

Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
AEMOND1 Aerodynamic MONDSP1	Component See Remark 36.	Blank	Blank
TRIM	AESTAT or AESURF entry ID	Blank	Blank
STABDER	AESTAT or AESURF entry ID	Restraint Flag. (Integer 0 or 1) See Remark 13.	Component
FLUTTER	Blank	Blank	See Remark 14.
DIVERG	Divergence Root Number See remark 38.	Blank	Mach No.
WMPID See Remarks 44. and 45.	MID	SEID	PID

Remarks:

1. Stress, strain, force, fatigue and ERP item codes can be found in [Item Codes, 1045](#). For stress or strain item codes that have dual meanings, such as von Mises or maximum shear, the option specified in the Case Control Section will be used; i.e., STRESS(VONM) or STRESS(MAXS).
2. RTYPE = "CSTRESS", "CSTRAIN", "CFAILURE", and "CSTRAT" are used only with the PCOMP/PCOMPG entry. "CSTRESS" and "CSTRAIN" item codes are described under Table 1 (Element Stress/Strain Item Codes) in [Item Codes, 1045](#). "CFAILURE" and "CSTRAT" item codes are described under Table 2 (Element Force Item Codes) in [Item Codes, 1045](#). Only force item codes that refer to failure indices of direct stress and interlaminar shear stress are valid.

The CFAILURE and CSTRAT response types requires the following specifications on the applicable entries:

- Failure theory in the FT field on PCOMP/PCOMPG entry.
 - Allowable bonding shear stress in the SB field on PCOMP/PCOMPG entry.
 - Stress limits in the ST, SC, and SS fields on all MATi entries.
 - When the PCOMPG is invoked, there is no default for the ATTB field.
3. ATTB can be used for responses of weight, composite laminae, dynamics, real and complex eigenvalues, WMPID and stability derivatives. For eigenvector responses, such as DISP, the ATTB identifies the mode of interest. For other responses, this field must be blank.
 4. All grids associated with a DRESP1 entry are considered to be in the same region for screening purposes. Only up to NSTR displacement constraints (see DSCREEN entry) per group per load case will be retained in the design optimization phase.

5. DRESP1 identification numbers must be unique with respect to DRESP2 identification numbers.
6. If PTYPE = "ELEM", the ATT_i correspond to element identification numbers.
7. If RTYPE = "DISP", "SPCFORCE", "GPFORCE", "TDISP", "TVELO", "TACCL" or "TSPCF", multiple component numbers (any unique combination of the digits 1 through 6 with no embedded blanks) may be specified on a single entry. Multiple response components may not be used on any other response types.
8. If RTYPE = "FRDISP", "FRVELO", "FRACCL", or "FRSPCF" only one component number may be specified in the ATT_A field. Numbers 1 through 6 correspond to real (or magnitude) components and 7 through 12 imaginary (or phase) components. If more than one component for the same grid is desired, then a separate entry is required.
9. Real/imaginary representation is the default for complex response types. Magnitude/phase representation must be requested by the corresponding Case Control command; e.g., DISP(PHASE) = ALL for FRDISP type responses.
10. REGION is used for constraint screening. The NSTR field on DSCREEN entries gives the maximum number of constraints retained for each region per load case.

IF RTYPE = "WEIGHT", "VOLUME", "LAMA", "EIGN", "FREQ", "CEIG", "TOTSE", "RMSDISP", "RMSVELO", "RMSACCL", no REGION identification number should be specified. If the region field is left blank for a grid response, one region is created for each DRESP1 ID. If the region field is left blank for an element response, one region is created for each property ID invoked. Usually, the default value is appropriate.

If the REGION field is not blank, all the responses on this entry as well as all responses on other DRESP1 entries that have the same RTYPE and REGION identification number will be grouped into the same region.
11. REGION is valid only among the same type of responses. Responses of different types will never be grouped into the same region, even if they are assigned the same REGION identification number by the user.
12. If RTYPE = "WEIGHT", "VOLUME", or "TOTSE" field ATT_i = "ALL" implies total weight/volume/total strain energy of all superelements except external superelements, 0 implies residual only and i implies SEID=i. Default="ALL". RTYPE="TOTSE" is not supported for shape optimization.
13. RTYPE = "STABDER" identifies a stability derivative response. ATT_B is the restraint flag for the stability derivative. ATT_B = 0 means unrestrained, and ATT_B = 1 means restrained. For example, ATT_A = 4000, ATT_B = 0, and ATT₁ = 3 reference the unrestrained C_z derivative for the AESTAT (or AESURF) entry ID = 4000.
14. RTYPE = "FLUTTER" identifies a set of damping responses. The set is specified by ATT_i:

ATT₁ = Identification number of a SET1 entry that specifies a set of modes.

ATT₂ = Identification number of an FLFACT entry that specifies a list of densities.

ATT₃ = Identification number of an FLFACT entry that specifies a list of Mach numbers.

ATT₄ = Identification number of an FLFACT entry that specifies a list of velocities.

If the flutter analysis is type PKNL, it is necessary to put PKNL in the PTYPE field of this entry.

15. For RTYPE = "FRXXXX", "PSDXXXX", "PRES" and "ERP" a real value for ATTB specifies a frequency value in cycles per unit time. If a real ATTB value is specified, then the responses are evaluated at the closest excitation frequency. The default for ATTB is all excitation frequencies. See Remark 20. for additional ATTB options. The OFREQ Case Control command has no effect on the selection of the frequencies.
16. For RTYPE = "TDISP", "TVELO", "TACCL", "TSPCF", "TFORC", and "TSTRE", ATTB specifies a time value. If ATTB is specified, then the responses are evaluated at the closest time selected by the OTIME command. The default for ATTB is all time steps selected by the OTIME command.
17. Intermediate station responses on CBAR elements due to PLOAD1 and/or CBARAO entries may not be defined on the DRESP1 entry.
18. RTYPE = "EIGN" refers to normal modes response in terms of eigenvalue (radian/time)**2 while RTYPE = "FREQ" refers to normal modes response in terms of natural frequency or units of cycles per unit time.
19. For RTYPE = LAMA, EIGN or FREQ, the response approximation used for optimization can be individually selected using the ATTB field when APRCOD = 1 is being used.

For RTYPE = LAMA, ATTB = blank or 1 selects direct linearization, ATTB = 2 = inverse linearization.

For RTYPE = EIGEN or FREQ, ATTB = blank = Rayleigh Quotient Approximation, = 1 = direct linearization, = 2 = inverse approximation.

The default Rayleigh Quotient Approximation should be preferred in most cases.

20. Character input for ATTB is available for RTYPE of "FRXXXX", "PSDXXXX", "TXXXX", "PRES" and "ERP". The character input represents a mathematical function and the options for character input are SUM, AVG, SSQ, RSS, MAX and MIN. The expression of mathematical function is shown as follows:

$$\text{SUM}(X_1, X_2, \dots, X_n) = \sum_{i=1}^n X_i$$

$$\text{AVG}(X_1, X_2, \dots, X_n) = \frac{\sum_{i=1}^n X_i}{n}$$

$$\text{SSQ}(X_1, X_2, \dots, X_n) = \sum_{i=1}^n X_i^2$$

$$\text{RSS}(X_1, X_2, \dots, X_n) = \sqrt{\sum_{i=1}^n X_i^2}$$

$$\text{MAX}(X_1, X_2, \dots, X_n) = \text{Maximum value among } X_i(i=1 \text{ to } n)$$

$$\text{MIN}(X_1, X_2, \dots, X_n) = \text{Minimum value among } X_i(i=1 \text{ to } n)$$

where X_i is the response for a forcing frequency or time step. For example

DRESP1,10,DX1,FRSTRE,ELEM,,3,AVG,10

yields a response which is equal to the average stress for element 10 across all forcing frequencies.

CAUTION!

1. If the ID of a DRESP1 with ATTB character input is referenced by a DRESP2 using the "DRESP2" flag, the results of the mathematical function are used to evaluate the DRESP2.
2. If the ID of a DRESP1 with ATTB character input is referenced by a DRESP2 using the "DRESP1" flag, the individual values of the DRESP1 are used to compute individual DRESP2 values and the mathematical function is not used by the DRESP2.

21. Element strain energy item codes can be found under [Table 7-13](#) in [Item Codes, 1045](#). Only element strain energy and element strain energy density can be referenced on a DRESP1 entry. RTYPE="ESE" is not supported for shape optimization.
22. For RTYPE=RMSDISP, RMSVELO, or RMSACCL the ATTB specifies the appropriate RANDPS ID.
23. Input other than 1 or 7 of ATTA field, acoustic pressure component, for PRES response type will be reset to 1 (if less than 7) or 7 (if greater than 6 and less than 13).
24. Design response weight is obtained from Grid Point Weight Generator for a reference point GRDPNT (see parameter GRDPNT). If GRDPNT is either not defined, equal to zero, or not a defined grid point, the reference point is taken as the origin of the basic coordinate system. Fields ATTA and ATTB refer to the row and column numbers of the rigid body weight matrix, which is partitioned as follows:

$$[W] = \begin{bmatrix} W_x & W_{12} & W_{13} & W_{14} & W_{15} & W_{16} \\ W_{21} & W_y & W_{23} & W_{24} & W_{25} & W_{26} \\ W_{31} & W_{32} & W_z & W_{34} & W_{35} & W_{36} \\ W_{41} & W_{42} & W_{43} & I_x & W_{45} & W_{46} \\ W_{51} & W_{52} & W_{53} & W_{54} & I_y & W_{56} \\ W_{61} & W_{62} & W_{63} & W_{64} & W_{65} & I_z \end{bmatrix}_{6 \times 6}$$

The default values of ATTA and ATTB are 3, which specifies weight in the Z direction. Field ATT1 = "ALL" implies total weight of all superelements except external superelements. SEIDI refers to a superelement identification number. SEIDI = "0" refers to the residual superelement. The default of ATT1 is blank which is equivalent to "ALL".

25. For RTYPE = GPFORCE, the PTYPE field is used to designate the GRID ID at which the force is defined. Output that is produced using PARAM NOELOF > 0 is not supported for the DRESP1 entry.

26. For RTYPE = GPFORCP, the PTYPE field is blank. The grid point force is for the sum of all elements from the GRID ID listed in ATTA to the GRID (orient ID) listed in ATTi. This response corresponds to that produced with PARAM NOELP > 0. It is not necessary to set PARAM NOELP > 0 to compute the GPFORCP response.
27. For RTYPE = PSDXXXX, the PTYPE field specifies the RANDPS ID.
28. RTYPE=COMP (compliance of structures = $P^T u$) and FRMASS (mass fraction of topology designed elements) entries are used for topology optimization or combined topology, sizing/shape optimization.
29. RTYPE=FRMASS can be used for topology and topometry optimization. For topology, FRMASS, is the mass divided by the mass calculated if all topology design variables are 1.0. FRMASS is calculated for designed elements only. FRMASS = 1.0 if all design variables are 1.0. For topometry, the initial FRMASS is defined as 1.0 for the initial design specified on the TOMVAR entries. For non-volume elements like CELAS, an artificial mass = 1.0 is assumed for each element.
30. Response type ABSTRESS is for element type code of 238 (CBAR) and 239 (CBEAM) only.
31. If a PSDxxxx or RMSxxxx response is being applied with RANDPS Bulk Data entries that include multiple subcases, the DESSUB or DESOBJ Case Control command that invokes the PSDxxxx or RMSxxxx response must be in the first subcase used by the RANDPS entry.
32. Response types, ESE, TOTSE, GPFORCE and GPFORCP are not supported for shape optimization.
33. For RTYPE=EIGN or FREQ, PTYPE field can be utilized to identify the source of the mode. Valid options are 'STRUC' or 'FLUID'. The default is 'STRUC'.
34. For RTYPE=ACPWR, PTYPE field can be utilized to identify the panel name. The default is 'total' which covers the whole interface between structural and fluid field.
35. For RTYPE=AFxxxx, PTYPE field can be utilized to specified the ID of Acoustic Field Point Mesh, AFPM.
36. For monitor point responses (RTYPE = STMONP1, STMOND1, MONPNT3 AEMONP1 or AEMOND1) the ATTA field specifies the components to be extracted. These can be any subset of the integers 1 through 6 that appear on the monitor quantity with the NAME provided in the PTYPE field. All of these responses can be invoked in a static aeroelastic (ANALYSIS=SAERO) subcase. STMOND1 and MONPNT3 can be invoked from a static (ANALYSIS=STAT) subcase. The responses are not available in a dynamic response or normal modes subcase. The response types have the following meaning:
 - a. STMONP1 – A structural MONPNT1
 - b. STMOND1 – A structural MONDSP1
 - c. MONPNT3 – A MONPNT3
 - d. AEMONP1 – An aerodynamic MONPNT1
 - e. AEMOND1 – An aerodynamic MONDSP1

For all but the STMONP1, the response is the elastic monitor point value. For the STMONP1, it is the elastic "minus" inertial "plus" elastic applied load value.

37. For RTYPE-FRMASS, if the PID in the ATTi field is not blank, the PTYPE field must specify the type of property the PID references.
38. RTYPE=DIVERG is for subcases with ANALYSIS=DIVERG (aeroelastic static divergence). Only a single Mach number can be specified for this RTYPE.
39. For RTYPE = FATIGUE (pseudo-static fatigue), PTYPE must be set to ELEM or PSOLID, PSHELL, PBAR, PBEAM, or PWELD. For RTYPE = FRFTG (frequency response fatigue), PTYPE must be set to ELEM or PSOLID or PSHELL. ATTB is the FID of a FATIGUE case control and must be the same FID for all RTYPES of type FATIGUE or FRFTG. RTYPES of type FATIGUE and FRFTG cannot be mixed; either all are RTYPE=FATIGUE or all are RTYPE=FRFTG.
40. If a FATIGUE case control references a SET ID, then ATTB must reference one of the IDs referenced by the SET. Design optimization using fatigue responses is currently limited to one fatigue analysis, that is, one loading sequence. Multiple fatigue responses referencing different loading sequences, i.e., FATIGUE case control IDs, will result in an error.
41. For the ERP response, the PTYPE field must be ERPPNL.
42. If the ATT1 field is blank for the ERP response, all panels are invoked.
43. Fatigue Item Codes can be found in [Fatigue Item Codes, 1104](#). Item codes that are available as design responses are 4-9 and 12 for the element centroid or first node/layer/location and the comparable items for ensuing nodes/layers/locations. The use of NODA on the FTGPARM entry is not supported.
44. For the WMPID (weight as a function of material and property ID), the following conventions apply:
 - a. ATTA is required, ATTB and ATTi are optional.
 - b. ATTB specifies the superelement with ATTB=0 indicating the residual, >0, indicates superelement ATTB and ATTB = ALL or blank means all superelements)
 - c. The ATTi fields designate the property ID's of interest. If ATTi is specified, the associated property type (e.g., PSHELL) must be specified by the PTYPE input of field 5 on the DRESP1 entry.
 - d. If there is no ATTi, all material with MID=ATTa will be used to generate the response.
 - e. If there are multiple ATTi, a single scalar weight is calculated that is the sum of the all the properties
 - f. Materials that are supported with this entry are: MAT1,MAT2, MAT3, MAT8, MAT9 and MAT10
 - g. Properties that are supported are: PSHELL, PROD, PBEAM, PTUBE, PSHEAR, PBAR, PSOLID, PBEND,PCOMP, PCOMPG, PBARL, PBEAML, PBRSECT and PBMSECT.
45. Certain restrictions apply for RTYPE=WMPID
 - a. The density of the material referenced on the ATTA field cannot also be designed using a DVMRELx entry
 - b. The property referenced on the ATTi field cannot also be invoked by a TOMVAR entry.
 - c. The WMPID response is supported for topology optimization or for shape sizing optimization, but not for both topology and shape sizing optimization.

DRESP2**Design Sensitivity Equation Response Quantities**

Defines equation responses that are used in the design, either as constraints or as an objective.

Format:

1	2	3	4	5	6	7	8	9	10
DRESP2	ID	LABEL	EQID or FUNC	REGION	METHOD	C1	C2	C3	
	"DESVAR"	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	"DTABLE"	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						
	"DRESP1"	NR1	NR2	NR3	NR4	NR5	NR6	NR7	
		NR8	-etc.-						
	"DNODE"	G1	C1	G2	C2	G3	C3		
		G4	C4	etc.					
	"DVPREL1"	DPIP1	DPIP2	DPIP3	DPIP4	DPIP5	DPIP6	DPIP7	
		DPIP8	DPIP9	-etc.-					
	"DVCREL1"	DCIC1	DCIC2	DCIC3	DCIC4	DCIC5	DCIC6	DCIC7	
		DCIC8	DCIC9	-etc.-					
	"DVMREL1"	DMIM1	DMIM2	DMIM3	DMIM4	DMIM5	DMIM6	DMIM7	
		DMIM8	DMIM9	-etc.-					
	"DVPREL2"	DPI2P1	DPI2P2	DPI2P3	DPI2P4	DPI2P5	DPI2P6	DPI2P7	
		DPI2P8	DPI2P9	-etc.-					
	"DVCREL2"	DCI2C1	DCI2C2	DCI2C3	DCI2C4	DCI2C5	DCI2C6	DCI2C7	
		DCI2C8	DCI2C9	-etc.-					
	"DVMREL2"	DMI2M1	DMI2M2	DMI2M3	DMI2M4	DMI2M5	DMI2M6	DMI2M7	
		DMI2M8	DMI2M9	-etc.-					
	"DRESP2"	NRR1	NRR2	NRR3	NRR4	NRR5	NRR6	NRR7	
		NRR8	-etc.-						
	"DVLREL1"	DLIL1	DLIL2	DLIL3	DLIL4	DLIL5	DLIL6	DLIL7	
		DLIL8	-etc.-						

Example:

DRESP2	1	LBUCK	5	3					
	DESVAR	101	3	4	5	1	205	209	
		201							
	DTABLE	PI	YM	L					
	DRESP1	14	1	4	22	6	33	2	

	DNODE	14	1	4	1	22	3		
		2	1	43	1				
	DVPREL1	101	102						
	DVCREL1	201	202						
	DVMREL1	301							
	DVPREL2	401	402						
	DVCREL2	501							
	DVMREL2	601	602	603					
	DRESP2	50	51						

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
LABEL	User-defined label. (Character, no default)
EQID	DEQATN entry identification number. (Integer > 0)
FUNC	Function to be applied to the arguments. See Remark 8. (Character)
REGION	Region identifier for constraint screening. See Remark 5. (Integer > 0)
METHOD	When used with FUNC = BETA, METHOD = MIN indicates a minimization task while MAX indicates a maximization task. (Default = MIN)
	When used with FUNCT = MATCH, METHOD = LS indicated a least squares while METHOD = BETA indicated minimization of the maximum difference. (Default = LS)
Ci	Constants used when FUNC = BETA or FUNC = MATCH in combination with METHOD = BETA. See Remark 8. (Real; Defaults: C1 = 1.0., C2 = .005, and C3=10.0)
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)
“DRESP1”	Flag indicating DRESP1 entry identification numbers. (Character). See Remark 13.
NRk	DRESP1 entry identification number. (Integer > 0)
“DNODE”	Flag indicating grid point and component identification numbers. (Character)
Gm	Identification number for any grid point in the model. (Integer > 0)
Cm	Component number of grid point Gm. ($1 \leq \text{Integer} \leq 3$)
“DVPREL1”	Flag indicating DVPREL1 entry identification number. (Character)
DPIPi	DVPREL1 entry identification number. (Integer > 0)

Descriptor	Meaning
“DVCREL1”	Flag indicating DVCREL1 entry identification number. (Character)
DCICi	DVCREL1 entry identification number. (Integer > 0)
“DVMREL1”	Flag indicating DVREL1 entry identification number. (Character)
DMIMi	DVMREL1 entry identification number. (Integer > 0)
“DVPREL2”	Flag indicating DVPREL2 entry identification number. (Character)
DPI2Pi	DVPREL2 entry identification number. (Integer > 0)
“DVCREL2”	Flag indicating DVCREL2 entry identification number. (Character)
DCI2Ci	DVCREL2 entry identification number. (Integer > 0)
“DVMREL2”	Flag indicating DVMREL2 entry identification number. (Character)
DMI2Mi	DVMREL2 entry identification number. (Integer > 0)
“DRESP2”	Flag indicating other DRESP2 entry identification number. (Character). See Remark 13.
NRRk	DRESP2 entry identification number. (Integer > 0)
“DVLREL”	Flag indicating DVLREL1 identification numbers. (Character)
DLILi	DVLREL1 entry identification number. (Integer > 0)

Remarks:

1. DRESP2 entries may only reference DESVAR, DTABLE, DRESP1, DNODE, DVPREL1, DVCREL1, DVMREL1, DVPREL2, DVCREL2, DVMREL2 and DVLREL1 entries. They may also reference other DRESP2 entries. However, a DRESP2 entry cannot reference itself directly or recursively.
2. a) If the referenced DRESP1 entries span subcases, the DRSPAN Case Control command is required to identify DRESP1 IDs for each subcase. DRESP2 entries that span subcases must be invoked above the subcase level by DESGLB on DESOBJ commands.
b) Referenced DRESP entries that span superelements are supported automatically.
c) Referenced DRESP2 entries cannot span subcases or superelements.
3. DRESP2 entries must have unique identification numbers with respect to DRESP1 entries.
4. The “DESVAR”, “DTABLE”, “DRESP1”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1”, “DVPREL2”, “DVCREL2”, “DVMREL2”, “DRESP2” and DVLREL1 flags in field 2 must appear in the order given above. Any of these words, along with the identification numbers associated with them, may be omitted if they are not involved in this DRESP2 relationship. However, at least one of these ten types of arguments must exist.
5. The REGION field follows the same rules as for the DRESP1 entries. DRESP1 and DRESP2 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP2 entry in the same region.

6. The variables identified by DVIDi, LABLj, NRk, the Gm, CMPM pairs, DPPIP, DCICm, DMIMn, DPI2Po, DCI2Cp, DMI2Mq, and NRRu are assigned (in that order) to the variable names (x1, x2, x3, etc.) specified in the left-hand side of the first equation on the DEQATN entry referenced by EQID. In the example below,

DESVARs 101 and 3 are assigned to arguments A and B.
 DTABLEs PI and YM are assigned to arguments C and D.
 Grid 14, Component 1 is assigned to argument R.

DRESP2	1	LBUCK	5	3					
	DESVAR	101	3						
	DTABLE	PI	YM						
	DNODE	14	1						
DEQATN	5	$F1(A, B, C, D, R)=A+B*C-(D^{**}3+10.0)+\sin(C*R)$							

7. (Gm, Cm) can refer to any grid component and is no longer limited to a designed grid component.
 8. The FUNC attributes can be used in place of the EQID and supports the functions shown in the following table:

Function	Description
SUM	Sum of the arguments
SFMAX	Maximization of stiffness and frequency. See remark 14.
AVG	Average of the arguments
SSQ	Sum of the squares of the arguments
RSS	Square root of the sum of the squares of the arguments
MAX	The maximum value of the argument list
MIN	The minimum value of the argument list
BETA	Minimize the maximum response. See Remark 10.
MATCH	Match analysis results with user specified values. See Remark 11.

When EQID has character input, the DEQATN entry is no longer needed. The functions are applied to all arguments on the DRESP2 regardless of the type. See Remark 20. of the DRESP1 entry for an explanation of these functions.

9. The number of arguments of a DEQATN can be more than the number of values defined on the DRESP2 if the DRESP1s referenced have RTYPE with 'FR' or 'PSD' prefix. Arguments are still positional. The extra arguments in the DEQATN must appear at the end of the argument list. The discrepancy is resolved internally with the forcing frequency(ies) associated with DRESP1s. An example is shown as follows:

DRESP1	10	FDISP1	FRDISP			1	10.	1001	
DRESP1	20	FDISP2	FRDISP			1	20.	1001	

DRESP2	30	AVGFD	100					
	DRESP1	10	20					
DEQATN	100	AVG(D1,D2,F1,F2) = (D1/F1+D2/F2)*0.5						

In the above example, the DEQATN has two more additional terms than have been defined on the DRESP2. The first additional term is the forcing frequency (in hertz) of the first DRESP1 ID on the DRESP2. The second additional term is the forcing frequency of second DRESP1 ID in the list. When all DRESP1s involved have the same frequency, the user is not required to name all the additional terms in the argument list of DEQATN.

10. FUNC = BETA facilitates a design task where the objective is to minimize the maximum response. Only DRESP1 entries can be invoked by DRESP2 and the DRESP's cannot span subcases. The BETA function creates the following design task:

$$\text{Minimize } \phi = C_1 X_\beta$$

$$\text{Subject to } g = \frac{\gamma X_\beta - r_j}{C_3} \geq 0$$

where γ is determined from

$$C_2 = (r_{j\max} - \gamma X_\beta) / C_3$$

User input parameters C_1 , C_2 , C_3 therefore have the following meaning:

C_1 (Default = 1.0) weights the spawned design variable, X_β , to create the objective. Since X_β starts at 1.0, C_1 is the initial objective.

C_2 sets the initial value of the maximum constraint created by this process. The default values of 0.005 is equal to DOPTPRM parameter GMAX.

C_3 (Default = 10.0) is an offset value to avoid dividing by zero when creating constraints.

11. FUNC = MATCH creates a response from the difference between analysis results, r_j , that are associated with DRESP1s and target values, r_j^T , that are input using DTABLE data. Only DRESP1 entries and DTABLE entries can be invoked by the DRESP2 entry.

When METHOD = LS, a least square minimization is performed where the response is

$$r_2 = \sum_{j=1}^m \left(\frac{r_j - r_j^T}{|r_j^T|} \right)^2$$

When METHOD = BETA, the design task becomes one of minimizing an objective that is the maximum normalized difference between the analysis and target values

$$\frac{r_j - r_j^T}{|r_j^T|}$$

in the same manner as outlined in Remark 10.

12. With FUNC=MATCH, if the DTABLE LABLj invokes an integer VALUi on the DTABLE entry, the integer points to a TABLEDi entry that provides tabular input as a function of frequency or time. This is used in conjunction with a single DRESP1 NRk that provides a response across a range of frequencies or times (i.e., RTYPE=FRxxxx, PSDxxxx, ACxxxx or Txxxx). When the TABLEDi option is invoked, the response can only be constrained, it cannot be the objective. This further implies that FUNC=MATCH, METHOD=BETA is not supported with TABLEDi.
13. If the mathematical function from a DRESP1 that has character input in the ATTB field (see remark 20 of the DRESP1) is to be used in the DRESP2 evaluation, the DRESP1 must be referenced under the “DRESP2” flag, not the “DRESP1” flag. If the “DRESP1” flag is used in this situation, it will result in a DRESP2 being evaluated for each of the individual values that contribute to the mathematical function.
14. The functions are applied to all DRESP1 arguments on the DRESP2: Maximization of stiffness and frequency FUNC=SFMAX, that refers multiple DRESP1 IDs where the fundamental frequency ID must be placed at last one. In addition, the ANALYSIS = MODES must be placed at the last subcase in Case Control Section. This is typically used in topology, topometry and topography optimization.

DRESP3

Design Sensitivity Response using user-supplied routine(s)

Defines an external response using user-supplied routine(s).

Format:

1	2	3	4	5	6	7	8	9	10
DRESP3	ID	LABEL	GROUP	TYPE	REGION				
	"DESVAR"	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	etc.						
	"DTABLE"	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	etc.						
	"DRESP1"	NR1	NR2	NR3	NR4	NR5	NR6	NR7	
		NR8	etc.						
	"DNODE"	G1	C1	G2	C2	G3	C3		
		G4	C4	etc.					
	"DVPREL1"	DPIP1	DPIP2	DPIP3	DPIP4	DPIP5	DPIP6	DPIP7	
		DPIP8	DPIP9	etc.					
	"DVCREL1"	DCIC1	DCIC2	DCIC3	DCIC4	DCIC5	DCIC6	DCIC7	
		DCIC8	DCIC9	-etc.-					
	"DVMREL1"	DMIM1	DMIM2	DMIM3	DMIM4	DMIM5	DMIM6	DMIM7	
		DMIM8	DMIM9	-etc.-					
	"DVPREL2"	DPI2P1	DPI2P2	DPI2P3	DPI2P4	DPI2P5	DPI2P6	DPI2P7	
		DPI2P8	DPI2P9	-etc.-					
	'DCREL2"	DCI2C1	DCI2C2	DCI2C3	DCI2C4	DCI2C5	DCI2C6	DCI2C7	
		DCI2C8	DCI2C9	-etc.-					
	"DVMREL2"	DMI2M1	DMI2M2	DMI2M3	DMI2M4	DMI2M5	DMI2M6	DMI2M7	
		DMI2M8	DMI2M9	-etc.-					
	"DRESP2"	NRR1	NRR2	NRR3	NRR4	NRR5	NRR6	NRR7	
		NRR8	-etc.-						
	DVLREL1	DLIL1	DLIL2	DLIL3	DLIL4	DLIL5	DLIL6	DLIL7	
		DLIL8	DLIL9	-etc.-					
	"USRDATA"	String							
		-etc.-							

Example:

DRESP3	1	LBUCK	TAILWNG	BUCK					
	DESVAR	101	3	4	5	1	205	209	

		201							
	DTABLE	PI	YM	L					
	DRESP1	14	1	4	22	6	33	2	
	DNODE	14	1	4	1	22	3		
		2	1	43	1				
	DVPREL1	101	102						
	DVCREL1	201	202						
	DVMREL1	301							
	DVPREL2	401	402						
	DVCREL2	501							
	DVMREL2	601	602	603					
	DRESP2	50	51						
	USRDATA	Constants: 12345.6 789.0 99.							

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
LABEL	User-defined label. (Character, no default)
GROUP	Group name the external response type belongs to (Character). See Remark 2.
TYPE	External response type (Character). See Remark 3.
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)
“DRESP1”	Flag indicating DRESP1 entry identification numbers. (Character)
NRk	DRESP1 entry identification number. (Integer > 0)
“DNODE”	Flag signifying that the following fields are grid points.
Gm	Grid point identification number. (Integer > 0)
Cm	Degree-of-freedom number of grid point Gm. ($1 \leq \text{Integer} \leq 3$)
“DVPREL1”	Flag indicating DVPREL1 entry identification number. (Character)
DPIPi	DVPREL1 entry identification number. (Integer > 0)
“DVCREL1”	Flag indicating DVCREL1 entry identification number. (Character)
DCICi	DVCREL1 entry identification number. (Integer > 0)
“DVMREL1”	Flag indicating DVMREL1 entry identification number. (Character)
DMIMi	DVMREL1 entry identification number. (Integer > 0)

Descriptor	Meaning
“DVPREL2”	Flag indicating DVPREL2 entry identification number. (Character)
DPI2Pi	DVPREL2 entry identification number. (Integer > 0)
“DVCREL2”	Flag indicating DVCREL2 entry identification number. (Character)
DCI2Ci	DVCREL2 entry identification number. (Integer > 0)
“DVMREL2”	Flag indicating DVMREL2 entry identification number. (Character)
DMI2Mi	DVMREL2 entry identification number (Integer > 0)
“DRESP2”	Flag indicating other DRESP2 entry identification number. (Character)
NRRk	DRESP2 entry identification number. (Integer > 0)
“DVLREL1”	Flag indicating DVLREL1 identification number (character)
DLILi	DVLREL1 entry identification number (integer)
“USRDATA”	Flag indicating user input data (Character). See Remark 8.

Remarks:

1. DRESP3 entries may reference DESVAR, DTABLE, DRESP1, DNODE, DVPREL1, DVCREL1, DVMREL1, DVPREL2, DVCREL2, DVMREL2, DRESP2 and DVLREL1 entries. However, a DRESP3 entry cannot reference another DRESP3 entry.
2. The group name must be referenced by an FMS CONNECT entry.
3. Multiple types of external responses can be defined in one group. Each type name identifies a specific external response evaluated in the user-supplied routines. See [Building and Using the Sample Programs MSC Nastran Utilities Guide](#) for a discussion of how to incorporate external responses.
4. a) Referenced DRESP2 entries cannot span subcases or superelements.
b) If referenced DRESP1 entries span subcases, the DRSPAN Case Control command is required to identify the DRESP1 IDs for each subcase. DRESP3 entries that span subcases must be invoked above the subcase level by DESGLB or DESOBJ commands.
c) Referenced DRESP1 entries that span superelements are supported automatically.
5. DRESP3 entries must have unique identification numbers with respect to DRESP2 and DRESP1 entries.
6. The “DESVAR”, “DTABLE”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1”, “DVPREL2”, “DVCREL2”, “DVMREL2”, “DRESP2”, “DVLREL1” and “USRDATA” keywords on the continuation entries must appear in the order given above. Any of these words, along with the subsequent data associated with them, may be omitted if they are not involved in this DRESP3 relationship. However, at least one of these types of arguments must exist.
7. The REGION field follows the same rules as for the DRESP1 entries. DRESP1 and DRESP3 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP3 entry in the same region.

8. The data in the USRDATA field is character string based. It provides a convenient way to pass constants to the external response server routines. The maximum number of characters allowed is 32000.

DSCREEN**Design Constraint Screening Data**

Defines screening data for constraint deletion.

Format:

1	2	3	4	5	6	7	8	9	10
DSCREEN	RTYPE	TRS	NSTR						

Example:

DSCREEN	STRESS	-0.7	2						
---------	--------	------	---	--	--	--	--	--	--

Descriptor	Meaning
RTYPE	Response type for which the screening criteria apply. See Remark 3. (Character)
TRS	Truncation threshold. (Real; Default = -0.5)
NSTR	Maximum number of constraints to be retained per region per load case. See Remark 3. (Integer > 0; Default = 20)

Remarks:

1. Grid responses associated with one particular load case are grouped by the specification of DRESP1 entries. From each group, a maximum of NSTR constraints are retained per load case.
2. Element responses are grouped by the property; i.e., all element responses for one particular load case belonging to the set of PIDs specified under ATTi on a DRESPi entry are regarded as belonging to the same region. In superelement sensitivity analysis, if the property (PID) is defined in more than one superelement, then separate regions are defined. A particular stress constraint specification may be applied to many elements in a region generating many stress constraints, but only up to NSTR constraints per load case will be retained.
3. For aeroelastic responses, that is RTYPE = "TRIM", "STABDER", and "FLUTTER", the NSTR limit is applied to all DRESP1 IDs that are the same RTYPE and have the same REGION specified.
4. For responses that are not related to grids or elements, that is RTYPE = "WEIGHT", "VOLUME", "EIGN", "FREQ", "LAMA", CEIG", FRMASS, COMP, and TOTSE", NSTR is not used. TRS is still applicable.
5. The RTYPE field is set to EQUA if constraints that are associated with DRESP2 entries are to be screened. The RTYPE field is set to DRESP3 if constraints that are associated with DRESP3 entries are to be screened. If the REGION field on the DRESP2 or DRESP3 is blank, one region is established for each DRESP2/DRESP3 entry.
6. If a certain type of constraint exists but no corresponding DSCREEN entry is specified, all the screening criteria used for this type of constraint will be furnished by the default values.
7. Constraints can be retained only if they are greater than TRS. See the Remarks under the [DCONSTR](#), [1706](#) entry for a definition of constraint value.
8. Constraint screening is applied to each superelement.

DTABLE**Table Constants**

Defines a table of real constants that are used in equations (see DEQATN entry).

Format:

1	2	3	4	5	6	7	8	9	10
DTABLE	LABL1	VALU1	LABL2	VALU2	LABL3	VALU3	LABL4	VALU4	
	LABL5	VALU5	LABL6	VALU6	LABL7	VALU7	LABL8	VALU8	
		-etc.-							

Example:

DTABLE	PI	3.142	H	10.1	E	1.0E6			
	G	5.5E5	B	100.					

Descriptor	Meaning
LABLi	Label for the constant. (Character)
VALUi	Value of the constant or TABLEDx ID. (Real or Integer, respectively)

Remarks:

1. Multiple DTABLE entries may be specified in the Bulk Data Section.
2. LABLi are referenced by the LABj on the DRESP2, DRESP3, DVCREL2, DVMREL2, or DVPREL2 entries.
3. Trailing blank fields are permitted at the end of each line of LABLi/VALUi pairs, but intermediate blanks are not. (See the example above for permitted trailing blanks.)
4. For PART SE, if LABLi is referenced on SEDRSP2 and/or SEDRSP3, DTABLE entries must be placed in a PART SE where companion design model entries, such as DESVAR, DRESP1 and etc, are available.
5. If the VALUi is an integer, this points to the ID of a TABLEDx Bulk Data entry that lists the constants as a function of frequency or time. See Remark 12. on the DRESP2 entry.
6. LABLi must be unique across all DTABLE and DTABLE2 entries.

DTABLE2**Table Constants**

Defines real constants from a field of property, material or connections bulk data entries which then can be invoked by a DVxREL2, DRESP2, or DRESP3 entry.

Format:

1	2	3	4	5	6	7	8	9	10
DTABLE2	LABL1	PNAME1	PID1	FNAME1	LABL2	PNAME2	PID2	FNAME2	
	LABL3	PNAME3	PID3	FNAME3					

Example:

DTABLE2	PTHK10	PSHELLL	10	T	MATIE	MT1	38	E	
	CBARX1	CBAR	3888	X1					

Descriptor	Meaning
LABLi	Label for the constant. (Character)
PNAMEi	Property, material or connection bulk data entry name. (Character)
PIDI	ID of PNAMEi entry. (Integer > 0)
FNAMEi	Field name of PNAMEi. (Character)

Remarks:

1. LABLi on DTABLE2 and DTABLE must be unique.
2. LABLi on DTABLE2 can be referenced under DTABLE flag of DVxREL2 (where x=P, M or C)/DRESP2/DRESP3.
3. Values for the FNAMEi field of the PNAMEi Bulk Data entry with the ID of PIDI are taken from analysis model before updating of analysis values with the designed value. If the updated value is desired, use the DVxREL2 flag on DRESP2 or DRESP3 entries instead.
4. FNAMEi must be the same as the character string that appears on the PNAMEi Bulk Data entry.
5. DTABLE2 must be utilized along with 'NASTRAN SYSTEM(444)=1' in input file or 'sys444=1' during job submittal.

DTI**Direct Table Input**

Defines table data blocks.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	NAME	"0"	T1	T2	T3	T4	T5	T6	
	V01	V02	-etc.-						
DTI	NAME	IREC	V1	V2	V3	V4	V5	V6	
	V7	V8	V9	V10	-etc.-	"ENDREC"			

Example: (The first logical entry is the header entry.)

DTI	XXX	0	3	4	4096		1	0	
	1.2	2.3							
DTI	XXX	1	2.0	-6	ABC	6.000	-1	2	
	4	-6.2	2.9	1	DEF	-1	ENDREC		

Descriptor	Meaning
NAME	Any character string that will be used in the DMAP sequence to reference the data block. See Remark 1. (Character; the first character must be alphabetic.)
Ti	Trailer values. (Integer ≥ 0 ; Default = 32767)
IREC	Record number. (Integer > 1)
V0i, Vi	Value. (Integer, Real, Character or blank)
"ENDREC"	Flags the end of the string of values (V0i or Vi) that constitute record IREC. (Character)

Remarks:

1. The user defines the data block and therefore must write a DMAP (or ALTER a solution sequence), which includes the DTIN modules, in order to use the DTI feature. See the [MSC Nastran DMAP Programmer's Guide](#). All of the rules governing the use of data blocks in DMAP sequences apply.
2. All fields following ENDREC must be blank.
3. The entry using IREC = 0 is called the header entry and is an optional entry. The values T1 through T6 go to a special record called the trailer. Other values on the optional continuation go to the header record. If the header entry or the trailer is not specified, T1 through T6 = 32767. On this entry, "ENDREC" may be used only if there is at least one continuation.
4. In addition to the optional header entry, there must be one logical entry for each record in the table. Null records require no entries.

5. “ENDREC” is used to input blank values at the end of a record. If “ENDREC” is not specified, the string for a record ends with the last nonblank field.
6. The maximum number of DMI and DTI data blocks is 1000.
7. If Ti is not an integer, a machine-dependent error will be issued that may be difficult to interpret.
8. If Modules are present then this entry may only be specified in the main Bulk Data section.

DTI,ESTDATA**Superelement Estimation Data Overrides**

Provides override data for time and space estimation for superelement processing operations.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	ESTDATA	"0"							
	kd1	vd1	kd2	vd2	-etc.-				

The next entries are repeated for any superelement for which estimate data overrides are desired. IREC must be incremented by 1.

DTI	ESTDATA	IREC	SEFLAG	SEID	k1	v1	k2	v2	
	k3	v3	-etc.-						

Example:

DTI	ESTDATA	0							
	NOMASS	-1							
DTI	ESTDATA	1	SE	10	C1	5.5	C3	4.5	
	C7	7.3							

Descriptor	Meaning
kdi	Keyword for estimation parameter. (Character from Table 9-9 .)
vdi	Value assigned to the estimation parameter kdi. (The type given in Table 9-9 .)
IREC	Record number beginning with 1. (Integer > 0)
SEFLAG	SEFLAG = "SE" or "SEID" indicates the next field containing a superelement identification number. (Character)
SEID	Superelement identification number. (Integer > 0)
ki	Keyword for override of estimation parameter for indicated superelement. (Character from Table 9-9 .)
vi	Value for keyword ki. (Type depends on ki as shown in the Table 9-9 .)

Table 9-9 DTI,ESTDATA Input Parameter Descriptions

Input Parameters				Meaning and Comments
Keyword	Type	Default Value	Math Symbol	
CRMS*	Real	-1.0	C	Number of active columns in $[K_{oo}]$.
FCRMS*	Real	0.10		If FCRMS ≤ 0.0 , FCRMS is used (c/o).
C1	Real	6.0	c_1	Average number of degrees-of-freedom per grid point in o-set.
C3	Real	8.0	c_3	Average number of connections per grid point.
C4	Real	0.15	c_4	I/O time (seconds) per block transferred.
C5	Real	6.0	c_5	Average number of effective degrees-of-freedom per grid point in a-set.
C6	Real	1.2	c_6	Total CPU factor.
C7	Real	6.0	c_7	Number of equivalent KGG data blocks for space estimation.
WF	Real	-1.0	W	If WF ≤ 0.0 then use available working storage in units of single-precision words.
NOMASS	Integer	1		If NOMASS $\neq 1$ then exclude mass terms from estimates.
TSEX	Real	0.5 (min)		Threshold limit for CPU.
SSEX	Real	50.0 (blocks)		Threshold limit for space.
TWALLX	Real	5.0 (min)		Threshold limit for wall time.
BUFSIZ	Integer	Machine Buffsize	B	Buffsize. See The NASTRAN Statement (Optional) (p. 14) in the <i>MSC Nastran Reference Guide</i> .
ML	Real	Machine Loop Time	M	Arithmetic time for the multiply/add loop. See the <i>SOL 700 Explicit Nonlinear User's Guide</i> .
CONIO	Integer	Machine I/O ratio		I/O count/CPU equivalence
PREC	Integer	1 or 2		Machine Word Length (1 = long, 2 = short). See The NASTRAN Statement (Optional) (p. 14) in the <i>MSC Nastran Reference Guide</i> .
NLOADS	Integer	1	N_L	Number of loading conditions
SETYPE	Character	"T"		Superelement type (T = Tip)
CMAX	Real	-1.0	C_{\max}	Maximum bandwidth

Parameters Obtained from SEMAP

NGI	Number of interior grid points.
NPE	Number of exterior grid points.
NS	Number of scalar points
NE	Number of elements.

Derived Parameters

$O = C1 + NGI$	Size of o-set.
$A = C5(NPE - NS) + NS$	Size of a-set.
$T = BUFSIZE/PREC$	Number of matrix terms in a buffer.

Estimation Equations

For each superelement, estimates of CPU time and disk space are made using the following equations.

Table 9-10 Equations Used for CPU Time and Disk Space Estimate

Printout Symbol	Math Symbol	Equations
TD	T_1	$T_1 = 1/2 \cdot M \cdot O \cdot C^2$
TFBS	T_2	$T_2 = 2 \cdot M \cdot C \cdot O \cdot a$
TMAA	T_3	$T_3 = M \cdot O \cdot a^2$ (set to 0.0 if NOMASS ≠ +1)
TSE	T_{SE}	$T_{SE} = C_6(T_1 + T_2 + T_3)$
SLOO	S_1	$S_1 = O \cdot C \cdot \frac{PREC}{WF}$
SGO	S_2	$S_2 = O \cdot a \cdot \frac{PREC}{B}$
SKGG	S_3	$S_3 = 36(NG_i + NG_e - NS)(C_3 + 1.0)\left(\frac{PREC}{B}\right)$
SSE	S_{SE}	$S_{SE} = S_1 + S_2 + C_7 \cdot S_3$
PASSES	p	FBS passes = $p = a \cdot O \cdot \frac{PREC}{WF}$
BKSTRN	BT	Blocks Transferred = $BT = 2 \cdot p \cdot S_1 + S_2 + p \cdot S_2$. (Last term omitted if NOMASS ≠ +1)
TWALL	T_W	Wall Time = $T_W = T_{SE} + C_4 \cdot BT$

Remarks:

1. In the superelement solution sequences, this data is stored automatically.
2. The header record continuation entries are optional if no global override data is to be specified. In this case, the complete header entry is optional.
 - Active column data can come from one of several places. The value for CRMS is determined as follows:

- RMS from the entry when IREC > 0 and field 4 is “SE”.
 - RMS from entries with IREC = 0.
 - Computed bandwidth when PARAM,OLDSEQ is specified.
 - If FCRMS is specified when IREC > 0 and field 4 is “SE”, then CRMS = FCRMS · O .
 - If FCRMS is specified when IREC = 0, then CRMS = FCRMS · O .
 - CRMS = 0.1 · O .
3. If CMAX is not specified, then it is defaulted to CRMS.
 4. In the example above, mass terms are excluded for all superelements and new values are given for parameters C1, C3, and C7 for Superelement 10 only.
 5. The estimates for TSEX, SSEX, and TWALLX are not printed unless at least one estimate exceeds the threshold.

DTI,INDTA**Stress, Strain and/or Force Sort/Filter Item Code Override**

Specifies or overrides default item codes for the sorting and filtering of element stresses, strains, and forces.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	INDTA	"0"							

To specify/override items for a sort of stress quantities:

DTI	INDTA	"1"	B1	C1	B2	C2	"ENDREC"		
-----	-------	-----	----	----	----	----	----------	--	--

To specify/override items for a sort of force quantities:

DTI	INDTA	"2"	B1	C1	B2	C2	"ENDREC"		
-----	-------	-----	----	----	----	----	----------	--	--

Examples:

DTI	INDTA	0							
-----	-------	---	--	--	--	--	--	--	--

To specify/override items for a sort of stress quantities:

DTI	INDTA	1	64	18	75	18	ENDREC		
-----	-------	---	----	----	----	----	--------	--	--

To specify/override items for a sort of force quantities:

DTI	INDTA	2	34	2	2	4	ENDREC		
-----	-------	---	----	---	---	---	--------	--	--

Descriptor	Meaning
Bi	Element type identification number. See the table in Item Codes, 1045 for allowable values. (Integer > 0)
Ci	Item code identification number for the stress, strain, or force quantity on which the sort or filter is to be performed. See the table in the Item Codes, 1045 for allowable values. (Integer)

Remarks:

1. This table is recognized only in SOLs 101, 103, 105, 106, 108, 109, 111, 112, 114, 115, 144, 153, and for stress quantities only. One or more of the user parameters S1, S1G, or S1M must be specified with a value greater than or equal to zero in order to request sorting and/or filtering. See user parameter S1 in [Parameters, 783](#). In order to sort force or strain quantities, a DMAP Alter is required.
2. If the Ci value is -1, the element type will be suppressed on the output file. An example of this feature could be as follows: If an element type is to be sorted on two different values and output twice, this can be accomplished by two calls to the STRSORT module with two unique DTI tables. However, other element types will be printed twice. This additional print can be suppressed by setting their sort codes to -1.

