{array} \right\}$$

**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, 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 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 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='extsldr.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='extsldr.op2', UNIT=45</pre> <p>In Case Control:</p> <pre>EXTDRIN (DMIGOP2=45)</pre>
MATOP4	<p>In FMS:</p> <pre>ASSIGN OUTPUT4='extsldr.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='extsldr.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 HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED STATICS
					MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED STATICS FOR BUCKLING ANALYSIS
2	MODES	7	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED NORMAL MODES - SOLVE ALL EIGENVALUES
3	BUCKLE	6	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED BUCKLING - SOLVE ALL EIGENVALUES
4	MODES	3	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED 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 HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED 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 HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED MODAL TRANSIENT - FIFTY TIME STEPS AND DELTA-T=.001
7	MTRAN	1	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED MODAL TRANSIENT - FIFTY TIME STEPS AND DELTA-T=.001
8	MCEIG	4	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED 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 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:  <i>unitop4</i> > 0: Store in sparse format  <i>unitop4</i> < 0: Store in non-sparse format on Fortran unit number given by  unitop4
<b>MATRIXDB or 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 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.  <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 (or MATRIXDB) = dbext (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> (or MATRIXOP4 = <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 (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 (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 (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:

$$\text{FBODYLD}(LID) = \left\{ \begin{array}{c} \text{ALL} \\ \text{name}_1, \text{name}_2, \text{name}_3, \dots \end{array} \right\}$$

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. 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. 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

Descriptor	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 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:

$$FRF \left[ \left( \begin{array}{c} GEN \\ ASM \\ GENASM \end{array} \right) \right] [COMPID = cpid][COMPNAME = cmpname]$$

$$[CONNPTS = setid] \left[ XITOUT = \left\{ \begin{array}{c} UNIT \\ UNITALL \\ USER \\ USERTOTL \end{array} \right\} \right] \left[ ASMOUT = \left\{ \begin{array}{c} COMP \\ ALL \\ ASSEMBLY \\ CONNINFO \\ n \\ cname \end{array} \right\} \right] \left[ LOADLBL = \left\{ \begin{array}{c} STD \\ ALT \\ ALTX \end{array} \right\} \right]$$

$$\left[ \left( \begin{array}{c} DB \\ OP2 = unit \end{array} \right) \right]$$

$$\left\{ \begin{array}{l} ICFGEN= \left\{ \begin{array}{c} ALL \\ n \\ compname \end{array} \right\} \\ ICFUSE= \left\{ \begin{array}{c} n \\ compname \end{array} \right\} \\ ICFAUTO= \left\{ \begin{array}{c} n \\ compname \end{array} \right\} \end{array} \right\} = \left\{ \begin{array}{c} ICFDB \\ ICFOP2=\{icfunit\} \end{array} \right\}$$

**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 ASMOUT = 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 (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 (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 (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 (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:  n > 0  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.  n < 0  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:  n > 0  First generate and then use ICF information in the FBA process only for those FRF components whose IDs are specified by SET ID n.  n < 0  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 ASMOUT = 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 ASMOUT = 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 ASMOUT 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:**

$$GPSDCON \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix} = \left\{ \begin{array}{l} ALL \\ n \end{array} \right\}$$

**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 (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:**

$$\text{HDOT} \left[ \left( \begin{bmatrix} \text{SORT1} \\ \text{SORT2} \end{bmatrix}, \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix} \right) \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Example:**

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 (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

Describer	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. (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. (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:**

$$\text{MAXMIN(DEF)} \{ \text{DISP, VELO, ACCE, MPCF, SPCF, OLOAD} \} \{ T1 T2 T3 R1 R2 R3 MAGT MAGR \},$$

$$\left[ \begin{array}{c} \text{ABSOLUTE}(p) \text{ MINALG}(q) \text{ MAXALG}(r) \\ \text{ALL}(p) \end{array} \right] [\text{RMS}] \left[ \begin{array}{c} \text{BRIEF} \\ \text{FULL} \end{array} \right],$$

$$\left[ \begin{array}{c} \text{CID} = \left\{ \begin{array}{c} \text{GLOBAL} \\ \text{BASIC} \\ \text{cid} \end{array} \right\} \end{array} \right]$$

**Format for element output:**

$$\text{MAXMIN(DEF)} \left\{ \begin{array}{c} \text{STRESS} \\ \text{STRAIN} \\ \text{FORCE} \end{array} \right\} \{ \text{eltype1}, \text{eltype2}, \dots \} \left\{ \begin{array}{c} \text{compl} \left[ \begin{array}{c} \text{CENTER} \\ / \text{ALL} \\ \text{GROUP} \\ \text{ENDS} \end{array} \right] [ , \text{comp2}, \dots ] \end{array} \right\}$$

$$\left[ \begin{array}{c} \text{ABSOLUTE}(p) \text{ MINALG}(q) \text{ MAXALG}(r) \\ \text{ALL}(p) \end{array} \right] [\text{RMS}] \left[ \begin{array}{c} \text{BRIEF} \\ \text{FULL} \end{array} \right]$$