3. [Table 13](#) lists the elements currently that are sortable. In addition, the element type identification number, the default stress output quantity, and the associated stress code identification numbers are provided. If this entry is not specified, then the stresses are sorted based on the default quantity given in [Table 13](#).

The following should be noted:

- a. The element type identification number is used internally by the program to differentiate element types.
- b. The stress code identification number is merely the word number in the standard printed output for the stress quantity of interest. For example, the thirteenth word of stress output for the CHEXA element is the octahedral shear stress. For this element type, the element identification number and the grid point ID each count as a separate word. Stress codes for the elements are tabulated in [Item Codes, 1045](#).
- c. By default, stress sorting for the membrane and plate elements will be performed on the Hencky-von Mises stress. For maximum shear stress, the STRESS (MAXS) Case Control command should be specified.

Table 13 Sortable Elements

Element	Element Type ID Number	Default Stress Output Quantity and Identification Number	
		Quantity	Stress Code ID Number
CBAR	34	Maximum stress at end B	14
CBEAM	2	Maximum stress at end B	108
CBEND	69	Maximum stress at end B	20
CONROD	10	Axial stress	2
CELAS1	11	Stress	2
CELAS2	12	Stress	2
CELAS3	13	Stress	2
CHEXA	67	Hencky-von Mises or Octahedral stress	13
CQUAD4	33	Maximum shear or Hencky-von Mises stress at Z_2	17
CQUAD4*	144	Maximum shear or Hencky-von Mises stress at Z_2	19
CQUAD8	64	Maximum shear or Hencky-von Mises stress at Z_2	19
CQUADR	82	Maximum shear or Hencky-von Mises stress at Z_2	19
CPENTA	68	Octahedral stress	13
CROD	1	Axial stress	2
CSHEAR	4	No default	---
CTETRA	39	No default	---
CTRIA3	74	Maximum shear or Hencky-von Mises stress at Z_2	17
CTRIA6	75	Maximum shear or Hencky-von Mises stress at Z_2	19
CTRIAR	70	Maximum shear or Hencky-von Mises stress at Z_2	19
CTRIAX6	53	No default	---
CTUBE	3	Axial stress	2

*CORNER output

DTI,SETREE**Superelement Tree Definition**

Defines a superelement tree that determines the superelement processing order.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	SETREE	"1"	SEUP1	SEDOWN1	SEUP2	SEDOWN2	SEUP3	SEDOWN3	
	SEUP4	SEDOWN4	SEUP5	SEDOWN5	-etc.-				

Example:

DTI	SETREE	1	1	14	2	14	3	14	
	4	14	14	0					

Descriptor	Meaning
SEUPI	Identification number of the superelement upstream from SEDOWNi. (Integer > 0)
SEDOWNi	Identification number of the superelement into which SEUPI is assembled. (Integer ≥ 0)

Remarks:

1. SETREE entries or the DTI,SETREE entry are required for multi-level superelement configurations.
2. If an DTI,SETREE entry is provided, then SETREE entries are not required.
3. If both SETREE entries and a DTI,SETREE entry exist, then the DTI,SETREE entry will be ignored.
4. If a superelement is not referenced on the DTI,SETREE or SETREE entry, then the manner in which it is handled depends on the type of that superelement. If it is a PART superelement, then the residual will be regarded as its downstream superelement and the undefined superelement will therefore be placed immediately above the residual in the tree. If it is a Main Bulk Data superelement, then it will also be handled like an undefined PART superelement as above *if all of its exterior points belong to the residual*. However, if one or more of its exterior points do not belong to the residual, then the program will terminate with a user fatal error complaining that one or more of the superelements are not in the same path.
5. If this entry is not present, the superelement tree and the processing order are determined automatically.
6. A superelement identification may appear only once in a SEUPI field.
7. On restart, if a superelement identification does not appear in a SEUPI field, its matrices will not be assembled, even though they may be present in the database.
8. See the *MSC Nastran Superelements and Modules User's Guide* for a description of user-designated trees.

9. This entry is stored in the database automatically. Once stored, the Bulk Data entry may be removed from the input file.
10. In the example above, the following superelement tree is defined:

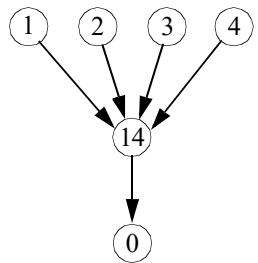


Figure 9-89 Sample Superelement Tree

DTI,SPECSEL**Response Spectra Input Correlation Table**

Correlates spectra lines specified on TABLED1 entries with damping values.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	SPECSEL	RECNO		TYPE	TID1	DAMP1	TID2	DAMP2	
	TID3	DAMP3	TID4	DAMP4	TID5	DAMP5	-etc.-		

Example:

DTI	SPECSEL	1		A	1	.02	2	.04	
		3	.06						
DTI	SPECSEL	3		V	4	.01			

Descriptor	Meaning
RECNO	Spectrum number. (Integer > 0)
TYPE	Type of spectrum. (Character: "A" for acceleration, "V" for velocity, or "D" for displacement.)
TID _i	TABLED1 entry identification number. (Integer > 0)
DAMP _i	Damping value assigned to TID _i . (Real)

Remarks:

1. The RECNO is the number of the spectrum defined by this entry. It is referenced on DLOAD Bulk Data entries.
2. The TID_i, DAMP_i pairs list the TABLED1 entry, which defines a line of the spectrum and the damping value assigned to it. The damping value is in the units of fraction of critical damping.
3. This entry is placed in the database automatically. Once stored, the Bulk Data entry may be removed from the input file.

DTI,SPSEL

Response Spectra Generation Correlation Table

Correlates output requests with frequency and damping ranges.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	SPSEL	RECNO	DAMPL	FREQL	G1	G2	G3	G4	
	G5	G6	G7	-etc.-					

Example:

DTI	SPSEL	1	2	1	11	12			
DTI	SPSEL	2	4	3	1	7	11	12	
	13	14							

Descriptor	Meaning
DAMPL	Identification number of the FREQ, FREQ1, or FREQ2 Bulk Data entry that specifies the list of damping values. (Integer > 0)
FREQL	Identification number of the FREQi Bulk Data entry that specifies the list of frequencies. (Integer > 0)
Gi	Grid point number where response spectra will be calculated. (Integer > 0)
RECNO	Record number of spectra to be generated. (Sequential integer beginning with 1.)

Remarks:

1. This table is used in SOLs 109 and 112.
2. Damping values are in the units of fraction of critical damping.
3. Output of response spectra requires the use of the XY PLOT...SPECTRA(RECNO)/Gi... command, where Gi is restricted to the grid points listed on the (RECNO) record of this entry.
4. The SPSEL table is stored in the database automatically in SOLs 109 and 112. Once stored, the Bulk Data entry may be removed from the input file.
5. There must be case control output request for displacements and velocities of the points to be output. For example you could use case control commands:

DISP(PLOT) = ALL

VELO(PLOT) = ALL

DTI,UNITS

Unit Definitions

Defines units necessary for conversion during the analysis for the Nastran/ADAMS interface or a Nastran fatigue analysis.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	UNITS	1	MASS	FORCE	LENGTH	TIME	STRESS		

Example:

DTI	UNITS	1	KG	N	M	S	MPA		
-----	-------	---	----	---	---	---	-----	--	--

Remarks:

1. The DTI,UNITS Bulk Data entry is required for a ADAMSMNF FLEXBODY=YES run. See the [ADAMSMNF* \(Case\)](#), 222 case control entry. ADAMS is not a unitless code (as is Nastran). Units must be specified. A DTI Bulk Data entry provides 'UNITS' (a unique identifier) input as the above example illustrates. Once identified, the units will apply to all superelements in the model. Acceptable character input strings are listed in the table below. MASS, FORCE, LENGTH, and TIME are required for ADAMS interface.
2. MSC Nastran is a unitless code and it is the user's responsibility to ensure compatible units. During a fatigue analysis, stress is converted to SI units of MPa because the fatigue material property stress parameters as defined on the MATFTG entry are internally converted to standard SI units of MPa. The stresses from the analysis must match. Thus it is necessary for the user to use DTI,UNITS to define the stress units to ensure proper conversion. The default units are MPa and this entry is only necessary if stresses are not in MPa.

Mass	Force	Length	Time	Stress*
KG - kilogram	N - newton	KM - kilometer	H - hour	MPA - megapascal
LBM - pound-mass	LBF - pounds-force	M - meter	MIN - minute	PA - pascal
SLUG - slug	KGF - kilograms-force	CM - centimeter	S - second	PSI - pound per square inch
GRAM - gram	OZF - ounce-force	MM - millimeter	MS - millisecond	KSI - kilo pound per square inch
OZM - ounce-mass	DYNE - dyne	MI - mile	US - microsecond	PSF - pound per square foot

Mass	Force	Length	Time	Stress*
KLBM - kilo pound-mass (1000 lbm)	KN - kilonewton	FT - foot	NANOSEC - nanosecond	KSF - kilo pound per square foot
MGG - megagram	KLBF - kilo pound-force (1000 lbf)	IN - inch	D - day	DYNECM2 - dyne per square centimeter
SLINCH - 12 slugs	MN - millinewton	UM - micrometer		BAR - bar
UG - microgram	UN - micronewton	NM - nanometer		ATM - physical atmosphere
NG - nanogram	NN - nanonewton	ANG - angstrom		
USTON - US ton		YD - yard		
		MIL - milli-inch		
		UIN - micro-inch		

* For fatigue analysis of spot welds, only MPA, PA, PSI, KSI, PSF, and KSF are supported.

For random vibration fatigue analysis using SOL 108 or 111, only MPA, PA, PSI, and KSI are supported.

DVBSHAP**Design Variable to Boundary Shapes**

Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model.

Format:

1	2	3	4	5	6	7	8	9	10
DVBSHAP	DVID	AUXMOD	COL1	SF1	COL2	SF2	COL3	SF3	

Example:

DVBSHAP	4	1	1	1.6					
---------	---	---	---	-----	--	--	--	--	--

Descriptor	Meaning
DVID	Design variable identification number of a DESVAR entry. (Integer > 0)
AUXMOD	Auxiliary model identification number. (Integer > 0)
COLi	Load sequence identification number from AUXMODEL Case Control command. (Integer > 0)
SFi	Scaling factor for load sequence identification number. (Real; Default = 1.0)

Remarks:

1. Design variable DVID must be defined on a DESVAR entry.
2. Multiple references to the same DVID and/or COLi will result in the vector addition of the referenced boundary shape vectors.
3. Multiple DVBSHAP entries may be specified.

DVCREL1**Design Variable to Connectivity Property Relation**

Defines the relation between a connectivity property and design variables.

Format:

1	2	3	4	5	6	7	8	9	10
DVCREL1	ID	TYPE	EID	CPNAME	CPMIN	CPMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	COEF3	-etc.-		

Example:

DVCREL1	5	CQUAD4	1	ZOFFS		1.0			
	1	1.0							

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of an element connectivity entry, such as "CBAR", "CQUAD4", etc. (Character)
EID	Element Identification number. (Integer > 0)
CPNAME	Name of connectivity property, such as "X1", "X2", "X3", "ZOFFS", etc. (Character)
CPMIN	Minimum value allowed for this property. If CPNAME references a connectivity property that can only be positive, then the default value of CPMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 4. (Real)
CPMAX	Maximum value allowed for this property. See Remark 4. (Real; Default =-1.0E+20)
C0	Constant term of relation. (Real; Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer > 0)
COEFi	Coefficient of linear relation or keyword = "PVAL". (If i = 1, Real or Character; if i > 1, Real)

Remarks:

1. The relationship between the connectivity property and design variables is given by:

$$CP_j = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. The fifth field of the entry, CPNAME, only accepts string characters. These string values must be the same as those given in the connectivity entry descriptions in this Guide. For example, if the plate offset is to be designed (CQUAD4, CTRIA3, etc), ZOFFS (case insensitive) must be specified on the CPNAME field.

4. The default values for CPMIN and CPMAX are not applied when the linear property is a function of a single design variable and C0=0. It is expected that the limits applied on the associated DESVAR entry will keep the designed property within meaningful bounds.
5. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the connectivity bulk data entry. If a DVCREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.
6. If the user inputs CQUAD4/CTRIA3 entries and then uses QRMETH = 5 to convert them to CQUADR/CTRIAR entries, the design of items on these entries using the DVCREL1 entry should refer to the converted type (i.e., CQUADR/CTRIAR). Similarly, if QRMETH=2 or 3 is used, the DVCREL1 entry should refer to CQUAD4/CTRIA3 types.

DVCREL2**Design Variable to Connectivity Property Relation**

Defines the relation between a connectivity property and design variables with a user-supplied equation.

Format:

1	2	3	4	5	6	7	8	9	10
DVCREL2	ID	TYPE	EID	CPNAME	CPMIN	CPMAX	EQID		
	"DESVAR"	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	"DTABLE"	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						

Example:

DVCREL2	1	CBAR	100	X1	0.05	1.0	100		
	DESVAR	1001							
	DTABLE	X10							

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of an element connectivity entry, such as "CBAR", "CQUAD4", etc. (Character)
EID	Element Identification number. (Integer > 0)
CPNAME	Name of connectivity property, such as "X1", "X2", "X3", "ZOFFS", etc. (Character)
CPMIN	Minimum value allowed for this property. If CPNAME references a connectivity property that can only be positive, then the default value of CPMIN is 1.0E-15. Otherwise, it is -1.0E35. (Real)
CPMAX	Maximum value allowed for this property. (Real; Default =1.0E+20)
EQID	DEQATN entry identification number. (Integer > 0)
"DESVAR"	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
"DTABLE"	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)

Remarks:

1. The variable identified by DVIDi and LABLi correspond to variable names (x1, x2, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x1 through xN (where N = m + n) are assigned in the order DVID1, DVID2, ..., DVIDm, LABL1, LABL2, ..., LABLn.
2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.
3. The fifth field of the entry, CPNAME, only accepts string characters. These string values must be the same as those given in the connectivity entry descriptions in this Guide. For example, if the plate offset is to be designed (CQUAD4, CTRIA3, etc.), ZOFFS (case insensitive) must be specified on the CPNAME field.
4. If the user inputs CQUAD4/CTRIA3 entries and then uses QRMETH = 5 to convert them to CQUADR/CTRIAR entries, the design of items on these entries using the DVCREL2 entry should refer to the converted type (i.e., CQUADR/CTRIAR). Similarly, if QRMETH=2 or 3 is used, the DVCREL2 entry should refer to CQUAD4/CTRIA3 types.

DVGRID

Design Variable to Grid Point Relation

Defines the relationship between design variables and grid point locations.

Format:

1	2	3	4	5	6	7	8	9	10
DVGRID	DVID	GID	CID	COEFF	N1	N2	N3		

Example:

DVGRID	3	108	5	0.2	0.5	0.3	1.0		
--------	---	-----	---	-----	-----	-----	-----	--	--

Descriptor	Meaning
DVID	DESVAR entry identification number. (Integer > 0)
GID	Grid point (GRID) or geometric point (POINT) identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0 ; Default = 0)
COEFF	Multiplier of the vector defined by Ni. (Real; Default = 0.0)
Ni	Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni $\neq 0.0$)

Remarks:

1. A CID of zero or blank (the default) references the basic coordinate system.
2. Multiple references to the same grid ID and design variable result in vectorial addition of the participation vectors defined by CID, COEFF, and Ni. There is no restriction on the number of DVGRID entries that may reference a given grid (GID) or design variable (DVID).
3. The coordinate update equation is given as

$$\{g\}_i - \{g\}_i^0 = \sum_j COEFF_j (X_{DVIDj} - X_{DVIDj}^0) \{N\}_j$$

where $\{g\}_i$ is the location of the i -th grid, $[g_x g_y g_z]^T$.

The vector $\{N\} = [N_x N_y N_z]^T$ is determined from CID and Ni. Note that it is a change in a design variable from its initial value X^0 , and not the absolute value of the design variable itself, that represents a change in a grid point location, $\{g\}_i - \{g\}_i^0$.

4. The DVGRID entry defines the participation coefficients (basis vectors) of each design variable for each of the coordinates affected by the design process in the relationship

$$\{\Delta g\}_i = \sum_j \{T\}_{ij} \cdot \Delta X_j$$

5. DVGRID entries that reference grid points on MPCs or RSSCON entries produce incorrect sensitivities. Often the sensitivities are 0.0 which may result in a warning message indicating zero gradients which may be followed by UFM 6499. Other rigid elements produce correct results.

DVLREL1

Defines the linear relation between analysis model loading and design variables in SOL 200 with Analysis = STATICS

Format:

1	2	3	4	5	6	7	8	9	10
DVLREL1	ID	TYPE	SID	LNAME	LMIN	LMAX	C0		
	ATT1	ATT2	ATT3	ATT4	ATT5				
	DVID	COEF	DVID2	COEF2	DVID3	Etc			

Example: Design the N1 FORCE at GRID 100 to be equal to DESVAR=10

DVLREL1	10	FORCE	300	N1					
	100								
	10	1.0							

Descriptor	Meaning
ID	Unique identification number (Integer>0)
TYPE	Name of Load, such as FORCE, see Remark 2. for supported types (Character)
SID	Load set ID (Integer>0)
LNAME	Load Name, such as F or N1 on the FORCE entry. See Remark 2. (Character)
LMIN	Minimum value for the load. See Remark 5. (Real, default=-1.0e35)
LMAX	Maximum value for the load (Real, Default=1.0e20)
C 0	Constant term of relation (Real, Default=0.0)
ATTi	Attributes of the designed load, see Remark 2. (Integer>0 or blank)
DVIDi	DESVAR entry identification number. (Integer>0)
COEFFi	Coefficient of linear relation or keyword="PVAL", See Remark 3. (If i=1, Real or Character; if i>1, Real)

Remarks:

1. The relationship between the analysis model load and the design variables is given by:

$$L_i = C_0 + \sum COEFF_i X_{DVID_i}$$

2. Supported TYPES and their ATT_i meaning are given in the following table:

Load Type	LNAME (character)	ATT1 (integer)	ATT2 (integer)	ATT3 (integer)	ATT4 (integer)	ATT5 (integer)
FORCE	F, N1, N2 or N3	G	CID (see remark 6)			
LOAD	S or Si	0 if LNAME is S, Li otherwise				
MOMENT	M, N1, N2 or N3	G	CID (see remark 6)			

3. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the load bulk data entry. If a DVLREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.
4. If there are multiple loads that satisfy the designation of the request (for example two FORCE entries with identical SID, GRID and CID values), it is a user input error.
5. The default values of LMIN and LMAX are not applied when the linear property is a function of a single design variable and C0=0. It is expected that the limits applied on the DESVAR entry will keep the designed property within reasonable bounds
6. Input of these data is optional. In most cases, leaving it off will result in a single load that qualifies with the remaining attributes. If there are multiple instances that qualify, it is a user input error.
7. The use of this entry is limited to statics and buckling analyses in SOL 200.

DVMREL1**Design Variable to Material Relation**

Defines the relation between a material property and design variables.

Format:

1	2	3	4	5	6	7	8	9	10
DVMREL1	ID	TYPE	MID	MPNAME	MPMIN	MPMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	COEF3	-etc.-		

Example:

DVMREL1	5	MAT1	1	RHO	0.05	1.0			
	1	1.0							

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of a material property entry, such as "MAT1", "MAT2", etc. (Character)
MID	Material Identification number. (Integer > 0)
MPNAME	Name of material property, such as "E" or "RHO". (Character)
MPMIN	Minimum value allowed for this property. If MPNAME references a material property that can only be positive, then the default value for MPMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 4. (Real)
MPMAX	Maximum value allowed for this property. See Remark 4. (Real; Default = 1.0E+20)
C0	Constant term of relation. (Real; Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer > 0)
COEFi	Coefficient of linear relation or keyword = "PVAL". (If i = 1, Real or Character; if i > 1, Real)

Remarks:

1. The relationship between the material property and design variables is given by:

$$MP_i = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. The fifth field of the entry, MPNAME, only accepts string characters. It must be the same as the name that appears in the [Bulk Data Entries, 1117](#) for various material properties. For example, if the isotropic material density is to be designed, RHO (case insensitive) must be specified on the MPNAME field.

4. The default value for MPMIN and MPMAX are not applied when the linear property is a function of a single design variable and C0=0.0. It is expected that the limits applied to the DESVAR entry will keep the designed property within reasonable bounds.
5. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the material bulk data entry. If a DVMREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.

DVMREL2**Design Variable to Material Relation**

Defines the relation between a material property and design variables with a user-supplied equation.

Format:

1	2	3	4	5	6	7	8	9	10
DVMREL2	ID	TYPE	MID	MPNAME	MPMIN	MPMAX	EQID		
	DESVAR	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	DTABLE	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						

Example:

DVMREL2	5	MAT1	1	E	0.05	1.0	100		
	DESVAR	1	2						
	DTABLE	E0							

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of a material property entry, such as "MAT1", "MAT2", etc. (Character)
MID	Material Identification number. (Integer > 0)
MPNAME	Name of material property, such as "E" or "RHO". (Character)
MPMIN	Minimum value allowed for this property. If MPNAME references a material property that can only be positive, then the default value for MPMIN is 1.0E-15. Otherwise, it is -1.0E35. (Real)
MPMAX	Maximum value allowed for this property. (Real; Default = 1.0E+20)
EQID	DEQATN entry identification number. (Integer > 0)
DESVAR	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
DTABLE	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)

Remarks:

1. The variables identified by DVID_i and LABL_i correspond to variable names (x₁, x₂, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x₁ through x_N (where N = m + n) are assigned in the order DVID₁, DVID₂, ..., DVID_m, LABL₁, LABL₂, ..., LABL_n.
2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.
3. The fifth field of the entry, MPNAME, only accepts string characters. It must be the same as the name that appears in the [Bulk Data Entries, 1117](#) for various material properties. For example, if the isotropic material density is to be designed, RHO (case insensitive) must be specified on the MPNAME field.

DVPREL1**Design Variable to Property Relation**

Defines the relation between an analysis model property and design variables.

Format:

1	2	3	4	5	6	7	8	9	10
DVPREL1	ID	TYPE	PID	PNAME/ FID	PMIN	PMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	-etc.-			

Example:

DVPREL1	12	PBAR	612	6	0.2	3.0			
	4	0.25	20	20.0	5	0.3			

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of a property entry, such as "PBAR", "PBEAM", etc. (Character)
PID	Property entry identification number. (Integer > 0)
PNAME/FID	Property name, such as "T", "A", or field position of the property entry, or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referred to by field position. (Character or Integer ≠ 0)
PMIN	Minimum value allowed for this property. If PMIN references a property that can only be positive, then the default value for PMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 6. (Real)
PMAX	Maximum value allowed for this property. (Real; Default = 1.0E+20)
C0	Constant term of relation. (Real; Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer > 0)
COEFi	Coefficient of linear relation or keyword = "PVAL". See Remark 7. (If i = 1, Real or Character; if i > 1, Real)

Remarks:

1. The relationship between the analysis model property and design variables is given by:

$$P_j = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. TYPE="PBEND" is not supported. TYPE="PBEAML" supports only PNAME and not FID.

4. FID may be either a positive or a negative number. If FID > 0, it identifies the field position on a property entry. If FID < 0, it identifies the word position of an entry in the element property table. For example, to specify the area of a PBAR, either PNAME=A, FID=+4 or FID=-3 can be used. In general, use of PNAME is recommended. For Type "PBUSH", PNAME is recommended, if FID is used it must be < 0.
5. Designing PBEAML or PBEAM requires specification of both property name and station. [Table 14](#) shows several examples.

Table 14

PTYPE	Property Name	END A	END B	i-th Station
PBEAML	DIM1	DIM1 or DIM1(A)	DIM1(B)	DIM1(i)
PBEAM	A	A or A(A)	A(B)	A(i)

Only stations that are input on a PBEAM or PBEAML entry can be referenced by a DVPREL1. For example, referencing an END B property name on a DVPREL1 entry when the referenced PBEAM does not explicitly specify the END B station, is not allowed.

6. The default values of PMIN and PMAX are not applied when the linear property is a function of a single design variable and C0=0. It is expected that the limits applied on the DESVAR entry will keep the designed property within reasonable bounds.
7. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the property bulk data entry. If a DVPREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.
8. With GPLY for TYPE field and GPLYID for PID field, a ply identified with GPLYID across all PCOMPG entries in the model can be designed. Internally, a DVPREL1 will be spawned for each PCOMPG has a ply ID of GPLYID. For TYPE=GPLY, the relationship between the analysis model property and design variables is given by.

$$P_i = C0 + (T0_i \text{ or } THETA0_i) \cdot \sum_j (COEF_j \cdot X_{DVID_j}) \text{ for PNAME=T or THETA}$$

Where T0 and THETA0 are value of thickness and theta angle on the original PCOMPG. Note that non-zero C0 is not recommended for TYPE=GPLY or PCOMPG. For THETA0 with original value equal to 0.0, THETA0 is taken as 1.0 and it is recommended to have XINIT of DVID set to 0.0.

DVPREL2**Design Variable to Property Relation**

Defines the relation between an analysis model property and design variables with a user-supplied equation.

Format:

1	2	3	4	5	6	7	8	9	10
DVPREL2	ID	TYPE	PID	PNAME/ FID	PMIN	PMAX	EQID		
	"DESVAR"	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	"DTABLE"	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						

Example:

DVPREL2	13	PBAR	712	5	0.2	0.4	50		
	DESVAR	4	11	13	5				
	DTABLE	PI	YM						

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of a property entry, such as PBAR, PBEAM, etc. (Character)
PID	Property entry identification number. (Integer > 0)
PNAME/FID	Property name, such as "T", "A", or field position of the property entry, or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referred to by field position. (Character or Integer ≠ 0)
PMIN	Minimum value allowed for this property. If FID references a stress recovery location field, then the default value for PMIN is -1.0+35. PMIN must be explicitly set to a negative number for properties that may be less than zero (for example, field ZO on the PCOMP entry). (Real; Default = 1.E-15)
PMAX	Maximum value allowed for this property. (Real; Default = 1.0E20)
EQID	DEQATN entry identification number. (Integer > 0)
"DESVAR"	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
"DTABLE"	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)

Remarks:

1. The variables identified by DVID_i and LABL_i correspond to variable names (x₁, x₂, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x₁ through x_N (where N = m+n) are assigned in the order DVID₁, DVID₂, ..., DVID_n, LABL₁, LABL₂, ..., LABL_m.
2. If both "DESVAR" and "DTABLE" are specified in field 2, "DESVAR" must appear first.
3. FID may be either a positive or a negative number. If FID > 0, it identifies the field position on a property entry. If FID < 0, it identifies the word position of an entry in EPT. For example, to specify the area of a PBAR, either PNAME=A, FID = +4 or FID = -3 may be used. In general, use of PNAME is recommended. For Type "PBUSH", PNAME is recommended, if FID is used it must be < 0.
4. Types "PBEND", "PBTRL" and "PBEAML" are not supported for the DVPREL2.
5. Designing PBEAM requires specification of both property name and station. [Table 15](#) shows one example.

Table 15

PTYPE	Property Name	END A	END B	i-th Station
PBEAM	A	A or A(A)	A(B)	A(i)

Only stations that are input on a PBEAM entry can be referenced by a DVPREL2. For example, referencing an END B property name on a DVPREL2 entry when the referenced PBEAM does not explicitly specify the END B station, is not allowed.

DVSHAP**Design Variable to Basis Vector(s)**

Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix.

Format:

1	2	3	4	5	6	7	8	9	10
DVSHAP	DVID	COL1	SF1	COL2	SF2	COL3	SF3		

Example:

DVSHAP	2	1	2.0	4	1.0				
--------	---	---	-----	---	-----	--	--	--	--

Descriptor	Meaning
DVID	Design variable identification number on the DESVAR entry. (Integer > 0)
COLi	Column number of the displacement matrix. See Remark 2. ($1 \leq$ Integer \leq maximum column number in the displacement matrix.)
SFi	Scaling factor applied to the COLi-th column of the displacement matrix. (Real; Default = 1.0)

Remarks:

1. DVID must be defined on a DESVAR entry.
2. COLi must be a valid column number in the displacement matrix.
3. Multiple references to the same DVID and/or COLi will result in a linear combination of displacement vectors. In the example above, the shape basis vector is a linear combination of the fourth column and twice the second column.
4. The displacement matrix must have been created by SOL 101 or 200 with analysis = statics and be available on a database, which is attached via the DBLOCATE FMS statement shown below:

```
ASSIGN    DISPMAT=' physical filename of MASTER DBset '
DBLOCATE DATABLK=(UG/UGD,GEOM1/GEOM1D,GEOM2/GEOM2D) ,
LOGICAL=DISPMAT
```

DVPSURF

Design Variable to Control Surface Setting Relation

Defines the relationship between a control surface setting in a particular subcase and a design variable.

Format:

1	2	3	4	5	6	7	8	9	10
DVPSURF	ID	AELABEL	TRIMID	DVID	COEF				

Example:

DVPSURF	10	OBDFLAP	1	100	0.01746				
---------	----	---------	---	-----	---------	--	--	--	--

Descriptor	Meaning
ID	Unique identification number (Integer>0)
AELABEL	LABEL of the AESURF entry that is being designed (Character ,no default)
TRIMID	Associated trim set identification number (Integer>0)
DVID	DESVAR entry identification number (Integer>0)
COEF	Coefficient of linear relation (Real)

Remarks:

1. The relationship between the control surface setting and the design variable is given by $\delta_{\text{SURF}} = \text{COEF} * X_{\text{DVID}}$
2. The surface called out by AELABEL must also appear on the trim entry specified by TRIMID. The value specified on the trim entry will be overwritten by the value obtained from the relationship of Remark 1.
3. Limits on the control deflection are not provided on this entry but can be specified on the underlying DESVAR
4. The DVID called out on this entry cannot be associated with any other designed property.
5. Note that since the DVPSURF calls out a TRIM ID, it is associated only with a single subcase.

DYFSISW

Fluid Structure Interaction Activation Switch

Allows activating or deactivating Fluid Structure Interaction and Eulerian solver. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
DYFSISW	SWID	TID	INITV						

Example:

DYFSISW	14	10							
---------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning				
SWID	Unique number of a DYFSISW entry. (Integer > 0; Required)				
TID	TABLED1 ID. Fluid Structure Interface and Euler solver are switched on and off, depending on the y-value of the table. The x-value of the table represents the time; the y-value denotes: <table> <tr> <td>ON</td><td>$y > 0.0$</td></tr> <tr> <td>OFF</td><td>$y < 0.0$</td></tr> </table>	ON	$y > 0.0$	OFF	$y < 0.0$
ON	$y > 0.0$				
OFF	$y < 0.0$				
INITV	Euler element initialization flag. See remark 3. (Character, Default=NO) <table> <tr> <td>NO</td><td>Element initialization at cycle 0</td></tr> <tr> <td>YES</td><td>Element initialization when the elements becomes active</td></tr> </table>	NO	Element initialization at cycle 0	YES	Element initialization when the elements becomes active
NO	Element initialization at cycle 0				
YES	Element initialization when the elements becomes active				

Remarks:

1. The default is that COUPLE, AIRBAG and Euler solver is active at all times.
2. When Euler solver is deactivated, no output will be generated for the Eulerian elements.
3. For simulations involving a prestress phase the Euler elements can be initially deactivated. When the Euler elements are activated the Eulerian masses still originate from the Euler initiation at cycle 0. If during prestressing the structure did not move much, then these Eulerian masses of cycle 0 can give a stable run. But if there has been substantial movement of the structure then Eulerian masses can be erroneously compressed. This shows up by large pressures and velocities in the Euler elements and a time step too small. To avoid this instability the Euler initialization can be postponed until the Euler elements become active.

DYPARAM, ATBAOUT

Output Frequency to Main Output File of ATB - SOL 700

Defines the frequency at which output is written to the main output file of ATB. Used in SOL700 only.

Format:

DYPARAM, ATBAOUT, value

Example:

DYPARAM, ATBAOUT, 5.0E-3

Option	Content
Value	Every multiple of ATBAOUT seconds, the main output file of ATB is updated. (Real > 0.0; default=10.0E-3)

Remarks:

1. Only active when field 3 on the A5 card of the ATB input file is set to a value of -1.
2. Controls the frequency of the output of segment acceleration, velocity and displacement, joint forces and moments.

DYPARAM, ATBHOUT

Write ATB Output to Time history files - SOL 700

A time-history file is created containing the output as requested in the ATB input file on cards H.1 to H.11.
Used in SOL700 only.

Format:

DYPARAM, ATBHOUT, option

Example:

DYPARAM, ATBHOUT, NO

Option	Content
Option	Flag of ATBHOUT. (Character; default=YES) YES: The time history files are created. NO: The time history files are not created.

DYPARAM, ATBTOUT

Output Frequency to Time-History Files of ATB - SOL 700

Defines the frequency at which output is written to the time-history files of ATB. Used in SOL700 only.

Format:

DYPARAM, ATBTOUT, value

Example:

DYPARAM, ATBTOUT, 1.0E-4

Option	Content
Value	Every multiple of ATBTOUT seconds, the time-history files of ATB are updated. (Real > 0.0; default=1.0E-3)

Remarks:

1. Only active when field 26 on the A5 card of the ATB input file is set to a value of -1.
2. Controls the frequency of all output requested on the H-cards, and of the tabular time-histories that are controlled by field 18 on the A5 card of the ATB input file.

DYPARAM,AUTOCOUP**Automatic Coupling**

Defines the automatic coupling algorithm. Used in Sol700 only.

Format:

DYPARAM,AUTOCOUP,ACTIVE,CLEAN,DUMMY, TOL_AREA, OUTPUT, FAIL

Example:

DYPARAM, AUTOCOUP, ON

Descriptor	Meaning
ACTIVE	Flag of activation of the sub element approach. (Character; default=OFF)
ON	Turn on the sub element approach.
OFF	Turn off the sub element approach.
CLEAN	Flag of cleaning obsolete sub elements. (Character; default=OFF)
ON	Cleans obsolete sub elements
OFF	Obsolete elements will not be removed from memory.
DUMMY	Flag of putting dummy segments on coupling surface archives. (Character; default=OFF)
ON	Puts dummy segments on the coupling surface archive
OFF	Does not put dummy segments on the coupling surface archive.
TOL_AREA	If the area of a hole is smaller than TOL_AREA a 1D flow computation method is used. (Real ≥ 0.0 ; default=0.0).
OUTPUT	Determines how results of Euler elements that are intersected by the structure are written to the Euler archive. (Character, Default = ZERO).
ZERO	Write zero for the results of these intersected elements.
AVERAGE	Average across sub elements. Each sub element has the same weight.
AVEREAGEU	Average across sub element using the uncover fraction as weights.
MAXUNC	The results are taken from the sub element with the largest volume uncover fraction.
COVER	Uses the cover field of COUPLE to select the proper sub element. Only supported for COVER=OUTSIDE or COVER=INSIDE. Not supported for COVER =NONE.
FAIL	Activates fluid flow through failed shell segments in the coupling surface. (Character; there is no default, if failure is not used, then FAIL should be left blank)
POROUS	Makes failed segment porous. This is the fast coupling approach.

REMOVE Removes the failed segments from the coupling surface.

Remarks:

1. Features that are not supported are:
 - Multiple Euler domain
 - Adaptive Euler meshes
 - Fluid flow through failed segments in the coupling surface failure can only be defined by option FAIL of DYPARAM, AUTOCOUP. Defining this failure by use of COUP1FL and COUPINT is not supported.
 - COUPLE with option AIRBAG
 - Graded meshes
 - Euler import
 - Viscosity
 - Porosity
 - Option FAIL=POROUS and REMOVE are not supported by DMP.
 - Markers
2. With automatic coupling fluid can be defined on both sides of the coupling surface. To do this option COVER of the COUPLE entry has to be set to NONE. When using COVER = OUTSIDE fluid is only initialized in the inside region of the coupling surface. For COVER = INSIDE the opposite applies. When the coupling surface is not closed, only COVER=NONE is allowed.
3. In the auto coupling approach holes in the coupling surface are meshed with dummy segments. For holes with large deformations PARAM DUMSEGS, ON can be used to maintain good dummy segments.
4. For adding FAIL = POROUS failed segments have to be made fully porous. This is straightforward.
5. FAIL = REMOVE. When the structure fails, holes in the coupling surface are formed. For a closed coupling surface each side of a segment is connected to another segment. Therefore, the segments have no free edges. But a segment adjacent to a hole has a free edge. At this edge it is not connected to any other segment. A list of free edges of the coupling surface specifies the holes. By maintaining a list of free edges auto coupling handles these holes automatically. If a segment fails, this list of free edges has to be updated. Also the failed segment has to be removed from the coupling surface.

Note:

This entry also supports DMP. While using for DMP, no changes to input is needed. In the auto coupling method Euler elements that are intersected by the structure are split into sub elements. For details refer to [1]. To enable dmp for auto coupling some of the properties of sub elements have to be communicated across CPU's. If holes in the coupling surface extend across multiple CPUs, also communication across CPU's is needed.

DYPARAM,AXIALSYM

Axial Symmetric Analyses - SOL 700

Enables an efficient and accurate 2-D axial symmetry for Eulerian materials. A much larger time step becomes possible by not taking into account the mesh-size in circumferential direction. Used in SOL 700 only.

Format:

DYPARAM,AXIALSYM,MESHTYPE,AXIALAXIS,SYMPLAN,PHI,ALIGN,PHI2

Example:

DYPARAM,AXIALSYM,RECT,X,XY,2.5,YES,0.0

Descriptor	Meaning	
MESHTYPE	Two types of Euler meshes are supported: (Character; Required)	
AXIAL	Axial symmetric meshes.	
RECT	Rectangular meshes	
AXIAL AXIS	X	X-Axis (Character; Required)
	Y	Y-Axis
	Z	Z-Axis
SYMPLAN	The approximate symmetry plan of the Euler mesh. On for MESHTYPE=AXIAL. See Remark 7. (Character; Required)	
XY	XY-Plane	
YZ	YZ-Plane	
ZX	ZX-Plane	
PHI	Only used for MESHTYPE = RECT. Used to create a 2D axial symmetric mesh with angles +PHI/2 and -PHI/2. (Real; Default = 0.0)	
ALIGN	Only used for MESHTYPE = AXIAL. (Character; Default = YES)	
YES	Align normals of oblique Euler element faces. This prevents errors in strains that can arise from small errors in Euler face normals.	
NO	Do not align normals.	
PHI2	As a final operation rotate the mesh around the axial axis by the angle PHI2. See Remark 6. (Real; Default = 0.0)	

Remarks:

1. Only available for Eulerian elements and does not support Lagrange elements. The effect of this parameter is not limited to the solvers. Also Euler archives will reflect the modified Euler mesh geometry.

2. The Euler mesh can already be symmetric but also a rectangular mesh comprising of one layer can be used. Using the angle specified by PHI this Euler mesh is mapped into a 2d axial symmetric mesh.
 3. The Euler mesh has to consist of one layer.
 4. Rectangular meshes that can be made 2d symmetric using the angle PHI should satisfy:
 - All boundary Euler faces are aligned with a coordinate direction
 - Only one layer thick.
 - The axial symmetry axis is either on the boundary of the Euler mesh or outside the Euler mesh. It is not allowed that the axial axis is inside the Euler mesh.
- Initialization of Euler element using geometric regions as defined by the TICEUL entry is carried out onto the transformed 2d axial mesh.
5. In the time step computation the circumferential mesh-size will not be taken into account.
 6. Use option PHI2 with caution. Euler initialization is done using the mesh rotated by the angle PHI2. So after including the angle PHI2 or modifying its value the Euler initialization should be revised.
 7. It is assumed that one of the coordinate planes is an approximate symmetry plane of the Euler mesh. Although approximate symmetry is sufficient, the coordinate plane can always be made an exact symmetry plane by the use of PHI2. If for example the Euler mesh has angles 0 and 2.5, PHI2 has to be set to -1.25 to get exact symmetry.

DYPARAM,AXREMAP

2D Axial symmetric Euler archive remap - SOL 700

Allows import of a 2D axial symmetric Euler archive into a 3D simulation.

Format:

1	2	3	4	5	6	7	8	9	10
DYPARAM	AXREMAP	x0	y0	z0	xn	yn	zn	range	

Example:

DYPARAM	AXREMAP	0.0	0.5	0.5	1.	0.	0.		
---------	---------	-----	-----	-----	----	----	----	--	--

Descriptor	Meaning
x0,y0,z0	x,y,z-coordinate of the point at which the 2D axial symmetric mesh is remapped. (Real; Default = 0.0)
xn,yn,zn	Unit vector that specifies the direction of the axial axis of the 2D axial symmetric mesh as viewed in the 3D mesh. (Real; Default = (1.0., 0.0, 0.0))
range	Only material whose distance from the axial axis is smaller than "range" will be initialized with the 2D axi-symmetric Euler archive. (Real, Default = 1e+20)

Remarks:

1. Since 2D axial symmetric simulations run much faster than 3D simulation it can save much cpu time to do the first part of the simulation with a 2D axial symmetric mesh. Afterwards the 2D-axial symmetric Euler archive is imported into the 3D simulation. By default the 2D axial symmetric archive will not be expanded in 3D. To enable this expansion DYPARAM, AXREMAP has to be used. It is useful for blast wave simulations. The 2D axial symmetric simulation has to be terminated before the blast wave approaches any 3D structure.
2. This import of Euler archives is done by means of the eid option in the pth file.
3. To generate axial symmetric meshes DYPARAM,AXIALSYM can be used.

DYPARAM,BULKL

Linear Bulk Viscosity Coefficient- SOL700

Defines the default value of the linear bulk viscosity coefficient.

Format:

DYPARAM, BULKL, VALUE

Example:

DYPARAM, BULKL, 0.1

Descriptor	Meaning
value	Value of the linear coefficient in the bulk viscosity equation. (Real ≥ 0.0 ; default=0.0)

Remarks:

1. The default value works well for the majority of problems.
2. The value defined on this entry is used as the default whenever BULKL is blank on the MATBV material entry.
3. When BULKL is specified on a material definition entry, the default value is overridden for that specific material.

DYPARAM,BULKQ

Quadratic Bulk Viscosity Coefficient - SOL700

Defines the default value of the quadratic bulk viscosity coefficient.

Format:

DYPARAM, BULKQ, VALUE

Example:

DYPARAM, BULKQ, 1.6

Descriptor	Meaning
value	Value of the quadratic coefficient in the bulk viscosity equation. (Real ≥ 0.0 ; default=0.0)

Remarks:

1. The default value works well in the majority of situations.
2. The value defined on this entry is used as the default whenever BULKQ is blank on the MATBV material entry.
3. When BULKQ is specified on a material definition entry, the default value is overridden for that specific material.

DYPARAM,BULKTYP

Bulk Viscosity Type - SOL700

Defines the default type of bulk viscosity.

Format:

DYPARAM, BULKTYP, option

Example:

DYPARAM, BULKTYP, DYNA

Describer	Meaning
Option	bulk viscosity type. (Character; Default=DYNA)
DYNA	Standard DYNA3D model
DYTRAN	Enhanced DYNA model

Remarks:

1832

DYPARAM,CFULLRIG

Converts 123456 Constraints to FULLRIG on RBE2 - SOL700

DYPARAM,CFULLRIG

Converts 123456 Constraints to FULLRIG on RBE2 - SOL700

Converts all 123456 constraints to the FULLRIG option on all entries. Used in SOL 700 only.

Format:

DYPARAM, CFULLRIG, value

Example:

DYPARAM, CFULLRIG, NO

Descriptor	Meaning
value	Activation flag. (Character; Default=YES)
YES	123456 constraints are converted to FULLRIG.
NO	123456 constraints are not converted to FULLRIG.

DYPARAM,CLUFLIM**Limiter of Volume Stain Rate for Clumps - SOL700**

In some cases, airbag runs become instable. Often, this is caused by a much too large volume strain rate in a clump that consists of too many elements. These clumps typically have a small average volume uncovered fraction. The large volume strain rate causes a huge compression work and this blows up the specific internal energy. When this happens it is clearly visible in the OUT file and in the results. This DYPARAM activates a limiter that scales down the volume strain rate for clumps with a small average uncovered fraction. It can keep an instable airbag run stable, just like PARAM, VELMAX can keep runs stable. Used in SOL 700 only.

Format:

DYPARAM, CLUFLIM, value

Example:

DYPARAM, CLUFLIM, 0.22

Descriptor	Meaning
value	The volume strain rate in a clump will be reduced when the average Uncovered Fraction of elements in a CLump falls below CLUFLIM. CLUFLIM has to be smaller than FBLEND. The default value of FBLEND is 0.66 giving a value of 0.22 for CLUFLIM. For more details on FBLEND refer to PARAM,FBLEND. (0<Real<FLBEND; default=FBLEND/3)

Remarks:

1. The Volume strain rate $\frac{DIV}{\Delta t}$ in clumps will be limited by

$$\left(\frac{DIV}{\Delta t} \right)_{Lim} = MIN\left(1, \frac{U}{CLUFLIM} \right) \left(\frac{DIV}{\Delta t} \right)$$

Here, U is the average uncovered fraction of elements in the clump as given by

$$U = \frac{\sum_{eleclump} Uncf_{el} \cdot Vol_{el}}{\sum_{eleclump}}$$

Here Uncf and Vol denote the uncovered fraction and volume of an element inside the clump. Therefore, only when the average uncover fraction falls below CLUFLIM, the volume strain rate is limited.

DYPARAM,CLUMPENR

Switch for Kinetic Energy Calculation Scheme of Blended Clumps - SOL700

Sets the definition of the kinetic energy calculation method for Eulerian blended clumps. Used in SOL 700 only.

Format:

DYPARAM, CLUMPENR, value

Example:

DYPARAM, CLUMPENR, SUM

Descriptor	Meaning
value	Flag for CLUMPENR.(Character; default=AVERAGE)
AVERAGE	The kinetic energy of a Eulerian blended clump is calculated from the average velocity of the clump. The average velocity of the blended clump is computed as the sum of the momentum of each member of the clump divided by the total clump mass.
SUM	The kinetic energy of a Eulerian blended clump is calculated as the sum of the kinetic energy of each member of the clump.

DYPARAM,CONN2OUT

CONN2 Summary Output - SOL700

Determines if a summary of concentrated masses and their energy and momentum is written to the output file. Used in SOL 700 only.

Format:

DYPARAM,CONN2OUT,value

Example:

DYPARAM,CONN2OUT,NO

Descriptor	Meaning
value	Flag for COMN2OUT. (Character; default=NO)
NO	No information about concentrated masses is written to the cycle and material summaries on the output file.
YES	A complete summary of concentrated masses including the associated mass, momentum, and energy is written to the output file.

Remarks:

1. When DYPARAM,CONN2OUT is set to NO, there is no summary of the concentrated mass. This means that the mass, momentum, and energy of the concentrated masses, is not added to the material and cycle summaries. Setting DYPARAM,CONN2OUT,NO saves memory and CPU time.

DYPARAM,CONTACT

Sets Defaults for CONTACT - SOL700

Defines certain defaults for the contact definitions. Used in SOL 700 only.

Format:

DYPARAM,CONTACT,option,value1,value2,value3,value4,value5,value6

Example:

DYPARAM,CONTACT,VERSION,V4

Descriptor	Meaning
VERSION	Defined the default version of contact. Only support value1. (Character; required)
V4	General Contact algorithm of SOL700
BELT	Suited for modeling contact between a belt element and a rigid structure. Master slave contact only. The contact logic doesn't apply a contact force, but applies an enforced displacement and velocity that keeps the slave nodes exactly on top of the master face. The slave node does not slide relative to the master face when the friction coefficient (FS) is set to 1E20.
BELT1	Identical to BELT algorithm, except that the slave nodes are initially repositioned on top of the closest master face.
DRAWBEAD	Suited for modeling a drawbead.
THICK	Defines the default value for THICK. Only support vaule1. (Real>0.0; required.)
GAP	Defines the default value for GAP. Only support vaule1. (Real>0.0; required.)
LIMITS	Definition of a three dimensional contact region where contact in the analysis model takes place. Significant CPU time savings can be achieved when used in adaptive contact. 6 values are used to define area. The format of this option is DYPARAM,CONTACT,LIMITS,xmin,xmax,ymin,ymax,zmin,zmax. (Real; default=1.0E20 for xmax, ymax and zmax and -1.0E20 for xmin, ymin and zmin) XMIN(value1) Lower limit in x-direction where main contact occurs XMAX(value2) Upper limit in x-direction where main contact occurs YMIN(value3) Lower limit in y-direction where main contact occurs YMAX(value4) Upper limit in y-direction where main contact occurs ZMIN(value5) Lower limit in z-direction where main contact occurs ZMAX(value6) Upper limit in z-direction where main contact occurs
DAMPING	Defines the default for usage of damping. Only support value1. (Character; required)
YES	Use damping in CONTACT

Descriptor	Meaning	
	NO	Not use damping in CONTACT
COPOR		Activates contact based porosity. Only support value1. (Character; default=NO)
	YES	Use porosity in CONTACT
	NO	Not use porosity in CONTACT
DAMPFOR		Defines whether the noncontact forces acting on the grid points need to be taken into account in the contact damping. This option is only used if DAMPING is set to YES. This option prevents large penetrations that might occur when the forces acting on the grid points tend to push them into the contact surface. This happens, for example, in airbag analyses, where a large pressure exists inside the bag. Only support value1. (Character; required)
	YES	Damping is considered in noncontact forces
	NO	Damping is not considered in noncontact forces
DYNA		The following parameters of the contact definition get the default values consistent with Dyna. (Character ; required)
	THICK	1.0
	THICKOF	0.0
INFO		Information on the contact state of grid points G1,G2,... is printed to ASCII files, named CNT. This information can be useful in debugging models with contacts. Supports many values as you want. (Integer > 0; required.)
EVIEW		Defines the default value of the view angle of Edge to Edge contacts. The value of the angle must be in degrees. Only support value1. (Real > 0.0; required)
FORCE		Controls the contact forces on the grid points. Supports value1, value2, value3. See Remark 1. (value1: Integer > 0; default=10, value2: Real > 0; default=10.0; value3: Character; default=ZERO)
	NMCYC (value1)	Frequency check of contact force; applied on each grid point. (Integer > 0; default = 100).
	SCALE (value2)	Scale factor for maximum allowable contact force. $F_{max} = SCALE \cdot F_{last_check}$ (Integer > 0; default = 10)
	TYPE (value3)	Contact force limitation. (Character; default=ZERO)
	ZERO	0.0
	FMAXF	Update contact limit forces using new contact forces (where non allowable forces are not taken into account)

Remarks:

1. The DYPARAM, FORCE check takes up some CPU time and, therefore, do not make this value too small. Furthermore, when the check is performed at each cycle, the force will be too limited and the bag will not unfold. Recommended values are between 5 and 200. The same problems can occur for SCALE (value2). In case this value is too small, the bag will not unfold either. The minimum value for air bags that should be used is about 5. The maximum is about 20. When this value is too big a difference will not be noticed. TYPE (value3)=ZERO is a bigger restriction. In some cases, TYPE=FMAXF might yield better results.

DYPARAM,COSUBMXT

Time-dependent Subcycle Limit in Euler/Lagrange Coupling - SOL700

Defines the maximum number of subcycles that can occur in Euler/Lagrange coupling. During a subcycle, the geometry of the coupling surface is not updated. This number can vary in time and is given by a table. Used in SOL 700 only.

Format:

DYPARAM,COSUBMXT,value

Example:

DYPARAM,COSUBMAT,10

Descriptor	Meaning
value	TABLED1 ID that specifies for each time the maximum number of time steps between updating the coupling surface geometry in the coupling calculations. (Integer > 0; required)

Remarks:

1. Updating the coupling geometry takes a lot of CPU time. Subcycling gives substantial savings in CPU time for coupled calculations.
2. The smaller the value of this parameter, the greater the accuracy of the analysis and the greater the cost. Conversely, larger values offer significant CPU savings, but very large values give incorrect results.
3. If the geometry of the coupling surface is changing rapidly, smaller values on the table should be used.

DYPARAM,COUFRIC

Coupling Surface Friction for Nonmetallic Eulerian Solids - SOL700

Defines the Coulomb friction scheme. Please check DYPARAM,COHESION too. Used in SOL 700 only.

Format:

DYPARAM, COUFRIC, value

Example:

DYPARAM, COUFRIC, NO-METAL

Descriptor	Meaning
value	Flag for COUFRIC usage. (Character; default=METAL)
METAL	A tensile condition will result in zero load on the structure part on cohesive coupling.
NO-METAL	a tensile load was applied in tensile condition on cohesive coupling.

Remarks:

1. Only used when Coulomb friction coefficients have been specified for a COUPLE entry.

DYPARAM,COHESION

Cohesion for Coulomb Friction - SOL 700

Allows friction and sticking during tensile conditions at the coupling surface.

Format:

DYPARAM,COHESION,MAXSTRS,FRIC,REFVEL

Example:

DYPARAM,COHESION,8.0e+10,8.0e+5,20.0

Descriptor	Meaning
MAXSTRS	Maximal normal stress. Allows tensile stresses at the coupling surface as long as the normal stress does not exceed MAXSTRS. (Real > 0)
FRIC	Friction stress under tensile conditions. (Real > 0)
REFVEL	Reference value for velocity. (Real > 0)

Remarks:

1. Only used when coulomb friction coefficients have been specified for a COUPLE entry.
2. During tension any relative tangential velocity between coupling surface and Eulerian material will yield a shear stress whose magnitude equals:

$$\text{FRIC} \cdot \min\left(1, \frac{V_{\text{rel,tangential}}}{\text{REFVEL}}\right)$$

This is a viscous-like friction law.

3. This shear force opposes the relative tangential movement along the coupling surface.

DYPARAM, DUMSEGS

Creating dummy segments for the auto coupling approach - SOL700

Specifies how holes in the coupling surface are meshed. It is only used for the auto coupling approach. This approach is activated by DYPARAM, AUTOCOUP.

Format and Example:

DYPARAM,DUMSEGS,INTERNAL

Descriptor	Meaning
INTERNAL	Meshes internal holes with a triangular dummy segment mesh [YES,NO] Default NO

Remarks:

- With the auto coupling method, holes are partially or completely meshed with dummy segments. With option NO the standard method of creating dummy segment is used. For details see chapter 8 of the SOL700 User's Guide. In this standard method holes are partially meshed with dummy segments. For an example refer to Figure 20 of Chapter 8. For holes that rotate, have arbitrary form or show large deformations, the standard method of creating dummy segments can cause the simulation to terminate with messages like

```
%E-P4307807-P4_SUBELM_MESH_CURVED_HOLES, , ,(13), DUMMY SURFACE  
CANNOT BE TRIANGULATED THIS IS PROGRAM ERROR. PLEASE CONTACT  
MSC. USING PARAM,DUMSEGS,ON CAN SOLVE THIS TRIANGULATION  
PROBLEM.
```

```
%E-P4309907-V4_SUBELEM_CONNECT_EXTENDED_GPS, , DUMMY SURFACE  
CANNOT BE TRIANGULATED THIS IS PROGRAM ERROR. PLEASE CONTACT  
MSC. USING PARAM,DUMSEGS,ON CAN SOLVE THIS TRIANGULATION  
PROBLEM.
```

In that case PARAM,DUMSEGS,ON can maintain a good dummy surface. The quality of the dummy surface can be checked by requesting a coupling surface archive. This is done by using the CPLSURFS command and by putting option DUMMY of DYPARAM, AUTOCOUP to ON. For all holes that are not internal holes, the standard method is used.

DYPARAM,ELDLTH

- SOL700

Print initial time step sizes for elements in the first cycle in SOL 700.

Format:

DYPARAM,ELDLTH,<value>

Example:

DYPARAM,ELDLTH,1

Describer	Meaning
value	Number of elements to be output. (Integer)
0	First 100 elements with the smallest time step sizes are printed.
1	The governing time step sized for all elements are printed. (Default)

DYPARAM,EULERCB

Euler Domain Decomposition Control - SOL700

Divides a Euler domain into several cubes. For SOL 700 only

Format:

DYPARAM,EULERCB,NELCUBE,NBX,NBY,NBZ

Example:

DYPARAM,EULERCB,2000,2,2,2

Descriptor	Meaning
NELCUBE	The number of elements per cube. (Integer; Default = 0)
NBX	The number of cubes in the x-direction. See Remark 2. (Integer; Default = 0)
NBY	The number of cubes in the y-direction. (Integer; Default = NBX)
NBZ	The number of cubes in the z-direction. (Integer; Default = NBX)

Remarks:

1. By setting NELCUBE equal to 2000 optimal use is made of memory caching during Euler computation. This may give a speedup of 1.5. When using adaptive meshing with dmp additional Euler cubes are created during the simulation. To keep the total number of cubes that are created within bounds the initial number of cubes should be limited to 100.
2. Defining NBX overrules the definition of NELCUBE.
3. Only supports Euler domains created by MESH,BOX. Limitations are:
 - No PEULERx/CHEXA's
 - No FLOW or BARRIER that use the BCID option are allowed. All FLOW and BARRIER entries must use either the DIR option or the XMIN,YMIN,ZMIN, XMAX,YMAX,ZMAX option to define the Euler MESH boundaries.

DYPARAM,EULERPR**Euler Cube Partition Control - SOL700**

Divides a Euler domain into several cubes. For SOL 700 only

Format:

DYPARAM,EULERPR,PROCDIR,NPX,NPY,NPZ

Example:

DYPARAM,EULERPR,USER,2,2,2

Descriptor	Meaning
PROCDIR	This directive controls the way cubes are distributed across processors. The effect can be checked by checking the Eulerian output variable PARTITION. (Character; Default = X)
X	Partition in global X direction first.
Y	Partition in global Y direction first.
Z	Partition in global Z direction first.
USER	Define user defined partitioning
SIMPLE	Partition Euler cubes in a simple pattern.
ORB	Partitions using orthogonal recursive bisection
NPX	The number of cubes in the x-direction. Required for PROCDIR=USER. (Integer; Default = 0)
NPY	The number of cubes in the y-direction. (Integer; Default = NPX)
NPZ	The number of cubes in the z-direction. (Integer; Default = NPX)

Remarks:

1. There are several ways to distribute cubes across processors. Some ways may lead to bad load balancing. To avoid this it is possible to control the way Euler cubes are distributed across processors by defining PROCDIR.
2. When option PROCDIR=USER, the values for NBX, NBY and NBZ must be such that NBX is equal or a multiple of NPX, NBY is equal or a multiple of NPY and NBZ is equal or a multiple of NPZ. Also for this option, NPX*NPY*NPZ must be equal to the number of processors used in the cluster.
3. For option PROCDIR=SIMPLE, the values NBX, NBY and NBZ on DYPARAM,EULERCB must be such that NBX*NBY*NBZ is equal or a multiple of the number of processors used. For instance, if the number of processors in the cluster is 4, NBX*NBY*NBZ must be equal to either 4 or 8 or 12, etc. Otherwise the calculation will terminate prematurely with an error message.
4. Only supports Euler domains created by MESH,BOX. Limitations are:
 - No MESH,ADAPT and no PEULERx/CHEXA's

- No FLOW or BARRIER that use the BCID option are allowed. All FLOW and BARRIER entries must use either the DIR option or the XMIN,YMIN,ZMIN, XMAX,YMAX,ZMAX option to define the Euler MESH boundaries.
5. With the orthogonal recursive bisection method the mesh is repeatedly cut in half. Each cut is along a coordinate direction and the direction of the cut is chosen to minimize the communication cost. To illustrate this method consider for example a mesh with 200,150 and 50 elements in respectively the x,y and z-direction. When split in x-direction the number of Euler faces at the split will be $150*50 = 7500$. When splitting across the y-direction it will be $200*50=10000$ and for the z-direction it will be $200*150=30000$. Since the x-split has the smallest number of faces at the split, the x-split has the smallest communication cost. Therefore the ORB scheme will select the x-split. At the x-split there are two mesh parts. Both have $100*150*50$ elements. Now a split in y-direction will give the minimal communication costs. This process of bisecting is continued until the number of sub meshes equals the number of CPU's.

DYPARAM,EULTRAN

Switch for Euler Transport Scheme of the Multi-Material Solver and the Single Material Strength Solver - SOL700

Sets the definition of the face velocity used in the transport scheme of the Multi-material solver and the single material strength solver. Used in SOL 700 only.

Format:

DYPARAM,EULTRAN,option1,option2

Example:

DYPARAM,EULTRAN,AVERAGE,FAIL

Descriptor	Meaning	
Option1	IMPULSE	The face velocity is impulse weighted. (Character; Default = IMPULSE)
	AVERAGE	The face velocity is a simple average.
Option2	NOFAIL	Failure is transported. See Remark 5. and 6. (Character; Default = NOFAIL)
	FAIL	Failure is transported. See Remark 5. and 6.

Remarks:

1. The default value of IMPULSE is sufficient for most Euler problems. Especially problems where the reference density of the different materials varies widely (e.g., orders of magnitude) are required to use the default option.
2. In case the IMPULSE option (default) is used, the Euler transport scheme computes that the face velocity uses an impulse weighted average of the material velocity in the left and the right element adjacent to the face.
3. In case the AVERAGE option is used, the Euler transport scheme computes the face velocity as one-half times the sum of the material velocity in the left and the right element adjacent to the face.
4. Does not apply to the single material hydrodynamic solver and the Roe solver.
5. The option FAIL requires a failure model for at least one Eulerian material. In case of the default NOFAIL then failed Euler material can support shear stress again as soon as new material enters the Euler element. Thus the information that part of the material inside the Euler element has failed is lost. The option FAIL activates transport of fail fraction and thereby keeps track of material that has failed. In this way only the failed part of the element can no longer supports shear stresses. In more detail, the yield stress in the material is scaled by $(1-failfrac)$, where failfrac denotes the fail fraction of the material. The fail fraction of the first material in an element can be retrieved from Euler archive or time-history results files in the variable DAMAGE. The value of fail fraction DAMAGE is between zero and one.
6. Option FAIL cannot be combined with the Johnson-Cook failure model (FAILJC).

DYPARAM,EUSUBMAX

- SOL700

Defines the maximum number of subcycles that can occur in the Euler solver. During a subcycle, the Euler computations are skipped. Used in SOL 700 only.

If coupling surface computations are more expensive than Euler computations than use of PARAM,COSUBMAX should be considered first. In that case the optimal setting of EUSUBMAX is LINKCS=BOTH and DFVUMAX is blank.

As with PARAM,COSUBMAX use of EUSUBMAX can lead to loss of accuracy for certain simulations. It is recommended to validate the use of EUSUBMAX by comparing the difference in results between using EUSUBMAX and not using EUSUBMAX for some typical target simulations.

Format

DYPARAM,EUSUBMAX,NSUBMAX,DFVUMAX,LINKCS

Example:

DYPARAM,EUSUBMAX,5

Descriptor	Meaning
NSUBMAX	NSUBMAX, The maximum number of time steps between updating Euler variables. (Integer ≥ 0 ; Default = 0)
DFVUMAX	Maximum allowed increase in uncovered volume fraction between to subsequent Eulerian computations. (Real > 0 ; Required when COSUBMAX is present. Leave blank when COSUBMAX is not present.)
LINKCS	Specifies interactions between Euler subcycling and coupling surface subcycling. EUSUBMAX can be used with COSUBMAX. LINKCS specifies how the two subcycling processes influence each other. (Character; Default = BOTH)
BOTH	The Euler computations will not be skipped when the coupling surface computations has been done. On the other hand if an Euler computation is to be done, also a coupling surface computation is done. The number of time that the Euler computation I skipped equals the number of times that the coupling surface computation has been done.
COUPLE	If an Euler computation is to be done, also a coupling surface computation is done.
EULER	If a coupling surface computation is to be done, also a Euler computation is done.
NONE	The Euler subcycling and coupling surface subcycling are independent. DFVUMAX has to be defined.

Remarks:

1. Updating the coupling geometry takes a lot of CPU time. Subcycling gives substantial savings in CPU time for coupled calculations.
2. The smaller the value of DFVUMAX, the greater the accuracy of the analysis and the greater the cost. Conversely, larger values offer significant CPU savings, but very large values give incorrect results.
3. When the Euler time step is considerably larger than the Lagrange time step it can be worthwhile to skip the Euler computation for several cycles. This can reduce computational costs considerable.
4. Skipping the Euler computation for several cycles, leads to postponed time steps. When the Euler computations are done these have to be taken into account by fluxing with an accumulated time step. Skipping the Euler computation is stopped as soon as the accumulated time step will become larger than the stable Euler time step. The total number of skipped cycles is limited by NSUBMAX. NSUBMAX is required input.
5. In skipping the Euler computation also the movement of the coupling surface has to be monitored. To estimate this movement the change in uncovered fraction of the elements is used. If the change in uncovered volume fraction from one cycle to the other is larger than DFVUMAX than the Euler computation is not skipped that cycle. DFVUMAX has to be left blank if coupling surface subcycling is used. Coupling surface subcycling already monitors the movement of the coupling surface. Choosing too large values for DFVUMAX can make results inaccurate.
6. EUSUBMAX is only supported by the multi-material Euler solver and the standard single material Euler solver. It is not supported by the Roe solver.

DYPARAM,EUSUBCYC

Growth or Subcycling Interval in Euler Computations - SOL700

Controls the growth of the subcycling interval in Euler computations. Used in SOL 700 only.

Format:

DYPARAM,EUSUBCYC,value

Example:

DYPARAM,EUSUBCYC,100

Descriptor	Meaning
value	Maximum growth of the subcycling interval. (Integer > 0; default=1)

Remarks:

1. The subcycling algorithm automatically estimates the number of subcycles to be used. This is updated throughout the calculation. This parameter controls how much the number of subcycles can grow. For example, EUSUBCYC is set to 1, and the current number of time steps between updates of the Euler variables. If Dytran estimates that the subcycling interval should be 7, the subcycling interval is increased by 1 until a value of 7 is reached.
2. There is no control on the amount by which the subcycling interval can decrease.

DYPARAM,FAILDT

Element Time-step Based Failure Model - SOL700

Defines the property of a failure model where element failure occurs when the element's time step falls below the specified limit. Used in SOL 700 only

Format:

DYPARAM, FAILDT, value

Example:

DYPARAM, FAILDT, 1.0E-3

Descriptor	Meaning
value	Minimum time-step. (Real > 0.0; default=1.0E-20)

Remarks:

1. This failure model is valid for all Lagrangian solid (CHEXA) and shell (CQUAD4) elements.
2. All elements for which the time step falls below the specified value are removed from the computation.
3. Although it is not usually necessary to limit the element time-step for Lagrangian elements, there are occasions where specifying a minimum time-step can be advantageous for computational performance, for example, when adaptive contact is used.
4. Note that this parameter should be used with care as you may influence the results of the analysis when you set the time-step criterion to a too high value. You then run the risk that elements are removed from the analysis while they may still be relevant.

DYPARAM,FAILOUT

Failed Element Output Parameter - SOL700

Defines whether failed elements are written to the output file (ARCHIVES). Used in SOL700 only.

Format:

DYPARAM, FAILOUT, value

Example:

DYPARAM, FAILOUT, NO

Descriptor	Meaning
value	Flag for FAILOUT. (Character; default=YES)
NO	Failed elements are not written to the archive files.
YES	Failed elements are written to the archive files.

Remarks:

1. When the NO option is chosen, the archives are written out as one file per requested time step regardless of the number set in the SAVE command for the archive files that appear in the Case Control Section. Please check \$S700.

DYPARAM, FLOWMETH**Flow-Method Between Two Euler Domains Across Open Areas in Coupling Surfaces - SOL700**

Defines the method for simulating material flow between two Euler domains across open areas in coupling surfaces. Used in SOL 700 only.

Format:

DYPARAM, FLOWMETH, value

Example:

DYPARAM, FLOWMETH, POLPACK

Descriptor	Meaning
value	Flag for FLOWMETH. (Character; default=NO)
POLPACK	The facets in the coupling surfaces that represent an open area are subdivided into smaller facets, with each connecting exactly to one Euler element in the first Euler domain and to exactly one Euler element in the second Euler domain. Material flow takes place across these smaller, subdivided facets (POLPACKs). This is the most accurate method.
FACET	The facets in the coupling surfaces that represent an open area are not subdivided. Material flow takes place across the original facets. If these facets are too large in relation with the Euler elements, the method becomes inaccurate. Material flow across one facet can involve several Euler elements on both sides of the hole and averaging occurs.

Remarks:

1. This parameter applies to simulations where:
 - Two coupling surfaces share a common set of facets.
 - Each coupling surface has own Euler domain.
 - Material flows from one Euler domain into the other through the open area represented by the common set of facets.

Flow only occurs if:

- The common facets are defined as 'open', using PORFCPL.
- The common facets open up due to failure of a shell structure, using COUP1INT.

Examples simulations are:

- Holes between air bag compartments.
- Holes between containers filled with gas or liquid.
- Open area between the top of a fuel-tank baffle and the fuel-tank skin.

- Open area in-between wide straps inside an air bag.
 - Failure of walls in between aircraft wing compartments.
 - Failure of tank armor by a blast wave.
 - Etc.
2. The following table summarizes what input cards support the simulation of material flow between two Euler domains across open areas in coupling surfaces:

Euler Solver	Material flow through a coupling surface	FLOW-METHOD=POLPACK	FLOW-METHOD=FACET	PORFLCPL (velocity based)	PORFLCPL (Pressure based)	COUP1INT/ COUP1FL (Failure of shell elements creates the opening)
HYDRO	YES	YES	NO	YES	YES	Only for flow method = polpack
HYDRO-Roe solver-1 st Order	YES	YES	YES	YES	YES	YES
HYDRO-Roe solver- 2 nd Order	YES	YES	YES	YES	YES	YES
MMHYDRO	YES	YES	NO	YES	NO	Only for flow method = polpack
STRENGTH	NO	-	-	-	-	-
MMSTREN	YES	YES	NO	YES	NO	Only for flow method = polpack

3. The Euler domains are shown below with the support types for each:

	FLOW-METHOD=POLACK	FLOW METHOD = FACET
MESH → TYPE= ADAPT	YES	No
MESH → TYPE= BOX	YES	YES
Modeling of CHEXA elements	No	YES

4. A Euler domain is associated with a coupling surface by specifying the MESHID or SET1ID on the COUPLE option.
5. FLOWMETH = POLPACK has the following limitations:

The entries NSTGP and NSTEL on all MESH entries should be left blank. It is not allowed to specify for any MESH entry the starting element number or starting grid point number.

- There are restrictions on output requests.
- Flow faces and wallets are not supported while FLOWDEF is supported.
- Viscosity is not supported

A case where these limitations require the use of FLOWMETH = FACET is when the Euler elements are generated in Patran, not using the MESH option, and one or more of the following options is used:

- FLOW boundaries are defined on some Euler faces.
- WALLET boundaries are defined on some Euler faces.
- Viscosity is defined.

DYPARAM,GEOCHECK

Define Geometry Consistency Check - SOL700

This parameter forces a check of the geometry for consistent connectivity of the defined hexagonal elements and correction if needed. Used in SOL 700 only.

Format:

DYPARAM, GEOCHECK, value

Example:

DYPARAM, GEOCHECK, ON

Descriptor	Meaning
value	Flag for activating GEOCHECK. (Character; default=OFF)
ON	Geometry consistency check is performed.
OFF	No geometry consistency check is performed.

Remarks:

1. The defined geometry is checked for consistent connectivity of the hexagonal elements. If an inconsistency is detected, the connectivity is corrected. CFACE entries with references to elements that have been corrected are corrected as well.
2. If a hexagonal mesh is generated with other commercial preprocessors, this parameter can correct the connectivity of the hexagonal elements in case problems are encountered with face generation or volume computation.

DYPARAM,FASTCOUP

Fast Coupling Algorithm - SOL700

Defines the fast coupling algorithm. Used in SOL 700 only.

Format:

DYPARAM,FASTCOUP,option1,option2,option3,option4,option5,option6,option7

Example:

DYPARAM,FASTCOUP,INPLANE,FAIL

Descriptor	Meaning
Option1	NO Fast coupling interaction is turned off. (Character; Default = INPLANE)
	INPLANE Small offset for inplane coupling surface segments.
	NOOFFSET No offset for inplane coupling surface segments
Option2	NOFAIL No failure of the coupling surface. (Character; Default = NOFAIL)
	FAIL Failure of the coupling surface will be taken into account.
Option3	Euler check algorithm. (Character; Default = NOCHKEUL)
	CHKEUL Checking whether all segments of the coupling surface are fully inside Euler elements.
	NOCHKEU No checking L
Option4	Euler update algorithm. (Character; Default = ALL)
	NEARONLY Do the full coupling surface only in the first cycle. In subsequent cycles only update elements and faces that are near the coupling surface.
	ALL Do the full coupling surface computation each cycle.
Option5	Controls coupling surface computations when there are multiple cubes defined with DYPARAM,EULERCB. (Character; Default = NONE)
	NONE All coupling surface computations are done for each cube.
	FACES The face cover fraction computation is done in one go for all cubes. The other coupling computations are still done per cube.
	POLPK The polpack computation efficiently handles multiple cubes. The other coupling computations are still done per cube.
ALL	All coupling surface computations efficiently handle cubes.

Descriptor	Meaning
Option6	In the coupling surface computations it is determined which structural segments intersect the Euler mesh. This is done by looping across all structural segments. For dmp each cpu has only part of the Euler mesh. Therefore for many segments it is known beforehand that they will not intersect the Euler mesh of the cpu and can be skipped at an early stage of the computation. Option 6 activates this skipping. (Character; Default = ON)
OFF	For dmp each CPU goes over all segments.
ON	Each CPU skips all segments that do not intersect the Euler mesh of the CPU.
Option7	Each coupling surface segment has to be intersected with the Euler elements and Euler faces. For large Euler meshes a substantial amount of checking has to be done to find the intersecting elements and faces. By dividing the Euler mesh in search boxes the costs of this checking is kept to a minimum. Especially for large Euler meshes combined with large coupling surface this checking can become expensive and then the use of search boxes can significantly reduce the costs. For Euler meshes of less than 100000 elements the use of search boxes have little effect. (Character; Default = ON)
OFF	Search boxes are not used.
ON	Search boxes are used.

Remarks:

1. The fast coupling algorithm is always turned on by default. In order to use general coupling, option1=NO should be used.
2. When option1 is set to INPLANE or when option 1 is blank, a small offset is given to coupling surface segments that are on top of a face of an Eulerian element. This is done because coupling surfaces segments on Eulerfaces make the Euler element volume computation invalid. Also boundary conditions on these segments are not correctly imposed. The net effect of these problems is unpredictable. The problem can either run correctly, or remain stable but give false results or become instable.
3. Option2 can only be used in combination with either PARAM,LIMITER,ROE or MMHYDRO or MMSTREN. The coupling surface must consist of CQUADS and/or CTRIAs and a failure model for the material of the surface must be defined.
4. This parameter can only be used when the Eulerian mesh is aligned with the basic coordinate system axes.

DYPARAM,HYDROBOD**Hydro Body Force - SOL700**

Defines a body force for single hydro material in Euler. Used in MSC Nasran SOL 700 only.

Format:

DYPARAM,HYDROBOD,XACC,YACC,ZACC

Example:

DYPARAM,HYDROBOD,-300.,0.,150.

Descriptor	Meaning
XACC	X-acceleration (Real; Default = 0.0)
YACC	Y-acceleration. (Real; Default = 0.0)
ZACC	Z-acceleration. (Real; Default = 0.0)

Remark:

1. This parameter defines a constant body force load in Euler for single hydro material only.

DYPARAM,HGCMEMShell Membrane Hourglass Damping Coefficient
Parameters - SOL700

Defines the default membrane damping coefficient for shell elements. Used in SOL 700 only.

Format:

DYPARAM, HGCMEM, value

Example:

DYPARAM, HGCMEM,0.07

Descriptor	Meaning
value	Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3.)

Remarks:

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for shell elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

DYPARAM,HGCOEFF**Hourglass Damping Coefficient - SOL700**

Defines the global default hourglass damping coefficient. Used in SOL 700 only.

Format:

DYPARAM, HGCOEFF, value

Example:

DYPARAM, HGCOEFF, 0.14

Descriptor	Meaning
value	Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3.)

Remarks:

1. The default applies to all types of hourglass suppression methods and should be used unless there is good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficients are not explicitly defined on HGSUPPR entries or on a HGCMEM HGCWRP, HGCTWS, or HGCSOL entry.
3. If this entry is omitted, the default value of the hourglass damping coefficients is either equal to the default value of 0.1 or is equal to the value specified on a HGCMEM, HGCTWS, HGCWRP, or HGCSOL DYPARAM entry.
4. The value of the coefficients can be explicitly defined for each property by using an HGSUPPR entry.

DYPARAM,HGCSOL

Solid Hourglass Damping Coefficient - SOL700

Define the default damping coefficient for solid elements. Used in SOL 700 only.

Format:

DYPARAM, HGCSOL, value

Example:

DYPARAM, HGCSOL, 0.11

Descriptor	Meaning
value	Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3.)

Remarks:

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for solid elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

DYPARAM,HGCTWS

Shell Twisting Hourglass Damping Coefficient - SOL700

Defines the default twisting damping coefficient for shell elements. Used in SOL 700 only.

Format:

DYPARAM, HGCTWS, value

Example:

DYPARAM, HGCTWS, 0.02

Descriptor	Meaning
value	Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3 .)

Remarks:

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for shell elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

DYPARAM,HGCWRP

Shell Warping Hourglass Damping Coefficient - SOL700

Defines the default warping damping coefficient for shell elements. Used in SOL 700 only.

Format:

DYPARAM, HGCWRP, value

Example:

DYPARAM, HGCWRP, 0.0

Descriptor	Meaning
value	Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3.)

Remarks:

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for shell elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

DYPARAM,HGSHELL**Shell Hourglass Suppression Method - SOL700**

Defines the default hourglass suppression method for shell elements. Used in SOL 700 only.

Format:

DYPARAM, HGSHELL, value

Example:

DYPARAM, HGSHELL, DYNA

Descriptor	Meaning
value	Type of HGSHELL. (Character; default=See remark 2.)
FBV	Flanagan-Belytschko viscous hourglass damping
DYNA	Viscous hourglass damping

Remarks:

1. The type of hourglass suppression method defined on this entry is used as the default whenever the type fields on the HGSUPPR for shell properties are left blank.
2. If this entry is omitted, the default suppression method used for shell elements is either FBV or the default method defined on the DYPARAM,HGTYPE entry.

DYPARAM,HGSOLID

Solid Hourglass Suppression Method - SOL700

Defines the default hourglass suppression method for solid elements. Used in SOL 700 only.

Format:

DYPARAM, HGSOLID, value

Example:

DYPARAM, HGSOLID, FBS

Descriptor	Meaning
value	Type of HGSOLID. (Character; default=See remark 2.)
FBS	Flanagan-Belytschko stiffness hourglass damping
DYNA	Viscous hourglass damping

Remarks:

1. The type of hourglass suppression method defined on this entry is used as the default whenever the type fields on the HGSUPPR for solid properties are left blank.
2. If this entry is omitted, the default suppression method used for solid elements is either FBS or the default method defined on the DYPARAM,HGTYPE entry.

DYPARAM,HGTYPE

Hourglass Suppression Method - SOL700

Defines the default type of hourglass suppression method. Used in SOL 700 only.

Format:

DYPARAM, HGTYPE, value

Example:

DYPARAM, HGTYPE, FBS

Descriptor	Meaning
value	Type of HGTYPE. (Character; default=See remark 2.)
FBS	Flanagan-Belytschko stiffness hourglass damping.
FBV	Flanagan-Belytschko viscous hourglass damping.
DYNA	Viscous hourglass damping.

Remarks:

1. The type of the hourglass suppression method defined on this entry is used as the default whenever the type fields in the HGSUPPR entries are left blank.
2. If this entry is omitted, the type can be defined on a DYPARAM,HGSHELL entry for shell elements, a DYPARAM,HGSOLID entry for solid elements, or on the HGSUPPR entries; otherwise the defaults apply. For shell elements the default is FBV; for solid elements, the default is FBS.

DYPARAM,HICGRAV

Gravity Used by HIC Calculations - SOL700

Defines the value of the gravity to be used by the HIC calculations. Used in SOL 700 only.

Format:

DYPARAM, HICGRAV, value

Example:

DYPARAM, HICGRAV, 980.7

Descriptor	Meaning
value	Gravity used by HIC Calculations. (Real > 0.0; default=9.80665)

Remarks:

1. The value set by this parameter will be used by all HIC output requests.
2. This parameter can only be set once in the input deck.

DYPARAM,HVLFAIL

Failure at Hydrodynamic Volume Limit - SOL700

Defines element failure on the hydrodynamic volume limit. Used in SOL 700 only.

Format:

DYPARAM, HVLFAIL, value

Example:

DYPARAM, HVLFAIL, YES

Descriptor	Meaning
value	Flag of HVLFAIL. (Character; default=NO)
YES	Element failure on hydrodynamic volume limit
NO	No element failure on hydrodynamic volume limit

Remarks:

1. Lagrangian elements (CHEXA) that have a material model with a failure model fail when the hydrodynamic volume limit is reached and the parameter is set to YES.
The elements can fail only when the following items are presented in the input:
 - a. The material model has a hydrodynamic volume limit (HVL).
 - b. A failure model is defined.
 - c. DYPARAM, HVLFAIL, YES
2. The hydrodynamic volume limit by default allows for 10% expansion.

DYPARAM,IMM

Initial Metric Method Formulation - SOL700

The option allows to specify the IMM method to be used. Used in SOL 700 only.

Format:

DYPARAM, IMM, OPTION1, OPTION2, VALUE1, VALUE2

Example:

DYPARAM, IMM, ZERO, YES, 1.0E-3, 1.0E-3

Descriptor	Meaning
OPTION1	Flag for element condition under compression. (Character; default=FULL)
FULL	While elements are under IMM condition, they will carry stresses when under compression.
REDUCED	While elements are under IMM condition they will carry a reduced stress when under compression. The relative area factor SMDFER is used to reduce the Young's modulus.
ZERO	While elements are under IMM condition they do not carry any compressive stresses. Use material damping to avoid excessive nodal velocities.
OPTION2	Flag for recalculation of IMM strain. (Character; default=See remark 2.)
NO or OFF	Do not recalculate IMM strains during the calculation.
YES or ON	Recalculate IMM strains during the calculation.
VALUE1	Start time of recalculation of IMM strains. See remark 3.(Real > 0.0; Default is 1.0E-3)
VALUE2	Times between recalculation of IMM strains. See remark 3.(Real > 0.0; Default is 1.0E-3)

Remarks:

- Method ZERO is best suitable when initially more than a couple of elements with zero or near zero area are present in the model.
- The default for Option2 depends on Option1.

Option1	Default for Option2
FULL	OFF
REDUCED	OFF
ZERO	ON

- When Option2 is OFF or NO, STREC and DTREC are neglected.

DYPARAM,INFOBJ

List the Generated BJOINs and Spotwelds - SOL700

Additional information about the BJOIN and spotweld connectivity will be listed in the output file. Used in SOL 700 only.

Format:

DYPARAM, INFOBJ, value

Example:

DYPARAM, INFOBJ, 0.14

Descriptor	Meaning
value	Flag of activation of writing BJOIN information. (Character; default=NO)
YES	Information is issued.
NO	Information is not issued.

Remarks:

1. The information listed is:
 - Grid point pairs forming a BJOIN or a spotweld.
 - BJOINs and spotwelds initially connected.

DYPARAM,INISTEP

Initial Time Step - SOL700

Defines the time step used at the start of the analysis. Used in SOL 700 only.

Format:

DYPARAM, INISTEP, value

Example:

DYPARAM, INISTEP, 1.0E-6

Descriptor	Meaning
value	Time step (in analysis time units) used for the first iteration. (Real > 0.0; required)

Remarks:

1. This parameter is required to start an analysis.

DYPARAM,LIMCUB

Contact Cube Sort Algorithm - SOL700

Defines the maximum number of cubes used to sort the grid points in a contact definition. Used in SOL 700 only.

Format:

DYPARAM, LIMCUB, value

Example:

DYPARAM, LIMCUB, 2300

Descriptor	Meaning
value	Maximum number of cubes. (Integer > 0; default=200)

Remarks:

1. Each slave node has to search for master nodes that are close enough to have potential contact. It is too expensive to have each slave node check each master node. To limit the number of checks, the space in which the nodes reside is subdivided into cubes. This subdivision is done so that the slave nodes have to check only the master nodes in their own cube and those in the neighboring cubes. The maximum number of cubes used to subdivide the space is equal to the value of LIMCUB.

DYPARAM,MATRMERG

Merges MATRIG and RBE2-FULLRIG Assemblies - SOL700

Merges MATRIG and/or RBE2-FULLRIG rigid bodies into a new FULLRIG assembly. Used in SOL 700 only.

Format:

```
DYPARAM, MATRMERG, FR<id1>, MR<id2>, MR<id3>, FR<id4>,...  
DYPARAM, MATRMERG, AUTO
```

Example:

```
DYPARAM, MATRMERG, FR1, MR2, MR6, MR7, FR4, MR8
```

Descriptor	Meaning
FR<id1>	Name of the new FULLRIG assembly. (Character; required.)
MR<idi> or FR<idi>	Names of MATRIG and/or RBE2-FULLRIG rigid bodies merged into a new FULLRIG assembly with name FR<id1>. No names can be supplied for the AUTO option.
AUTO	Automatic merge. See remark 2. (Character; required.)

Remarks:

1. FR<id1> must be a nonexisting RBE2-FULLRIG. The properties of FR<id1> (as mass, center of gravity, and moments of inertia) are computed by solver from the properties of each rigid body mentioned on the entry. Rigid body output can be asked for FR<id1>, and loads or rigid body constraints can be applied to FR<id1>. The other MATRIGs and RBE2-FULLRIGs mentioned on the MATRMERG entry disappear after they have been merged.
2. Instead of supplying rigid body names, the AUTO option can be used. After all the normal DYPARAM,MATRMERG and DYPARAM,MATRMRG1 entries have been applied, a DYPARAM,MATRMERG,AUTO merges all the resulting MATRIGs and RBE2-FULLRIGs which have common grid points into a new rigid assembly called FM<id>, where the id is a new FM number starting from 1. As it is not known at the start of an analysis how many FM assemblies will be created, no rigid body output can be asked for FM<id>, and no constraints or loads can be applied to FM<id>. The MATRIGs and RBE2-FULLRIGs, which have been merged by the AUTO option into a new FM<id> assembly, disappear.
3. To supply predefined properties for the merged assembly, DYPARAM,MATRMRG1 can be used, where the first rigid body mentioned on the entry must be an existing RBE2-FULLRIG or MATRIG.

DYPARAM,MATRMRG1

Merges MATRIG and RBE2-FULLRIG Assemblies - SOL700

Merges MATRIG and/or RBE2-FULLRIG rigid bodies into one existing MATRIG or RBE2-FULLRIG assembly with predefined properties. Used in SOL 700 only.

Format:

```
DYPARAM,MATRMRG1,MR<id1>,MR<id2>,MR<id3>,FR<id4>,...  
DYPARAM,MATRMRG1,FR<id1>,MR<id2>,MR<id3>,FR<id4>,...
```

Example:

```
DYPARAM,MATRMRG1,MR1,MR2,MR6,MR7,FR4,MR8
```

Descriptor	Meaning
MR<id1> or FR<id1>	Name of the existing MATRIG or FULLRIG assembly (must be an existing one). (Character; required.)
MR<idi> or FR<idi>	Names of MATRIG and/or RBE2-FULLRIG rigid bodies, which are merged with the existing MR<id1> or FR<id1> into a new MATRIG assembly, with name MR<id1> or FR<id1>. (Character; required.)

Remarks:

1. MR<id1> or FR<id1> must be an existing MATRIG or RBE2-FULLRIG, respectively. For a FULLRIG, the properties of FR<id1> (as mass, center of gravity and moments of inertia) are computed by solver from the properties of each rigid body mentioned on the entry. For a MATRIG, the mass of MR<id1> is either the predefined mass on the MATRIG (id1) entry or the predefined density on the MATRIG (id1) entry times the total volume of all MATRIG members in the MATRMRG1 entry. The center of gravity and moments of inertia of MR<id1> are either predefined on the MATRIG (id1) entry, or are otherwise computed from the properties of each rigid body on the entry. The other MATRIGs and RBE2 FULLRIGs mentioned on the MATRMRG1 entry disappear after they have been merged.

DYPARAM,MAXSTEP

Maximum Time Step - SOL700

Defines the maximum allowable time step. Used in SOL 700 only.

Format:

DYPARAM, MAXSTEP, value

Example:

DYPARAM, MAXSTEP, 1.E-3

Descriptor	Meaning
value	The maximum time step. (Real > 0.0; default=1.0E20)

Remarks:

1. If the time step calculated by solver is greater than MAXSTEP, the time step is set to MAXSTEP.

DYPARAM,MESHPLN

Mesh Density for Covering Rigid Planes - SOL700

Defines Mesh density for covering rigid planes. Used in SOL 700 only.

Format:

DYPARAM, MESHPLN, value

Example:

DYPARAM, MESHPLN, 4

Descriptor	Meaning
value	Rigid planes will be meshed with MESHPLN times MESHPLN dummy quad elements. (Integer > 0; default=3)

Remarks:

1. The default is sufficient in most cases.

DYPARAM,MINSTEP

Minimum Time Step - SOL700

Defines the minimum time step that causes the analysis to terminate. Used in SOL 700 only.

Format:

DYPARAM,MINSTEP,value

Example:

DYPARAM,MINSTEP,1.E-6

Descriptor	Meaning
value	When the time step is less than the MINSTEP value, the analysis terminates. (Real > 0.0; default=10% of DYPARAM,INISTEP)

Remarks:

1. When the elements become very distorted, in a poorly designed mesh for example, or when they have endured a very large distortion, the time step may drop dramatically. The analysis continues, however, and a lot of computer resources may be wasted. This option allows you to specify a minimum time step that causes the analysis to terminate.

DYPARAM,MIXGAS

Controls Updating of Gas Fractions - SOL700

Specifies whether the gas constants of the Euler material or of gas bags are updated based on the gas composition and temperature. Used in SOL 700 only.

Format:

DYPARAM,MIXGAS,value

Example:

DYPARAM,MIXGAS, YES

Descriptor	Meaning
value	Flag of MIXGAS. (Character; default=NO)
YES	The gas constants for the Euler material and any gas bags are recalculated based on temperature and gas composition.
NO	Euler and gas bag gas constants are not recalculated.

Remarks:

1. This parameter is only defined for use with GBAG gas bag definitions and/or the single-material Euler solver.
2. This parameter can be used in conjunction with INFLTR and INFLHB inflator definitions and with PORHOLE, PERMEAB, PORFGBG, and PERMGBG porosity definitions.
3. By default, DYPARAM,MIXGAS is set to YES if any INFLHB or INFLGAS entries are present.

DYPARAM,NZEROVEL

Auto Constrain Failed Nodes - SOL700

Set the velocity of a node to zero in case all attached elements have failed. Used in SOL 700 only.

Format:

DYPARAM, NZEROVEL, value

Example:

DYPARAM, NZEROVEL, YES

Descriptor	Meaning
value	Flag of NZEROVEL. (Character; default=NO)
YES	Perform check and set the velocity to zero if all attached elements have failed.
NO	Do not perform check.

Remarks:

1. This parameter applies only to nodes of Lagrangian elements.
2. Specifying NO reduces the CPU overhead time.
3. When the velocity of a node is set to zero, effectively the node is constraint, like an SPC or SPC1.
4. Special attention is necessary for the contact definition. If the failed node is not taken out of the contact, it behaves as a rigid boundary constraint.

DYPARAM,OLDLAGT

Use Collapsed Hexahedron Scheme for CTETRA - SOL700

Activate the collapsed hexahedron scheme as default for lagrangian CTETRA elements. Used in SOL 700 only.

Format:

DYPARAM,OLDLAGT,value

Example:

DYPARAM,OLDLAGT,1

Descriptor	Meaning
value	Flag of OLDSLGT. (Integer ≥ 0 ; default=See remark 1.)
0	Deactivate collapsed hexahedron scheme
1	Activate

Remarks:

1. The current default integration scheme for Lagrangian CTETRA elements use linear tetrahedron FE one. It is more consistent (in terms of accuracy) and efficient (both in memory and CPU time) compared with the collapsed hexahedron scheme. The old scheme based on collapsed hexahedron with reduced integration is deactivated. If the old scheme is activated, it is used as default. But, it is still possible to use the new scheme for CTETRA by using separate PSOLID with IN = 1 and ISOP = 1 combination.

DYPARAM,JWLDET

Hydro Body Force - SOL700

Specifies whether the blast wave of one explosive can ignite another explosive. Here it assumed that the explosives are modeled by a combination of EOSJWL and DETSPH entries.

Format:

PARAMJWLDET,OPTION

Example:

PARAM,JWLDET,NOLINK

Descriptor	Meaning
LINK	Multiple denotations with EOSJWL are LINKED. The detonation wave of one explosive can ignite another explosive.
NOLINK	Multiple denotations with EOSJWL are NOT LINKED. The detonation wave of one explosive cannot ignite another explosive
Default	NOLINK

Remark:

1. Option NOLINK: TDET is set to -1 for elements that have no JWL material. The "NOLINK" option is only valid with true JWL materials - not valid for the "Static Detonation/Ideal Gas" model.
2. Setting this parameter as "NOLINK" will prevent "sympathetic ignition". Each charge will ignite at the specified "TDET" in its own DETSPH card.

DYPARAM,LIMITER

Original Roe-Type Euler Solver Scheme - SOL700

Defines the type and the spatial accuracy of scheme used in the Euler solver based on the ideas of Prof. Philip Roe. Used in SOL 700 only.

Format:

DYPARAM,LIMITER,TYPE,OPTION

Example:

DYPARAM,LIMITER,ROE

Descriptor	Meaning
TYPE	Type of scheme. (Character; Default = ROE)
	ROE Roe solver for single hydrodynamic materials.
OPTION	blank Second order in space. (Character; Default = blank)
	NONE First order in space.

Remarks:

1. By default, when the parameter is not set, the solver is used that is defined on the PEULER or the PEULER1 entry. In case “2ndOrder” or “1stOrder” was defined on the PEULER or PEULER1 entry, the parameter setting has no effect.
2. By default, second order spatial accuracy is used. The temporal accuracy is automatically defined according to the spatial accuracy that you select.
3. Note that 2nd order spatial accuracy in the Roe solver involves the internal flow field only. We recommend that you use the full 2nd order improved fluid-and gas Euler solver. You can activate the improved solver by putting the “2ndOrder” field on the PEULER or PEULER1 entry.
4. When type ROE is defined, no void elements are allowed and it cannot be used in combination with EOSJWL. Also options concerning air-bag analyses are not supported.
5. For more details on fluid- and gas Euler solvers, refer to the Getting Started and the Theory Manuals.