**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: 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 algebraically in ascending order. 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:

$$\text{MEFFMASS} \left[ \begin{bmatrix} \text{PRINT} \\ \text{NOPRINT} \end{bmatrix} \begin{bmatrix} \text{PUNCH} \\ \text{NOPUNCH} \end{bmatrix}, \text{GRID} = \text{gid}, \begin{bmatrix} \text{SUMMARY}, \text{PARTFAC}, \\ \text{MEFFM}, \text{MEFFW}, \\ \text{FRACSUM}, \text{ALL} \end{bmatrix} \right] = \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\}$$

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 0.0	1.409326E-02 0.0	1.065749E-02 0.0
2	2.049217E-05 0.0	2.465558E-03 0.0	1.864483E-03 0.0
3	5.175167E-04 0.0	6.226612E-02 0.0	4.708635E-02 0.0
4	1.732168E-05 0.0	2.084095E-03 0.0	1.576016E-03 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 0.0	1.000000E+00 0.0	9.924866E-01 0.0
2.000000E+00	4.066132E-03 0.0	1.000000E+00 0.0	9.936857E-01 0.0
3.000000E+00	1.411670E-02 0.0	1.000000E+00 0.0	9.955440E-01 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:

$$\begin{aligned}
 & \text{MODALSE} \left[ \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right] \left[ \begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right] \left[ \begin{array}{l} \text{PUNCH} \\ \text{REAL or IMAG} \end{array} \right] \left[ \begin{array}{l} \text{PHASE} \\ \text{ESORT} = \left\{ \begin{array}{l} \text{MODE} \\ \text{ASCEND} \\ \text{DESCENT} \end{array} \right\} \end{array} \right] \\
 & [\text{THRESH} = e] \left[ \begin{array}{l} \left\{ \begin{array}{l} \text{TIME} \\ \text{FREQ} \end{array} \right\} \\ r \end{array} \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\} \left( \left[ \begin{array}{l} \text{AVERAGE} \\ \text{AMPLITUDE} \\ \text{PEAK} \end{array} \right] \right) = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}
 \end{aligned}$$

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 0.0	1.409326E-02 0.0	1.065749E-02 0.0
2	2.049217E-05 0.0	2.465558E-03 0.0	1.864483E-03 0.0
3	5.175167E-04 0.0	6.226612E-02 0.0	4.708635E-02 0.0
4	1.732168E-05 0.0	2.084095E-03 0.0	1.576016E-03 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 0.0	1.000000E+00 0.0	9.924866E-01 0.0
2.000000E+00	4.066132E-03 0.0	1.000000E+00 0.0	9.936857E-01 0.0
3.000000E+00	1.411670E-02 0.0	1.000000E+00 0.0	9.955440E-01 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:  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:

$$\text{MONITOR}[\text{REAL or IMAG}, \text{PHASE}, \text{NODSP1}, \text{NOPNT1}, \text{NOPNT2}, \text{NOPNT3}] = \left\{ \begin{array}{l} \text{ALL} \\ \text{NONE} \end{array} \right\}$$

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. 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:  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. 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

Describer	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:

$$RCROSS \left[ \begin{array}{c} \left[ \begin{array}{c} REAL \text{ or } IMAG \\ PHASE \end{array} \right] \left[ \begin{array}{c} PRINT \\ NOPRINT \end{array} \right], [PUNCH], [PSDF, CORF, RALL] \end{array} \right] = n$$

**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/ NOINRL	Controls calculation of residual vectors based on inertial forces due to rigid-body motion (Default =INRLOD).
APPLOD/ NOAPPL	Controls calculation of residual vectors based on applied loads (Default = APPLOD).
ADJLOD/ NOADJLOD	Controls calculation of residual vectors based on adjoint load vectors (SOL 200 only; Default = ADJLOD).
RVDOF/ NORVDOF	Controls calculation of residual vectors based on RVDOFi entries (Default = RVDOF).
DAMPLOD/ NODAMP	Controls calculation of residual vectors based on viscous damping (Default = DAMPLOD).
DYNRSP/ 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/ 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/ 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 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:**

$$\text{SDISPLACEMENT} \left[ \begin{bmatrix} \text{SORT1} \\ \text{SORT2} \end{bmatrix}, \text{PRINT}, \text{PUNCH}, \begin{bmatrix} \text{REAL or IMAG} \\ \text{PHASE} \end{bmatrix} \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**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.