DYPARAM,PARALLEL

Parallel Execution Information - SOL700

The option allows you to gather information on the parallel section. Used in SOL 700 only.

Format:

DYPARAM, PARALLEL, INFPAR, value

Example:

DYPARAM, PARALLEL, INFPAR, ON

Descriptor	Meaning
INFPAR	A report is written on the actual amount of work done at the reported parallel levels. (Character; required)
Value	Flag for INFPAR. (Character; default=OFF)
	ON Activate writing information.
	OFF Deactivate writing information.

Remarks:

1. A summary on the parallel operation when using the shared-memory mode can be requested by including a DYPARAM,PARALLEL,INFPAR,ON entry in the input file.
2. Currently, the information on the parallel sections is available for the shell solver only.

DYPARAM,PLCOVCUT**Pressure Cut Off Time - SOL700**

Defines time when PLCOVER is cut off. Used in SOL 700 only.

Format:

DYPARAM, PLCOVCUT, value

Example:

DYPARAM, PLCOVCUT, 3.E-3

Descriptor	Meaning
value	If there are one or more COUPLE definitions with a PLCOVER specified on the COUOPT entry, a cut off is applied to the PLCOVER until time = PLCOVCUT. From time = 0 to time = PLCOVCUT, the PLCOVER is cut off to the pressure in the intersected Eulerian element. For times greater than PLCOVCUT, the full PLCOVER is applied to the coupling surface. This parameter is useful in air-bag analyses, where PLCOVER is used to model the environment pressure. During the early stages of the deployment of the air bag, the pressure inside the bag may drop. Applying the full PLCOVER may lead to an unstable deployment of the air bag. (Real ≥ 0.0 ; default=0).

Remarks:

1. See also the COUPLE and COUOPT Bulk Data entries.

DYPARAM,PMINFAIL

Switches Failure at Spall Limit - SOL700

Defines Lagrangian solid element failure on reaching the spall limit. Used in SOL 700 only.

Format:

DYPARAM, PMINFAIL, value

Example:

DYPARAM, PMINFAIL, YES

Descriptor	Meaning
VALUE	Flag of PMINFAIL. (Character; default=NO)
YES	Element failure on spall limit
NO	No element failure on spall limit

Remarks:

1. Lagrangian elements (CHEXA) that have a material definition with a failure model will fail when the parameter is set to YES and the spall limit (minimum pressure) is reached, even when the other failure criterion is not yet reached.
2. The spall limit is set on the PMINC entry.

DYPARAM,RBE2INFO

Lists MATRIG and RBE2 Grid Points - SOL700

The grid points attached to MATRIG and RBE2 assemblies are listed to the output file. Used in SOL 700 only.

Format:

DYPARAM, RBE2INFO, value

Example:

DYPARAM, RBE2INFO, GRIDON

Descriptor	Meaning
VALUE	Flag of PBE2INFO. (Character; default=GRIDOFF)
GRIDON	Information is issued C
GRIDOFF	No information is issued

DYPARAM,RHOCUT

Global Density Cutoff Value - SOL700

Defines the minimum density for all Eulerian elements. Used in SOL 700 only.

Format:

DYPARAM, RHOCUT, value

Example:

DYPARAM, RHOCUT, 1.E-10

Descriptor	Meaning
value	Density cutoff. (Real > 0.0; default=See remark 4.)

Remarks:

1. Any Eulerian element with a density less than RHOCUT is considered to be empty. All of its variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
 - a. If the density of element B after transport is less than RHOCUT, then no transport is done.
 - b. If the density of element A after transport is less than RHOCUT, then all of the mass is transported to element B.
3. A reasonable value of RHOCUT is 1.E-5 times the initial density.
4. If only RHOCUT is defined, all Eulerian elements use the RHOCUT value as cutoff density. If RHOCUT is omitted, all Eulerian elements use a cutoff density automatically set to 1.E-5 times a characteristic density. For single-material Eulerian elements, this characteristic density is the reference density.

DYPARAM,RJSTIFF**Rigid-joint Stiffness - SOL700**

Defines the stiffness of a rigid joint. Used in SOL 700 only.

Format:

DYPARAM, RJSTIFF, value

Example:

DYPARAM, RJSTIFF, 100.

Descriptor	Meaning
value	Multiplication factor for the stiffness of all rigid joints. (Real > 0.0; default=1.0)

Remarks:

1. The absolute stiffness of rigid joints is calculated automatically by solver. The stiffness of joints is taken so that a stable solution is guaranteed. The stiffness calculation takes into account the fact that a rigid body can be constrained by more than one joint.
2. This parameter can be used to increase or decrease the stiffness of the joints. Care must be taken because too high a value may lead to an unstable calculation.

DYPARAM,SCALEMAS

Mass Scaling Definition - SOL 700

Defines the activation of mass scaling.

Format:

DYPARAM, SCALEMAS, DTMIN, MXPERC, STEPS.

Example:

DYPARAM, SCALEMAS, 1.0E-6, 100.0, 1

Descriptor	Meaning
DTMIN	Minimum allowable time step. (Real > 0.0; required)
MXPERC	Maximum percentage of added numerical mass with respect to original mass. (Real > 0.0; required)
STEPS	Number of steps. (Integer > 0; required)

Remarks:

1. Numerical mass is added to all Lagrangian solid, triangular, quadrilateral, rod, bar and beam elements such that its time step never becomes less than:

$$dt = STEPFACT * DTMIN.$$

Where,

dt = time step of calculation

STEPFACT = time step safety factor (see PARAM,STEPFACT)

DTMIN = value specified on the PARAM,SCALEMAS entry

If the added mass of a certain element exceeds the maximum percentage (MXPERC) of its original mass, no more mass will be added, and subsequently, the time step may decrease again.

2. The value of STEPS determines the checking frequency against the mass scaling criterion; the check is done for every defined number of STEPS. STEPS = 1 is recommended.
3. By requesting MSMASS in an ELOUT output request, the ratio of scaled mass to original mass of the elements can be retrieved. By making fringe plots of this parameter, a check can be made if mass has not been added in a critical area.
4. See *MSC Nastran Explicit Nonlinear (SOL 700) User's Guide*, Chapter 4: Special Modeling Techniques, [Mass Scaling](#) Definition for instructions on how to use this entry

DYPARAM,SHELLFRM

Sets the Default of the Shell Formulation - SOL700

Sets the default for the shell formulation for quadrilateral elements. Used in SOL 700 only.

Format:

DYPARAM, SHELLFORM, value

Example:

DYPARAM, SHELLFORM, BLT

Descriptor	Meaning
value	Flag for default of shell formulation. (Character; default=KEYHOFF)
BLT	The shell-formulation default is BLT.
KEYHOFF	The shell-formulation default if KEYHOFF.

Remarks:

1. The DYPARAM,SHELLFORM changes the default formulation for quadrilateral shell elements. All shell properties entries that do not explicitly define the formulation, use the default as specified on the DYPARAM entry.
2. Triangular shell elements have only one formulation (C0-TRIA). Therefore, the DYPARAM is ignored for triangular elements.

DYPARAM,SHELMSYS

Shell Element System Definition - SOL700

Defines the shell element system for the BLT shells. Used in SOL 700 only.

Format:

DYPARAM, SHELMSYS, value

Example:

DYPARAM, SHELMSYS, SIDE21

Descriptor	Meaning
value	Type of shell element system. (Character; default=MIDSIDES)
SIDE21	x-axis along side21.
MIDSIDES	x-axis connecting midpoints.

Remarks:

1. SIDE21 puts the x-axis along side21 of the element, whereas MIDSIDES puts the x-axis along the vector connecting the midpoints of the side14 and side32.
2. Using the SIDE21 option for the BLT shell will result in the same Belytschko-Lin-Tsay implementation as BELY.

DYPARAM,SHPLAST

Type of Plane-Stress Plasticity for Shells - SOL700

Specifies the type of calculation used to determine the plane-stress plasticity method for shells. Used in SOL 700 only.

Format:

DYPARAM, SHPLAST, value

Example:

DYPARAM, SHPLAST, VECT

Descriptor	Meaning
value	Type of Plain-stress plasticity for shells. (Character; default=ITER)
RADIAL	Noniterative, approximate radial return.
VECT	Iterative, vectorized with three iterations.
ITER	Nonvectorized iterations.

Remarks:

1. The RADIAL approach does not require iterations and, therefore, is the most efficient. It is, however, an approximation.
2. The other two approaches iterate to find the solution. ITER is the best since it takes as many iterations as are necessary. On vector machines, such as CRAY, this is inefficient since it cannot be vectorized. VECT always performs three vectorized iterations, which is more efficient. However, three iterations may not be enough, and inaccuracies could occur.

DYPARAM,SHSTRDEF

Composite Shell Stress and Strain Output Definition - SOL700

Specifies the default coordinate system for the stress and strain output of composite shells. Used in SOL 700 only.

Format:

DYPARAM, SHSTRDEF, value

Example:

DYPARAM, SHSTRDEF, ELEM

Descriptor	Meaning
value	Type of output. (Character; default=FIBER)
FIBER	Stresses and strains are output in the fiber and matrix directions.
ELEM	Stresses and strains are output in the element coordinate system.

Remarks:

1. The default setting can be overruled per property on a PCOMPA entry on the STRDEF field.

DYPARAM,SHTHICK

Shell-Thickness Modification Option - SOL700

Specifies whether or not the thickness of the shell changes with membrane straining. Used in SOL 700 only.

Format:

DYPARAM, SHTHICK, value

Example:

DYPARAM, SHTHICK, YES

Descriptor	Meaning
value	Flag of shell thickness modification. (Character; default=YES)
YES	Shell thickness is modified according to the membrane strain.
NO	Shell thickness is constant.

Remarks:

1. The YES option gives a true large-strain shell but requires some extra computation.
2. The NO option should give adequate results as long as the membrane strains are not very large (i.e., not more than 5–10%).
3. This option applies to all the formulations of the shell elements, except for the PCOMP. The thickness of PCOMP shell elements will always remain constant.

DYPARAM,SLELM

Store Shell Sublayer Variables - SOL700

Defines whether shell sublayer variables are to be stored in the element arrays. Used in SOL 700 only.

Format:

DYPARAM, SLELM, value

Example:

DYPARAM, SLELM, NO

Descriptor	Meaning
value	Flag of storing shell sublayer variables. (Character; default=YES)
YES	Store as an element variable.
NO	Do not store as an element variable.

Remarks:

1. This parameter applies only to shell elements.
2. The shell sublayer variables are primarily stored in sublayer arrays. They can be copied into the element arrays only for specific output purposes.
3. Specifying NO reduces the CPU overhead time.
4. Irrespective of the entry on this parameter, sublayer variables are accessible in the sublayer arrays. For example, requesting TXX1 retrieves the stress from the element array, whereas TXX01 retrieves it from the sublayer arrays.

DYPARAM,SMP,BATCHSIZ**SMP Batchsize and CPULOOP Definition - SOL700**

Define batch size and number of CPU loops, per entity type. Used in SOL 700 only.

Format:

DYPARAM, SMP, <entity>, BATCHSIZ, <value1>, CPULOOPS, <value2>

Example:

```
DYPARAM, SMP, BEAMS, BATCHSIZ, AUTOMAT
DYPARAM, SMP, BEAMS, BATCHSIZ, 256
```

Descriptor	Meaning
ENTITY	Define the batchsize for the following entities. (Character; required)
BEAMS	Beam elements
SHELLS	Shell elements
SOLIDs	Solid elements
CONTACT	Contacts
VALUE1	The batchsize for this entity (Integer number) or AUTOMAT (character). See Remarks 1.and 3. (Character or Integer >0; default = 63)
VALUE2	If value1 is defined as AUTOMATIC, the user may define the number of loops per CPU See Remark 2.(Integer > 0; default=2)

Remarks:

1. The batchsize may be defined per entity type. The SMP parts uses the batchsize to assign a certain number of batches to each CPU. Example:
 - Batchsize = 63 (default)
 - Number of Beam elements = 580
 - Number of CPUs = 4

As a results,

- Needs to process 10 batches ($9 \times 63 + 1 \times 13$)
- To spread this evenly over four CPUs, a solver increases this to 12 batches because then each CPU can process 3 batches.
- As a result, the batch size is recalculated for 12 batches to: batchsize = 49
- This means that:

CPU# 1,2,3 process three batches of 49 elements each ($3 \times 3 \times 49 = 441$ elements)

CPU# 4 processes two batches of 49 elements and one batch of 41

It is clear that the user-defined batchsize may not be used ultimately. The reason for this is that each CPU is allocated the same number of batches in order to optimize the work done by each processor.

2. When the batchsize is set to AUTOMAT, a solver computes an initial batchsize based on the number of batches processed by each CPU. By default, processes two batches per CPU.

In the example above, this means that based on:

- Batchsize = AUTOMAT
- Number of batches per CPU = 2 (default)
- Number of Beam elements = 580
- Number of CPUs = 4

As a result,

- Computes an initial batchsize of 72 ($=580/(4*2)$)
- Increases this batchsize to a higher number to make sure the number of batches fits in the fixed number of CPU's: batchsize = $72 + (580 - 4*2*72) = 72 + 4 = 76$ elements
- Based on this initial batchsize, the number of batches would be: $1 + (580-1)/76 = 8$ batches
- This fits nicely on four CPU's as expected
- The CPUs process:

CPU# 1,2,3 processes two batches of 76 elements each ($3 \times 2 \times 76 = 456$ elements)

CPU# 4 processes one batch of 76 elements and 1 batch of 48 elements.

3. By default, a solver does not use the AUTOMAT batchsize algorithm. Testing has shown that an initial batchsize of 63 yields the fastest results. This may vary depending on the problem simulated and the number of elements and may be worth adjusting.

The reason for this may be found in the fact, that with many elements in the model, the number of elements per batch increases significantly when the AUTOMAT batchsize algorithm is used. Each CPU will have to process two batches (by default) with many elements. Based on the cache memory, this is usually not optimal for a CPU processor. Shorter batchsizes (< 128) will do a better job.

DYPARAM,SMP,CPUINFO**SMP CPU Information - SOL700**

Define the CPU information per entity type. Used in SOL 700 only.

Format:

DYPARAM, SMP, <entity>, CPUINFO, <value1>

Example:

DYPARAM, SMP, CONTACT, CPUINFO, ON

Descriptor	Meaning
ENTITY	Define the CPUINFO for the following entities. (Character; required)
BEAMS	Beam elements
SHELLS	Shell elements
SOLIDs	Solid elements
CONTACT	Contacts
MEMBR	Membrane elements
EULT9	Eulerian Roe Solver
FSCOUP	Fast coupling algorithm
ALL	Output for all of the above
VALUE1	Flag for writing CPU information. (Character; required)
ON	Ask for parallel CPU time information C
OFF	No CPU time information (default)

Remarks:

1. By default, a solver does not give any SMP CPU timings per entity. In order to judge the efficiency of the different algorithms, this parameter may be used to judge the speedup acquired by processing on multiple CPUs.

DYPARAM,SNDLIM

Sound Speed Minimum Value - SOL700

Defines the minimum value for the speed of sound. Used in SOL 700 only.

Format:

DYPARAM, SNDLIM, value

Example:

DYPARAM, SNDLIM, 1.E-6

Descriptor	Meaning
value	Minimum value of speed of sound. (Real > 0.0; default=1.0E-3)

Remarks:

1. This parameter is used to avoid the possibility of division by zero in the time step calculation.
2. SNDLIM has the units of velocity.

DYPARAM,SPREMAP**1-D Spherical Symmetric Euler Archive Remap - SOL700**

Allows a 1-D spherical symmetric Euler archive importation to a 3-D simulation. Used in SOL 700 only.

Format:

DYPARAM, SPREMAP, X0, Y0, Z0, range

Example:

DYPARAM, SPREMAP, 0.0, 0.5, 0.5

Descriptor	Meaning
X0, Y0, Z0	X, Y, Z coordinate of the point at which the 1-D spherical symmetric mesh is remapped. (Real; default=0.0)
RANGE	Only material whose distance to (X0,Y0,Z0) is smaller than “range” will be initialized with the 1-D spherical symmetric Euler archive. (Real > 0.0; default = 1.0E20)

Remarks:

1. Since 1-D spherical symmetric simulations run much faster than 3-D simulation, it can save much CPU time to do the first part of the simulation with a 1-D axial symmetric mesh. Afterwards, the 1-D spherical symmetric Euler archive is imported into the 3-D simulation. By default, the 1-D spherical symmetric archive will not be expanded in 3-D. To enable this expansion, DYPARAM,SPREMAP has to be used. It is useful for blast wave simulations. The 1-D spherical symmetric simulation has to be terminated before the blast wave approaches any 3-D structure.
2. This import of Euler archives is done by means of the EID option in the pth file.
3. To generate axial symmetric meshes DYPARAM,SPHERSYM can be used.
4. The remap of 1-D spherical symmetric to axial symmetric Euler meshes is supported.

DYPARAM,SPHERSYM

Spherical Symmetric Analyses - SOL700

Enables an efficient and accurate 1D spherical symmetric solution for Eulerian materials. A much larger time step becomes possible by basing the time step only on the mesh-size in radial direction. Used in SOL 700 only.

Format:

```
DYPARAM, SPHERSYM, MESHTYPE, RADAXIS, PHI
```

Example:

```
PARAM, SPHERSYM, RECT, X, 2.0
```

Descriptor	Meaning
MESHTYPE	Two types of Euler meshes are supported. (Character; required)
SPHERIC	Axial symmetric meshes
RECT	Rectangular meshes
RADAXIS	Radial Axis. (Character; required.)
X	Global x direction
Y	Global y direction
Z	Global z direction
PHI	Only used for MESHTYPE = RECT. Used to creates a 1d Spherical mesh with angles +PHI/2 and -PHI/2. (Real ≥ 0 ; default=0.0)

Remarks:

1. Only available for Eulerian elements and does not support Lagrange elements. The effect of this parameter is not limited to the solvers. Also Euler archives will reflect the modified Euler mesh geometry.
2. The Euler mesh can already be symmetric but also a rectangular mesh comprising of one row of elements can be used. Using the angle specified by PHI this Euler mesh is mapped into a 1D spherical symmetric mesh.
3. The Euler mesh has to consist of one row of elements.
4. In the time step computation only the mesh-size in radial direction will be taken into account.

DYPARAM,STRNOUT

Shell Sublayer Strain Output - SOL700

Saves the total strains and equivalent effective stress (von Mises stress) at shell sublayers for output. Used in SOL 700 only.

Format:

DYPARAM, STRNOUT, value

Example:

DYPARAM, STRNOUT, YES

Descriptor	Meaning
value	Flag for writing strain results. (Character; default=YES)
YES	Save
NO	Do not save.

Remarks:

1. A limited output set saves memory.
2. Perfectly elastic materials only have the limited output set.
3. Total strain output for shell composite materials can be requested from the PCOMPA Bulk Data entry.

DYPARAM,TOLCHK

Tolerance for Fast Coupling - SOL700

Fast coupling requires that Euler elements are orthogonal. This means that normals of Euler element faces have to be in a coordinate direction. In practice there can be small errors in the geometry of the Euler element and Euler face normals do not always exactly point into a coordinate direction. Small deviations from the coordinate direction do not give problems and are allowed by fast coupling.

To check the direction of an Euler face, the face normal vector is projected onto the closest coordinate direction. If this projection is 1, the normal is exactly in the coordinate direction. When this projection is within a sufficient small tolerance of 1, the face can be handled by fast coupling. The tolerance used is TOLCHK. If the projection is smaller than 1-TOLCHK, the face cannot be handled by fast coupling and the analysis terminates. Options are then to slightly increase TOLCHK , write out double precision format in PATRAN, use general coupling, or use the MESH entry. Increasing TOLCHK too much can make the coupling surface computation less accurate. To keep the computation accurate, the maximal allowed value of TOLCHK is 1e-6. If DYPARAM,TOLCHK is not used, the tolerance used is 1e-14. Used in SOL 700 only.

Format:

DYPARAM, TOLCHK, value

Example:

DYPARAM, TOLCHK, 1e-10

Descriptor	Meaning
value	Tolerance in accepting faces by fast coupling. (Real > 0.0; required.)

DYPARAM,VDAMP

Dynamic Relaxation Parameter - SOL700

Controls the global damping in the dynamic relaxation. Used in SOL 700 only.

Format:

DYPARAM, VDAMP , value

Example:

DYPARAM, VDAMP , 0.001

Descriptor	Meaning
value	Dynamic relaxation parameter. (Real ≥ 0.0 ; default=0.0)
0.0	No dynamic relaxation.

Remarks:

1. The dynamic relaxation parameter is connected to the system natural frequency, ?, as $?=s??t$, where s denotes a percentage of critical damping. The damping occurs by factoring the velocities every time step as follows:

$$F_1 = (1-\beta)/(1+\beta)$$

$$F_2 = 1/(1+\beta)$$

$$v^{n+1/2} = F_1 v^{n-1/2} + F_2 a^n \beta t^n$$

where v is the velocity, a is the acceleration, and ? is the dynamic relaxation parameter.

DYPARAM,VELMAX

Maximum Velocity - SOL700

Defines the maximum velocity in Eulerian and Lagrangian meshes. Used in SOL 700 only.

Format:

DYPARAM, VELMAX, value, option

Example:

DYPARAM, VELMAX, 1.E6

Descriptor	Meaning
VALUE	Maximum velocity. (Real > 0.0; default=1.0E10)
OPTION	Flag for removing Euler elements. (Character; default=YES)
YES	Remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.
NO	Do not remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.

Remarks:

1. For Eulerian elements, VELMAX is applied to the velocity components separately. Therefore, the magnitude of the velocity vector is limited to VELMAX. For Lagrangian elements, all components are scaled down by the same factor to ensure that the maximal magnitude of the velocity vector does not exceed VELMAX.
2. Although it is not usually necessary to limit the velocity in Eulerian meshes, there are occasions in regions of near-vacuous flow for example, where specifying a maximum velocity can be advantageous. The same applies to Lagrangian meshes in contact regions for example. This parameter should be used with care.
3. Because very high velocities occur mostly in Eulerian elements with very small mass, the mass in these elements can be removed to keep the analysis stable. This option is not available for Lagrangian solid elements.
4. VELMAX must be greater than the minimum velocity specified by PARAM,VELCUT.

DYPARAM,VELMAX1**Maximum Translational and Angular Velocity - SOL700**

Defines the maximum translational and angular velocity in Eulerian and Lagrangian meshes. Used in SOL700 only.

Format:

DYPARAM, VELMAX1, VELMAXT, option, VELMAXA

Example:

DYPARAM, VELMAX1, 1.E6

Descriptor	Meaning	
VELMAXT	Maximum translational velocity. (Real > 0.0; default=1.0E20)	
OPTION	Flag for removing Euler elements. (Character; default=YES)	
	YES	Remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.
	NO	Do not remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.
VELMAXA	Maximum angular velocity. (Real > 0.0; default=1.0E20)	

Remarks:

- For Eulerian elements, VELMAXT is applied to the velocity components separately. Therefore, the magnitude of the velocity vector is limited to VELMAXT. For Lagrangian elements, all components are scaled down by the same factor to ensure that the maximal magnitude of the velocity vector does not exceed VELMAXT.
- VELMAXA is used to limit the angular velocity of grid points. By default, the angular velocities are not limited. In general, there is no need to limit angular grid point velocities.
- Although it is not usually necessary to limit the velocity in Eulerian meshes, there are occasions in regions of near-vacuous flow for example, where specifying a maximum velocity can be advantageous. The same applies to Lagrangian meshes in contact regions for example. This parameter should be used with care.
- Because very high velocities occur mostly in Eulerian elements with very small mass, the mass in these elements can be removed to keep the analysis stable. This option is not available for Lagrangian solid elements.
- VELMAXT must be greater than the minimum velocity specified by PARAM, VELCUT.
- DYPARAM, VELMAX, and DYPARAM, VELMAX1 can be used in the same input deck. The values set by DYPARAM, VELMAX1 will overrule the values set by DYPARAM, VELMAX.

For example, when using

DYPARAM, VELMAX, 1000, YES

```
DYPARAM,VELMAX1,,YES,1e+10
```

In the OUT file shows:

```
%I-INIT 02-p3_set_velmaxat, , ,(13),  
SUMMARY VELMAX SETTINGS:  
MAXIMAL VALUE TRANSLATIONAL VELOCITY = 0.100000E+04  
MAXIMAL VALUE ANGULAR VELOCITY = 0.100000E+11  
REMOVE EULERIAN MASS WHEN VELOCUITY EXCEEDS LIMIT: YES
```

DYPARAM,VISCPLAS

Use Overstress Formula to Update Strain-rate Dependent Plasticity - SOL700

Activate the overstress formula to update strain-rate dependent plasticity. This formula is normally used for viscous-plastic material. Used in SOL 700 only.

Format:

DYPARAM, VISCOPLAS, value

Example:

DYPARAM, VISCOPLAS, 1

Descriptor	Meaning
VALUE	Flag of updating overstress formula. (Integer ≥ 0 ; default=0)
0	Use scaling-up scheme
1	Use overstress formula

Remarks:

1. The strain rate dependent plasticity is normally calculated by scaling up the basic yield stress without strain rate effect. Then the trial stresses are mapped back to the scaled-up yield surface. This algorithm may lead to premature instability. Another technique is to calculate the so-called viscous-plastic strain rate using “overstress” formula. And then the stresses are updated based on this viscous-plastic strain. This technique seems to be more stable than the previous one. For shell elements, this option works when combined with DYPARAM,SHPLAST,RADIAL. Only MATEP and Johnson-Cook models are supported. For solid elements, this option works only for MATEP.
2. For shell elements, DYPARAM,VISCOPLAS,1, in combination with DYPARAM,SHPLAST,RADIAL, will use consistent plane stress plasticity algorithm both for strain rate dependent and independent plasticity. This new algorithm is more accurate than 3-D approach.

ECHOOFF Deactivate Printed Echo

Marks the point or points in the input file to deactivate printed echo of the input file.

Format:

1	2	3	4	5	6	7	8	9	10
ECHOOFF									

Example:

ECHOOFF									
---------	--	--	--	--	--	--	--	--	--

Remarks:

1. This entry may also be used in the Executive Control and Case Control Sections. It is described in the Case Control command, [ECHO \(Case\), 310](#).
2. The companion to this entry is the ECHOON entry.
3. The ECHOOFF command is BEGIN BULK or BEGIN SUPER = n dependent. Its last time used does carry over between changes in BEGIN BULK or BEGIN SUPER = n commands. Also, the Case Control ECHO command should be above subcase level and the following is recommended: ECHO=UNSORT.

ECHOON**Activate Printed Echo**

Marks the point or points in the input file to activate printed echo of the input file.

Format:

1	2	3	4	5	6	7	8	9	10
ECHOON									

Example:

ECHOON									
--------	--	--	--	--	--	--	--	--	--

Remarks:

1. This entry may also be used in the Executive Control and Case Control Sections. It is described in the Case Control command, [ECHO \(Case\), 310](#).
2. The companion to this entry is the ECHOOFF entry.
3. The ECHOON command is BEGIN BULK or BEGIN SUPER = n dependent. Its last time used does carry over between changes in BEGIN BULK or BEGIN SUPER = n commands. Also, the Case Control ECHO command should be above subcase level and the following is recommended: ECHO=UNSORT.

EIGB**Buckling Analysis Set**

Defines data needed to perform buckling analysis.

Format:

1	2	3	4	5	6	7	8	9	10
EIGB	SID	METHOD	L1	L2	NEP	NDP	NDN		
	NORM	G	C						

Example:

EIGB	13	INV	0.1	2.5	2	1	1		
		MAX							

Descriptor	Meaning	
SID	Set identification number. (Unique Integer > 0)	
METHOD	Method of eigenvalue extraction. (Character: “INV” for inverse power method or “SINV” for enhanced inverse power method.)	
L1, L2	Eigenvalue range of interest. (Real, L1 < L2)	
NEP	Estimate of number of roots in positive range not used for METHOD = “SINV”. (Integer > 0)	
NDP, NDN	Desired number of positive and negative roots. (Integer>0; Default = 3*NEP)	
NORM	Method for normalizing eigenvectors. (Character: “MAX” or “POINT”; Default = “MAX”)	
	MAX	Normalize eigenvectors to the unit value of the largest component in the analysis set. (Default).
	POINT	Normalize eigenvectors to the unit value of the component defined in G and C fields. The value for NORM defaults to MAX if the defined component is zero.
G	Grid or scalar point identification number. Required only if NORM=“POINT”. (Integer > 0)	
C	Component number. Required only if NORM=“POINT” and G is a geometric grid point. (1 ≤ Integer ≤ 6)	

Remarks:

1. The EIGB entry must be selected with the Case Control command METHOD = SID.
2. Each eigenvalue is the factor by which the prebuckling state of stress is multiplied to produce buckling in the shape defined by the corresponding eigenvector.

3. The continuation entry is optional. If the continuation is not specified, than NORM = “MAX” normalization is performed.
4. If NORM = “MAX”, components that are not in the analysis set may have values larger than unity.
5. The SINV method is an enhanced version of the INV method. It uses Sturm sequence techniques to ensure that all roots in the range have been found. It is generally more reliable and more efficient than the INV method.
6. Convergence is achieved at 10^{-6} .
7. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the L1 and L2 fields are left blank, an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If NDP and NDN are set to 1, there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.
8. If Modules are present then this entry may only be specified in the main Bulk Data section.

EIGC**Complex Eigenvalue Extraction Data**

Defines data needed to perform complex eigenvalue analysis.

Format:

1	2	3	4	5	6	7	8	9	10
EIGC	SID	METHOD	NORM	G	C	E	ND0		

The following continuation is repeated for each desired search region. (J = 1 to n, where n is the number of search regions.)

	ALPHAAJ	OMEGA AJ	ALPHABJ	OMEGABJ	LJ	NEJ	NDJ		
--	---------	----------	---------	---------	----	-----	-----	--	--

Alternate Format for Continuation Entry for Block Complex Lanczos:

	ALPHAAJ	OMEGA AJ	MBLKSZ	IBLKSZ	KSTEPS		NJi		
--	---------	----------	--------	--------	--------	--	-----	--	--

Examples:

EIGC	14	CLAN							
		+5.6					4		
EIGC	15	HESS					6		

Descriptor	Meaning
SID	Set identification number. (Unique Integer > 0)
METHOD	Method of complex eigenvalue extraction. (Character: "INV", "HESS", or "CLAN")
INV	Inverse power.
HESS	Upper Hessenberg. For linear perturbation of ANALYSIS= DCEIG with large displacement, please don't use HESS. See Remarks 2. and 3.
CLAN	Complex Lanczos. For linear perturbation of ANALYSIS= DCEIG with large displacement, CLAN is recommended. See Remark 8.
IRAM	Implicitly Restarted Arnoldi Method. See Remark 13.
NORM	Method for normalizing eigenvectors. See Remark 7. (Character: "MAX" or "POINT"; Default = "MAX")
MAX	Normalize the component having the largest magnitude to a unit value for the real part and a zero value for the imaginary part.
POINT	Normalize the component defined in fields 5 and 6 to a unit value for the real part and a zero value for the imaginary part. The value for NORM defaults to "MAX" if the magnitude of the defined component is zero.
G	Grid or scalar point identification number. Required if and only if NORM = "POINT". (Integer > 0)

Descriptor	Meaning
C	Component number. Required if and only if NORM = "POINT" and G is a geometric grid point. ($0 \leq \text{Integer} \leq 6$)
E	Convergence criterion. (Real ≥ 0.0 . Default values are: 10^{-4} for METHOD = "INV", 10^{-15} for METHOD = "HESS", E is machine dependent for METHOD = "CLAN".)
MBLKSZ	Maximum block size. See Remark 11. (Default = 7; Real ≥ 0.0)
IBLKSZ	Initial block size. See Remark 11. (Default = 2; Real ≥ 0.0)
KSTEPS	Frequency of solve. (Default = 5; Integer > 0)

Table 9-11 Relationship Between METHOD Field and Other Fields

Field	METHOD Field		
	HESS	INV	CLAN
NDj (Integer > 0)	Desired number of eigenvalues and eigenvectors. (No Default)	Desired number of roots and eigenvectors in j-th search region. (Default = $3 * NEj$)	Desired number of roots and eigenvectors to be extracted at j-th shift point. (No Default)
ALPHAAj OMEGAAj Real and imaginary parts of Aj in radians per unit time. (Real).	Not used	End point Aj of j-th search region in complex plane. (Default = 0.0)	j-th shift point. (Default = 0.0)
ALPHABj OMEGABj Real and imaginary parts of Bj in radians per unit time. (Real).	Not used	End point Bj of j-th search region in complex plane. (Default = 0.0)	See alternate definitions in fields MBLKSZ and IBLKSZ below; see also Remark 11.
Lj (Real > 0.0)	Not used	Width of j-th search region. (Default = 1.0)	See alternate definitions in fields MBLKSZ and IBLKSZ below; see also Remark 11.
NEj (Integer > 0)	Not used	Estimated number of roots in j-th search region.	Not used
MBLKSZ For block CLAN only			Maximum Block Size Default = 7
IBLKSZ For block CLAN only			Initial Block Size Default = 2

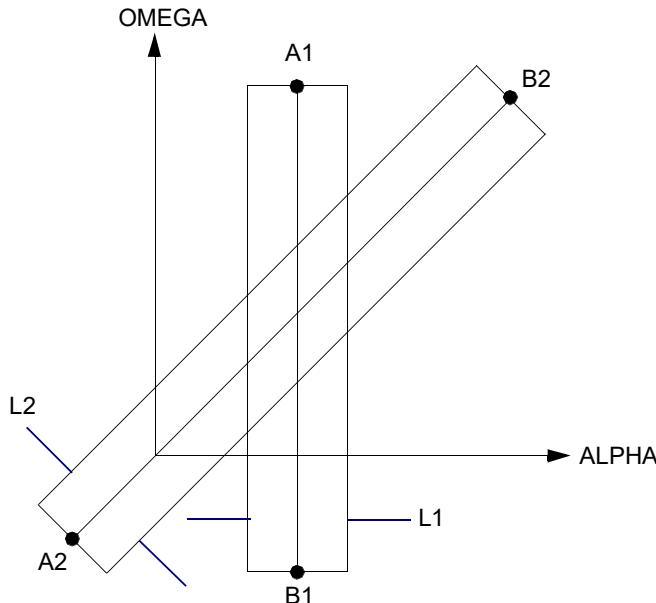


Figure 9-90 Sample Search Regions

Remarks:

1. The EIGC entry must be selected in the Case Control Section with the command CMETHOD = SID. Methods of solution are also controlled by SYSTEM(108) on the NASTRAN statements (described in [Executing MSC Nastran, 1](#)).

SYSTEM(108)	Specification
0 (Default)	QZ HESS method or CLAN block Lanczos, as selected by EIGC entry or equivalent PARAM input. Default value.
1	Force Householder QR (Hessenberg with spill, mass matrix must be nonsingular)
2	Force old single vector complex Lanczos
4	Force new block complex Lanczos
8	Debugging output for Lanczos methods
16	Turn off block reduction in block complex Lanczos
32	Turn off block augmentation in block complex Lanczos
64	Turn off full orthogonality in block complex Lanczos

SYSTEM(108)	Specification
128	Turn off preprocessing of initial vectors in block complex Lanczos
256	Force LR method (Hessenberg, no spill, mass matrix must be invertible)
512	Force QZ method
65536	Use semi-algebraic sort on imaginary part of roots (pre-V70.6 sort)

The word “force” above implies that the method selected by the system cell will be used even when another method has been selected on an EIGC entry. Sums of these values will produce two or more actions at once, when feasible. As the numbers get larger, the function is more developer-oriented than user-oriented.

- When using METHOD = “HESS”, the following should be noted:

The “HESS” method is generally more reliable and economical for small and moderate-size problems. It computes ND eigenvalues and eigenvectors.

For linear perturbation of ANALYSIS= DCEIG with large displacement (param, lgdisp, 1) in SOL 400, HESS may yield incorrect results. Therefore, please avoid to use HESS in linear perturbation analysis with large displacement.

- The EIGC entry may or may not require continuations as noted below.

- For the “HESS” method, continuations are not required; and their contents are ignored when present, except for ND1. However, it is recommended that continuations are not used.
- For the “CLAN” method when the continuation entry is not used a shift is calculated automatically. When a shift is input on the first continuation entry it is used as the initial shift. Only one shift is used. Data on other continuation entries is ignored.
- For METHOD = “INV”, each continuation defines a rectangular search region. Any number of regions may be used and they may overlap. Roots in overlapping regions will not be extracted more than once.
- For all methods, if no continuation is present, then ND0 must be specified on the first entry. If a continuation is present, then NDj must be specified on the continuation and not on the first entry.

- The units of ALPHAAJ, OMEGAAJ, ALPHABJ, and OMEGABJ are radians per unit time.
- DIAG 12 prints diagnostics for the inverse power method, the complex Lanczos method and the QZ HESS method.
- If METHOD = “HESS” and the LR or QR methods (non-default methods) are selected by system cell 108 the mass matrix must be nonsingular. The default QZ method does not require a nonsingular mass matrix.
- The normalized eigenvectors may be output with the SDISPLACEMENT and/or DISPLACEMENT Case Control commands.

8. When using METHOD = CLAN, the following should be noted. The modern CLAN method (default for METHOD entry of CLAN) has been enhanced to include a block complex Lanczos approach. This method is more reliable and will not accept inaccurate roots which the old method had a tendency to do. Thus, given the same input, the new method may often accept fewer roots. For continuity the old method has been maintained and may be selected by setting SYSTEM(108). For linear perturbation of ANALYSIS= DCEIG with large displacement (param, lgdisp, 1) in SOL 400, CLAN is recommended.
9. The SVD method is provided for DMAP applications. If used in solution 107 or 110, and mass or damping terms are present, a user fatal exit is taken. See the MSC Web site for the Flight Loads Product examples on the use of the SVD method. The SVD operation decomposes the input stiffness matrix K into the factors U, S, and V. U and V are collections of vectors of orthogonal functions. S is a rectangular matrix with terms on the diagonal of its left partition. The factors satisfy the equation $K = U^*S^*V'$, where “ * ” implies complex conjugate transpose. The ND1 value has a meaning for the SVD functions which differs from eigensolution.

ND1	OUTPUT
>0	All vectors of U and V are output.
=0	U and V are returned in a purged state.
<0	S is returned as a square matrix whose number of columns is equal to the minimum number of rows or columns of the input matrix. U and V are truncated to be commensurate with S. This is a method to reduce the costs of solving very rectangular input matrices by providing a partial solution for the most interesting vectors.

10. For DMAP applications there are input parameters, not present in the solution sequences, that may be used to replace the function of the EIGC and CMETHOD entries. See the MSC Software Web site for details.
11. The MBLKSZ and IBKLSZ parameters are integers in concept, but must be input at real numbers (that is, with a decimal point.) They represent maximum sizes, and may be reduced internally for small size problems.
12. The IRAM method is an out-of-core implementation of the public domain ARPACK complex eigenvalue solver. To use IRAM, the mass matrix has to be symmetric or Hermitian, and the number of modes desired can be at most the order of the generalized eigenproblem minus one. IRAM does not compute the left eigenvectors and thus is not suitable for SOL200 design optimization analysis.
13. For the "IRAM" method, only the fields that contain EIGC, SID, METHOD, NORM, ND0, NDj are meaningful. The user selects the method by specifying IRAM in the METHOD field, the eigenvector normalization scheme in the NORM field, and the number of desired modes in the ND0 or the NDj field. The other fields are irrelevant.
14. If Modules are present then this entry may only be specified in the main Bulk Data section.

EIGP**Poles in Complex Plane**

Defines poles that are used in complex eigenvalue extraction by the Determinant method.

Format:

1	2	3	4	5	6	7	8	9	10
EIGP	SID	ALPHA1	OMEGA1	M1	ALPHA2	OMEGA2	M2		

Example:

EIGP	15	-5.2	0.0	2	6.3	5.5	3		
------	----	------	-----	---	-----	-----	---	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
ALPHAi,OMEGAi	Coordinates of point in complex plane. (Real)
Mi	Multiplicity of complex root at pole defined by point at ALPHAi and OMEGAi. (Integer > 0)

Remarks:

1. The EIGP entry defines poles in the complex plane that are used with an associated EIGC entry having the same set number.
2. The units of ALPHAi and OMEGAi are radians per unit time.
3. Poles are used only in the determinant method. (METHOD = "DET" on the EIGC entry).
4. One or two poles may be defined on a single entry.
5. See *The NASTRAN Theoretical Manual*, Section 10.3.4, for details.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

EIGR**Real Eigenvalue Extraction Data**

Defines data needed to perform real eigenvalue analysis.

Format:

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	F1	F2	NE	ND			
	NORM	G	C						

Example:

EIGR	13	LAN				12			
------	----	-----	--	--	--	----	--	--	--

Descriptor	Meaning
SID	Set identification number. (Unique Integer > 0)
METHOD	Method of eigenvalue extraction. (Character) Modern Methods: LAN Lanczos Method AHOU Automatic selection of HOU or MHOU method. See Remark 13. Obsolete Methods: INV Inverse Power method. SINV Inverse Power method with enhancements. GIV Givens method of tridiagonalization. MGIV Modified Givens method. HOU Householder method of tridiagonalization. MHOU Modified Householder method. AGIV Automatic selection of METHOD = "GIV" or "MGIV". See Remark 13.
NORM	Method for normalizing eigenvectors. (Character: "MASS," "MAX," or "POINT"; Default = "MASS") MASS Normalize to unit value of the generalized mass. (Default) MAX Normalize to unit value of the largest component in the analysis set. POINT Normalize to a positive or negative unit value of the component defined in fields 3 and 4. The POINT option is not supported for METHOD=LAN. (Defaults to "MASS" if defined component is zero.)

Descriptor	Meaning
G	Grid or scalar point identification number. Required only if NORM = "POINT". (Integer > 0)
C	Component number. Required only if NORM = "POINT" and G is a geometric grid point. (1 ≤ Integer ≤ 6)

Table 9-12 Relationship Between METHOD Field and Other Fields for Obsolete Methods

Field	METHOD Field	
	INV or SINV	GIV, MGIV, HOU, or MHOU
F1, F2	Frequency range of interest. F1 must be input. If METHOD = "SINV" and ND, is blank, then F2 must be input. See also Remark 21. (Real ≥ 0.0)	Frequency range of interest. If ND is not blank, F1 and F2 are ignored. If ND is blank, eigenvectors are found with natural frequencies that lie in the range between F1 and F2. (Real ≥ 0.0; F1 < F2)
NE	Estimate of number of roots in range (Required for METHOD = "INV"). Not used by "SINV" method. (Integer > 0)	Not used.
ND	Desired number of roots. If this field is blank and METHOD = "SINV", then all roots between F1 and F2 are searched and the limit is 600 roots. (Integer > 0, Default is $3 \cdot NE$ for METHOD = "INV" only.)	Desired number of eigenvectors. If ND is zero, the number of eigenvectors is determined from F1 and F2. If all three are blank, then ND is automatically set to one more than the number of degrees-of-freedom listed on SUPPORTi entries. (Integer ≥ 0; Default = 0)

Remarks:

1. The EIGR entry must be selected with the Case Control command METHOD = SID.
2. See [Real Eigenvalue Analysis](#) in the *MSC Nastran Dynamic Analysis Guide* for a discussion of method selection.
3. The units of F1 and F2 are cycles per unit time.
4. The continuation entry is optional. If the continuation entry is not specified, then mass normalization is performed.
5. The contemporary methods are LAN and AHOU. The other methods are in a maintenance-only status, with no enhancements planned for them. They may be eliminated in a future release of Nastran.

6. The LAN method is the most general-purpose method, and may be used on both small- and large-size problems. It takes advantage of sparsity of input matrices, leading to greater efficiency on large-size problems. Because Lanczos performance is tuned for medium to large problems, this has caused difficulties with very small problems. Thus, by default, on problems with fewer than 20 degrees-of-freedom when the LAN method is selected, the method is switched to AHOU. The criteria for automatic switching is controlled by SYSTEM(359) on the NASTRAN entry. The NE, G, and C fields are ignored for the LAN method. The NORM field may be set to MASS (the default value) or MAX. The conventions used when both the Fi and ND fields are specified are described in [Table 9-13](#) of the EIGRL entry description. The EIGRL entry is an alternate method to select the LAN method. It has several other input options for special cases. When both EIGRL and EIGR have the same SID and that SID is selected by a METHOD command the EIGRL entry takes precedence.
7. The AHOU method is competitive with the LAN method when there are small, dense matrices and many eigenvectors are required. This most commonly occurs when static or dynamic reduction is performed. The AHOU method does not take advantage of matrix sparsity, so that computation cost rises with the cube of the number of DOFs. The AHOU method responds to all permitted values for all the other fields except NE, which is ignored.
8. All methods require a positive semi-definite (psd) mass matrix for stable solutions. The mass matrix may be tested for this condition for all methods of solution by setting SYSTEM(303). A value of “-4” should be sufficient to identify problem matrices. A fatal error exit is taken when it is not met. All Nastran metric elements are designed to produce psd mass matrices. CMASSi elements, DMIG matrices selected by the M2GG command, and matrices input via INPUTT4 are special methods that allow addition of non-psd terms by use of non-metric element input. If none of this type of special input is present and the fatal error exit is taken you may have encountered an error in a metric element. Contact your local MSC technical representative for corrective action in this case.
9. The LAN and AHOU methods allow singular but positive semi-definite mass matrices.
10. The tridiagonal methods include the xGIV and xHOU methods, where “x” is described in the following comments. All tridiagonal methods compute all eigenvalues, and the number of eigenvectors specified by the Fi and Nd fields, as described in [Table 9-12](#).
11. If “x” is blank (for example, the HOU method is selected) the mass matrix must be non-singular.
12. If “x” is M (for example, the MHOU method is selected) the mass matrix may be singular. A modified, shifted problem is solved in an inverse basis with this method. Some precision in the solution and longer computation time is exchanged for a more stable solution.
13. If “x” is A (for example, the AHOU method is selected) an attempt is made to solve the problem without shifting, in the interest of cost reduction and maximum precision. If the mass matrix is determined to be poorly conditioned for inversion the basis is automatically shifted with the modified method.
14. If NORM = “MAX”, components that are not in the analysis set may have values larger than unity.
15. If NORM = “POINT”, the selected component should be in the analysis set (a-set). (The program uses NORM = “MAX” when it is not in the analysis set.) The displacement value at the selected component will be positive or negative unity.

16. The “SINV” method is an enhanced version of the “INV” method. It uses Sturm sequence number techniques to make it more likely that all roots in the range have been found. It is generally more reliable and more efficient than the “INV” method.
17. For the “INV” and “SINV” methods, convergence is achieved at 10^{-6} . Convergence is tested by other criteria for the other methods.
18. For the “SINV” method only, if F2 is blank, the first shift will be made at F1, and only one eigensolution above F1 will be calculated. If there are no modes below F1, it is likely that the first mode will be calculated. If there are modes below F1 (including rigid body modes defined by SUPORT entries), a mode higher than the first mode above F1 may be calculated.
19. When F1, F2, and ND are all zero or blank, ND is reset to 1. A User Warning Message is produced for this condition, which is interpreted as likely to be due to an inadvertent omission by the user.
20. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the F1 and F2 fields are left blank, an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If ND is set to 1, there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.
21. F2 must be specified if DOMAIN SOLVER ACMS or DOMAIN SOLVER MODES is also specified in the Executive Control Section.
22. For large sized problems, Lanczos is the most efficient and is the recommended method for large problems.
23. It is not recommended to use AHOU, HOU, MHOU if RIGID=LAGRAN case control.
24. If Modules are present then this entry may only be specified in the main Bulk Data section.

EIGRL

Real Eigenvalue Extraction Data, Lanczos Method

Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

Format:

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND	MSG_LVL	MAXSET	SHFSCL	NORM	
option_1 = value_1 option_2 = value_2, etc.									

Example:

EIGRL	1	0.1	3.2	10					
NORM=MAX NUMS=2									

Descriptor	Meaning
SID	Set identification number. (Unique Integer > 0)
V1, V2	For vibration analysis: frequency range of interest. For buckling analysis: eigenvalue range of interest. See Remark 4. (Real or blank, $-5 \times 10^{16} \leq V1 < V2 \leq 5. \times 10^{16}$)
ND	Number of roots desired. See Remark 4. (Integer > 0 or blank)
MSG_LVL	Diagnostic level. ($0 \leq \text{Integer} \leq 4$; Default = 0)
MAXSET	Number of vectors in block or set. Default is machine dependent. See Remark 14.
SHFSCL	Estimate of the first flexible mode natural frequency. See Remark 10. (Real or blank)
NORM	Method for normalizing eigenvectors (Character: "MASS" or "MAX") <ul style="list-style-type: none"> MASS Normalize to unit value of the generalized mass. Not available for buckling analysis. (Default for normal modes analysis.) MAX Normalize to unit value of the largest displacement in the analysis set. Displacements not in the analysis set may be larger than unity. (Default for buckling analysis.)
ALPH	Specifies a constant for the calculation of frequencies (F_i) at the upper boundary segments for the parallel method based on the following formula. See Remark 13. (Real > 0.0; Default = 1.0): $\text{ALPH} < 1.0, F_i = V1 + (V2 - V1) \frac{1 - \text{ALPH}^i}{1 - \text{ALPH}^{\text{NUMS}}}$ $\text{ALPH} = 1.0, F_i = V1 + (V2 - V1) \frac{i}{\text{NUMS}}$
NUMS	Number of frequency segments for the parallel method. (Integer > 0; Default = 1)

Descriptor	Meaning
Fi	Frequency at the upper boundary of the i-th segment. See Remark 13. (Real or blank; $V1 < F1 < F2 < \dots < F15 < V2$)
option_i= value_i	Assigns a value to the fields above except for SID. ALPH, NUMS, and Fi must be specified in this format. V1, V2, ND, MSGLVL, MAXSET, SHFSCL, and NORM may be specified in this format as long as their corresponding field is blank in the parent entry.

Remarks:

1. Real eigenvalue extraction data sets must be selected with the Case Control command METHOD = SID.
2. The units of V1 and V2 are cycles per unit time in vibration analysis, and are eigenvalues in buckling analysis. Each eigenvalue is the factor by which the prebuckling state of stress is multiplied to produce buckling in the shape defined by the corresponding eigenvector.
3. NORM = "MASS" is ignored in buckling analysis and NORM = "MAX" will be applied.
4. The roots are found in order of increasing magnitude; that is, those closest to zero are found first. The number and type of roots to be found can be determined from [Table 9-13](#).

Table 9-13 Number and Type of Roots Found with EIGRL Entry

V1	V2	ND	Number and Type of Roots Found
V1	V2	ND	Lowest ND or all in range, whichever is smaller.
V1	V2	blank	All in range
V1	blank	ND	Lowest ND in range $[V1, +\infty]$
V1	blank	blank	Lowest root in range $[V1, +\infty]$
blank	blank	ND	Lowest ND roots in $[-\infty, +\infty]$
blank	blank	blank	Lowest root. See Remark 11.
blank	V2	ND	Lowest ND roots below V2
blank	V2	blank	All below V2

5. In vibration analysis, if $V1 < 0.0$, the negative eigenvalue range will be searched. (Eigenvalues are proportional to V_i squared; therefore, the negative sign would be lost.) This is a means for diagnosing improbable models. In buckling analysis, negative V1 and/or V2 require no special logic.
6. Eigenvalues are sorted on order of magnitude for output. An eigenvector is found for each eigenvalue.
7. MSGLVL controls the amount of diagnostic output during the eigenvalue extraction process. The default value of zero suppresses all diagnostic output. A value of one prints eigenvalues accepted at each shift. Higher values result in increasing levels of diagnostic output.

8. MAXSET is used to limit the maximum block size. It is otherwise set by the region size or by ND with a maximum size of 15. It may also be reset if there is insufficient memory available. The default value is recommended.
9. In vibration analysis, if V1 is blank, all roots less than zero are calculated. Small negative roots are usually computational zeroes which indicate rigid body modes. Finite negative roots are an indication of modeling problems. If V1 is set to zero, negative eigenvalues are not calculated.
10. A specification for SHFSCL may improve performance, especially when large mass techniques are used in enforced motion analysis. Large mass techniques can cause a large gap between the rigid body and flexible frequencies. If this field is blank, a value for SHFSCL is estimated automatically.
11. On occasion, it may be necessary to compute more roots than requested to ensure that all roots in the range have been found. However, this method will not output the additional roots.
12. NASTRAN SYSTEM(146) (also known as FBSMEM) provides options for I/O reduction during FBS operations according to the following table.

Table 16 SYSTEM(146) Options

SYSTEM(146) (default=0)	Description
-1 or 0	Use available memory to store as much of the factor matrix as possible during FBS operations. System Information Message 4199 is printed in the F04 file.
1	Disable all special memory operations for FBS.
2	Increase memory reserved for sparse method by approximately 200%.
3	Increase memory reserved for sparse method by approximately 300%.
4	Increase memory reserved for sparse method by approximately 400%.

13. For the distributed parallel method, the frequency range between V1 and V2 may be subdivided into segments that can then be analyzed in parallel. V1 and V2 must be specified for the parallel method. NUMS must be specified greater than 1 to take advantage of the parallel method. NUMS may also be specified on the NUMSEG keyword of the NASTRAN statement. Currently, NUMSEG must equal the number of processors and by default NUMSEG is set to the number of processors requested by the DMP keyword. If both are specified, then NUMS takes precedence.
The upper frequencies of each segment may be generated automatically by ALPH or specified directly in Fi. If both are specified, then Fi takes precedence over ALPH as long as they are consistent. ALPH if multiplied by 100 may also be specified on FRQSEQ keyword of the NASTRAN statement.
14. Increasing MAXSET may improve performance for large problems where a large number of eigenvalues are being found. The default is 7 on all machines. SYSTEM(263) may be set in an rcfle to effectively modify the default; however the setting on the EIGRL entry always takes precedence.

15. SYSTEM(196), keyword SCRSAVE, controls reuse of scratch files when segment logic is invoked. SYSTEM(196) is useful only when multiple frequency segments are requested on a Lanczos run. (Multiple frequency segments can be requested via the NUMS field in the EIGRL entry and by SYSTEM(197).) Each frequency segment requires a minimum of three scratch files. When multiple frequency segments are used on a single processor computer then each frequency segment is solved serially. In this case, it makes sense to let segment #2 use the scratch files which were used by segment #1 since work for segment #1 has been completed (otherwise it wouldn't be working on #2). Similarly, when work for segment #2 is finished, segment #3 should be able to use #2's scratch files. SYSTEM(196)=1 allows such file reuse and is considered a safe default on Version 70 and later systems.
16. The new buckling shift logic in Version 70.5 tends to shift to 1.0 first. The logic may have difficulty finding the lowest ND roots if a problem requests a small number of roots (ND) when there are thousands of roots below 1. In this case either the loading should be scaled, SHFSCL specified, or a smaller frequency range requested.
17. Because Lanczos performance is tuned for medium to large problems, this has caused difficulties with very small problems. Thus, by default, on problems with fewer than 20 degrees-of-freedom when the LAN method is selected, the method is switched to AHOU. The criteria for automatic switching is controlled by SYSTEM(359) on the NASTRAN entry.
18. V2 must be specified if DOMAINSOLVER ACMS or DOMAINSOLVER MODES is also specified in the Executive Control Section.
19. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the V1 and V2 fields are left blank, an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If ND is set to 1, there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.
20. For buckling solutions, if V1 is explicitly set to 0.0, negative eigenvalues will be calculated based on a negative shift value. This is done to avoid singularities resulting from a shift at 0.0. If V1 is set to a small positive value, then negative eigenvalues will not be calculated. This behavior was changed with the MSC Nastran 2010 release whereas before setting V1 to 0.0 yielded no negative eigenvalues. The best approach, however, is to follow the suggestion from Remark 19.
21. If Modules are present then this entry may only be specified in the main Bulk Data section.

ELEMUDS

Element Property User Defined Service or Subroutine

Allows the user to provide element property routines for use with specified Nastran elements.

Format:

1	2	3	4	5	6	7	8	9	10
ELEMUDS	PID	PTYPE	GROUP	UNAME	DEPEN	NMLSPD			
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	IDATAn			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	RDATAn			
	“CHAR”	CDATA1	CDATA2	CDATAn			

Examples:

In FMS Section of Nastran input stream:

```
CONNECT SERVICE ELEMENT 'SCA.MDSolver.Util.Ums'
```

In Bulk Data:

ELEMUDS	17	PBUSH2D	ELEMENT	THPAD	FREQ				
---------	----	---------	---------	-------	------	--	--	--	--

Descriptor	Meaning
PID	Element property identification number that matches the identification number on a PBUSH2D element property entry. (Integer > 0)
PTYPE	The name of the element entry. Currently supported element property entry is: PBUSH2D. (Character; no Default)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. See Remark 6.(Character)
DEPEN	Specifies if element is frequency dependent. DEPEN=NOM or blank; user element is not frequency dependent. DEPEN=FREQ; user element is frequency dependent. (Character default=NOM)
NMLSPD	Nominal speed (cycles per second) used by the CBUSH2DA to calculate nominal stiffness, mass and damping terms. Nominal terms will be used during reduction and real eigenvalue solution. In a frequency-dependent problem, updated values will be calculated at each frequency.
“INT”	Keyword indicating that the following data is integer. (Character)
IDATAi	Additional user supplied Integer data not already existing on the specified property entry. (Integer; no Default)
“REAL”	Keyword indicating that the following data is real. (Character)

RDATAi	Additional user supplied Real data not already existing on the specified property entry. (Real; no Default)
“CHAR”	Keyword indicating that the following data is Character. (Character)
CDATAAi	Additional user supplied Character data not already existing on the specified property entry. (Character; no Default)

Remarks:

1. This entry triggers the call to a user element property subroutine for advanced nonlinear materials. The GROUP must match the GROUP field of the CONNECT SERVICE FMS statement.
2. On the FMS CONNECT statement, only the CONNECT SERVICE can be used with this entry.
3. PID must match an existing PID.
4. A CDATAi entry cannot be the Character “REAL”, “INT”, or “CHAR”.
5. Certain user subroutines may require integer, real, or character data input as specified in the User Defined Services manual.
6. UNAME must be truncated to 8 characters in the bulk data field
7. If the ROMAC interface is being used then UNAME=THPAD is required. See the THPAD Bulk Data entry (p. 3244) for a description of ROMAC service. (<http://www.virginia.edu/romac/>)

ELEMUDS

Element Property User Defined Service or Subroutine - SOL 600

Allows the user to provide element property routines for use with enhanced nonlinear elements in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
ELEMUDS	PID	PTYPE	GROUP	UNAME					
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	IDATAn			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	RDATAn			
	“CHAR”	CDATA1	CDATA2	CDATAn			

Examples:

ELEMUDS	17	PSHELL	UELEM	RTN1					
ELEMUDS	17	PSHELL	UELEM1	RTN3					
	REAL	.00134	1.467+4	.03					
	INT	8	3						

Descriptor	Meaning
PID	Element property identification number that matches the identification number on a PSHELL, PSOLID, PCOMP, PCOMPG, PBAR, PBEAM, PSHEAR, or PROD element property entry. (Integer > 0)
PTYPE	The name of the element entry. Currently supported element property entries include: PSHELL, PSOLID, PCOMP, PCOMPG, PBAR, PBEAM, PSHEAR, or PROD. (Character; no Default)
GROUP	The group name used for the FMS section CONNECT SERVICE entry. (Character; no Default)
UNAME	A primary or secondary name to identify the user subroutine. See Remark 6. (Character; Default blank)
“INT”	Keyword indicating that the following data is integer. (Character)
IDATAi	Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)
“REAL”	Keyword indicating that the following data is real. (Character)
RDATAi	Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)

“CHAR”	Keyword indicating that the following data is Character. See Remark 7. (Character)
CDATAAi	Additional user supplied Character data not already existing on the specified MAT entry. (Character; no Default)

Remarks:

1. This entry triggers the call to a user element property subroutine for advanced nonlinear materials.
2. PID must match an existing PID.
3. A CDATAi entry cannot be the Character “REAL”, “INT”, or “CHAR”.
4. Certain user subroutines may require integer, real, or character data input as specified in the User Defined Services manual.
5. The ELEMUDS entry may be used instead of the USRSUB6 entry. Both entries should not be used in the same run. EVAL will be stored as a character*16 name in common block /userch/.
6. UNAME identifies the user subroutine name to be called.
7. Character fields CDATAi are ignored in SOL 600 are not passed to the user subroutine.

ELIST**Element List**

Defines a list of CQUAD4 and CTRIA3 structural elements for virtual fluid mass.

Format:

1	2	3	4	5	6	7	8	9	10
ELIST	LID	E1	E2	E3	E4	E5	E6	E7	
	E8	E9	E10	-etc.-					

Example:

ELIST	3	51	-62	68	THRU	102	122		
-------	---	----	-----	----	------	-----	-----	--	--

Descriptor	Meaning
LID	Identification number of list. (Integer > 0)
Ei	Identification number of a structural element. See Remark 1. for the meaning of the negative sign. The string “THRU” may be used to indicate that all existing elements between those referenced in the preceding and succeeding fields are in the list. (Integer ≠ 0 or “THRU”)

Remarks:

1. If the ELIST entry is referenced by field 6 of an MFLUID entry, the wetted side of the element is determined by the presence or absence of a minus sign preceding the element's ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element's positive normal as determined by applying the right-hand rule to the sequence of its corner points. If the “THRU” option is used, then immediately preceding and succeeding elements must have the same sign.
2. Large open “THRUs” should be avoided.
3. The word “THRUs” must not appear in field 2 or 9 on the parent entry or on any continuations.
4. If any ELIST entry is changed or added on restart then a complete re-analysis may be performed. Therefore, ELIST entry changes or additions are not recommended on restart.

ENDDATA

Bulk Data Delimiter

Designates the end of the Bulk Data Section.

Format:

ENDDATA

Remark:

1. ENDDATA is optional.

ENDDYNA

Defines the End of Direct Text to Dytran

All entries between TODYNA and ENDDYNA will be passed directly by SOL 700 to Dytran. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
ENDDYNA									

Example:

TODYNA									
MAT1	345	29.0E6	0.285	0.0004					
ENDDYNA									

Descriptor	Meaning
TODYNA	
MAT1	
ENDDYNA	

Remarks:

See [TODYNA, 3272](#) for details of how this entry is used.

ENTUDS

User Defined Logic at Entry Point

Calls user defined logic within a SCA service at the point specified within the solution sequence.

Format:

1	2	3	4	5	6	7	8	9	10
ENTUDS	ENTID	ENTPNT	GROUP						
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	IDATA _n			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	RDATA _n			
	“CHAR”	CDATA1	CDATA2	CDATA _n			

Example:

ENTUDS	1	RCENT1	MY_FUNC						
	INT	2	17						
	REAL	.5	.25						

Descriptor	Meaning
ENTID	Entry point identification number (Integer > 0)
ENTPNT	The point of entry in the solution sequence. Acceptable values for now: “RCENT1”, “RCENT2”, “RCOUT1”, “RCEEXEC1”. (Character; Required)
GROUP	The SCA group name used to identify the service (Character; Required)
“INT”	Key word indicating that the following data is integer. (Character)
IDATA _i	Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)
“REAL”	Key word indicating that the following data is real. (Character)
RDATA _i	Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)
“CHAR”	Key word indicating that the following data is Character. (Character)
CDATA _i	Additional user supplied Character data not already existing on the specified MAT entry. (Character; no Default)

Remarks:

1. This entry is for RC Network solver only.
2. In SINDA input file, there are 4 entry points for users to input customized logics.
 - a. RCENT1 ---- Variable 1 block

- b. RCENT2 ---- Variable 2 block
- c. RCOUT1 ---- Output block
- d. RCEEXEC1 ---- Execution block

EOSDEF

Deflagration - SOL 700 only

EOSDEF defines the properties of the deflagration equation of state, and the reaction rate to model the burning of solid propellants. The burning of the solid propellant produces hot gas.

Format

1	2	3	4	5	6	7	8	9	10
EOSDEF	MID	GAMMA	B	R	C _v	C _p	E	RHOS	+EOSDEF1
EOSDEF	3	1.123	0.001	304			1E+6	1600	+EOSDEF1

Example

1	2	3	4	5	6	7	8	9	10
+EOSDEF1	RHOF	W	BETA	SAVR	X	Y			
+EOSDEF1	0.1	4E-8	0.85	6000	0.6	0.0			

Field	Contents	Type	Default
MID	Unique material number.	I > 0	Required
GAMMA	Constant γ	R>0	Required
B	Gas co-volume	R≥ 0	Required
R	Gas constant.	R>0	Refer to remark 2.
CV	Specific heat at constant volume	R>0	Refer to remark 3.
CP	Specific heat at constant pressure	R>0	Refer to remark 3.
E	Chemical energy per unit burned mass	R>0.0	Required
RHOS	Reference density of propellant	R>0	Required
RHOF	<u>Density of powder</u> reference density of propellant	R>0	Refer to remark 5.
W	Burning rate coefficient	R≥ 0.0	Required
BETA	Burning rate exponent	R≥ 0.0	Required
SAVR	Initial surface area divided by volume	R>0	Required
X	Parameter form function	R≥ 0.0	Required
Y	Parameter form function	R≥ 0.0	Required

Remarks

1. EOSDEF – The deflagration Equation of State is explained in *Dytran Theory Manual*, Chapter 4: Models.
2. This equation of state is used with Eulerian elements.
3. The temperature of gas is calculated when one of the constants R, C_v or C_p are specified. When temperature is not mentioned in an output request, omit the he constants .
4. The pressure in the reaction products is defined by the Noble-Abel equation of state as follows:

$$\rho = (\gamma - 1) \frac{\rho}{1 - b\rho} e \text{ for reacted product,}$$

$$T = \frac{(\gamma - 1)e}{R} \text{ where } \gamma, b \text{ are constants and } R \text{ is the gas constant.}$$

The chemical reaction rate for conversion of un-reacted explosive to reaction products is mentioned the relation below:

$$\xi = w SAVR$$

vivacity

$$\phi = (1 - F)^X + YF$$

form function

$$\frac{dF}{dt} = \xi \phi p^\beta$$

time derivative of burn fraction

where;

w = burning rate coefficient

β = burning rate exponent

SAVR = initial surface area divided by volume

Y= parameter form function

S= parameter form function

5. RHOF allows taking into account the air between gunpowder grains. Also, it allows taking into account air inside the combustion chamber. For example If the chamber is filled with 10% real propellant and the rest is filled with air then RHOF = 0.1.
6. To indicate what Eulerian regions can burn, the variable DEFMAT can be used. In regions that can burn DEFMAT=1. DEFMAT can be set on the TICVAL card:
TICVAL,14,,DENSITY,1.14,SIE,300000.2,DEFMAT,1.0
7. Ignition of the propellant approached by setting SIE to the flame temperature in specific areas:
TICVAL,15,DENSITY,1.14,SIE,4231908.591,DEFMAT,1.0
Where SIE_{ignition} = Flame Temp x Cv
Flame Temp can be extracted from Chemical Energy per unit burned mass and Gas constant: T_{flame} = E / R .

EOSGAM**Gamma Law Gas Equation of State - SOL 700 only**

Defines the properties of a Gamma Law equation of state where the pressure p is defined as:

$$p = (\gamma - 1)\rho e$$

where:

e = specific internal energy per unit mass

ρ = overall material density

γ = A constant

Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
EOSGAM	EID	GAMMA	R	CV	CP	VISC			

Example:

EOSGAM	2	1.4	287.0						
--------	---	-----	-------	--	--	--	--	--	--

Descriptor	Meaning
EID	Unique equation of state number. (Integer > 0, Required)
GAMMA	Constant γ . (Real ≥ 0 , default=blank)
R	Gas constant. See Remarks 1. and 3. (Real > 0, default=blank)
CV	Specific heat at constant volume. See Remarks 1. and 3. (Real > 0, default=blank)
CP	Specific heat at constant pressure. See Remarks 1. and 3. (Real > 0, default=blank)
VISC	Viscosity coefficient. If zero, no viscosity. See Remarks 4., 5., and 6. (Real ≥ 0 , default=blank)

Remarks:

1. The temperature of the gas will be calculated when one of the gas constants, R , C_v or C_p is specified.
2. The Euler variable name for temperature is TEMPTURE.
3. γ , R , C_v and C_p have the following relationships:

$$\gamma = \frac{C_p}{C_v} \quad R = C_p - C_v$$

4. The viscosity coefficient is the dynamic viscosity. It is the ratio between shear stress and velocity gradient. The SI-unit of viscosity is

$$\text{Pa} \cdot \text{s} = \frac{\text{N}_\text{s}}{\text{m}^2} = \frac{\text{kg}}{\text{ms}}$$

5. If possible, use (in coupled analysis) the FASTCOUP coupling algorithm because viscous fluxes are computed more accurately for fast coupling than for general coupling.
6. For the single mat solver, viscous stresses can be requested by the use of TXX through TZx. Also, EFFSTS is available. For the multi-material solver, viscous stresses are stored in TXX-VIS, TYY-VIS, TZZ-VIS, TXY-VIS, TYZ-VIS, TZx-VIS. This viscous stresses depend only on the current velocity gradients. The stresses like TXX are elastic-plastic stresses and depend on past stress. The total stress tensor in the element is given by the average of the viscous stress and elastic-plastic stress. The weight factors are the material fraction of viscous fluid and the remaining materials.

EOSIG**Ignition and Growth Equation of State - SOL 700 only**

Defines the properties of Ignition and Growth equation of state and the reaction rate equation used to model high explosives. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
EOSIG	EID	UNITDEF	DBEXPL	UNITCNV					+
+		AE	BE	R1E	R2E	OMGE	I	G	+
+	A	AP	BP	R1P	R2P	OMGP	X	Y	+
+	Z	R	ECHEM	PRSTOL	ITRMAX				

Example:

EOSIG	10		LX17	SI					
-------	----	--	------	----	--	--	--	--	--

Descriptor	Meaning																		
EID	Unique equation of state number referenced from MATDEOL. (Integer > 0; Required)																		
UNITDEF	User-defined default unit for the inputs: See Remarks 2. and 3. (Character) <table> <tr> <td>CCGMS</td><td>m/g/μs units</td></tr> <tr> <td>SI</td><td>International System units</td></tr> <tr> <td>METRIC</td><td>Metric units</td></tr> <tr> <td>IMPER</td><td>imperial units</td></tr> <tr> <td>MMMG</td><td>mm/mg/μs units</td></tr> </table>	CCGMS	m/g/ μ s units	SI	International System units	METRIC	Metric units	IMPER	imperial units	MMMG	mm/mg/ μ s units								
CCGMS	m/g/ μ s units																		
SI	International System units																		
METRIC	Metric units																		
IMPER	imperial units																		
MMMG	mm/mg/ μ s units																		
DBEXPL	Use explosive material from the database (See Remarks 4. and 6.) (Character, NO). The following detonation materials are available in the data base: <table> <tr> <td>NO</td><td>The database is not used. See Remark 5.</td></tr> <tr> <td>P94A</td><td>PBX-9404 (a) explosive</td></tr> <tr> <td>TATB</td><td>TATB explosive</td></tr> <tr> <td>PENT</td><td>PETN explosive</td></tr> <tr> <td>CTNT</td><td>Cast TNT explosive</td></tr> <tr> <td>LCOMPB</td><td>LANL COMP B explosive</td></tr> <tr> <td>MCOMPB</td><td>Military COMP B explosive</td></tr> <tr> <td>P94B</td><td>PBX-9404 (b) explosive</td></tr> <tr> <td>LX17</td><td>LX-17 explosive</td></tr> </table>	NO	The database is not used. See Remark 5.	P94A	PBX-9404 (a) explosive	TATB	TATB explosive	PENT	PETN explosive	CTNT	Cast TNT explosive	LCOMPB	LANL COMP B explosive	MCOMPB	Military COMP B explosive	P94B	PBX-9404 (b) explosive	LX17	LX-17 explosive
NO	The database is not used. See Remark 5.																		
P94A	PBX-9404 (a) explosive																		
TATB	TATB explosive																		
PENT	PETN explosive																		
CTNT	Cast TNT explosive																		
LCOMPB	LANL COMP B explosive																		
MCOMPB	Military COMP B explosive																		
P94B	PBX-9404 (b) explosive																		
LX17	LX-17 explosive																		
UNITCNV	User defined conversion units: (Character; see Remarks 2. and 3.) <table> <tr> <td>CGMS</td><td>cm/g/μs units</td></tr> </table>	CGMS	cm/g/ μ s units																
CGMS	cm/g/ μ s units																		

Descriptor	Meaning
SI	International System units
METRIC	Metric units
IMPER	Imperial units
MMMGSS	mm/mg/ μ s units
AE	Constant A_e for un-reacted explosive. See Remark 5. (Real; Required)
BE	Constant B_e for un-reacted explosive. See Remark 5. (Real; Required)
R1E	Constant R_{1e} for un-reacted explosive. See Remark 5. (Real; Required)
R2E	Constant R_{2e} for un-reacted explosive. See Remark 5. (Real; Required)
OMGE	Constant ω_e for un-reacted explosive
I	First ignition coefficient. See Remark 5. (Real; Required)
G	Second ignition coefficient. See Remark 5. (Real; Required)
A	Density ignition coefficient. See Remark 5. (Real; Required)
AP	Constant A_p for reacted product. See Remark 5. (Real; Required)
BP	Constant B_p for reacted product. See Remark 5. (Real; Required)
R1P	Constant R_{1p} for reacted product. See Remark 5. (Real; Required)
R2P	Constant R_{2p} for reacted product. See Remark 5. (Real; Required)
OMGP	Constant ω_p for reacted product. See Remark 5. (Real; Required)
X	Surface burning exponent. See Remark 5. (Real, 2./9.)
Y	Surface burning exponent. See Remark 5. (Real, 2./3.)
Z	Pressure exponent. See Remark 5. (Real; Required)
R	Relative density exponent. See Remark 5. (Real, .4)
ECHEM	Chemical energy of high explosive per unit mass. See Remark 5. (Real; Required)
PRSTOL	Tolerance for pressure equilibrium iterations in mixed phase elements. (Real > 0, 1.E-6)
ITRMAX	Maximum number of iterations in pressure equilibrium iterations. (Integer > 0, 16)

Remarks:

1. This equation of state can only be used with solid Eulerian elements.
2. The definition of the unit system in which the input values are defined is required information only in case you wish to have an automatic conversion to a different unit system as defined by the UNITCNV field. In case you are using the conversion mechanism, note that the density RHO in the corresponding MATDEUL entry will be interpreted in the unit system defined here. [Table 9-14](#) defines sets of units available:

Table 9-14 Sets of Units Used in the IG Model

Quantity	CGμs	SI	Metric	Imperial	MMMGμs
Length	Centimeter (cm)	Meter (m)	Centimeter (cm)	Inch (in)	Millimeter (mm)
Time	Microsecond (μs)	Second (s)	Second (s)	Second (s)	Microsecond (μs)
Mass	Gram (g)	Kilogram (kg)	Gram (g)	Slug (lbf-s ² /in)	Milligram (mg)
Force	Teradyne	Newton (N)	Dyne	Pound force (lbf)	kN
Density	g/cm ³	Kg/m ³	g/cm ³	lbf-s ² /in ⁴	mg/mm ³
Stress	Mbar	Pascal (Pa)	μbar	Lbf/in ²	GPa
Energy	1012 erg (Mbars-cm ³)	Joule (J)	Erg	Lbf-in	J
Temperature	Kelvin (K)	Kelvin (K)	Kelvin (K)	Kelvin (K)	Kelvin (K)

- The UNITCNV field defines the unit system to which the material parameters are converted. In case you are not using one of the database material models, you also have to define the default unit system (UNITDEF) in which you supplied the data.
- You can use the database containing several detonation materials to start the analysis. The material data are taken from Lee/Tarver (Ref. 1) and Murphy/Lee (Ref. 2) papers in the Theory Manual. The equations of state parameters are given in the [Table 9-15](#).
- The default setting for DBEXPL is NO, which means you should define the values in the input fields (fields 12 to 36). If the database material name is defined, all values in the input fields will be overridden. The reference density RHO defined on the corresponding MATDEUL entry will be set to the value from the database.
- The default unit system for the material database parameters is the CGMS unit system. If you wish to use the material base data in a different unit system, you can specify this by defining the target unit system in the UNITCNV field.
- You can define the shear property and yield model of the material with respectively SHXXX and YLDXX entry. Note that the unit system of data required in these entries should be consistent with the unit system defined in the UNITCNV field.
- The IG equation of state cannot be used in combination with a spallation model.
- The following JWL equation of state is used to calculate the pressure of the un-reacted explosive (in “solid” state):

$$p_e = A_e \left(1 - \frac{\omega_e \eta_e}{R_{1e}}\right) e^{-\frac{R_{1e}}{\eta_e}} + B_e \left(1 - \frac{\omega_e \eta_e}{R_{2e}}\right) e^{-\frac{R_{2e}}{\eta_e}} + \omega_e \eta_e \rho_e E_e$$

where:

$$\eta_e = \rho_e / \rho_0$$

the relative density of the unreacted explosive.

$$E_e$$

the specified internal energy per unit mass of the unreacted explosive

$$\rho_0$$

the initial density of the explosive

$$A_e, B_e, \omega_e, R_{1e}, R_{2e}$$

the input constants of the unreacted explosive

Similarly, the pressure in the reaction products (in “gas” state) is defined by another JWL form as follows:

$$p_p^1 = A_p \left(1 - \frac{\omega_p \eta_p}{R_{1p}}\right) e^{-\frac{R_{1p}}{\eta_p}} + B_p \left(1 - \frac{\omega_p \eta_p}{R_{2p}}\right) e^{-\frac{R_{2p}}{\eta_p}} + \omega_p \eta_p \rho_p E_p$$

where:

$$\eta_p = \rho_p / \rho_0$$

= the relative density of the unreacted explosive

$$E_p$$

= the specified internal energy per unit mass of the unreacted explosive

$$A_p, B_p, \omega_p, R_{1p}, R_{2p}$$

= the input constants of the reaction product. The chemical reaction rate for conversion of un-reacted explosive to reaction products is described by the following reaction rate equation:

$$\frac{\partial F}{\partial t} = I(1 - F)^x (\eta_e - 1 - a)^r + G(1 - F)^x F^y (P)^z$$

here F denotes the burn fraction that is defined as the fraction of the explosive that has already reacted. For more details concerning the implementation of this equation of state, please refer to the Theory Manual

10. You can access the results of the un-reacted explosive and reaction products for IG elements. These EOSIG specific output variables are

Keyword	Description
SIE-E	Specific internal energy per unit mass of un-reacted explosive part
SIE-P	Specific internal energy per unit mass of reaction products part
FMAT	Volume fraction
RHO-E	Density of un-reacted explosive part
RHO-P	Density of reaction products part
MASS-E	Mass of un-reacted explosive part
MASS-P	Mass of reaction products part

The output variables for the burn fraction are

Keyword	Type of Elements	Description
FBURN	Solid Lagrangian Elements	Burn fraction of EOSIG material
	Euler Elements	Not applicable for EOSIG materials. Burn fraction for EOSJWL material
IGBURN	Solid Lagrangian Elements	Not available
	Euler Elements	Burn fraction of EOSIG MATERIAL

11. The ignition of IG material can be initiated by:

- Compression of the IG material in a small region, where the compression originates from outside that region. This is the most physical method to initiate ignition. Examples are a shock wave entering the region, a flow boundary that supplies mass to the region and a plate or other structural part that compresses the region. In all these cases the IG material should be initialized with zero pressures. This can be achieved by not specifying the specific energy on the TICVAL entry that prescribes the initial state of the IG material. The specific energy will be computed such that the initial pressure is zero.
- Compression of the IG material in a small region where the compression originates within that region. This can be done by specifying either a density that exceeds the compression limit or a specific energy that gives rise to a sufficiently large pressure.

Table 9-15 Coefficients for the IG Model of Several Explosions in the Database.

Explosive	PBX-9404 (a)	TATB	PETN	Case TNT	LANL COMP B	Military COMP B	PBX-9404 (b)	LX-17
Unreacted Equation of State and Constitutive Values:								
RHO (g/cm ³)	1.842	1.90	1.842	1.61	1.712	1.630	1.842	1.903
AE (Mbar)	69.69	108.2	37.42	17.98	778.1	1479.	9522.	778.1
BE (Mbar)	-1.727	-2.406	-1.313	-0.931	-0.05031	-0.05261	-0.5944	-0.05031
R1E	7.8	8.2	7.2	6.2	11.3	12.	14.1	11.3
R2E	3.9	4.1	3.6	3.1	1.13	1.2	1.41	1.13
OMGE	0.8578	1.251	1.173	0.8926	0.8938	0.9120	0.8867	0.8938
Reacted Product Equation of State Values:								
AP (Mbar)	8.524	6.5467	6.17	3.712	5.242	5.5748	8.524	6.5467
BP (Mbar)	0.1802	0.071236	0.16926	0.032306	0.07678	0.0783	0.1802	0.071236
R1P	4.6	4.45	4.4	4.15	4.2	4.5	4.6	4.45
R2P	1.3	1.2	1.2	0.95	1.1	1.2	1.3	1.2
OMGP	0.38	0.35	0.25	0.30	0.34	0.34	0.38	0.35
ECHEM (Mbar-cm ³ /g)	0.0554	0.0363	0.0548	0.0433	0.0496	0.04969	0.0554	0.03626
Reaction Rate Parameters:								
I (μ s ⁻¹)	44.0	50.0	20.0	50.0	44.0	44.0	44.0	50.0
G (Mbar-z μ s ⁻¹)	200.0	125.0	400.0	40.0	414.0	514.0	850.0	500.0
A	0.01	0.01	0.01	0.01	0.01	0.01	0.01	0.0
Z	1.6	2.0	1.4	1.2	2.0	2.0	2.0	3.0
X	2/9	2/9	2/9	2/9	2/9	2/9	2/9	2/9
Y	2/3	2/3	2/3	2/3	2/3	2/3	2/3	2/3
R	4.0	4.0	4.0	4.0	4.0	4.0	4.0	4.0

12. EID must unique among all EOSxx entries in one model.

EOSJWL**JWL Explosive Equation of State - SOL700**

Defines the properties of a JWL equation of state commonly used to calculate the pressure p of the detonation products of high explosives

$$p = p_0 + A \left(1 - \frac{\omega\eta}{R_1}\right) e^{-\frac{R_1}{\eta}} + B \left(1 - \frac{\omega\eta}{R_2}\right) e^{-\frac{R_2}{\eta}} + \omega\eta\rho_0 e + \frac{\omega\rho\lambda Q}{\rho_0}$$

$$\frac{d\lambda}{dt} = a(1 - \lambda)^m p^n$$

e = specific internal energy per unit mass

ρ_0 = reference density

ρ = overall material density

η = ρ / ρ_0

p_0 = initial pressure

λ = fraction that describes the afterburning. It ranges from 0 to 1

A, B, R₁, R₂, a, m, n and Q are constants.

The last term: $\frac{\omega\rho\lambda Q}{\rho_0}$ models afterburning.

Format and Example

EOSJWL	MID	A	B	R1	R2	OMEGA	P0	Q	+
EOSJWL	37	5.2E11	0.77E11	4.1	1.1	0.34		0.0	+
+ a m n									
+ 0.0 0.0 0.0									

Descriptor	Meaning	Type	Default
EID	Unique equation of state number referenced from MATDEUL. (Integer > 0; Required)	I > 0	Required
A	Constant A. (Real, 0.0)	R	0.0
B	Constant B. (Real, 0.0)	R	0.0
R1	Constant R1. (Real, 0.0)	R	0.0
P2	Constant R2. (Real, 0.0)	R	0.0

Descriptor	Meaning	Type	Default
OMEGA	Constant ω . (Real, 0.0)	R	0.0
P0	Initial pressure. See Remark 3. (Real, 0.0)	R	0.0; see Remark 4.
Q	Initial pressure	$R \geq 0.0$	0.0; see Remark 6.
a	Initial pressure	$R \geq 0.0$	0.0; see Remark 6.
m	Initial pressure	$R \geq 0.0$	0.0; see Remark 6.
n	Initial pressure	$R \geq 0.0$	0.0; see Remark 6.

Remarks:

1. This equation of state can be used only with Eulerian elements.
2. A [DETSPH](#) entry must be used to specify the detonation model.
3. This equation of state is discussed in *Dytran Theory Manual*, Chapter 4: Models, [EOSJWL – JWL Equation of State](#).
4. In simulations with ideal gases, the JWL material needs to have an initial pressure to counter balance the pressure of the ideal gas. Similarly, in case of under calculations where an explosive is located at a certain depth, P0 should be preset to equal the initial hydrostatic pressure.
5. The [DETSPH](#) definition of one EOSJWL material also applies to all other EOSJWL materials. Therefore, the blast wave of one explosive can ignite other explosives. To prevent this linked ignition, [PARAM,JWLDET,NOLINK](#) can be used.
6. By default the constants a, m, n and Q are zero, and afterburning is not taken into account. To model afterburning all the variables Q, a, m and n have to be set to positive constants. The extent of the afterburning can be checked by requesting the Euler element variable AFTERBURN. This variable equals λ . If the afterburning in an Euler element is complete then AFTERBURN = 1.0.

EOSMG**Mie-Gruneisen Equation of State - SOL 700 only**

Defines the properties of a Mie-Gruneisen equation of state commonly used to calculate the pressure p in high strain rate processes. Used in SOL 700 only.

$$p = \frac{\rho_0 c^2 n}{(1 - s\eta)^2} \left(1 - \frac{\Gamma_0 \eta}{2}\right) + \Gamma_0 \rho_0 e$$

$$\eta = 1 - \frac{\rho_0}{\rho_1}$$

$$\rho_1 = \min(\rho, RM)$$

where

e	= specific internal energy per unit mass. For material at zero pressure, e has to be initialized as zero.
ρ_0	= reference density
ρ	= overall material density
Γ_0	= Gruneisen parameter at reference density.
s	= defined by $U_s = c_0 + sU_p$, where U_s and U_p are respectively the linear shock velocity and particle velocity as obtained from shock data.
c	= sound speed at reference density
RM	= Cut-off value for density.

Format:

1	2	3	4	5	6	7	8	9	10
EOSMG	EID	c	S	R1	RM				

Example:

EOSMG	37	2000	1.5	2.0	3000				
-------	----	------	-----	-----	------	--	--	--	--

Descriptor	Meaning
EID	Unique equation of state number referenced from MATDEUL. (Integer > 0; Required)
c	Sound speed at reference density. (Real; Required)
s	Constant s . (Real; Required)
Γ_0	Gruneisen gamma. (Real; Required)
RM	Cut off value for density. (Real; Required)

Remarks:

1. This equation of state can be used only with Eulerian elements.
2. This equation of state is discussed in Equations of State.
3. The cut off value RM is only used for limiting the pressure. To prevent division by zero RM should be less than $s/s - 1 \rho_{ref}$. RM can be set slightly below this value. In case the simulation gets instable because of too large pressures RM can be decreased.
4. EID must unique among all EOSxx entries in one model.

EOSNA**Noble-Abel equation of state - SOL700**

Defines the properties of Noble-Abel equation of state where the pressure p is defined as:

$$p = (\gamma - 1) \frac{\rho}{1 - b\rho} e$$

where:

- e = specific internal energy per unit mass
- ρ = overall material density
- γ = a constant
- b = Gas co-volume

Format and Example

1	2	3	4	5	6	7	8	9	10
EOSNA	EID	GAMMA	B	R	CV	CP			
EOSNA	35	1.2363	0.001	314.46					

Field	Contents	Type	Default
EID	Unique equation of state number.	I > 0	Required
GAMMA	Constant γ .	R ≥ 0.0	Required
B	Gas co-volume .	R ≥ 0.0	0.0 Refer to Remark 3.
R	Gas constant.	R > 0.0	Refer to remark 4.
CV	Specific heat at constant volume.	R > 0.0	Refer to remark 4.
CP	Specific heat at constant pressure	R > 0.0	Refer to remark 4.

Remarks

1. Refer to *Dytran Theory Manual*: Chapter 4-Models, for EOSNA – Noble-Abel Equation of State.
2. EOSNA – Noble-Abel Equation of State is used with Eulerian elements only.
3. The parameter B models the interactions between gas particles.
4. The temperature of the gas is calculated when one of the gas constants, R , C_v , and C_p is mentioned.
5. The Euler variable name for temperature is TEMPTURE.
6. The relation between Gamma, R , C_v , and C_p is shown below:

$$\gamma = \frac{C_p}{C_v} \quad R = C_p - C_v$$

EOSPOL**Polynomial Equation of State for Solids - SOL 700 only**

Defines the properties of a polynomial equation of state where the pressure p is defined in SOL 700 as follows:

In compression ($\mu > 0$),

$$p = a_1\mu + a_2\mu^2 + a_3\mu^3 + (b_0 + b_1 + b_2\mu^2 + b_3\mu^3)\rho_0e$$

In tension ($\mu < 0$),

$$p = a_1\mu + (b_0 + b_1\mu)\rho_0e$$

Where

$$\mu = \eta - 1$$

$$\eta = \rho/\rho_0$$

p = overall material density

ρ_0 = reference density

e = specific internal energy per unit mass

Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
EOSPOL	EID	A1	A2	A3	B0	B1	B2	B3	
	HVL	VISC							

Example:

EOSPOL	100	80.E6							
	1.1								

Descriptor	Meaning
EID	Unique equation of state member. (Integer > 0; Required)
A1	Coefficient a1 or Bulk Modulus. (Real; Default = 0.0)
A2	Coefficient a2. (Real; Default = 0.0)
A3	Coefficient a3. (Real; Default = 0.0)
B0	Coefficient b0. (Real; Default = 0.0)
B1	Coefficient b1. (Real; Default = 0.0)
B2	Coefficient b2. (Real; Default = 0.0)

Descriptor	Meaning
B3	Coefficient b3. (Real; Default = 0.0)
HVL	Hydrodynamic volume limit. (Real > 1.0; Default = 1.1)
VISC	Viscosity coefficient.(Real; Default = 0.0)

Remarks:

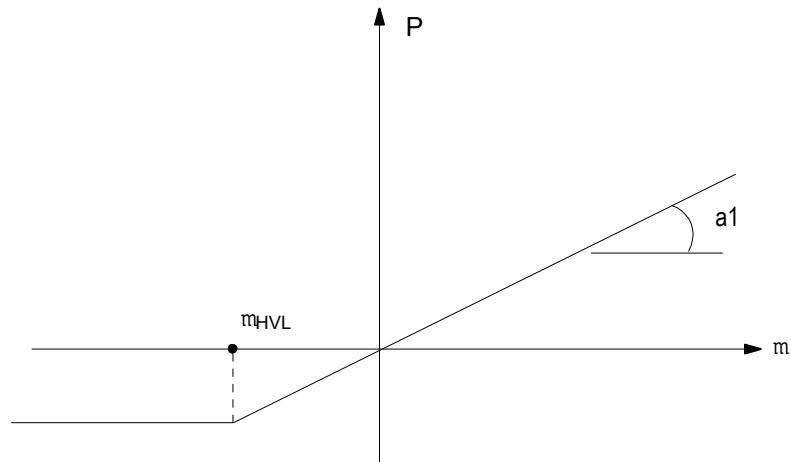
- When the relative volume (ρ_0/ρ) exceeds HVL, the pressure is cut off to

$$P_{HVL} = f(\mu_{HVL})$$

with

$$\mu_{HVL} = \frac{1}{HVL} - 1$$

e.g., for $p = a_1 \cdot \mu$, the pressure behavior is as follows:



- When the PARAM,HVLFAIL is set to YES, the elements where the relative volume (ρ_0/ρ) exceeds HVL fail completely. Their stress state is zero.

EOSUDS**User-defined Equation of State - SOL 700 only**

Defines the frequency dependent properties for an isotropic poroelastic material. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
EOSUDS	EID	GROUP	UNAME	VISC					

Example:

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE water 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

In Bulk Data:

1	2	3	4	5	6	7	8	9	10
EOSUDS	12	WATER	EXEOS	.01					

Descriptor	Meaning
EID	Unique equation of state number. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. (Character; default=EXEOS)
VISC	Viscosity coefficient R > 0 No viscosity. See Remarks 4.and 5.

Remarks:

1. The EID must be referenced by a MATDEUL or MAT1 entry.
2. The equation of state name is passed to method usrEOS of the SCAIMDSolver700 interface and can be used to identify the equation of state.
3. The viscosity coefficient is the dynamic viscosity. It is the ratio between shear stress and velocity gradient.
4. For the single mat solver viscous stresses will be stored in the output files by the use of TXX through TZx. Also, EFFSTS is available. For the multi-material solver viscous stresses are stored in TXX-VIS, TYY-VIS, TZZ-VIS, TXY-VIS, TYZ-VIS, TZx-VIS. These viscous stresses only depend on the current velocity gradients. The stresses like TXX are elastic-plastic stresses and depend on past stresses. The total stress tensor in the element is given by the average of the viscous stress and elastic-plastic stress. The weight factors are the material fraction of viscous fluid and the remaining materials.
5. UNAME can be:

Subroutine Name	Function
EXEOS	Standard user defined Equation of State

EPOINT

Extra Point List

Defines extra points for use in dynamics problems.

Format:

1	2	3	4	5	6	7	8	9	10
EPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	

Example:

EPOINT	3	18	1	4	16	2			
--------	---	----	---	---	----	---	--	--	--

Alternate Format and Example:

EPOINT	ID1	"THRU"	ID2						
EPOINT	17	THRU	43						

Descriptor	Meaning
IDi	Extra point identification number. (100,000,000 > Integer > 0; for "THRU" option, ID1 < ID2).

Remarks:

1. All extra point identification numbers must be unique with respect to all other structural, scalar, and fluid points for direct methods of solution. For modal methods, they must be larger than the number of eigenvectors retained for analysis.
2. EPOINT is used to define coordinates used in transfer function definitions (see the TF and DMIG entries).
3. If the alternate format is used, extra points ID1 through ID2 are also defined to be extra points.
4. See the *MSC Nastran Dynamic Analysis User's Guide* for a discussion of extra points.

ERPPNL**Equivalent Radiated Power Definition**

Defines one or more panels by referencing sets of elements or properties.

Format:

1	2	3	4	5	6	7	8	9	10
ERPPNL	NAME1	SETID1	NAME2	SETID2	NAME3	SETID3	NAME4	SETID4	
	NAME5	SETID5							

Example:

ERPPNL	ROOF	1	DOORLF	16					
--------	------	---	--------	----	--	--	--	--	--

Descriptor	Meaning
NAMEi	Panel label. (CHAR)
SETIDi	Identification number of a SET3 Bulk Data entry that lists the panel property entries or the panel elements. (Integer > 0)

Remarks:

1. The SET3 entries can only refer to 2D and/or 3D structural element types, such as QUAD4, TRIA3, QUADR, CTRIAR, QUAD8, TRIA6, HEXA, PENTA and TETRA, or PSHELL, PCOMP, PCOMPG and PSOLID entry associated with those element types.
2. NAMEi are used in a Case Control SET definition defining *setp* to select the panels in the Case Control command ERP.
3. Duplicate Element IDs are removed if they occur on the referencing SET3 entries prior to the ERP calculation.
4. User does not need to specify the outer face(s) of 3D elements. The outer face(s) or exposed face(s) of 3D elements will be determined automatically. Interior 3D element(s) which has no exposed face(s) will not be included for ERP computation.

EULFOR

Body Force Loading on Euler Elements - SOL 700 only

Defines a body force loading (acceleration) on Euler elements per unit mass. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
EULFOR	BID	CID	VALUE	TID	N1	N2	N3		

Example:

EULFOR	100	5		13	1.	0.	0.		
--------	-----	---	--	----	----	----	----	--	--

Descriptor	Meaning
BID	Unique body force number. (Integer > 0; Required)
CID	ID of a coordinate system. (Integer ≥ 0 ; Default = 0)
VALUE	Scale factor for the vector. See Remark 1. (Real ≥ 0.0)
TID	TABLED1 ID describing the scale factor for the load as function of time. See Remark 1. (Integer)
N1, N2, N3	Components of a vector giving the load (acceleration) direction defined in coordinate system CID. At least one must be nonzero. See Remark 2. (Real; Default = 0.0)

Remarks:

1. Either VALUE or TID must be nonzero.
2. By default the components are zero, but at least one of them should be nonzero.

EULFOR1

Body force for Eulerian Regions - SOL700

Alternative way to define an acceleration within a geometric region of the Euler model regions are defined by geometric shapes which are defined by EULFREG entries.

Format:

EULFOR1	SID	ESID						
---------	-----	------	--	--	--	--	--	--

Example:

EULFOR1	300	1						
---------	-----	---	--	--	--	--	--	--

Descriptor	Meaning
SID	Unique EULFOR1 number referenced from a PEULER1 entry .(Integer >0, required)
ESID	Group of geometric region EULFREG ID. (Integer > 0; Required)

Remarks:

1. EULFOR1 and EULFREG are only available for the multi-material Euler solver.
2. The combination of Eulerian region and material ID determines where the acceleration field is applied.
3. It is allowed to cover only part of the Euler domain with EULFOR1 definitions.
4. To increase the accuracy of the region definition parameter MICRO can be used.

EULFREG

Body force for Eulerian Regions - SOL700

Defines the acceleration field for sets of Eulerian regions. The Eulerian regions are defined by geometric shapes. For each coordinate direction a time-depended acceleration can be defined.

Format:

EULFREG	ERID	ESID	TYPE1	VID1	MID1	ACCX	ACCY	ACCZ	
	LEVEL								

Example:

EULFREG	300	1	BOX	400	100	100	200	300	
	0.0								

Descriptor	Meaning
ERID	Unique ERID number. (Integer > 0; Required)
ESID	ID of group of Euler regions referenced from the EULFOR1 entry. (Integer > 0; Required)
TYPEi	The type of Eulerian region. (Character; Required)
	SURF Region inside or outside a multifaceted surface.
	SPHERE Region inside a sphere.
	CYLINDER Region inside a cylinder.
	BOX Region inside a box.
	ELEM Region defined by element list.
VIDi	Number of a geometric entity, a SET1 number, or number of a SURFINI entry. (Integer > 0; Required)
MIDI	Number of a MATDEUL entry to which the acceleration field will be applied. (Integer > 0; Required)
ACCi	Unique table number that defines the variation of acceleration in time. ACCX, ACCY and ACCZ respectively denote the acceleration in the x-, y- and z-direction. (Integer > 0; Required)
LEVELi	Level indicator for this material and initial values. (Real; default=0.0)

Remarks:

1. EULFOR1 and EULFREG are only available for the multi-material Euler solver.
2. The combination of Eulerian region and material ID determines where the acceleration field is applied.
3. It is allowed to cover only part of the Euler domain with EULFOR1 definitions.

4. All level indicators LEVELi must have different values. The level indicator can be negative.
5. To increase the accuracy of the region definition parameter MICRO can be used.

EXCLUDE

Ignore Bulk Data entries from primary to secondary Module

Specifies Bulk Data entries in the primary Module to be ignored in the secondary (or copied) Module.

Format:

1	2	3	4	5	6	7	8	9	10
EXCLUDE	ENTRY1	ENTRY2	ENTRY3	-etc.,					

Example:

EXCLUDE	SPC	FORCE							
---------	-----	-------	--	--	--	--	--	--	--

Descriptor	Meaning
ENTRYi	Name of a Bulk Data entry to be ignored.

Remarks:

1. A secondary Module may have its own BEGIN MODULE section but it is not required as long as the user is willing to accept all modeling data from the reference Module because, by default, all entries in the reference Module will be copied to the secondary Module. However, if the user wishes to exclude and/or replace some of the entries from the reference Module in the copy then he simply specifies at least one instance of the entry to be ignored. For example, if the user wants to ignore all FORCE entries from the reference Module then specify a single FORCE entry in the secondary Module's Bulk Data section. Alternatively, the user may specify a list of Bulk Data entries to ignore on the EXCLUDE Bulk Data entry in the secondary Module's Bulk Data section.
2. EXCLUDE must be specified in the secondary (or copied) Module's Bulk Data section.
3. GRID entries cannot be ignored or replaced and EXCLUDE,GRID will cause a fatal error.
4. See [MDBULK](#) for the definition of secondary (or copied) Modules.

EXTRN**Partitioned External Superelement Connection**

Defines a boundary connection for an external superelement.

Format:

1	2	3	4	5	6	7	8	9	10
EXTRN	GID1	C1	GID2	C2	GID3	C3	GID4	C4	
	-etc.-		GID6	"THRU"	GID7	C6	-etc.-		

Example:

EXTRN	1001	123	1120	123456	1201	123			
-------	------	-----	------	--------	------	-----	--	--	--

Descriptor	Meaning
GID _i	Grid identification number to which the exterior superelement matrices will be connected.
C _i	Component numbers. (Integer 0, blank, or 1 for scalar points; Integers 1 through 6 with no embedded blanks for grids.)

Remarks:

1. EXTRN can only be specified in partitioned Bulk Data Sections and is ignored in the main Bulk Data Section.
2. Connection grids must be specified in the partitioned Bulk Data Section following BEGIN SUPER = SEID.
3. "THRU" may be specified only in fields 3, 5, or 7.
4. Pairs of blank fields may be entered to allow easier modification of the EXTRN entry.
5. The order of the GID_i and C_i pairs must be in the internal order of the grids set in the creation run. But since Nastran no longer re-sequences grids then the pairs are specified in ascending GRID ID order. However, if PARAM,OLDSEQ or the SEQGP entry is specified then the proper order may not be ascending. To determine the internal grid order, add PARAM,USETPRT,0 and PARAM,USETSTR1,A.

Entries F - L

FAILJC

Johnson-Cook Failure Model - SOL 700 only

Defines the properties of the Johnson-Cook failure model. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FAILJC	FID	D1	D2	D3	D4	D5	$\dot{\varepsilon}_{pl}^0$	TROOM	
	TMELT	CP	MTH						

Example:

FAILJC	1	.05	3.44	-2.12	0.002	0.16	1.0	297.0	
	1495	450	CONT						

Descriptor	Meaning
FID	Unique failure model number. Referenced from MATDEUL. (Integer > 0; Required)
D1...D5	Parameters. See Remark 4. (Real; Default = 0.0)
$\dot{\varepsilon}_{pl}^0$	Reference plastic strain rate. (Real; Default = 1.0)
TROOM	Room temperature. (Real; 0.0)
TMELT	Melt temperature. (Real; 1.E+20)
CP	Heat capacity. (Real; 1.E+20)
MTH	Specifies how failure is applied. (Character; CONT)
	CONT Continuous failure
	DISC Discrete failure
	NOFAIL Damage is not used for failure

Remarks:

1. This failure model is only available for Eulerian materials using the multi-material solver with strength.
2. The use of coupling surfaces is not supported.
3. The variable D can be visualized by adding DAMAGE to the Output request for Euler elements.
4. Defines the properties of a failure model where failure is determined by a damage model. The damage model is given by:

$$D = \sum_{time} \frac{\Delta \varepsilon_p}{\varepsilon^{frac}}$$

$$\varepsilon^{frac} = (D_1 + D_2 \exp(D_3 \sigma^*)) \left(1 + D_4 \ln \frac{\dot{\varepsilon}_{pl}}{\dot{\varepsilon}_{pl}^0} \right) (1 + D_5 T^*)$$

$$\sigma^* = \frac{\sigma_m}{\bar{\sigma}}$$

$$T^* = \frac{T - T_{room}}{T_{melt} - T_{room}}$$

The summation is performed over all past time increments. The variable D measures the damage; T is the temperature, σ is the mean stress, σ^* is the effective stress and $\dot{\varepsilon}_{pl}$ is the fracture strain. The fracture strain depends on a non-dimensional plastic strain rate $\dot{\varepsilon}_{pl}/\dot{\varepsilon}_{pl}^0$. If D exceeds one it is set equal to one. The damage variable D is transported along with the Eulerian material.

There are two methods to determine when elements fail:

- Continuous failure: The yield stress is reduced by a factor (1-D). When D exceeds 1 the yield stress equals zero and the element fails.
- Discrete failure: the element fails when D equals one.

This failure model applies to high-strain rate deformation of metals. It is less suitable for quasi-static problems.

FAILMPS

Maximum Plastic Strain Failure Model - SOL 700 only

Defines the properties of a failure model where failure occurs when the equivalent plastic strain exceeds the specified value. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FAILMPS	FID	MPS							

Example:

FAILMPS	1	.15							
---------	---	-----	--	--	--	--	--	--	--

Descriptor	Meaning
FID	Unique failure model number referenced from MATDEUL. (Integer > 0; Required)
MPS	Maximum plastic strain that causes failure. (Real; Required)

FAILUDS**User Defined Failure For Nonlinear Explicit Analysis - SOL 700 only**

User defined simple failure of Eulerian materials. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FAILUDS	FID	GROUP	UNAME						

Example:

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE usrfail 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

In Bulk Data:

1	2	3	4	5	6	7	8	9	10
FAILUDS	200	usrfail	EXFAIL						

Descriptor	Meaning
FID	Unique output number. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. (Character; default=EXFAIL)

Remarks:

1. Only can be used for SOL 700.
2. FID has to be referenced by a MATDEUL, MAT1 or MATORT entry.
3. UNAME can be:

Subroutine Name	Function
EXFAIL	Standard user defined failure
EXFAIL1	Alternative 1 user defined failure
EXFAIL2	Alternative 2 user defined failure

4. For option UNAME=EXFAIL2, for each material and for each Euler element a variable will be created that monitors the degree of failure of the material. This variable is denoted by DAMAGE and is between 0 and 1. The EXFAIL2 routine allows updating this damage variable due to the plastic strain increment of the current cycle.
5. For option UNAME=EXFAIL2 there are three ways in which this damage variable can model failure. These are:

- Continuous failure: The yield stress is reduced by a factor (1-D). When D exceeds 1.0, the yield stress equals zero and the element fails.
 - Discrete failure: the element fails when D equals one.
 - No failure: positive damage values will not lead to failure. This is useful if the failure modeling is done by an YLDUDS routine. Then the yield stress can be reduced depending on the magnitude of the damage variable.
6. UNAME=EXFAIL2 is only supported by the multi-material Euler solver with strength.
 7. UNAME=EXFAIL1 is only supported with MATORT.
 8. For Lagrangian materials, UNAME=EXFAIL or EXFAIL1 are only available.

FBADLAY**Dynamic Load Time Delay for FRF Based Assembly (FBA)**

Defines the time delay term τ in the equations of motion of the dynamic loading function for frequency dependent loads in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FBADLAY	SID	COMPID/ COMPNAME	PNTID	C	DELAY				

Examples:

FBADLAY	15	BODY	10	3	0.5				
---------	----	------	----	---	-----	--	--	--	--

FBADLAY	25	30	5	2	0.1				
---------	----	----	---	---	-----	--	--	--	--

Descriptor	Meaning
SID	Identification number. See Remark 1. (Integer > 0)
COMPID	Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed)
PNTID	Grid or scalar point identification number. See Remark 3. (Integer > 0)
C	Component number. See Remark 3. (Integer 1 through 6 for grid point; blank or 0 for scalar point)
DELAY	Time delay term τ . See Remark 4. (Real)

Remarks:

1. SID is referenced by RLOAD1, RLOAD2 and ACSRCE entries.
2. This entry is ignored if the specified COMPID/COMPNAME is not part of the FBA process. A user warning message is issued in this case.
3. The component C of the point PNTID specified in this entry must be among the excitation degrees of freedom of the FBA process. If not, the program terminates the job with a user fatal message.
4. Refer to RLOAD1, RLOAD2 and ACSRCE entries for the formulas that define the time delay term τ in frequency response analysis.
5. All FBADLAY entries specified in an FBA process are automatically converted internally by the program to equivalent DELAY entries by replacing the grid/scalar point IDs referenced in these entries by equivalent internal point IDs.

FBALOAD

Load Scale Factor Specification for FRF Based Assembly (FBA) Process

Defines the scale factor for frequency dependent loads in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FBALOAD	SID	COMPID/ COMPNAME	PNTID	C	A				

Examples:

FBALOAD	10	WING	20	3	2.5				
FBALOAD	20	30	25	1	1.5				

Descriptor	Meaning
SID	Identification number. See Remark 1. (Integer > 0)
COMPID	Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Optional name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed).
PNTID	Grid or scalar point identification number. See Remark 3. (Integer > 0)
C	Component number. See Remark 3. (Integer 1 through 6 for grid point; blank or 0 for scalar point)
A	Scale factor. See Remark 4. (Real)

Remarks:

1. SID is referenced by RLOAD1, RLOAD2 and ACSRCE entries.
2. This entry is ignored if the specified COMPID/COMPNAME is not part of the FBA process. A user warning message is issued in this case.
3. The component C of the point PNTID specified in this entry must be among the excitation degrees of freedom of the FBA process. If not, the program terminates the job with a user fatal message.
4. Refer to RLOAD1, RLOAD2 and ACSRCE entries for the formulas that define the scale factor A in frequency response analysis.
5. All FBALOAD entries specified in an FBA process are automatically converted internally by the program to equivalent DAREA entries by replacing the grid/scalar point IDs referenced in these entries by equivalent internal point IDs.

FBAPHAS**Dynamic Load Phase Lead for FRF Based Assembly (FBA)**

Defines the phase lead term θ in the equations of motion of the dynamic loading function for frequency dependent loads in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FBAPHAS	SID	COMPID/ COMPNAME	PNTID	C	PHASE				

Examples:

FBAPHAS	25	FRAME	40	1	10.0				
FBAPHAS	30	50	10	2	5.0				

Descriptor	Meaning
SID	Identification number. See Remark 1. (Integer > 0)
COMPID	Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed)
PNTIDi	Grid or scalar point identification numbers. See Remark 3. (Integer > 0)
C	Component number. See Remark 3. (Integers 1 through 6 for grid points; blank or 0 for scalar points.)
PHASE	Phase lead term θ . See Remark 4. (Real)

Remarks:

1. SID is referenced by RLOAD1, RLOAD2 and ACSRCE entries.
2. This entry is ignored if the specified COMPID/COMPNAME is not part of the FBA process. A user warning message is issued in this case.
3. The component C of the point PNTID specified in this entry must be among the excitation degrees of freedom of the FBA process. If not, the program terminates the job with a user fatal message.
4. Refer to RLOAD1, RLOAD2 and ACSRCE entries for the formulas that define the phase lead term θ in frequency response analysis.
5. All FBAPHAS entries specified in an FBA process are automatically converted internally by the program to equivalent DPHASE entries by replacing the grid/scalar point IDs referenced in these entries by equivalent internal point IDs.

FBODYLD

Equilibrated Free-Body Applied Load Case Definition

Defines an equilibrated free-body applied load case.

Format:

1	2	3	4	5	6	7	8	9	10
FBODYLD	NAMEL	FBODYSB							
	LABEL								

Example:

FBODYLD	WINGLD	WINGSB							
	LOAD ON THE RIGHT WING								

Descriptor	Meaning
NAMEL	User defined name identifying the load case. (Character; Required)
FBODYSB	Name of a FBODYSB Bulk Data entry that defines the subsystem for this load. (Character; Required)
LABEL	A string comprising no more than 64 characters (fields 2 through 9) that identifies and labels the load case. (Character; optional)

Remarks:

1. NAMEL must be unique.
2. The Label is optional.

FBODYSB**Equilibrated Free-Body Subsystems Definition**

Defines an equilibrated free-body subsystem.

Format:

1	2	3	4	5	6	7	8	9	10
FBODYSB	NAMES	GRIDSET	ELEMSET	XFLAG					
	LABEL								

Example:

FBODYSB	WING	1	1	ADM					
	RIGHT WING								

Descriptor	Meaning	
NAMES	User defined name identifying the submodel. (Character; Required)	
GRIDSET	Identification number of a SET1 entry that has a list of Grid Point Force grids to include in defining the subsystem. (Integer > 0)	
ELEMSET	Identification number of a SET1 entry that has a list of elements to include in the system (Integer > 0 or blank)	
XFLAG	Exclusion flag. Exclude the indicated Grid Point Force types.	
	Default	Blank (no type excluded)
	S	SPC forces
	M	MPC forces
	A, L, or P	Applied loads
	D	DMIG's (and any other type not described above)
Label	An optional string of up to 64 characters (fields 2 through 9) that identifies the subsystem.	

Remarks:

- Only those Grid Point Forces which have both an included grid point and element (or Grid Point Force type) will be taken into account.
- If ELEMSET is blank, no contributions are made from the set of elements attached to the grid.
- Fictitious grids or elements do not produce error or warning messages.
- The XFLAG data can be any combination of the letters S,M,A,L,P and D (e.g., MAD).
- The continuation is optional.

FEEDGE**Finite Element Edge Definition**

Defines a finite element edge and associates it with a curve.

Format:

1	2	3	4	5	6	7	8	9	10
FEEDGE	EDGEID	GRID1	GRID2	CIDBC	GEOMIN	ID1	ID2		

Example:

FEEDGE	101	123	547		GMCURV	12			
--------	-----	-----	-----	--	--------	----	--	--	--

Descriptor	Meaning			
EDGEID	Unique identification number.	Integer > 0	Required	
GRIDi	Identification number of end GRIDs defining this edge.	Integer > 0	Required	
CIDBC	Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined. See Remark 1.	Integer ≥ 0	Required	
GEOMIN	Type of entry referenced by IDi; “GMCURV” or “POINT”. See Remark 2.	Character	POINT	
IDi	Identification number of a POINT or GMCURV entry. See Remarks 2., 3., and 4.	Integer ≥ 0		

Remarks:

- If CIDBC is not blank then it overrides the CIDBC specified on the GMSURF or FEFACE entries for this particular edge. A fatal message will be issued when more than one CIDBC is associated with any entity.
- The Bulk Data entries referenced by ID1 and ID2 depends on the GEOMIN field:

GEOMIN	ID1	ID2
POINT	POINT	POINT
GMCURV	GMCURV	not applicable

- When GEOMIN = “GMCURV”

- FEEDGE associates the finite element model and the geometric information.
- GRID1 and GRID2 are the end points of the edge, and the edge is on the CURVID curve. A locally parametric cubic curve is fit to the geometric curve such that the two have the same tangent at GRIDi (C1 continuous).

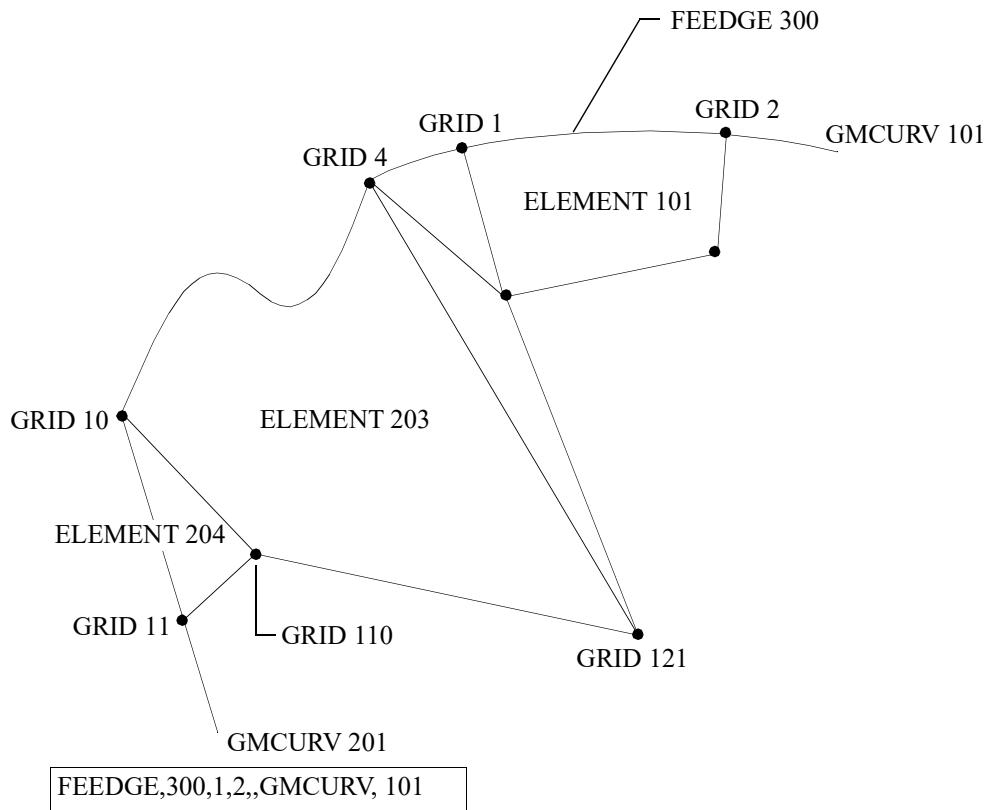


Figure 9-91 Specifying Geometry Using GEOMIN=GMCURV Method

4. When GEOMIN = “POINT”

- The edge passes through the points defined on the POINT entries referenced by ID1 and ID2.
- The shape of the edge is selected as follows:

ID1	ID2	Shape of the FEEDGE
Blank or 0	Blank or 0	Linear
>0	Blank or 0	Quadratic
>0	>0	Cubic
Blank or 0	>0	Not allowed

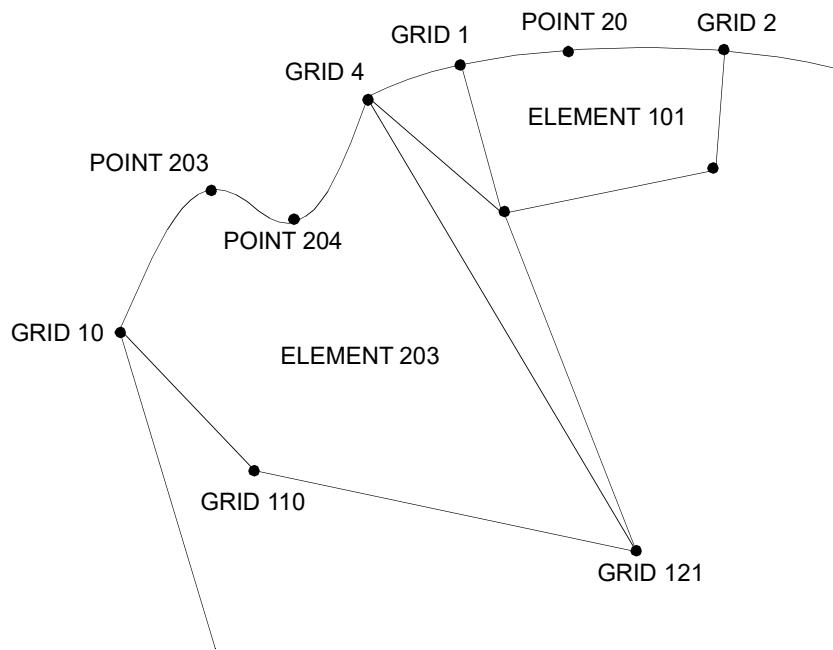


Figure 9-92 Specifying Geometry Using GEOMIN=POINT Method

5. A local coordinate system can be associated with an edge using the GMCORD entry.
6. The hierarchy set to resolve the conflicts arising in the Global System input data is described under Remark 10 of the [GMBC, 2072](#) entry description.

FEFACE**Finite Element Face Definition**

Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.

Format:

1	2	3	4	5	6	7	8	9	10
FEFACE	FACEID	GRID1	GRID2	GRID3	GRID4	CIDBC	SURFID		

Example:

FEFACE	101	123	547	243	295	12			
--------	-----	-----	-----	-----	-----	----	--	--	--

Descriptor	Meaning		
FACEID	Unique identification number. See Remark 1.	Integer > 0	Required
GRIDi	Identification number of end GRIDs defining a triangular or quadrilateral face. See Remark 2.	Integer > 0	Required
CIDBC	Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined. See Remark 3.	Integer ≥ 0	Remark 3.
SURFID	Alternate method used to specify the geometry of the edges of the face. See Remarks 4. and 5.	Integer ≥ 0	0

Remarks:

1. An FEFACE entry is required if any of the following situations exist:
 - The geometry of the surface defined by SURFID is to be used by a finite element;
 - CIDBC is specified for a face or surface; or
 - If loads or constraints or enforced boundary conditions are applied to a surface.
2. The shape (geometry) of the face is defined by the shape of the edges. The points defined by GRIDi must be specified in either a clockwise or counterclockwise order.
3. If CIDBC is not blank, then it overrides the CIDBC specified on the GMSURF entry for this particular face. A fatal message will be issued when more than one CIDBC is associated with any entity.
4. When SURFID is blank or 0, the edges will be considered linear unless there is an FEEDGE entry for the given edge.
5. When SURFID > 0 ,
 - FEFACE associates the finite element model and the geometric information specified on the GMSURF entry.

- GRIDi defines a finite element face (clockwise or counter clockwise in order) that is on the SURFID surface.
 - For the edges of this face, which are not defined by an FEEDGE entry, locally parametric cubic curves are fit to the geometric surface such that the two have the same tangent at GRIDi (C1 continuous).
6. Whenever a given edge of a face is common to two or more surfaces (i.e., lies on the intersecting curve), then the user must supply GMCURV and FEEDGE entries in order to resolve the conflict in the input geometry. A fatal message is issued if an edge is not uniquely defined.

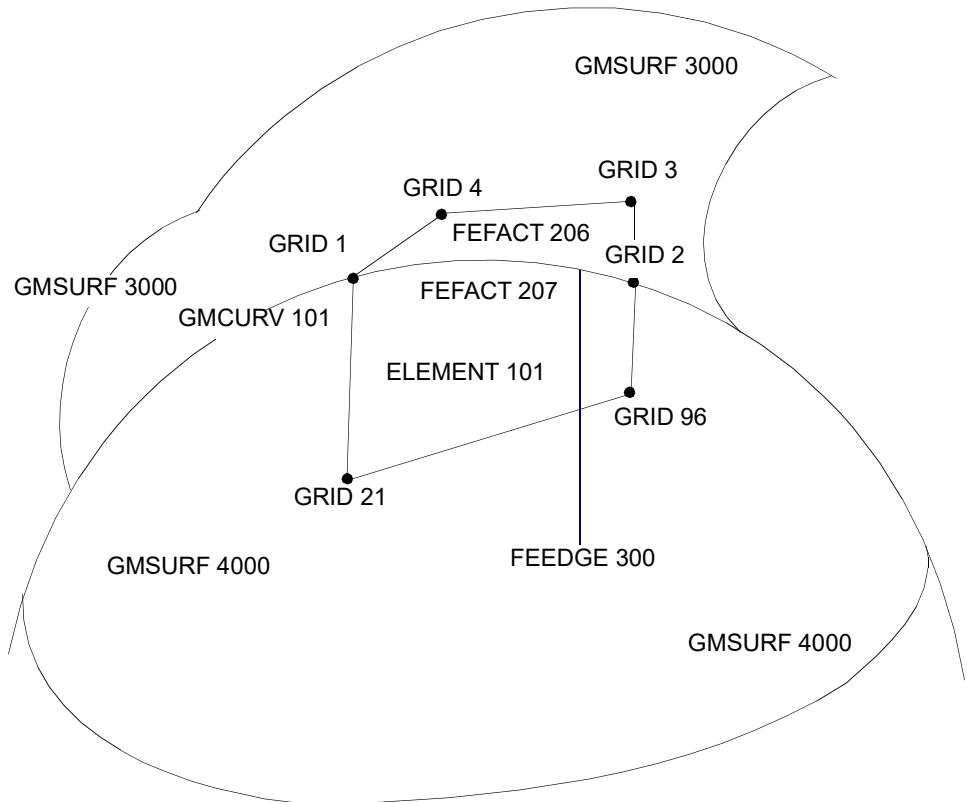


Figure 9-93 Face Edge Common to Two Surfaces

7. The hierarchy set to resolve the conflicts arising in the Global System input data is described under Remark 10 of the [GMBC, 2072](#) entry description.

FFCONTR**Closed Volume Intended for Fluid Filled Containers**

Defines the pressure within a closed volume. Intended for the use in (partially) filled containers, where dynamic fluid effects are negligible, e.g. top loading and hot filling. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FFCONTR	FFID	SID	FVOL	PATM	TEMPTAB	DENSTAB	TACTIVE		

Example:

FFCONTR	1	2	1.50E-03	0.1E6	10	20			
---------	---	---	----------	-------	----	----	--	--	--

Descriptor	Meaning
FFID	Unique FFCONTR identification number. (Integer > 0; Required)
SID	Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG entry defining the closed surface. (Integer > 0; Required)
FVOL	Fluid volume in the container. (Real ≥ 0.0 ; Required)
PATM	Atmospheric pressure. Used for determination of the constant C for $p \cdot V = C$. (Real > 0.0 ; Required)
TEMPTAB	A reference to a TABLED1 ID that specifies how temperature of the container changes in time. (Integer ≥ 0 ; Blank)
DENSTAB	A reference to a TABLED1 ID that specifies how density of the container changes with temperature. (Integer ≥ 0 ; Blank)
TACTIVE	Time at which the pressure computation of the bottle is started. Until this time the pressure inside the bottle equals the ambient pressure. The volume of the bottle at TACTIVE will be used for the initial pressure computation of the gas in the bottle.

Remarks:

1. If TEMPTAB is not set, the gas above the fluid is assumed to be an ideal, iso-thermal gas: $p \cdot V = C$, where C is a constant. If TEMPTAB is set, the temperature is applied to both the fluid as well as the gas. Then the gas satisfies $p \cdot V/T = C$, where T is the temperature of the fluid.
2. The fluid is assumed incompressible.
3. The pressure is based on the uniform pressure gasbag algorithm, where the pressure is uniform in the volume, but variable in time.
4. Output for the fluid-filled container is available through a SURFOUT definition. The available variables are: PRESSURE, VOLUME, TEMPTURE, VOLGAS, VOLFLUID, GAUGEPRES and RHOFLUID.
5. The normals of the surface referenced by SID are reversed automatically if required.

6. Modeling guidelines are described in the “Getting Started” Section.
7. If DENSTAB is set then volume of the fluid changes according to

$$V^{Fluid} = \frac{\rho(T_0)V_0^{Fluid}}{\rho(T)}$$

Here T_0 and V_0^{Fluid} are initial values for temperature and fluid volume, ρ is the fluid density and T denotes the current temperature. If TEMPTAB is not set the DENSTAB entry will not be used.

8. At time=TACTIVE the gas is assumed to be in contact with the ambient pressure for the last time. This means that at Time = TACTIVE the pressure in the bottle equals the ambient pressure. After TACTIVE the bottle has been closed and there is no longer contact between ambient and gas inside the bottle. Any change in volume of the bottle or temperature or fluid will result in change of pressure of the gas inside the bottle.

FLFACT**Aerodynamic Physical Data**

Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.

Format:

1	2	3	4	5	6	7	8	9	10
FLFACT	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	-etc.-						

Example:

FLFACT	97	.3	.7	3.5					
--------	----	----	----	-----	--	--	--	--	--

Alternate Format and Example:

FLFACT	SID	F1	"THRU"	FNF	NF	FMID			
FLFACT	201	.200	THRU	.100	11	.133333			

Descriptor	Meaning
SID	Set identification number. (Unique Integer > 0)
Fi	Aerodynamic factor. (Real)
FNF	Final aerodynamic factor. (Real)
NF	Number of aerodynamic factors. (Integer > 0)
FMID	Intermediate aerodynamic factors. See Remark 4. (Real)

Remarks:

- Only the factors selected by a FLUTTER entry will be used.
- Embedded blank fields are not allowed in the first format above.
- The factors must be specified in the order in which they are to be used within the looping of flutter analysis.
- $FMID$ must lie between $F1$ and FNF ; otherwise, $FMID$ will be set to $(F1 + FNF)/2$. Then

$$F_i = \frac{F1(FNF - FMID)(NF - i) + FNF(FMID - F1)(i-1)}{(FNF - FMID)(NF - i) + (FMID - F1)(i-1)}$$

where $i = 1, 2, \dots, NF$

The use of $FMID$ (middle factor selection) allows unequal spacing of the factors.

$FMID = \frac{2 \cdot F1 \cdot FNF}{F1 + FNF}$ gives equal values to increments of the reciprocal

of F_i .

5. If method = PK and this entry specifies velocities, then the velocities must be non-zero. Input of negative values produces eigenvector results at a velocity equal to the positive value of the input. Input of positive values provide eigenvalues results without eigenvectors.

FLOW**Flow Boundary Condition**

Defines the properties of a material for the boundaries of an Eulerian mesh. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FLOW	LID	BCID	MESH	DIR					
	XMIN	XMAX	YMIN	YMAX	ZMIN	ZMAX			
	TYPE1	VALUE1	TYPE2	VALUE2	TYPE3	VALUE3	TYPE4	VALUE4	
	TYPE5	VALUE5							

Example:

FLOW	120	122							
	XVEL	100.0							

Descriptor	Meaning	
LID	Number of a set of flow boundary conditions. (Integer > 0; Required)	
BCID	Number of a set of segments, specified by BCSEG entries, where the flow boundary is located. See Remark 5. (Integer ≥ 0)	
MESH	Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 6. (Integer ≥ 0)	
DIR	Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 7. (Character)	
XMIN-ZMAX	Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 8. (Real)	
TYPEi	The flow boundary property being defined. (Character)	
	Material	The MATDEUL ID number.
	XVEL	The material velocity in the x-direction.
	YVEL	The material velocity in the y-direction.
	ZVEL	The material velocity in the z-direction.
	PRESSURE	The pressure of the material at the boundary.
	DENSITY	The density of the material at inflow.
	SIE	The specific internal energy at inflow

Descriptor	Meaning
	FLOW The type of flow boundary required.
	HYDSTAT A Hydrostatic pressure profile using a HYDSTAT entry.
TYPEi	The value for the property defined. (Real; Integer or Character; Required) For TYPEi set to FLOW, the value is a character entry being either IN, OUT, BOTH or SYM defining that the flow boundary is defined as an inflow, outflow, possibly an in- or outflow or symmetry boundary. The default is BOTH. TYPEi is required data only if one or more of the TYPEi entries are defined. The TYPEi entries are not required. Thus, a flow boundary by default allows for in- or outflow of the material adjacent to the boundary. For TYPE = HYDSTAT, the value is an integer entry denoting the HYDSTAT entry to be used.

Remarks:

1. LID must be referenced by a TLOAD1 entry.
2. Any material properties not specifically defined have the same value as the element with the flow boundary condition.
3. TLOAD entries referencing FLOW entries must have the TID field blank or zero.
4. In the case of material flow into a multi-material Euler mesh, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions.
5. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
6. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
7. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
8. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1E+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the Min value.
9. Prescribing both pressure and velocity may lead to the instabilities.
10. For TYPE = HYDSTAT, the pressure is set using HYDSTAT, the velocity equals the element velocity. In case of inflow the density follows from the hydrostatic pressure by using the equation of state.

FLOWC**Cyclic Flow Boundary Condition**

Defines the properties of a material for the boundaries of a Eulerian mesh.

Inflow values can be taken from another boundary condition. This allows cyclic or periodic boundary conditions. Likewise, the outflow of material goes into the other boundary condition.

FLOWC entries have to be defined in pairs. The FID on one entry has to be equal to FID2 of the other entry. For example

`FLOWC, 10, 30, 20`

`FLOWC, 20, 40, 10`

For FLOWC boundaries, mass flow summaries can be created as time history. Used in SOL700 only

Format:

1	2	3	4	5	6	7	8	9	10
FLOWC	FID	BCID	TYPE	MESH	DIR	FID2			+
+	XMIN	XMAX	YMIN	YMAX	ZMIN	ZMAX			+
+	METHOD	TID							

Example:

FLOWC	1	122	2						
+									
+	2	1							

Field	Contents
FID	Unique number of a FLOWC entry. (Integer > 0; Required)
BCID	Number of a set of segments specified by the BCSEG entries where the flow boundary is located. See Remark 3. (Integer > 0)

Field	Contents
TYPE	<p>Flow boundary types. See Remarks 2.and 3. (Character; Required)</p> <p>IN: Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.</p> <p>OUT: Only outflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use material mixture as present in the adjacent Euler element.</p> <p>BOTH: Material is allowed to flow in or out. In or outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be optionally defined. If not given the pressure in the adjacent Euler element will be taken.</p>
MESH	Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 4. (Integer > 0)
DIR	Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 5. (Character; no default)
FID2	Referenced FLOWC id from which inflow and outflow values will be taken from. (Integer > 0; Required)
XMIN-ZMAX	Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 6. (Real))
METHOD	<p>Method describing how material properties and pressure are mapped going from one boundary to the other. (Integer > 0; Default=1)</p> <ol style="list-style-type: none"> 1. Map both velocity, material flow properties, and pressure loads one-to-one. 2. Map velocity, material flow one-to-one. Pressure is given by element pressures. 3. Inflow is taken as the average of outflow properties. This applies to both velocity and material properties. Pressure is given by element pressures.
TID	TABLED1 ID that specifies a time dependent scale factor by which the amount of inflow is multiplied. This will not conserve total mass. (Integer > 0. Default = not used).

Remarks:

1. LID must be referenced by a TLOAD1 entry.
2. TLOAD entries referencing FLOW entries must have the TID field blank or zero.
3. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
4. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.

5. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
6. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1E+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the Min value.
7. FLOWC can be used to specify flow boundaries for CHEXA's and also for Euler element created by the MESH,BOX option.
8. FLOWC is only supported by the multi-material Euler solver.
9. To enable using the outflow values of FID2 as the inflow values for FID, the boundary faces of FID and FID2 are mapped onto each other. For this mapping, only translations and rotations around coordinate axes are permitted.
10. Consider a cubic Euler mesh and that material flows from the left to the right. At the right side, the boundary condition is imposed

FLOWC,4,30,,5

and at the left side

FLOWC,5,40,,4

These definitions cause all material that flows out of the right side boundary into the left side boundary. Moreover, the Euler element pressures on the right side are put on the Euler elements of the left boundary condition.

In practice, it may be useful to skip the coupling between the two boundaries with regard to pressure. This can be done by setting METHOD equal to 2.

With these definitions, material from boundary 4 still flows into boundary 5 but pressure boundaries are transmitting.

11. TID is useful if several objects have identical outflow that is used as inflow by another object. Then only one object has to be modeled. To account for the other objects when defining inflow, the scale factor can be used. It can also be used to turn off in and outflow. When TID is set, either METHOD=2 or METHOD=3 are recommended and METHOD=1 should not be used.
12. It is allowed that the definition of the FLOWC entry overlaps with FLOW definitions. In that case, the FLOWC definition overrules the other ones

FLOWDEF

Default Flow Boundary

Definition of default Eulerian flow boundary condition. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FLOWDEF	FID		TYPEM						
	TYPE1	VALUE1	TYPE2	-etc.-					

Example:

FLOWDEF	25		HYDRO						
	DENSITY	1000							

Descriptor	Meaning																		
FID	Unique FLOWDEF number. (Integer > 0; Required)																		
TYPEM	HYDRO, STRENGTH, MMHYDRO, or MMSTREN. (Character, HYDRO)																		
TYPEi	The flow boundary property being defined. (Character) <table> <tr> <td>Material</td> <td>The MATDEUL ID number.</td> </tr> <tr> <td>XVEL</td> <td>The material velocity in the x-direction.</td> </tr> <tr> <td>YVEL</td> <td>The material velocity in the y-direction.</td> </tr> <tr> <td>ZVEL</td> <td>The material velocity in the z-direction.</td> </tr> <tr> <td>PRESSURE</td> <td>The pressure of the material at the boundary.</td> </tr> <tr> <td>DENSITY</td> <td>The density of the material at inflow.</td> </tr> <tr> <td>SIE</td> <td>The specific internal energy at inflow</td> </tr> <tr> <td>FLOW</td> <td>The type of flow boundary required.</td> </tr> <tr> <td>HYDSTAT</td> <td>A Hydrostatic pressure profile using a HYDSTAT entry.</td> </tr> </table>	Material	The MATDEUL ID number.	XVEL	The material velocity in the x-direction.	YVEL	The material velocity in the y-direction.	ZVEL	The material velocity in the z-direction.	PRESSURE	The pressure of the material at the boundary.	DENSITY	The density of the material at inflow.	SIE	The specific internal energy at inflow	FLOW	The type of flow boundary required.	HYDSTAT	A Hydrostatic pressure profile using a HYDSTAT entry.
Material	The MATDEUL ID number.																		
XVEL	The material velocity in the x-direction.																		
YVEL	The material velocity in the y-direction.																		
ZVEL	The material velocity in the z-direction.																		
PRESSURE	The pressure of the material at the boundary.																		
DENSITY	The density of the material at inflow.																		
SIE	The specific internal energy at inflow																		
FLOW	The type of flow boundary required.																		
HYDSTAT	A Hydrostatic pressure profile using a HYDSTAT entry.																		
VALUEi	The value for the property defined. (Real; Integer or Character; Required) <p>For TYPEi set to flow the value is a character entry being either IN, OUT, or BOTH, defining that the flow boundary is defined as an inflow, outflow, or possibly an in- or outflow boundary. The default is BOTH.</p> <p>VALUEi is required data only if one or more of the TYPEi entries are defined. The TYPEi entries are not required. Thus, a flow boundary by default allows for in- or outflow of the material adjacent to the boundary.</p> <p>For TYPE = HYDSTAT, the value is an integer entry denoting the HYDSTAT entry to be used.</p>																		

Remark:

1. If this entry is not specified, a default wall boundary condition is applied to all Eulerian free faces.
2. For TYPE = HYDSTAT, the pressure is set using HYDSTAT, the velocity equals the element velocity. In case of inflow the density follows from the hydrostatic pressure by using the equation of state.

FLOWT

Time Dependent Flow Boundary

Defines the material properties for the in- or outflow of material through the boundary of an Euler mesh. Inflow velocity and material properties can be chosen time dependent. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FLOWT	FID	BCID	TYPE	MESH	DIR				
	XMIN	XMAX	YMIN	YMAX	ZMIN	ZMAX			
	VELTYPE	VELOCITY	PRESTYP	PRES					
	MID	DENSTYP	DENSITY	SIETYPE	SIE				

Example:

FLOWT	2	122	IN						
	TABLE	101	TABLE	102					
	91	TABLE	104	TABLE	107				

Descriptor	Meaning						
FID	Unique number of a FLOWT entry. (Integer > 0; Required)						
BCID	Number of a set of segments specified by the BCSEG entries where the flow boundary is located. See Remark 3. (Integer ≥ 0)						
TYPE	<table> <tr> <td>IN</td> <td>Inflow boundary. See Remarks 2 and 3. (Character; Required) Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.</td> </tr> <tr> <td>OUT</td> <td>Only outflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use material mixture as present in the adjacent Euler element.</td> </tr> <tr> <td>BOTH</td> <td>Material is allowed to flow in or out. In or outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be optionally defined. If not given the pressure in the adjacent Euler element will be taken.</td> </tr> </table>	IN	Inflow boundary. See Remarks 2 and 3. (Character; Required) Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.	OUT	Only outflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use material mixture as present in the adjacent Euler element.	BOTH	Material is allowed to flow in or out. In or outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be optionally defined. If not given the pressure in the adjacent Euler element will be taken.
IN	Inflow boundary. See Remarks 2 and 3. (Character; Required) Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.						
OUT	Only outflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use material mixture as present in the adjacent Euler element.						
BOTH	Material is allowed to flow in or out. In or outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be optionally defined. If not given the pressure in the adjacent Euler element will be taken.						
MESH	Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 4. (Integer ≥ 0)						
DIR	Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 5. (Character)						

Descriptor	Meaning
XMIN-ZMAX	Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 6. (Real)
VELTYPE	Type of velocity definition. (Character, Element) ELEMENT Value of Euler Element CONSTANT Value is constant in time TABLE Value varies in time
VELOCITY	Value of inflow or outflow velocity. If VELTYPE = TABLE it refers to a TABLED1. The velocity direction is normal to the coupling surface or subsurface. A positive velocity corresponds with inflow. See Remark 6. (Integer or Real)
PRESTYP	Type of pressure definition (Character, Element) ELEMENT Value of Euler element CONSTANT Value is constant in time TABLE Value varies in time
PRES	Value of inflow or outflow pressure. If PRESTYPE = TABLE it refers to a TABLED1 ID. (Integer or Real)
MID	MATDEUL ID of inflowing material. Input is not allowed for TYPE = OUT. When MID is specified, it is required to also define density and sie for the inflowing material. (Integer)
DESTYP	Type of density definition. (Character, Default is required when MID is given.) ELEMENT Value of Euler element CONSTANT Value is constant in time TABLE Value varies in time
DENSITY	Value of density. If DENSTYP = TABLE it refers to a TABLED1 ID. (Character; Default is required when MID is given.)
SIETYPE	Type of density definition. (Character; Default is required when MID is given.) ELEMENT Value of Euler element CONSTANT Value is constant in time TABLE Value varies in time.
SIE	Value of specific internal energy. If SIETYPE = TABLE it refers to a TABLED1 ID. (Integer or Real; Default is required when MID is given.)

Remarks:

1. LID must be referenced by a TLOAD1 entry.
2. TLOAD entries referencing FLOW entries must have the TID field blank or zero.

3. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
4. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
5. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
6. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1E+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the Min value.
7. Any material properties not specifically defined have the same value as the element that with the boundary conditions.
8. In the case of material flow into a multi-material Euler mesh, the material number, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions
9. The boundary condition initiates/determines a wave in compressible material like gas and water. This can be either an outgoing or an ingoing wave. For stability it is important that the waves created are compatible with the flow type near the boundary. Relevant flow types are subsonic inflow, subsonic outflow, supersonic inflow and supersonic outflow. For example for subsonic inflow prescribing both pressure and velocity would initiate outgoing waves. Outgoing waves for an inflow boundary condition is known to be instable. However, for supersonic inflow one can specify both pressure and velocities there are no outgoing waves at a supersonic inflow boundary.

FLOWUDS

User Defined Flow Boundary.

Defines a flow boundary on an Eulerian mesh specified by a user subroutine. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FLOWUDS	LID	GROUP	UNAME	BCID	MESH	DIR			
+	XMIN	XMAX	YMIN	YMAX	ZMIN	ZMAX			

Example:

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE myflow 'SCA.MDSolver.Obj.Uds.Dytran.Flow'
```

In Bulk Data:

1	2	3	4	5	6	7	8	9	10
FLOWUDS	12	MYFLOW	EXFLOW	300					

Descriptor	Meaning
LID	Unique output number. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. (Character; default=EXFLOW)
BCID	Number of a set of segments, specified by BCSEG entries, where the flow boundary is located. See Remark 6. (Integer > 0)
MESH	Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 7. (Integer > 0)
DIR	Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 8. (Character)
XMIN-ZMAX	Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 9. (Real)

Remarks:

1. Only can be used for SOL 700.
2. UNAME can be:

Subroutine Name	Function
EXFLOW	Standard user defined flow boundary on the Euler mesh

3. For multi material models EXFLOW allows for the definition of any material to flow into the Eulerian mesh. The outflow can only be of materials present in the mesh.
4. TLOAD1 entries referencing FLOWUDS entries must have the TID field blank or zero.
5. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
6. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
7. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
8. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1E+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the MIN value.

FLSYM**Axisymmetric Symmetry Control**

Defines the relationship between the axisymmetric fluid and a structural boundary having symmetric constraints. The purpose is to allow fluid boundary matrices to conform to structural symmetry definitions.

Format:

1	2	3	4	5	6	7	8	9	10
FLSYM	M	S1	S2						

Example:

FLSYM	12	S	A						
-------	----	---	---	--	--	--	--	--	--

Descriptor	Meaning
M	Number of symmetric sections of structural boundary around the circumference of the fluid being modeled by the set of structural elements. (Even Integers ≥ 2)
S1, S2	Description of boundary constraints used on the structure at the first and second planes of symmetry. (Character: "S" means symmetric, "A" means antisymmetric.)

Remarks:

1. This entry is allowed only if an AXIF entry is also present.
2. Only one FLSYM entry is allowed.
3. This entry is not required if there are no planes of symmetry.
4. First plane of symmetry is assumed to be at $\phi = 0$. Second plane of symmetry is assumed to be at $\phi = 360^\circ/M$.
5. Symmetric and antisymmetric constraints for the structure must, in addition, be provided by the user.
6. The solution is performed for those harmonic indices listed on the AXIF entry that are compatible with the symmetry conditions.
7. For example, if FLSYM is used to model a quarter section of structure at the boundary, M = 4. If the boundary constraints are "SS", the compatible cosine harmonics are 0, 2, 4, ..., etc. If "SA" is used, the compatible cosine harmonics are 1, 3, 5, ..., etc.

FLUTTER**Aerodynamic Flutter Data**

Defines data needed to perform flutter analysis.

Format:

1	2	3	4	5	6	7	8	9	10
FLUTTER	SID	METHOD	DENS	MACH	RFREQ	IMETH	NVALUE/ OMAX	EPS	

Example:

FLUTTER	19	K	119	219	319	S	5	1.-4	
---------	----	---	-----	-----	-----	---	---	------	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
METHOD	Flutter analysis method. (Character: "K" for K method, "PK" for PK method, "PKNL" for PK method with no looping, "PKS" for PK sweep method, "PKNLS" for PK sweep method with no looping, "KE" for the K method restricted for efficiency.) See Remark 9.
DENS	Identification number of an FLFACT entry specifying density ratios to be used in flutter analysis. (Integer > 0)
MACH	Identification number of an FLFACT entry specifying Mach numbers (m) to be used in flutter analysis. (Integer > 0)
RFREQ (or VEL)	Identification number of an FLFACT entry specifying reduced frequencies (k) to be used in flutter analysis; for the "PKx" methods, the velocities FLFACT entry is specified in this field. (Integer > 0)
IMETH	Choice of interpolation method for aerodynamic matrix interpolation. See Remark 6. (Character: "L" = linear, "S" = surface; "TCUB" = termwise cubic; Default = "L".)
NVALUE	Number of eigenvalues beginning with the first eigenvalue for output and plots. (Integer > 0; Default is the number of modal degrees-of-freedom (u_h)).
OMAX	For the PKS and PKNLS methods, OMAX specifies the maximum frequency, in Hz., to be used in the flutter sweep. (Real > 0.0; Default = maximum normal mode eigenfrequency)
EPS	Convergence parameter for k . Used in the PK and PKNL methods only. See Remark 4. (Real > 0.0; Default = 10^{-3} .) See Remark 9. for the meaning of EPS when the PKS or PKNLS methods are being used.

Remarks:

1. The FLUTTER entry must be selected with the Case Control command FMETHOD = SID.
2. The density is given by DENS · RHOREF, where RHOREF is the reference value specified on the AERO entry and DENS is the density ratio specified on the FLFACT entry.

3. The reduced frequency is given by $k = (REFC \cdot \omega / 2 \cdot V)$, where REFC is given on the AERO entry, ω is the circular frequency, and V is the velocity. If $k = 0.0$, as specified on the FLFACT entry, then only the K method may be specified and the Inverse Power method of eigenvalue extraction (INV on the EIGC entry) must be used. Aeroelastic divergence analysis is more appropriately performed using one of the “PKx” methods.
4. For the PK and PKNL methods, an eigenvalue is accepted when:

$$|k - k_{estimate}| < EPS \quad \text{for } k_{estimate} < 1.0$$

$$|k - k_{estimate}| < EPS \cdot k_{estimate} \quad \text{for } k_{estimate} \geq 1.0$$
5. When one of the “PKx” methods is selected, physical displacements will only be generated for the velocities on the FLFACT that are specified as negative values of the requested velocity. Also, structural damping as specified on the GE field of MATi entries is ignored.
6. If IMETH = “L”, a linear interpolation is performed on reduced frequencies at the Mach numbers specified on the FLFACT entry using the MKAEROi entry Mach number that is closest to the FLFACT entry Mach number. For IMETH = “S”, a surface interpolation is performed across Mach numbers and reduced frequencies. For IMETH = “TCUB” a termwise cubic interpolation on reduced frequency is used. IMETH = “S” is only available for the “K” and “KE” flutter methods. IMETH = “TCUB” is only available for the “PKx” methods. For the “PKx” methods, IMETH = “S” or “L” or blank provides linear interpolation while “TCUB” provides a termwise cubic interpolation.
7. For the “K”, “KE”, “PK”, and “PKS” methods, all combinations of the FLFACT entry are analyzed. For the “PKNL” and “PKNLS” methods, only ordered pairs are analyzed; i.e., $(\rho_1, M_1, V_1), (\rho_2, M_2, V_2) \dots (\rho_n, M_n, V_n)$. For the PKNL and PKNLS methods, equal number of densities, Mach numbers and velocities must be specified.
8. “K” and “KE” methods are not supported for design sensitivity and optimization.
9. The PKS and PKNLS methods determine flutter eigenvalues by performing a sweep of equally spaced reduced frequencies ranging from $k_{est} = 0.0$ through $k_{est} = \pi \cdot REFC \cdot OMAX / Velocity$. The number of intervals is calculated using $NINT = INT(1.0/EPS)$.
10.
 - The PK method uses only real matrix terms for computing the flutter solution. This means any imaginary terms in any of the matrices, M, B, or K are ignored with the PK method and the imaginary part of the aerodynamic matrix is added as a real matrix to the viscous damping matrix B.
 - For the KE method, the B matrix is ignored while complex stiffness forms of structural damping are supported. Modal viscous damping (TABDMP1 input) will be included when the KDAMP parameter is set to -1.
 - The K method supports all forms of damping.
11. METHOD=K requires a CMETHOD in Case Control pointing to an EIGC Bulk Data entry. It is recommended that the number of roots requested on the EIGC entry be twice the number of normal modes used in the modal flutter analysis.

FORCE**Static Force**

Defines a static concentrated force at a grid point by specifying a vector.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		

Example:

FORCE	2	5	6	2.9	0.0	1.0	0.0		
-------	---	---	---	-----	-----	-----	-----	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0 ; Default = 0)
F	Scale factor. (Real)
Ni	Components of a vector measured in coordinate system defined by CID. (Real; at least one Ni $\neq 0.0$. unless F is zero)

Remarks:

1. The static force applied to grid point G is given by

$$\vec{f} = F \vec{N}$$

where \vec{N} is the vector defined in fields 6, 7 and 8. The magnitude of \vec{f} is equal to F times the magnitude of \vec{N} .

2. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. A CID of zero or blank (the default) references the basic coordinate system.
4. For scalar points see SLOAD.
5. For TYPE=12 or TYPE=13 on the TLOAD1, G is the ID of a rigid body: the MID of a rigid material MATRIG or the EID of a RBE2. The MID of a rigid material and the EID of RBE2 must be different when both of a RBE2 and a rigid material are used with these TYPES. SOL 700 only.
6. For axisymmetric elements, the point loads to be entered on the card should be obtained by integrating over 1 radian of the circumference.

FORCE1**Follower Force, Alternate Form 1**

Defines a concentrated force at a grid point by specification of a magnitude and two grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				

Example:

FORCE1	6	13	-2.93	16	13				
--------	---	----	-------	----	----	--	--	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
F	Magnitude of the force. (Real)
G1, G2	Grid point identification numbers. (Integer > 0; G1 and G2 may not be coincident.)

Remarks:

1. The force applied to grid point G is given by

$$\vec{f} = F \vec{n}$$

where \vec{n} is a unit vector parallel to a vector from G1 to G2.

2. In the static solution sequences, SID must be selected by the LOAD Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, 400, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
4. For axisymmetric elements, the point loads to be entered on the card should be obtained by integrating over 1 radian of the circumference.

FORCE2

Follower Force, Alternate Form 2

Defines a concentrated force at a grid point by specification of a magnitude and four grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE2	SID	G	F	G1	G2	G3	G4		

Example:

FORCE2	6	13	-2.93	16	13	17	13		
--------	---	----	-------	----	----	----	----	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
F	Magnitude of the force. (Real)
Gi	Grid point identification numbers. (Integer > 0; G1 and G2 may not be coincident; G3 and G4 cannot be coincident.)

Remarks:

1. The direction of the force is parallel to the cross product of vectors from G1 to G2 and G3 to G4.
2. In the static solution sequences, SID must be selected by the LOAD Case Control command.
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, and 400 if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
4. For axisymmetric elements, the point loads to be entered on the card should be obtained by integrating over 1 radian of the circumference.

FORCEAX**Conical Shell Static Force**

Defines a static concentrated force on a conical shell ring.

Format:

1	2	3	4	5	6	7	8	9	10
FORCEAX	SID	RID	HID	S	FR	FP	FZ		

Example:

FORCEAX	1	2	3	2.0	0.1	0.2	0.3		
---------	---	---	---	-----	-----	-----	-----	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
RID	RINGAX entry identification number. (Integer > 0)
HID	Harmonic identification number or a sequence of harmonics. See Remark 5. (Integer ≥ 0)
S	Scale factor for the force. (Real)
FR, FP, FZ	Force components in r, ϕ, z directions. (Real)

Remarks:

1. FORCEAX is allowed only if an AXIC entry is also present.
2. Axisymmetric shell loads must be selected with the Case Control command LOAD = SID.
3. A separate entry is needed for the definition of the force associated with each harmonic.
4. See [Conical Shell Element \(RINGAX\)](#) (Ch. 3) in the *MSC Nastran Reference Guide* for further discussion.
5. If a sequence of harmonics is to be placed in HID, the form is as follows: "Sn1Tn2" where n1 is the start of the sequence and n2 is the end of the sequence (e.g., for harmonics 0 through 10, the field would contain "S0T10").

FORCUDS**User-defined Enforced Motion at Grid Points**

Defines enforced motion at grid points specified by a user subroutine. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FORCUDS	LID	GROUP	UNAME						
+	G1	G2	THRU	G3					

Example:

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE extvel 'SCA.MDSolver.Obj.Uds.Dytran.Loads'
```

In Bulk Data:

FORCUD	1	EXTVEL							
+	1	2	3	5					

Descriptor	Meaning
LID	Number of a set of loads.(Integer>0;required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; required)
UNAME	User subroutine name associated with the entry. (Character; default=EXTVEL)
Gi	Numbers of the grid points that are constrained. If the word THRU appears between two numbers, all the numbers in the range are included in the list. BY indicates the increment to be used within this range. (Integer or character; required.)

Remarks:

1. LID must be referenced by a TLOAD1 entry.
2. FORCUDS can only be used to specify enforced velocities for grid points. The TYPE field on the TLOAD1 entry must be set to two. The TID on the TLOAD1 entry must be set to zero or blank (no time variation).
3. The GROUP name must be unique from all other GROUP names if more than one FORCUDS is used. Therefore each FORCUDS must have its own group name in a FMS section CONNECT SERVICE statement.
4. The constraint name is passed to the subroutine and can be used to identify the constraint.
5. A THRU specification, including the start and finish points in the range, must be on one line.
6. If the THRU specification is used, all the points in the sequence do not have to exist. Those that do not exist are ignored. The first point in the THRU specification must be a valid grid point. BY can be used to exclude grid points.

7. None of the fields in the list of grid points can be blank or zero, since this designation marks the end of the list.
8. Any number of continuation lines can be used to define the list of grid points.
9. UNAME can be:

Subroutine Name	Function
EXTVEL	user defined velocities on grid points

FREEPT

Fluid Free Surface Point

Defines the location of points on the surface of a fluid for recovery of surface displacements in a gravity field.

Format:

1	2	3	4	5	6	7	8	9	10
FREEPT	IDF	IDP1	PHI1	IDP2	PHI2	IDP3	PHI3		

Example:

FREEPT	3	301	22.5	302	90.0	303	370.0		
--------	---	-----	------	-----	------	-----	-------	--	--

Descriptor	Meaning
IDF	RINGFL entry identification number. (Integer > 0)
IDPi	Free surface point identification number. (Integer > 0)
PHIi	Azimuthal position on fluid point (RINGFL entry) in the fluid coordinate system. (Real)

Remarks:

1. FREEPT is allowed only if an AXIF entry is also present.
2. All free surface point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. The free surface points are used for the identification of output data only.
4. Three points may be defined on a single entry.
5. The referenced fluid point (IDF) must be included in a free surface list (FSLIST entry).
6. Output requests for velocity and acceleration can be made at these points.

FREQ**Frequency List**

Defines a set of frequencies to be used in the solution of frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	F10	-etc.-					

Example:

FREQ	3	2.98	3.05	17.9	21.3	25.6	28.8	31.2	
	29.2	22.4	19.3						

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
Fi	Frequency value in units of cycles per unit time. (Real ≥ 0.0)

Remarks:

1. Frequency sets must be selected with the Case Control command FREQUENCY = SID.
2. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|,$$

where DFREQ is a user parameter, with a default of 10^{-5} . f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

3. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.

FREQ1

Frequency List, Alternate Form 1

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment, and the number of increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ1	SID	F1	DF	NDF					

Example:

FREQ1	6	2.9	0.5	13					
-------	---	-----	-----	----	--	--	--	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
F1	First frequency in set. (Real ≥ 0.0)
DF	Frequency increment. (Real > 0.0)
NDF	Number of frequency increments. (Integer > 0 ; Default = 1)

Remarks:

1. FREQ1 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and DF are cycles per unit time.
3. The frequencies defined by this entry are given by

$$f_i = F1 + DF \cdot (i-1)$$

where i = 1 to (NDF + 1).

4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$|f_N - f_{N-1}| < DFREQ \cdot |f_{MAX} - f_{MIN}|,$$

where DFREQ is a user parameter, with a default of 10^{-5} . f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

FREQ2**Frequency List, Alternate Form 2**

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, final frequency, and the number of logarithmic increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ2	SID	F1	F2	NF					

Example:

FREQ2	6	1.0	8.0	6					
-------	---	-----	-----	---	--	--	--	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
F1	First frequency. (Real > 0.0)
F2	Last frequency. (Real > 0.0, F2 > F1)
NF	Number of logarithmic intervals. (Integer > 0; Default = 1)

Remarks:

1. FREQ2 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and F2 are cycles per unit time.
3. The frequencies defined by this entry are given by

$$f_i = F1 \cdot e^{(i-1)d}$$

where $d = \frac{1}{NF} \ln \frac{F2}{F1}$ and $i = 1, 2, \dots, (NF + 1)$

In the example above, the list of frequencies will be 1.0, 1.4142, 2.0, 2.8284, 4.0, 5.6569 and 8.0 cycles per unit time.

4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$|f_N - f_{N-1}| < DFREQ \cdot |f_{MAX} - f_{MIN}| ,$$

where DFREQ is a user parameter, with a default of 10^{-5} . f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

6. If Modules are present then this entry may only be specified in the main Bulk Data section.

FREQ3**Frequency List, Alternate 3**

Defines a set of excitation frequencies for modal frequency-response solutions by specifying number of excitation frequencies between two modal frequencies.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ3	SID	F1	F2	TYPE	NEF	CLUSTER			

Example:

FREQ3	6	20.0	2000.0	LINEAR	10	2.0			
-------	---	------	--------	--------	----	-----	--	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
F1	Lower bound of modal frequency range in cycles per unit time. (Real ≥ 0.0 for TYPE = LINEAR and Real = 0.0 for TYPE = LOG)
F2	Upper bound of modal frequency range in cycles per unit time. (Real > 0.0 ; F2 \geq F1; Default = F1)
TYPE	LINEAR or LOG. Specifies linear or logarithmic interpolation between frequencies. (Character; Default = "LINEAR")
NEF	Number of excitation frequencies within each subrange including the end points. The first subrange is between F1 and the first modal frequency within the bounds. The second subrange is between first and second modal frequencies between the bounds. The last subrange is between the last modal frequency within the bounds and F2. (Integer > 1; Default = 10)
CLUSTER	Specifies clustering of the excitation frequency near the end points of the range. See Remark 6. (Real > 0.0; Default = 1.0)

Remarks:

1. FREQ3 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.
2. FREQ3 entries must be selected with the Case Control command FREQUENCY = SID.
3. In the example above, there will be 10 frequencies in the interval between each set of modes within the bounds 20 and 2000, plus 10 frequencies between 20 and the lowest mode in the range, plus 10 frequencies between the highest mode in the range and 2000.
4. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

5. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter, with a default of 10^{-5} . f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

6. CLUSTER is used to obtain better resolution near the modal frequencies where the response varies the most. CLUSTER > 1.0 provides closer spacing of excitation frequency towards the ends of the frequency range, while values of less than 1.0 provide closer spacing towards the center of the frequency range. For example, if the frequency range is between 10 and 20, NEF = 11, TYPE = "LINEAR"; then, the excitation frequencies for various values of CLUSTER would be as shown in [Table 17](#).

$$\hat{f}_k = \frac{1}{2}(\hat{f}_1 + \hat{f}_2) + \frac{1}{2}(\hat{f}_2 - \hat{f}_1)|\xi|^{1/\text{CLUSTER}} \cdot \text{SIGN}(\xi)$$

where

$\xi = -1 + 2(k - 1)/(NEF - 1)$ is a parametric coordinate between -1 and 1

k = varies from 1 to NEF ($k = 1, 2, \dots, NEF$)

\hat{f}_1 = is the lower limit of the frequency subrange

\hat{f}_2 = is the upper limit of the subrange

\hat{f}_k = is the k-th excitation frequency

\hat{f} = is the frequency, or the logarithm of the frequency, depending on the value specified for TYPE

Table 17 CLUSTER Usage Example

Excitation Frequency Number	ξ	CLUSTER				
		c=0.25	c-0.50	c-1.0	c-2.0	c-4.0
		Excitation Frequencies in Hertz				
1	-1.0	10.00	10.0	10.0	10.00	10.00
2	-0.8	12.95	11.8	11.0	10.53	10.27
3	-0.6	14.35	13.2	12.0	11.13	10.60
4	-0.4	14.87	14.2	13.0	11.84	11.02
5	-0.2	14.99	14.8	14.0	12.76	11.66
6	0.0	15.00	15.0	15.0	15.00	15.00
7	0.2	15.01	15.2	16.0	17.24	18.34

Table 17 CLUSTER Usage Example

Excitation Frequency Number	ξ	CLUSTER				
		c=0.25	c=0.50	c=1.0	c=2.0	c=4.0
		Excitation Frequencies in Hertz				
8	0.4	15.13	15.8	17.0	18.16	18.98
9	0.6	15.65	16.8	18.0	18.87	19.40
10	0.8	17.05	18.2	19.0	19.47	19.73
11	1.0	20.00	20.0	20.0	20.00	20.00

7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
9. If Modules are present then this entry may only be specified in the main Bulk Data section.

FREQ4

Frequency List, Alternate Form 4

Defines a set of frequencies used in the solution of modal frequency-response problems by specifying the amount of “spread” around each natural frequency and the number of evenly spaced excitation frequencies within the spread.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ4	SID	F1	F2	FSPD	NFM				

Example:

FREQ4	6	20.0	2000.0	0.30	21				
-------	---	------	--------	------	----	--	--	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
F1	Lower bound of frequency range in cycles per unit time. (Real ≥ 0.0 ; Default = 0.0)
F2	Upper bound of frequency range in cycles per unit time. (Real > 0.0 ; F2 > F1; Default = 1.0E20)
FSPD	Frequency spread, +/- the fractional amount specified for each mode which occurs in the frequency range F1 to F2. (1.0 > Real > 0.0; Default = 0.10)
NFM	Number of evenly spaced frequencies per “spread” mode. (Integer > 0; Default = 3; If NFM is even, NFM + 1 will be used.)

Remarks:

1. FREQ4 applies only to modal frequency-response solutions (SOLs 111, 146, and 200 and is ignored in direct frequency-response solutions.
2. FREQ4 entries must be selected with the Case Control command FREQUENCY = SID.
3. There will be NFM excitation frequencies between $(1 - FSPD) \cdot f_N$ and $(1 + FSPD) \cdot f_N$, for each natural frequency in the range F1 to F2.
4. In the example above there will be 21 equally spaced frequencies across a frequency band of $0.7 \cdot f_N$ to $1.3 \cdot f_N$ for each natural frequency that occurs between 20 and 2000. See [Figure 9-94](#) for the definition of frequency spread.

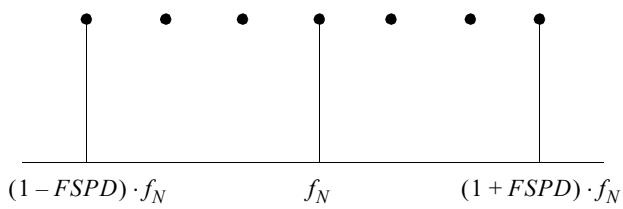
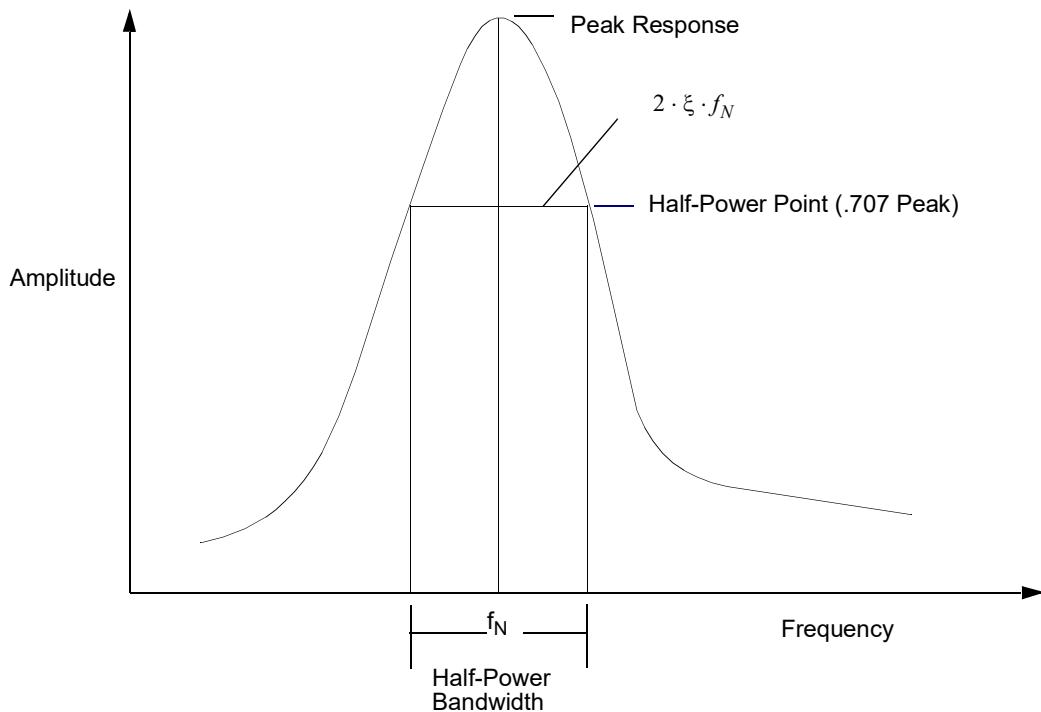


Figure 9-94 Frequency Spread Definition

Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.

- The frequency spread can be used also to define the half-power bandwidth. The half-power bandwidth is given by $2 \cdot \xi \cdot f_N$, where ξ is the damping ratio. Therefore, if FSPD is specified equal to the damping ratio for the mode, NFM specifies the number of excitation frequency within the half-power bandwidth. See [Figure 9-95](#) for the definition of half-power bandwidth.

**Figure 9-95 Half-Power Bandwidth Definition**

- Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.
- All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$|f_N - f_{N-1}| < DFREQ \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter with a default of 10^{-5} . The values f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

8. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
9. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
10. If Modules are present then this entry may only be specified in the main Bulk Data section.

FREQ5**Frequency List, Alternate Form 5**

Defines a set of frequencies used in the solution of modal frequency-response problems by specification of a frequency range and fractions of the natural frequencies within that range.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ5	SID	F1	F2	FR1	FR2	FR3	FR4	FR5	
	FR6	FR7	-etc.-						

Example:

FREQ5	6	20.0	2000.0	1.0	0.6	0.8	0.9	0.95	
	1.05	1.1	1.2						

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
F1	Lower bound of frequency range in cycles per unit time. (Real ≥ 0.0 ; Default = 0.0)
F2	Upper bound of frequency range in cycles per unit time. (Real > 0.0 ; F2 > F1; Default = 1.0E20)
FRi	Fractions of the natural frequencies in the range F1 to F2. (Real > 0.0)

Remarks:

1. FREQ5 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.
2. FREQ5 entries must be selected with the Case Control command FREQUENCY = SID.
3. The frequencies defined by this entry are given by

$$f_i = FRi \cdot f_{N_i}$$

where f_{N_i} are the natural frequencies in the range F1 through F2.

4. In the example above, the list of frequencies will be 0.6, 0.8, 0.9, 0.95, 1.0, 1.05, 1.1, and 1.2 times each natural frequency between 20 and 2000. If this computation results in excitation frequencies less than F1 and greater than F2, those computed excitation frequencies are ignored.
Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.
5. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

6. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter with a default of 10^{-5} . The values f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
9. If Modules are present then this entry may only be specified in the main Bulk Data section.

FRFCOMP**Frequency Response Function (FRF) Component Specification for FRF Based Assembly (FBA)**

Specifies the FRF components that are to be assembled as part of an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FRFCOMP	COMPID	COMPNAME	MEDIUM	UNITNO	LSCALFAC	FSCALFAC			

Examples:

FRFCOMP	10	LEFTWING	OP2	25					
FRFCOMP	20	FRAME	UF	30	1000.0				

Descriptor	Meaning
COMPID	ID of the component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Name of the COMPID FRF component. See Remark 1. (Up to 8 characters).
MEDIUM	Medium on which the FRF matrices and other related data are stored. Acceptable character values are DB (for database), OP2 (for OUTPUT2 file) or UF (for Universal file). See Remarks 2., 3. and 4.
UNITNO	Fortran unit number for the OP2 and UF options. (Integer > 0). See Remarks 3. and 4.
LSCALFAC	Length scale factor for the UF option. (Real > 0.0; Default = 1.0). See Remarks 7., 8., 9., 10. and 11.
FSCALFAC	Force scale factor for the UF option. (Real > 0.0; Default = 1.0). See Remarks 7., 8., 9., 10., and 11.

Remarks:

1. COMPNAME may be referenced by the ASMOUT keyword in the FRF Case Control command in order to request output for a specific FRF component in the FBA process. Accordingly, COMPNAME may not have the values of CONNINFO, ALL, COMP or ASSEMBLY as these are all reserved words for use with the ASMOUT keyword in the FRF Case Control command.
2. If the DB option is specified, then the following type of ASSIGN should be specified in the FMS section of the FBA job to access information on the database for the specified FRF component:

```
ASSIGN dbname = 'frfgen_job.MASTER'
DBLOCATE DATABLK = (FRFDB) LOGICAL = dbname
```

3. If the OP2 option is specified, then the following type of ASSIGN should be specified in the FMS section of the FBA job to access information on the OUTPUT2 file for the specified FRF component:
ASSIGN INPUTT2 = 'frfgen_job_op2' UNIT = 25
4. If the UF option is specified, then the following type of ASSIGN should be specified in the FMS section of the FBA job to access information on the Universal file for the specified FRF component:
ASSIGN UNVFILE = 'frfgen_job_unv' UNIT = 26
5. Component ID of 0 is assigned to the assembled FRF configuration resulting from the FBA process.
6. An FRF generation job using an FRF Case Control command with GEN/GENASM and COMPID and COMPNAME keywords specified in it will automatically generate and save an FRFCOMP Bulk Data entry on the assembly punch (.asm) file for that FRF component for subsequent use in an FBA process.
7. LSCALFAC and FSCALFAC are meaningful only for the UF option. They are ignored for the DB and OP2 options.
8. The FRF and other information on the Universal File (UF) is grouped by so-called Universal Dataset Numbers (UDNs). The heart of the information on the UF for a test FRF component is in UDN 58 which contains FRF data for that component. Also of interest, if present, is UDN 15 which contains grid point coordinate data in single precision or UDN 2411 which contains grid point coordinate data in double precision. Further, if UDN 15 or UDN 2411 is present, the data therein may reference definition or displacement coordinate systems. Such coordinate system data is resident in UDN 18.

Details of the various UDNs and their formats can be obtained from the following websites:
<http://www.sdrl.uc.edu/universal-file-formats-for-modal-analysis-testing-1>
<http://www.sdrl.uc.edu/universal-file-formats-for-modal-analysis-testing-1/file-format-storehouse/file-formats>
9. The points of a test FRF component that are considered by the FBA process are those that are defined in UDN 58 either as response points or as excitation points. Any such point that appears both as a response point and as an excitation point is regarded as a potential connection point of the test FRF component.
10. Connections between test FRF components and other FRF components are determined by what is supplied on the UF as indicated below.
 - a. If UDN 15 or UDN 2411 is supplied on the UF, then connections between the test FRF component grid points and those of other FRF components will be based on *matching coordinates*.
 - b. If neither UDN 15 nor UDN 2411 is supplied, then normally, connections between the test FRF component grid points and those of other FRF components will be based on *matching grid point IDs*. However, the user can override this by specifying FRFCCONN entries, thereby causing connections between *user specified grid points*. In any case, when the connections are based on matching grid point IDs or based on FRFCCONN entries, it is the user's responsibility to ensure that the connection grid points have the same physical locations in space so that correct results are obtained from the FBA process. The program clearly cannot check for this condition since the coordinates of one or more of the points are not available.
11. For the UF option, LSCALFAC and FSCALFAC are used as follows:

- For Universal Dataset No. (UDN) 15 or 2411 (if present) on the Universal File:
The X, Y, and Z coordinate values in the UDN are multiplied by LSCALFAC before they are used in the FBA process.
- For Universal Dataset No. (UDN) 58 on the Universal File:
The FRF response quantities in the UDN are multiplied by appropriate factors as shown in the following table before they are used in the FBA process.

Table 9-16 Multiplication Factors for FRF Response Quantities in UDN 58 of the Universal File

FRF Response Quantity	Type of Excitation	Factor by Which FRF Response Quantity is Multiplied Before It is Used in the FBA Process
Translational displacement, velocity or acceleration at a structural point	Unit force at an excitation point	LSCALFAC/FSCALFAC
	Unit moment at an excitation point	1.0/FSCALFAC
Rotational displacement, velocity or acceleration at a structural point	Unit force at an excitation point	1.0/FSCALFAC
	Unit moment at an excitation point	1.0/(LSCALFAC*FSCALFAC)
Pressure at an acoustic point	Unit force at an excitation point	1.0/(LSCALFAC**2)
	Unit moment at an excitation point	1.0/(LSCALFAC**3)

FRFCONN

FRF Component Explicit Connection for FRF Based Assembly (FBA)

Defines explicit connection data for FRF components in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FRFCONN	CONNID	COMPID1/ COMPNAME1	POINT1	COMPID2/ COMPNAME2	POINT2				

Examples:

FRFCONN	10	5	100	15	200				
FRFCONN	20	STRUT	25	WING	35				

Descriptor	Meaning
CONNID	Unique identification number of the FRFCONN entry. (Integer > 0)
COMPID _i	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Integer > 0)
COMPNAME _i	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Up to 8 characters; no blank allowed)
POINT _i	Grid or scalar point identification number. See Remarks 2. through 6. (Integer > 0)

Remarks:

1. If neither of the FRF components COMPID1/COMPNAME1 and COMPID2/COMPNAME2 is part of the FBA process, then this entry is ignored. However, if one of them is part of the FBA process but not the other, the program terminates the job with a user fatal message.
2. POINT_i must be among the connection points of the corresponding FRF component COMPID_i/COMPNAME_i. If not, the program terminates the job with a user fatal message.
3. It is, in general, not necessary to have FRFCONN entries when the connections in the FBA process involve only grid points. In the FBA process, all connection grid points of FRF components that have the same basic coordinates are normally connected automatically without any user intervention or specification.
4. If the connection points of an FRF component in the FBA process consist of coincident grid points, the program identifies such points via a user information message. All such coincident grid points must be referenced on FRFCONN or FRFRELS entries in order to ensure proper connections in the FBA process. In the absence of such specifications for coincident connection grid points, the program terminates the job with a user fatal error.
5. FRFCONN entries are required if the user wants to specify explicit connections between test FRF component grid points whose coordinates are not available and those of other FRF components. See Remark 10. under the description of the FRFCOMP entry for further details.

6. FRFCONN entries are also required if the user wants to combine scalar points of FRF components in the FBA process.

FRFFLEX**FRF Component Flexible Connection Specification for FRF Based Assembly (FBA)**

Defines properties for flexible connections between FRF components in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FRFFLEX	FLEXID	C	COMPID1/ COMPNAME1	POINT1	COMPID2/ COMPNAME2	POINT2	KVALUE/ KTABID	BVALUE/ BTABID	
	GEVALUE/ GETABID								

Examples:

FRFFLEX	10	1	100	15	200	25	100.0	150	
	0.02								

FRFFLEX	20	4	STRUT	120	WING	260	10	0.25	
---------	----	---	-------	-----	------	-----	----	------	--

Descriptor	Meaning
FLEXID	Unique identification number of the FRFFLEX entry. (Integer > 0)
C	A single component number. (Any integer between 1 and 6 for grid points; integer 0 or blank for scalar points.)
COMPIDI	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Integer > 0)
COMPNAMEi	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Up to 8 characters; no blank allowed)
POINTi	Grid or scalar point identification number. See Remark 2. (Integer > 0)
KVALUE	Elastic property value (force per unit displacement). See Remark 3. (Real > 0.0 or blank)
KTABID	Identification number of a TABLEDi entry that defines the elastic property value (force per unit displacement) as a function of frequency. See Remark 3. (Integer > 0 or blank)
BVALUE	Damping property value (force per unit velocity). See Remark 3. (Real > 0.0 or blank)
BTABID	Identification number of a TABLEDi entry that defines the damping property value (force per unit velocity) as a function of frequency. See Remark 3. (Integer > 0 or blank)
GEVALUE	Damping coefficient value. See Remarks 4., 5. and 6. (Real > 0.0 or blank)
GETABID	Identification number of a TABLEDi entry that defines the damping coefficient value as a function of frequency. See Remarks 4., 5. and 6. (Integer > 0 or blank)

Remarks:

1. If neither of the FRF components COMPID1/COMPNAME1 and COMPID2/COMPNAME2 is part of the FBA process, then this entry is ignored. However, if one of them is part of the FBA process but not the other, the program terminates the job with a user fatal message.
2. POINTi must be among the connection points of the corresponding FRF component COMPIDi/COMPNAMEi. If not, the program terminates the job with a user fatal message.
3. The KVALUE/KTABID and BVALUE/BTABID fields may not both be blank.
4. The continuation entry is not needed if GEVALUE/GETABID is not to be defined.
5. GEVALUE/GETABID may not be specified unless KVALUE/KTABID is specified.
6. To obtain the damping coefficient, multiply the critical damping ratio C/C_0 by 2.0.
7. It is important to note that this entry by itself does *not* define a connection between the specified points. It merely defines properties for flexible connections between two points whose connection is established either explicitly via an FRFCOMP entry or is implied by automatic connection.
8. The flexible connection properties for component C of connection points POINT1 and POINT2 may be defined on more than one FRFFLEX entry.
9. In the absence of FRFFLEX data, the program assumes rigid connections between the corresponding components.
10. A grid point component may not appear on both an FRFFLEX entry and an FRFRELS entry. If it does, the program identifies such usage and terminates the job with a user fatal message.

FRFRELS

FRF Component Grid Point Release for FRF Based Assembly (FBA)

Defines the degrees-of-freedom of FRF component connection grid points that are not to be connected in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FRFRELS	SID	C	COMPID1/ COMPNAME1	GRIDPNT1	COMPID2/ COMPNAME2	GRIDPNT2	COMPID3/ COMPNAME3	GRIDPNT3	

Examples:

FRFRELS	100	45	10	15	BODY	20	FRAME	30	
---------	-----	----	----	----	------	----	-------	----	--

FRFRELS	20	1	WING	25	NACELLE	35			
---------	----	---	------	----	---------	----	--	--	--

Descriptor	Meaning
SID	Identification number of the FRFRELS entry. (Integer > 0)
C	Component number(s). See Remark 1. (Any unique combination of the integers 1 through 6 with no embedded blanks.)
COMPID _i	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Integer > 0)
COMPNAME _i	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Up to 8 characters; no blank allowed)
GRIDPNT _i	Grid point identification number. See Remarks 3. and 4. (Integer > 0)

Remarks:

1. The grid point component(s) specified by C will not be connected in the FBA process.
2. If FRF component COMPID_i/COMPNAME_i is not part of the FBA process, then the release data for that FRF component is ignored.
3. GRIDPNT_i must be among the connection points of the corresponding FRF component COMPID_i/COMPNAME_i. If not, the program terminates the job with a user fatal message.
4. If GRIDPNT_i is not connected to any other grid point in the FBA process, the program ignores the FRFRELS data for this point and issues a warning message indicating this to the user.
5. If the connection points of an FRF component in the FBA process consist of coincident grid points, the program identifies such points via a user information message. All such coincident grid points must be referenced on FRFCCONN or FRFRELS entries in order to ensure proper connections in the FBA process. In the absence of such specifications for coincident connection grid points, the program terminates the job with a user fatal error.

FRFSPC1**FRF Component Single-Point Constraint for FRF Based Assembly (FBA)**

Defines single-point constraints for FRF component connection points in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FRFSPC1	SID	C	COMPID1/ COMPNAME1	POINT1	COMPID2/ COMPNAME2	POINT2	COMPID3/ COMPNAME3	POINT3	

Examples:

FRFSPC1	100	4	10	12	ENGINE	23	FRAME	31	
FRFSPC1	20	1	STRUT	25	NACELLE	35			

Descriptor	Meaning
SID	Identification number of the single-point constraint set. See Remark 1. (Integer > 0)
C	Component number(s). (Any unique combination of the integers 1 through 6 with no embedded blanks for grid points; integer 0 or blank for scalar points.)
COMPID _i	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Integer > 0)
COMPNAME _i	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Up to 8 characters; no blank allowed)
POINT _i	Grid or scalar point identification number. See Remarks 3. and 4. (Integer > 0)

Remarks:

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. If FRF component COMPID_i/COMPNAME_i is not part of the FBA process, then the single-point constraint data for that FRF component is ignored.
3. POINT_i must be among the connection points of the corresponding FRF component COMPID_i/COMPNAME_i. If not, the program terminates the job with a user fatal message.
4. If POINT_i is not connected to any other point in the FBA process, the program ignores the FRFSPC1 data for this point and issues a warning message indicating this to the user.
5. Unlike in non-FBA jobs, wherein a degree-of-freedom that has SPCs specified for it yields exact zero results, a degree-of-freedom that has an FRFSPC1 specified for it in an FBA job will, in general, not give exact zero results, but will give results that are nearly zero or very close to zero, usually around 1.0E-12 or so.

FRFXT

Unit Load Degree-of-Freedom Specification for Frequency Response Function (FRF) Computations

Specifies a single degree-of-freedom where unit loads are to be applied for Frequency Response Function (FRF) generation.

Format:

1	2	3	4	5	6	7	8	9	10
FRFXT	PNTID	C	LABEL						

Example:

FRFXT	10	3	UNIT LOAD AT LEFT CORNER						
-------	----	---	--------------------------	--	--	--	--	--	--

Descriptor	Meaning
PNTID	Grid or scalar point identification number. (Integer > 0)
C	A single component number. (Integer 0 or blank for a scalar point; any integer between 1 and 6 for a grid point.)
LABEL	A string comprising no more than 48 characters (fields 4 through 9) that will be used in the label portion of the FRF output to identify this unit load specification. See Remarks 1. and 2.

Remarks:

1. The small field format must be employed for this entry. If the free field or large field format is employed, the results are unpredictable and in many cases may lead to fatal errors and subsequent termination of the job.
2. The LABEL data must have a non-blank entry in field 4.
3. The FRFXT1 Bulk Data entry and the DLOAD Case Control request provide alternate means of unit load specification for FRF generation.
4. Redundant unit load specifications are ignored.
5. If Modules are present then this entry may only be specified in the main Bulk Data section.

FRFxit1**Unit Load Degrees-of-Freedom Specification for Frequency Response Function (FRF) Computations**

Specifies degrees-of-freedom where unit loads are to be applied for Frequency Response Function (FRF) generation.

Format:

1	2	3	4	5	6	7	8	9	10
FRFxit1	C	PNTID1	PNTID2	PNTID3	PNTID4	PNTID5	PNTID6	PNTID7	

Example:

FRFxit1	123	10	20	30	40				
---------	-----	----	----	----	----	--	--	--	--

Alternate Format and Example:

1	2	3	4	5	6	7	8	9	10
FRFxit1	C	PNTID1	THRU	PNTID2					
FRFxit1	123	5	THRU	15					

Descriptor	Meaning
C	Component numbers. (Any unique combination of integers 1 through 6 with no embedded blanks for grid points; integer 0 or blank for scalar points.)
PNTIDi	Grid or scalar point identification numbers. See Remark 1. (Integer > 0)

Remarks:

1. Points in the THRU range need not all exist.
2. The FRFxit Bulk Data entry and the DLOAD Case Control request provide alternate means of unit load specification for FRF generation.
3. Redundant unit load specifications are ignored.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.

FSICTRL

Defines the analysis type for OpenFSI fluid structure analysis simulations in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
FSICTRL	SERV_ID	TYPE	FREQ	ANALYSIS					

Examples:

FSICTRL	scafsi	EXPLICIT	1						
---------	--------	----------	---	--	--	--	--	--	--

Descriptor	Meaning	Character	None
SERV_ID	OpenFSI SCA service name associated with the wetted surface loads (see WETLOAD, 3327). The OpenFSI SCA service is defined using the CONNECT SERVICE file management section statement.	Character	None
TYPE	Type of solution strategy coupling between the external code and SOL 700. TYPE can be either EXPLICIT or IMPLICIT.	Character	EXPLICIT
FREQ	External force and displacement update frequency per time step, for the exchange with the external code using the IMPLICIT solution strategy TYPE.	Integer > 0	1
ANALYSIS	OpenFSI service type, it is either DV or U. The DV indicates the service implements OpenFSI interface, the U means the service implements OpenFSI_Ex interface.	Character	DV

Remarks:

1. This entry is used for nonlinear SOL 400 analysis.
2. In the FMS Section:
CONNECT SERVICE SERV_ID 'ExternalCodeVendor.OpenFSI '
3. The SERV_ID string must be 8 characters or less.

FSLIST**Free Surface List**

Defines the fluid points (RINGFL entry) that lie on a free surface boundary.

Format:

1	2	3	4	5	6	7	8	9	10
FSLIST	RHO	IDF1	IDF2	IDF3	IDF4	IDF5	IDF6	IDF7	
	IDF8	-etc.-							

Examples:

FSLIST	1.0-4	AXIS	432	325	416	203	256	175	
--------	-------	------	-----	-----	-----	-----	-----	-----	--

Descriptor	Meaning
RHO	Mass density at the surface. (Real > 0.0 or blank; the default is taken from DRHO on the AXIF entry.)
IDF1	Identification number of RINGFL entry. (Integer > 0 or Character = “AXIS” or “LAXIS”) See Remark 5.
IDF2-IDFi	Identification number of additional RINGFL entries. (Unique Integers > 0)

Remarks:

1. This entry is allowed only if an AXIF entry is also present.
2. The order of the points must be sequential with the fluid on the right with respect to the direction of travel.
3. The word “AXIS” defines an intersection with the polar axis of the fluid coordinate system.
4. If the fluid density varies along the boundary, there must be one FSLIST entry for each interval between fluid points.
5. If the polar axis of the fluid coordinate system is to occur at the first point use AXIS. If the polar axis of the fluid coordinate system is to occur at the last point use LAXIS. See Remark 2.

FTGDEF**Fatigue Element Definitions**

Defines elements and their associated fatigue properties to be considered for fatigue analysis for time domain SOLutions 101, 103, 112 and frequency domain SOLutions 108 and 111.

Format (SOL 101/103/112):

1	2	3	4	5	6	7	8	9	10
FTGDEF	ID	TOPSTR	PFTGID	TOPDMG	NENTS	maxENTS			
	"ELSET"	ELSID1	PFTGID1	ELSID2	PFTGID2	ELSID3	PFTGID3		
		ELSID4	PFTGID4		
		-etc.-							
	"SPOTW"	ELSID1	PFTGID1	ELSID2	PFTGID2	ELSID3	PFTGID3		
		ELSID4	PFTGID4		
		-etc.-							
	"SEAMW"	ELSID1	PFTGID1	NDSID1	WELD1	TYPE1			
		ELSID2	PFTGID2	NDSID2	WELD2	TYPE2			
		-etc.-							
	"XELSET"	XELSID1	XELSID2	XELSID3	XELSID4	XELSID5	XELSID6	XELSID7	
		XELSID8	
		-etc.-							

Format (SOL 108/112):

1	2	3	4	5	6	7	8	9	10
FTGDEF	ID	TOPRMS							
	"ELSET"	ELSID1		ELSID2		ELSID3			
		ELSID4				
		-etc.-							
	"XELSET"	XELSID1	XELSID2	XELSID3	XELSID4	XELSID5	XELSID6	XELSID7	
		XELSID8	
		-etc.-							

Examples:

FTGDEF	22	100.0	3						
FTGDEF	22								
	ELSET	14	3	15	4				

FTGDEF	22								
	SPOTW	44	1	45	2	46			

FTGDEF	22								
	SEAMW	44	99	31	FILLET	TOE			
		45	99		LASER	ROOT			

Descriptor	Meaning
ID	Unique identification number called out by the FATIGUE case control. (Integer > 0).
TOPSTR	SOL 101, 103, 112: Top stress percentage. Only entities with combined stress in this top percentage are retained and report results. Should not be used with SOL 200 or for fatigue analysis of spot and seam welds; leave blank.
TOPRMS	SOL 108, 111: Top RMS (root mean square) stress percentage. Only entities with RMS stress levels in this top percentage are retained and report results. (0.0 < Real ≤ 100.0; Default = blank - 100% will be used) See Remark 8.
PFTGID	SOL 101, 103, 112 only. ID of a PFTG entry for associating fatigue properties to all elements of the model. Ignored if ELSET flag is present and should be left blank in this case. (Optional, Integer > 0). See Remark 2.
TOPDMG	SOL 101, 103, 112 only. Top damage percentage. Only elements with damage in this top percentage will be retained and report results. (0.0 < Real ≤ 100.0; Default = blank). Should not be used with SOL 200 or for fatigue analysis of spot and seam welds; leave blank. See Remark 8.
NENTS	SOL 101, 103, 112 only. Specifies the number of actual entities to output in order to limit or further reduce output file sizes based on most damage or maximum stress/strain range. A positive value indicates the criterion for entity filtering is based on the most damaged entities. A negative number indicates the filtering criteria is based on maximum stress/strain range. (Integer != 0; Default = blank). Not valid for SOL 200; leave blank. See Remark 10.
maxENTS	SOL 101, 103, 112 only. This is used only if STROUT=4 is specified in the FATIGUE case control. If the number of entities requested is greater than maxENTS, the job stops to avoid performance issues. Setting this to a large number can severely affect performance. When using STROUT=4, the number of entities should be limited. (Integer > 0; Default = 100).
ELSET	Flag indicating that a list of element set and property pairs will follow, defining the elements and their associated properties for consideration in the fatigue analysis. (Optional, Character = ELSET) See Remark 9.
ELSIDi	ID of a SET1, SET3, or SET4 entry listing entities of the model (elements) to be included in the fatigue analysis. (Integer > 0). See Remark 3.
PFTGIDI	ID of a PFTG entry, which indicates the fatigue property associated to the preceding entities defined by ELSIDi. (Optional, Integer > 0). See Remark 2.

Descriptor	Meaning
SPOTW	SOL 101, 103, 112 only. Flag indicating that a list of element set and property pairs will follow, defining the elements and their associated properties for fatigue analysis of spot welds. See Remark 5. and 6.
ELSIDi	Same as ELSIDi under ELSET above. See Remark 3.
PFTGIDi	Same as PFTGIDi under ELSET above.
SEAMW	SOL 101, 103, 112 only. Flag indicating that a list of element set and property pairs will follow, defining the elements and their associated properties for fatigue analysis of seam welds. See Remark 5.
ELSIDi	Same as ELSIDi under ELSET above. These are to elements that make up the seam weld toe, root, or throat. See Remark 3.
PFTGIDi	Same as PFTGIDi under ELSET above.
NDSIDi	ID of a SET1 or SET3 entry listing grids of the elements defined by ELSIDi to be retained in the analysis. These grids define the seam line of the seam weld. If left blank, all nodes of the elements are retained. Nodes defined that are not part of ELSIDi are ignored. (Optional, Integer >0).
WELD <i>i</i>	Seam weld definition. One of the following: FILLET, OVERLAP, LASER, EDGE, or GENERIC, which define either a fillet, overlap, laser overlap, laser edge overlap or generic seam weld, respectively. (Character; Default=GENERIC). See Remark 7.
TYPE <i>i</i>	The type location on the seam weld that this set of elements represent. One of the following: TOE, ROOT, or THROAT. (Character; Default = TOE for all but WELD <i>i</i> = LASER where Default = ROOT). See Remark 7.
XELSET	Flag indicating that sets of elements to be excluded from the fatigue analysis will follow. (Optional, Character). See Remark 4.
XELSID <i>i</i>	ID of a SET1 or SET3 entry listing elements of the model to be excluded from the fatigue analysis. (Integer>0). See Remark 3.

Remarks:

1. **FTGDEF** bulk data entries are ignored if not selected by a **FATIGUE** case control. If no **FTGDEF** is present for a given fatigue analysis, all elements of the model that have fatigue material properties defined will be used with default properties except for fatigue analysis of spot and seam welds. See Remark 5. below.
2. If no PFTGID or PTFGIDi is specified, default properties will be assigned to the entities.
3. If a SET3 is specified, field 3 of the SET3 entry must be set to "ELEM". The SET4 entry must be specified to select elements by property ID. The following elements referenced by PSHELL, PSHEAR, and PSOLID properties are supported for standard S-N and ϵ -N fatigue analysis: CQUAD4, CQUADR, CQUAD8, CSHEAR, CTRIA3, CTRIAR, CTRIA6, CHEXA, CPENTA,

and CTETRA. The following elements referenced by PBAR, PBEAM, PSOLID, and PWELD properties are supported for fatigue analysis of spot welds: CBAR, CBEAM, CHEXA, CWELD. Only elements referenced by PSHELL properties are available for fatigue analysis of seam welds, excluding TRIA3 as no corner stresses are available from this element.

4. If only the XELSET flag is present, then the entire model is included in the fatigue analysis less the excluded entities.
5. For fatigue analysis of spot or seam welds the FTGDEF card is required with the corresponding keyword and line defining the elements of interest.
6. Elements used to represent spot welds are typically very stiff bar or beam (CBAR/CBEAM) elements, weld (CWELD) elements, or individual solid (CHEXA) elements. These elements connect the two metal sheets defined by shell elements (CSHELL). Bars and beams must connect directly to the grids of the shells, whereas welds and fasteners only need to pierce the shell elements. If individual solid elements are used, face G1-G2-G3-G4 must have its grids connected to shell elements that define the top sheet via RBE3 rigid elements. And face G5-G6-G7-G8 must have corresponding RBE3 elements connecting the bottom sheet. Some CWELD options auto-generate grids on the top and bottom metal sheets (when only GS grid is defined on CWELD entry with no GA/GB grids); the fatigue results will be associated to these generated grids.
7. The normals of the throat elements should point outward toward the welder, except for laser overlap, in which case the normals just need to be consistently the same direction. The elements defining the toe and root of the weld must have the top of the shell (Z2 layer) be the side where the crack is expected to develop. For full descriptions of the throat, root, and toe elements for the various seam welds, please see the *MSC Nastran Embedded Fatigue User's Guide*, which show proper modeling techniques. The WELDi and TYPEi entries are used for labeling purposes only and have no effect on internal calculations.
8. Use of TOPSTR/TOPRMS and/or TOPDMG in the same analysis allows the user to perform a 2-pass or a 3-pass run to speed up the analysis times of very large models and more quickly determine the critical damage locations.
 - If TOPSTR/TOPRMS is used alone (TOPDMG is left unspecified), than a 2-pass analysis is performed where the first (1st) pass determines the elements with the highest combined stresses based on the stress combination (COMB field on [FTGPARM](#) entry) and the largest range of the combined loading time histories. The second (2nd) pass computes damage on only the remaining elements.
 - If TOPDMG is used alone (TOPSTR=100% or left unspecified), then a 2-pass analysis is performed where the first (1st) pass determines the elements with the highest damage based on a set of reduced loading time histories. The second (2nd) pass computes damage on only the remaining elements using the fully populated time history loading.
 - If TOPSTR and TOPDMG are both specified, then a 3-pass analysis is performed where the first (1st) pass eliminates all but the highest stressed elements and the second (2nd) pass eliminates more elements based on damage due to a reduced set of time histories. The third (3rd) pass computes damage on the remaining element with the fully populated time history loading.

- If neither TOPSTR/TOPRMS or TOPDMG are specified (both are blank), then a 1-pass analysis is done on all the specified elements with fully populated loading time histories. This is the default scenario.

When using TOPDMG, the method in which the reduced time histories are created is determined by the settings on the RAINFLOW line of the [FTGPARM](#) entry. If this entry is absent or the RAINFLOW line is not present, then the default is RTYPE=LOAD with PCTRED=50%.

9. Element sets must be the same for all FATIGUE case control IDs that request surface resolved stresses through the FTGPARM entry (SRESOLVE field = YES). Otherwise a fatal condition is flagged and the analysis stops.
10. This parameter can be used by itself or in conjunctions with TOPSTR and/or TOPDMG. Only entities that pass this filter are reported in the f06 file and/or output to the OUTPUT2 and/or HDF5 files. If FATIGUE(STROUT=4) case control is used it is highly recommended that NENTS be used to limit the output. Temporary CSV files are created for each entity and F06 output can be enormous if not filtered using NENTS. If NENTS is used with the SRESOLVE option on the [FTGPARM](#) entry, then the number of entities printed may be more as the SRESOLVE option saves the number of grids and each element associated with those grids.

FTGEVNT**Fatigue Loading Events**

Groups simultaneously applied loads into loading events for pseudo-static fatigue analysis using SOL 101 or modal analysis using SOL 103 or random vibration fatigue using SOL 108 or SOL 111 by referencing FTGLOAD entries.

Format (SOL 101, 103, 108, 111):

1	2	3	4	5	6	7	8	9	10
FTGEVNT	ID	FLOAD1	FLOAD2	FLOAD3	FLOAD4	FLOAD5	FLOAD6	FLOAD7	
		FLOAD8	
		-etc-							
	“NAME”				EVNTNAM				

Example:

FTGEVNT	22	4	11						
	NAME								rough_road

Descriptor	Meaning
ID	Unique ID. See Remark 1.
FLOADi	ID of a FTGLOAD entry (Integer > 0, no default). See Remark 5.
NAME	Flag indicating that a name is to be associated with this load event. See Remark 2.
EVNTNAM	Event name associated with this event (Character). Can span fields 3 through 9, however for practical purposes, the name is truncated in f06 output. No spaces are allowed. See Remark 3. and 4.

Remarks:

1. Each **FTGEVNT** ID must be unique relative to all other **FTGEVNT** and **FTGSEQ** IDs.
2. Event names are passed to the f06 print file and displayed as the name of the event as opposed to the event ID. If no name is given, the event is referred to by its ID.
3. EVNTNAM cannot have numbers at the start of any field. For example, using 8 character formatted fields, the first character of fields 3-9 cannot be numeric (columns 1, 9, 17, etc. of the EVNTNAM field). These would be acceptable:
“Flights” or “F_l_i_g_h_t_s”.
But these would not be acceptable:
“1_Flights” or “F_1_i_g_2ts”
because the “1” and the “2” are in columns 1 and 9 of the EVNTNAM field.

4. EVNTNAM cannot handle spaces within its field as one might expect. If you wish to use spaces, make sure that once a space is used the next character does not start until the start of the next field. For example, using 8 character formatted fields, the following would be acceptable:
“My_flights” or “My Flights”.
But this would not be acceptable:
“My Flights”
and would end up just being displayed as “Myts” because once a space is encountered within the 8 character field, the code interprets that as no more character data until it processes the next 8 character field. It is best not to use spaces in EVNTNAM.
5. For random vibration fatigue (**SOL 108 and 111**),
 - Only one FTGLOAD of TYPE=PSD can be referenced
 - Only one FTGLOAD of TYPE=STATIC can be referenced
 - A FTGEVNT referencing a FTGLOAD of TYPE=STATIC by itself is not allowed
 - A FTGEVNT referencing a FTGLOAD of TYPE=SINE or NARROW by itself (or themselves) is allowed for pure deterministic and/or sine sweeps
 - Only single load PSD random input allows mixed deterministic, sine sweep, and static offsets on random loading. Only a static offset (FTGLOAD of TYPE=STATIC) can be mixed with multi-load PSD random input.
 - FTGSEQ entry referencing FTGEVNT entries cannot mix events containing single load and multi-load PSDs.
 - All multi-load PSD events must all have the same number of channels (input loads).
 - When only FTGLOAD entries of TYPE=SINE or NARROW are referenced (i.e., only deterministic loading with no random loading input), it is recommended that the ATYPE field of the FTGPARM entry be set to SINE.

FTGLOAD**Fatigue Cyclic Loading Variation**

Defines cyclic loading variation for pseudo-static fatigue analysis using SOL 101 or modal analysis using SOL 103 or random PSD, deterministic, sine sweep and/or static offset loading for SOL 108 and SOL 111.

Format (SOL 101 and 103):

1	2	3	4	5	6	7	8	9	10
FTGLOAD	ID	TID	LCID	LDM	SCALE/ MAX	OFFSET/ MIN	TYPE	CHNL	
	"UNITS"	EQUIV	EQNAME						

Format (SOL 108 and 111):

1	2	3	4	5	6	7	8	9	10
FTGLOAD	ID	TID	LCID		SCALE	OFFSET	TYPE		
	"DETLOAD"	F1	G1/H1	B1	F2	G2/H2	B2		
		F3	G3/H3	B3		
		-etc.-							
	"SWEEP"	F1	G1/H1		Fn	Gn/Hn			
		SWTYPE	SWNUM	SWRATE	NSWEEP	FRFUNIT	LDUNIT		

Examples:

FTGLOAD	55	4	2	1.0	1.0	0.0			
FTGLOAD	23	4	5	1.0	1.0	0.0	DB		
	UNITS	5.5	Flights						

Descriptor	Meaning
ID	Unique ID which is referenced by a FTGEVNT entry or directly by a FATIGUE case control in the case of SOL 101 only for a single load (Integer > 0). See Remark 1.
TID	<ul style="list-style-type: none"> ■ SOL 101, 103: Table ID of a TABLFTG (or TABLED1) entry that defines the time variation of the loading or the ID of a UDNAME entry for external definition of the loading time variation (Integer > 0). See Remarks 2. through 6. Ignored if TYPE=CONST or STATIC. ■ SOL 108, 111: leave blank unless: <ul style="list-style-type: none"> ■ TYPE=STATIC: UDNAME ID referencing an external OP2 file containing stress results for static loading offsets. Leave blank (and set LCID = 0 if a constant offset defined solely by the OFFSET field is desired). ■ TYPE = PSD with TIM2PSD: UDNAME ID referencing an external RPC or CSV file when time history to PSD conversion is requested. A TIM2PSD entry must be present in this case, otherwise TID is ignored.
LCID	<ul style="list-style-type: none"> ■ SOL 101: Subcase ID of a linear static solution SUBCASE (Integer > 0, no default). ■ SOL 103: Mode number for SOL 103 (Integer > 0, no default). ■ SOL 108, 111: <ul style="list-style-type: none"> ■ TYPE = PSD with no TIM2PSD: SID of RANDPS entry(ies). TID should be left blank in this case. ■ TYPE = PSD with TIM2PSD: -1 for fully correlated multi-input loading, -2 for uncorrelated multi-input loading. ■ TYPE = STATIC: SUBCASE ID of a static SUBCASE (freq=0.0). A TID must also be defined in this case to point to an external OP2 file. Set to zero (0) if a constant offset defined solely by the OFFSET field is desired. TID should be left blank in this case. ■ TYPE=SINE or NARROW: SUBCASE ID of transfer function (TF) used for deterministic or sweep loading. TID should be left blank in this case.
LDM	SOL 101, 103 only. Largest magnitude of the applied load (in the same units used to define the load time variation in field TID) used to normalize the load (Real, default=1.0). This effectively scales the stress to simulate a stress state due to a unit load. See Remark 7.
SCALE/ MAX	<p>SOL 101, 103: Scale factor applied to the load time history (Real, default=1.0). See Remark 7. Or the 1st peak level of a constant amplitude signal if TYPE=CONST (Real, default=1.0).</p> <p>SOL 108, 111: Acts as an additional scale factor on the input PSD. This applies only to TYPE=PSD (Real, default=1.0).</p>

Descriptor	Meaning
OFFSET/ MIN	SOL 101, 103: Offset applied to the load time history (Real, default=0.0). See Remark 7. Or the 2nd peak level of a constant amplitude signal if TYPE=CONST (Real, default = -1.0). MAX can be < or > than MIN for TYPE=CONST. SOL 108, 111: Offset of resultant stress PSD in stress units. This has the effect of adding a mean stress effect to the random load response. This only applies to TYPE=STATIC and if TID is blank, this is only a constant offset. With TID a variable offset plus this constant offset is applied (Real, default = 1.0).
TYPE	Flag indicating the type of load being defined. Values can be: blank, "DB", "DAC", "RPC", "CONST", "PSD", "SINE", "NARROW" or "STATIC". Default is blank. See Remarks 2. through 6.
CHNL	SOL 101 and 103 only. Channel of referenced RPC file (TYPE=RPC). (Integer >0, Default is blank). See Remark 12.
"UNITS"	SOL 101 and 103 only. Flag indicating that a fatigue equivalent unit name is defined for this loading. See Remarks 9. and 10.
EQUIV	Number of equivalent units (Real>0.0, default=1.0). See Remarks 9. and 10.
EQNAME	Equivalent name of this loading history. EQNAME can span across fields 4 through 9. If not defined it will be called <i>Repeats</i> (Character). No spaces are allowed. See Remarks 9. and 10.
"DETLOAD"	SOL 108 and 111 only. Flag indicating that deterministic load input is to follow for TYPE=SINE or NARROW. Fi Frequency of sine wave for TYPE=SINE or center frequency of narrow band frequency block for TYPE=NARROW. (Real>0.0; no Default). Gi Height of sine wave (Gi) in units of peak amplitude (FE stress units) for TYPE=SINE. Hi Height of narrow band PSD block (Hi) in units of (stress) ² /Hz for TYPE=NARROW. (Real>=0.0; no Default). Bi Width of narrow band frequency block. Only supplied for TYPE=NARROW. (Real>=0.0; no Default).
"SWEEP"	SOL 108 and 111 only. Flag indicating that sine sweep load input is to follow for TYPE=SINE (not supported for NARROW). F1/Fn F1/Fn Frequency of 1st and last sine waves for TYPE=SINE. (Real>0.0; no Default). G1/H1 Height of 1st (G1) and last (Gn) sine waves in units of peak amplitude (FE stress units) for TYPE=SINE. (Real>=0.0; no Default). Gn/Hn Gn/Hn Height of 1st (G1) and last (Gn) sine waves in units of peak amplitude (FE stress units) for TYPE=SINE. (Real>=0.0; no Default). SWTYPE The sweep type: Decibel (DB), Octave (OCT), or Linear (HZ). (Character; Default = HZ) SWNUM Number of sine/narrow bands to use (Integer>0 where N=SWNUM+1; Default = 50) SWRATE Sweep rate per second in HZ (default), DB or OCT. (Real>0.0; No Default) NSWEEP Number of sweep passes (Integer>0; No Default).

Descriptor	Meaning
FRFUNIT	Units of loading used to create FRF - can be acceleration (A), velocity (V), displacement (D), or force (F). Used for TYPE=SINE only. (Character; Default = A)
LDUNIT	Units of loading used to define sweep - can be acceleration (A), velocity (V), displacement (D), or force (F). Used for TYPE=SINE only. (Character; Default = A)

Remarks:

1. If a FATIGUE case control command invokes an ID that is present on both an FTGSEQ entry and a FTGLOAD entry, the FTGSEQ request will be honored and the FTGLOAD entry will be ignored unless it is referenced on a FTGEVNT entry.
2. For SOL 101 and 103, if the TYPE field is blank then TID references a TABLFTG or TABLED1 entry. For SOL 108 or 111, TYPE must be supplied.
3. For SOL 101 and 103, if TYPE=DB, DAC or RPC, then TID references a UDNAME ID. A UDNAME entry must be supplied in this case to specify the file and path of the externally defined load time variation. This file is expected to be in standard DAC file format or channel data in the form of a RPC file. TYPE=DB is the same as TYPE=DAC. You cannot mix TYPE=RPC designations with any other TYPES within the same fatigue analysis (FATIGUE=FID). If RPC is used, all FTGLOAD entries must use TYPE=RPC. The same RPC file must be referenced via UDNAME for all FTGLOADs of the same event (FTGEVNT). For SOL 108 and 111, if TYPE=STATIC or if TYPE=PSD with TIM2PSD entry active, the UDNAME references an external OP2 file or RPC/CSV channel data file, respectively.
4. For SOL 101 and 103, if TYPE is STATIC, the TID field should be left blank as it will be ignored. STATIC indicates that the stress state from the specified LCID is to act as a static offset with no load time variation when performing the linear superposition, which will give every element a different offset defined by the stress state at each element of the specified subcase, as opposed to simply specifying the OFFSET field, which gives every element the same offset. If the "STATIC" flag is specified, there must be at least two FTGLOAD entries defined and called out by a FTGEVNT entry, one of which must be time varying (see Remark 7.). For SOL 108 and 111, if TYPE is STATIC, the TID field must be supplied.
5. For SOL 101 and 103, if TYPE is CONST, the TID, LDM, and CHNL fields are ignored. The MAX and MIN fields are used to define the maximum and minimum values of a constant amplitude cycle. TYPE=CONST cannot be mixed with any other TYPE values for a given fatigue analysis (FATIGUE=FID). All loading must be of TYPE=CONST. This is sometimes referred to as block loading.
6. For modal analysis using SOL 103, the referenced load variation defines the modal participation factors for the referenced mode.
7. For SOL 101 and 103, the LDM, SCALE, and OFFSET are used together in the following manner to scale/modify the stress state in order to determine the resulting stress time variation:

$$\sigma_{ij}(t) = \left(P(t) \times \frac{\sigma_{ij,l}}{LDM} \times \text{SCALE} \right) + \text{OFFSET}$$

8. where $\sigma_{ij}(t)$ is the resulting stress tensor at time t, $\sigma_{ij,l}$ is the stress tensor from the subcase or mode defined by the LCID field, and P(t) is the y value of the load-time history at time t as defined by the TID field. For multiple loads, the principle of linear superposition is used to combine all loads for a single event. For SOL 108 and 111, LDM and SCALE act as a divisor and multiplier, respectively, to the defined loading. OFFSET is only used for TYPE=STATIC and acts as an additional constant offset over all entities.
9. If the "UNITS" flag is absent, the default fatigue equivalent unit is 1.0 *Repeats* of the stress time history. If this FTGLOAD is referenced by a FTGEVNT, then the equivalent units on this entry are ignored and those on the FTGSEQ entry take precedent. Only if a FTGLOAD is directly referenced by a FATIGUE case control are the fatigue equivalent units used as defined on the FTGLOAD entry.
10. Example of using equivalent units: If one repeat of the defined time history is equivalent to 5 times around a test track, the equivalent unit name, EQNAME, might be "*laps*," and the equivalent unit, EQUIV, would be 5.0. Fatigue life will be reported in these units if defined, otherwise they are reported as *Repeats* of the loading. Life output is reported in both *Repeats* and the fatigue equivalent units, if defined.
11. All FTGLOAD entries referenced by a FTGEVNT should reference different SUBCASEs for SOL 101 (or modes for SOL 103) and must have time variations consisting of the same number of points.
12. CHNL is only used for TYPE=RPC for SOL 101 and 103. If supplied, the specified channel of the referenced RPC file is used. If it is left blank, the next available channel sequentially from the last one referenced will be used. For example, if there are three FTGLOAD entries for a specific event and CHNL is blank for all three, the 1st one will use channel 1, the 2nd one will use channel 2 and the 3rd will use channel 3. If in this example the 1st specifies CHNL=11 and the others are blank, then the channels used will be 11, 12, and 13. If the 1st is left blank and the 2nd references CHNL=12, then the channels used will be 1, 12, and 13.

FTGPARM**Fatigue Parameters**

Defines parameters for a fatigue analysis in time domain SOLs 101, 103, and 112 and in frequency domain SOLs 108 and 111.

Format (SOLs 101, 103, 112):

1	2	3	4	5	6	7	8	9	10
FTGPARM	ID	TYPE	FACTOR	NTHRD	LOGLVL	LAYER	STROUT		
	"STRESS" or "STRAIN"	COMB	CORR	PLAST	LOC	INTERP	RECOVER	SRESOLVE	
		NANGLE							
	"RAINFLOW"	RTYPE	GATE	PCTRED					
	"CERTNTY"	SURV							
	"FOS"	OPTION	LIFE	BACKACC	MAXFAC	MINFAC			
	"DAMAGE"	CHECK	FLOOR	MAXDAM	FAILDAM				
	"SPOTW"	COMB	CORR	NANGLE	SWLOC	MIDDLE	TORSION		
	"SEAMW"	COMB	CORR	THICK	LOCSM	RESENT			
	"MULTI"	MMTHD	NONLWR	NONUPR	BIAXLWR	BIAXMID	BIAXUPR	ZERO	
		GATE							
	"NAVG"	MTHD	OUTPUT	NORMAL					

Format (SOLs 108, 111):

1	2	3	4	5	6	7	8	9	10
FTGPARM	ID	TYPE			LOGLVL	LAYER			
	"STRESS"	COMB	CORR	PLAST	LOC		RECOVER		
	"FOS"		LIFE						
	"VIBFTG"	ATYPE	MAXSTR	CLIPLVL	MAXPEAK	STRBINS	MAXFREQ	NCALC	

Examples:

FTGPARM	22	EN							
---------	----	----	--	--	--	--	--	--	--

FTGPARM	22	SN							
	STRESS	SGVON	NONE		NODA		CUBIC		

FTGPARM	22	SN							
	SEAMW		FKM	0					

FTGPARM	22	SN							
	SPOTW	STNDRD	SIMPLE	18	3	0	0		

FTGPARM	22	EN							
	MULTI	AUTO	0.25	0.5	-0.6	0.25	0.6	10.0	
		20.0							

Descriptor	Meaning
ID	Unique ID of the FTGPARM entry called out by a FATIGUE case control (Integer > 0). See Remark 1.
TYPE	SOL 101, 103, 112: Type of fatigue analysis: "SN" or "EN" (Character, Default=SN).
	SOL 108, 111: Type of fatigue analysis: "SN" or "EN" (Character, Default=SN). This is used only when there is a conflict with the defined material properties, i.e., both S-N and ε-N properties exist for the same material MID. Otherwise it is ignored and the material properties define the type of fatigue analysis to be performed..
FACTOR	SOL 101, 103, 112 only. Global scale factor to be applied to combined resultant stress output (Real>0.0, Default=1.0).
NTHRD	SOL 101, 103, 112 only. Number of threads to use for parallel processing for this fatigue analysis (Integer ≥ 0 ; Default = 1). Zero (0) is used as a flag to tell the code to automatically determine the number of threads to use.
LOGLVL	SOL 101, 103, 112: Level of messaging sent to the log file (Integer ≥ 0 , Default = 0; 0=None, 1=Error, 2=Info, 3=Low, 4=Medium, 5=High). Note that LOGLVL >3 can result in a significant performance penalty and should be used for debugging purposes only and limited to the entities of interest.
	SOL 108, 111: Level of output to generate (Integer ≥ 0 , Default = 0; 0=Standard, 1=Output all PSD responses, 2=Additionally output all Rainflow data). Note that LOGLVL >0 can result in a significant performance penalty and should be used for debugging purposes only and limited to the entities of interest only. See Remark 16.
LAYER	For shell elements, the output results layer to print to the f06 file. Values can be 0=Worst, 1=Top(Z2), 2=Bottom(Z1). (Integer, Default = 0). This is for printed output only. The analysis produces results for both layers, which are always available through the MASTER/DBALL, Output2, or other files for graphical postprocessors. See the FATIGUE case control entry.
STROUT	SOL 101, 103, 112 only. Requests output of the actual stresses (or strains) used in standard SN and eN fatigue analyses. Values can be 0=No Output, 1=Print Output, 2=Plot Output. (Integer, Default = 0). Printed output is placed in the F06 file and also available in an OESFTG data block for postprocessing purposes. Plot only places the data in the OESFTG data block. This request is overridden by the FATIGUE case control if it also includes a STROUT request.

Descriptor	Meaning
"STRESS" or	Flag indicating that stress is used in the fatigue calculation. See Remark 2.
"STRAIN"	SOL 101, 103, 112 only. Flag indicating that strain is used in the fatigue calculation. Not valid for TYPE = SN. See Remark 2.
COMB	Stress/strain combination to use in the fatigue analysis. Acceptable values are listed in Table 9-17 after the Remarks below (Character; Default= ABSMAXPR).
CORR	Mean stress correction to use in the fatigue analysis. Acceptable values are listed in Table 9-18 and Table 9-19 after the Remarks below (Character; Default=None).
PLAST	SOL 101, 103, 112: Plasticity correction for TYPE = EN. Value can be "NEUBER," or "SEEGER" for Neuber or Hoffmann-Seeger methods, respectively (Character; Default=NEUBER). See Remark 3. SOL 108, 111: Only NEUBER correction is available.
LOC	SOL 101, 103, 112: Location to report fatigue lives. Valid values are "NODA", "NODE", or "ELEM" based on nodal averaged, element nodal, or element centroid stresses or strains, respectively. (Character; Default = NODA). See the RECOVER field and Remark 4. Also see the "NAVG" field and parameters. SOL 108, 111: Location to report fatigue lives. Valid values are "NODE", "NODA", or "ELEM" based on element nodal, nodal averaged, or element center stresses, respectively. (Character; Default = NODA). See the RECOVER field and Remark 4.
INTERP	SOL 101, 103, 112 only. Interpolation limit for multi-curve mean stress correction method (Integer ≥ 0 ; 0=Use Max Curve, 1=Extrapolate; Default = 0).
RECOVER	Stress recovery method to determine stresses/strains for the fatigue calculation. Valid values are "SGAGE", "CORNER", "BILIN", "CUBIC" or "CENTER" (Character; Default = CORNER if LOC=NODA or NODE; Default = CENTER if LOC=ELEM and any other setting is ignored). These are the same values used for the STRESS (or STRAIN) case control output request. See LOC field and Remark 4.
SRESOLVE	SOL 101, 103, 112 only. Request for surface resolved, 2D stress state. Used only for 3D solid elements. (Character YES or NO; Default = NO). See Remark 5.

Descriptor	Meaning
NANGLE	The number of calculation angles for 2D critical plane analysis (COMB=CRITICAL) ($1 \leq \text{Integers} \leq 360$; Default = 36, i.e. every 10 degrees). Practical values are 360, 180, 120, 90, 72, 60, 45, 40, 36, 30, 24, 18, 15, 12, 10, 9, 8, 6, 5, 4 for every 1, 2, 3, 4, 5, 6, 8, 9, 10, 12, 15, 20, 24, 30, 36, 40, 45, 60, 72, 90 degrees, respectively. The more angles computed, the more compute intensive the analysis will be. In actuality, only half the angles are computed since the principals are the same 180 degrees opposite any computed angle.
"RAINFLOW"	SOL 101, 103, 112 only. Flag indicating that parameters that follow define rainflow cycle counting parameters for rainflow data reduction. See Remark 5.
RTYPE	Method of rainflow data reduction (Time History Compression). Value can be "LOAD" for load time history data reduction on each load time history or "FAST" for performing a less accurate, but computationally faster method (Character; no default). See footnote in Table 9-17.
GATE	Load value used as gate range. This load is used for gating out small disturbances of "noise" in the time history using a peak-valley extraction method to speed up the analysis. Only used if RTYPE=LOAD. GATE and PCTRED are mutually exclusive. PCTRED ignored if GATE is supplied. (Real ≥ 0.0 , Default=0.0)
PCTRED	Percent reduction value based on the maximum load range used to reduce the load time history using a peak-valley extraction method to speed up the analysis. Only used if RTYPE=LOAD. Ignored if GATE is supplied. GATE and PCTRED are mutually exclusive. (0.0 \leq Real ≤ 100.0 , Default=50)
"CERTNTY"	SOL 101, 103, 112 only. Flag indicating that the parameter that follows defines the certainty of survival in fatigue analysis.
SURV	Certainty of survival based on the scatter in the S-N or e-N curves. (0.1 \leq Real ≤ 99.9 ; Default = 50). See Remark 7.
"FOS"	SOL 101, 103, 112 : Flag indicating that parameters that follow are used in a factor of safety analysis. The presence of this flag triggers a factor of safety analysis. See Remark 8. and 14.
	SOL 108, 111 : Flag indicating that the total signal time duration desired is to be specified for use in calculating margin of safety. Only the LIFE field is required in this case and defaults to 1.0. A margin of safety is always calculated.
OPTION	Supported option is LIFE, requesting a life-based factor of safety analysis or NONE. (Character; default = LIFE).

Descriptor	Meaning
LIFE	SOL 101, 103, 112: The targeted design life given in user defined life units (such as laps, miles, etc.) as defined by “UNITS” line on FTGSEQ or FTGLOAD entry, or <i>Repeats</i> of the cyclic loading if no “UNITS” line exists (Real>0, no default). SOL 108, 111: Total time duration required for margin of safety calculation in time units specified by TUNIT field on FTGSEQ entry.
BACKACC	SOL 101, 103, 112 only. The back calculation accuracy used to control back calculation iterations that determine the scale factor on the applied stress level to achieve the target design life. Defined as a percentage error on the target design life. (0.01 < Real ≤100.0, Default = 1.0).
MAXFAC	SOL 101, 103, 112 only. The maximum safety factor to calculate and report. When this threshold is exceeded, the analysis will go on to the next element and report the maximum for the exceeded element (2.0 ≤ Real ≤ 5.e6, Default = 5.0).
MINFAC	SOL 101, 103, 112 only. The minimum safety factor to calculate and report. If the result is below this threshold, the analysis will report MINFAC as the safety factor for this element and go on to process the next element. (0.0<Real≤0.5, Default = 0.2).
"DAMAGE"	SOL 101, 103, 112 only. Flag indicating that parameters for static failure check are to follow.

Descriptor	Meaning
CHECK	Action to take on static failure. 0, 1, or -1 for Warn, Stop, or None, respectively. Default = 0. See Remark 9 . <ul style="list-style-type: none"> ■ 0=Warn: the calculation continues but a warning message is issued. The damage for this entity is also assigned a numeric value equal to the value of the FAILDAM field, 2.0, by default. Not recommended for fatigue analysis of spot welds. ■ 1=Stop: The analysis stops the damage calculation for an entity with static failure and continues on to the next entity. At the end of the entire analysis of all entities, if there has been a static failure, a fatal message is issued and the job is terminated. This setting is not recommended for fatigue analysis of spot welds. Fatigue output (CSV, FER, etc) that was requested is still kept, and entities that failed the UTS test have damage/life set to 2/0.5 (assuming the FTGPARM/FAILDAM is set to default value). ■ -1=False: No static failure check is made. Damage will be calculated up to the numerical limit defined by the MAXDAM field. Note that this may be using an extrapolation of the S-N curve and may predict damage values per cycle greater than 1.0. This also has an additional effect using standard SN parameters. In this case, the adjustment of the S-N curve below 1000 cycles to take into account the static strength of the material is disabled.
FLOOR	This parameter sets a lower limit on calculated damage. If a cycle has less predicted damage, the damage value is set to zero. This parameter is somewhat redundant as similar functionality may be accessed using the fatigue cutoff Nfc in the material definition. ($0.0 \leq \text{Real} \leq \text{MAXDAM}$; Default = 0.0).
MAXDAM	This parameter sets a numerical upper limit on the damage that can be predicted for each cycle. Setting this to >1 shows how much unacceptable damage has accumulated. ($\text{FLOOR} \leq \text{Real} \leq \text{FAILDAM}$; Default = 1.0).
FAILDAM	This parameter assigns the numerical value of damage for any entity (node or element) when the stress exceeds the UTS. This parameter is used to identify static failure from a single cycle when looking at damage. ($1.0 \leq \text{Real} \leq 1e30$ and $\text{MAXDAM} \leq \text{Real}$; Default = 2.0).
"SPOTW"	SOL 101, 103, 112 only. Flag indicating that parameters for fatigue analysis of spot welds are to follow. See Remark 10 .
COMB	Stress combination to use in the fatigue analysis. Acceptable values are listed in Table 9-17 after the Remarks below (Character; Default=STNDRD, which is basically a critical plane analysis).

Descriptor	Meaning
CORR	Mean stress correction to use in the fatigue analysis of spot welds. Only NONE or SIMPLE are valid for fatigue analysis of spot welds. See Table 9-18 and Table 9-19 below. (Character; Default = NONE).
NANGLE	The number of calculation angles in 360 degrees around the spot weld. (10≤Integer≤360; Default = 18, i.e., every 20 degrees)
SWLOC	Location on the spot welds to report fatigue life. Zero (0) reports worst case angle and location (top/bottom sheet or nugget); one (1) reports worst case angle for each location; two (2) reports worst case location for each angle; three (3) reports all locations and angles. (Integer 0, 1, 2, or 3, Default=0).
MIDDLE	Whether to process middle sheets if there are more than two sheets in the weld. (Integer 0 or 1, Default = 0 - do not process middle sheets). See Remark 12 .
TORSION	Whether to calculate torsion in the spot weld. (Integer 0 or 1, Default = 0 - do not calculate torsion). See Remark 13 .
"SEAMW"	SOL 101, 103, 112 only. Flag indicating that parameters for fatigue analysis of seam welds are to follow. See Remark 10 .
COMB	Stress/strain combination to use in the fatigue analysis of seam welds. Only ABSMAXPR and CRITICAL are supported. See Table 9-17 below. (Character; Default = ABSMAXPR).
CORR	Mean stress correction to use in the fatigue analysis of seam welds. Only NONE or FKM are valid for fatigue analysis of seam welds. See Table 9-18 and Table 9-19 below. (Character; Default=NONE).
THICK	Thickness correction to be applied. (Integer 0 or 1; Default =0 - no correction)
LOCSM	Location on seam weld to report life. Valid values are "NODE", "SGAGE", "CORNER", "BILIN", or "CUBIC" (Character; Default = NODE). If "STRESS" line is also included, LOCSM must be the same as LOC. LOC=ELEM is not valid for fatigue analysis of seam welds and cannot be mixed with LOC = NODE. See Remark 4 .
RESENT	Result entity type used in the fatigue analysis of seam welds; only STRESS is currently supported. (Character; Default = STRESS).
"MULTI"	SOL 101, 103, 112 only. Flag indicating that parameters for biaxial/multiaxial assessment are to follow. See Remark 11 . and 14 .

Descriptor	Meaning
MMTHD	Can be set to the following: <ul style="list-style-type: none"> ■ NONE = No multiaxial assessment is done. ■ SIMPLE = Calculates simple biaxiality ratios only. ■ STANDard =Standard method of assessment, which merely returns the results of the assessment. ■ AUTO = Performs the standard method, but then may recalculate fatigue damage depending on the results of the assessment.
NONLWR NONUPR	Used only when MMTHD = AUTO. Lower and Upper thresholds used to check if the loading is proportional. This is used in combination with the biaxiality ratio thresholds. NONLWR: (0<Real≤NONUPR, Default = 0.25) NONUPR: (NONLWR<Real, Default = 0.5) NONUPR not used if TYPE=SN
BIAXLWR BIAXMID BIAXUPR	Used only when MMTHD = AUTO. Lower, Middle, and Upper threshold ratios used to check if the loading is proportional. This is used in combination with the non-proportionality factor thresholds. For TYPE=EN: BIAXLWR: (-1≤Real≤BIAXMID, Default = -0.6) BIAXMID: (BIAXLWR≤Real≤BIAXUPR, Default = 0.25) BIAXUPR: (BIAXMID≤Real≤1, Default = 0.6) For TYPE=SN: BIAXLWR: (-1≤Real≤0, Default = -0.6) BIAXMID: Not used BIAXUPR: (0<Real≤1, Default = 0.6)
ZERO	Used only when MMTHD = AUTO. Stress range below which damage is assumed zero and therefore no recalculation is performed. Specified as a percentage of UTS. (0<Real≤100, Default=10.0).
GATE	Used only when MMTHD = AUTO. This gate value is used to prevent small stresses from adversely affecting the biaxiality calculation. Stresses below this value are not included in the biaxiality calculation; their biaxiality ratio will be set to zero (0). The value is set as a percentage of UTS. (0<Real≤100, Default=20.0).

Descriptor	Meaning
"NAVG"	SOL 101, 103, 112 only. Flag indicating that parameters for nodal (grid point) stress (or strain) averaging are to follow. The LOC field must be set to NODA, otherwise this line is ignored. These parameters are the same as those specified in the SURFACE and VOLUME case control for requesting grid point stresses/strains and the user is referred to the OUTPUT(POST) section for more details. All defaults are used if this line is omitted. See Remark 15.
MTHD	Method to calculate average grid point stresses/strains: <ul style="list-style-type: none"> ■ TOPO = Topological method (default) ■ GEOM = Geometric method.
OUTPUT	Coordinate system in which to translate stresses/strains before averaging: <ul style="list-style-type: none"> ■ -1 : Specifies the element coordinate system for output ■ 0 : Specifies the basic coordinate system for output (default) ■ CID : Specifies the coordinate system defined on a CORDij bulk data entry for output <p>The X-axis of this system is used to define the X stress direction. For shell elements the X-axis of the specified coordinate system is projected onto the surface or, that is, the resulting averaged stresses are a projection onto the surface.</p>
NORMAL	For shell elements defining a surface, this specifies the method to define the surface normal. For solid elements defining a volume, this specifies the method to define the stress Z-axis. Indirectly this also sets the reference direction for positive fiber and shear stress output, but has no effect when OUTPUT=-1. Can be set to R (default), X1, X2, or X3.
	M Specifies the reverse of the direction given by R, X1, X2, or X3 and must be entered as MR, MX1, MX2, or MX3 with no space between the M and the following letter. R Specifies the radius vector from the origin of reference coordinate system to the grid point.
	The Y stress direction is the cross product of the Z and X stress directions defined by the NORMAL and the X-axis of the OUTPUT coordinate system.
"VIBFTG"	SOL 108, 111 only. Flag indicating that vibration fatigue parameters follow. The line is optional and defaults are used if not present.
ATYPE	Analysis method to use to generate the PDF of rainflow ranges: DIRLIK, NARROW, STEIN, SINES. (Character; default = DIRLIK)

Descriptor	Meaning
MAXSTR	Maximum stress to use as a function of the number of RMS stress levels to determine maximum stress in the rainflow cycle count. (Real>0.0; Default=10.0).
CLIPLVL	Value of stress where all stresses are "clipped" (kept at that level) when doing fatigue life calculation. Real>0.0; (Default CLIPLVL=MAXSTR). CLIPLVL=6 means +/- 3*RMS amplitudes or 6*RMS in terms of range.
MAXPEAK	Used to calculate an estimation of maximum and minimum stress (number of RMS levels) in the random response. It is not used in any way to calculate damage or life. This is also used when the maximum elastic-plastic peak strain is calculated. Where a mean load is included the peak stress is calculated as the mean_stress + MAXPEAK*RMS and the minimum stress is calculated as the mean_stress - MAXPEAK*RMS. This is similar to the time domain approach where the maximum and minimum stress are retained from the rainflow cycle count. (Real>0; Default=CLIPLVL/2 or, if CLIPLVL not defined, MAXSTR/2).
STRBINS	Number of bins to generate in rainflow cycle count. Default = 1280 for ATYPE=SINES otherwise 32. Not recommended to use anything over 5000.
MAXFREQ	Percentage of frequency content retained as a function of the first spectral moment m_0 of the PSD defining the maximum frequency used to integrate the spectral moments for m_1 , m_2 , and m_4 . 0<=Real<=100.0; Default=99.9. 100% is not a recommended setting as it can cause numerical instability related to the calculation of the 4th spectral moment.
NCALC	Alternative options for the way "N" (life to failure) is calculated for a given stress bin. Valid values are MID, UPR, or AVG for middle, upper or average. (Character; Default=AVG). See Table 9-20 .

Remarks:

1. FTGPARM bulk data will be ignored if not selected by a FATIGUE case control entry. If a FTGPARM entry is not defined, default properties are used for the requested fatigue analysis.
2. **SOL 101, 103, 112:** For total life or stress-life (TYPE=SN), only STRESS results can be used. For crack initiation or strain-life (TYPE=EN), the fatigue analyzer may use either STRESS or STRAIN results from the finite element analysis. This selection should make no difference to the final results of a crack initiation calculation, as strains will always be calculated. The exception is when shell results are used. In this case, STRESS should be selected because only 2D results are available and the absence of the out-of-plane strains will cause incorrect calculation of combined parameters. It is an error to have both STRESS and STRAIN lines. If both are missing, then STRESS will be assumed with its default values.
3. **SOL 101, 103, 112:** PLAST can be set to NEUBER or SEEGER. Please note that NEUBER can be used universally for uniaxial stress states. SEEGER requires a 2D stress state and is generally used when the stress state is not purely uniaxial. PLAST is only valid for TYPE=EN.

4. If LOC=ELEM (element center), fatigue lives are calculated based on stresses/strains at element centroids (not recommended for anything but shell or 2D solid elements (plane stress/strain and axisymmetric) and the RECOVER field is ignored or automatically set to CENTER.

If LOC=NODE, the fatigue lives are calculated from the stresses/strains at the element nodes, resulting in multiple damage values per node. Note that for LOC=NODE, the RECOVER field can be set to SGAGE, BILIN, CUBIC, or CORNER to control which stresses are used in the fatigue analysis for element nodal stresses and correspond to the same options as on the STRESS case control.

If LOC is set to NODA, the same RECOVER options as LOC=NODE apply, however:

SOL 101, 103, 112: With LOC=NODA, fatigue lives are based on stresses/strains from nodal averaged (grid point) stresses, resulting in a single damage value at each node as opposed to using LOC=NODE where the fatigue lives are calculated from the stresses/strains at the element nodes, resulting in multiple damage values per node. The grid point stresses are generated using Nastran's GPSTRESS output request internally (it is not required to include the GPSTRESS case control unless you wish to view the grid point stress also). See the NAVG line settings also on how to control grid point stress calculations.

SOL 108, 111: With LOC=NODA, after the nodal transfer function (TF) stresses for each element contribution are converted to the requested stress combination (e.g. COMB=SGVON), the stresses are summed and averaged at each node. This is done at every frequency to give a single TF of stress at each node. Thus a single damage and fatigue life per node is reported as opposed to slightly different values when LOC=NODE. Please note that the coordinate system in which the elements are defined should be consistent (the same). Otherwise the nodal averaging will not be done using consistent coordinate systems. Thus, for solid element, setting CORDM on the PSOLID entry to the Basic coordinate system or a specified CID is required and setting CID=-1 or -2 (element systems) is not recommended. For shell elements, MCID should be set on the respective element entry.

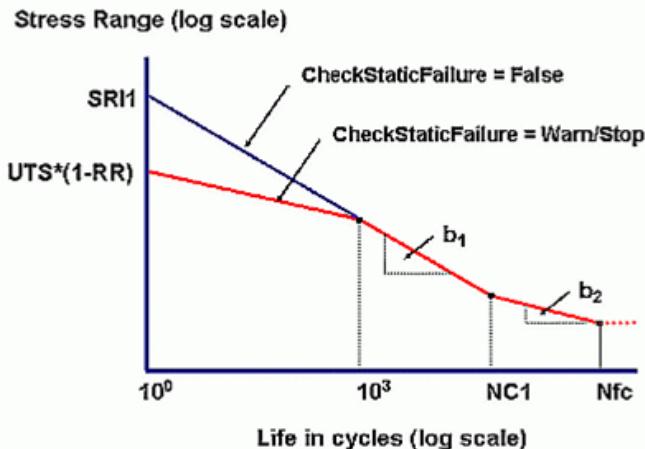
This parameter cannot be changed on a RESTART since it would require a different set of stress/strain data, which is not allowed on a RESTART for a pure fatigue analysis.

LOC=NODE generally gives more conservative answers than LOC=NODA since no averaging of the stresses/strains is done from the contributions of the surrounding elements.

5. **SOL 101, 103, 112:** SRESOLVE is applicable for all three values of LOC, i.e.: LOC = ELEM, NODE or NODA. SRESOLVE is an option to evaluate surface stresses instead of volume stresses. When surface resolved stresses are requested from 3D solid elements, a thin layer of 2D shell elements is internally created encompassing the volume of the elements defined on the FTGDEF entry from which the 2D stress state is then determined. This parameter is necessary if a multiaxial assessment of a model made of 3D elements is necessary, otherwise 3D elements are ignored during a multiaxial assessment. Fatigue life and multiaxial statistics are only reported for these newly defined 2D elements (or reduced set of nodes) on the surface of the volume (free surface). Interior nodes are not included in the fatigue calculation. The internal element IDs generated by this option can be controlled by using MSC Nastran SYSTEM cell 183 (Default is set to 200000000).

Element sets must be the same for all FATIGUE case control IDs that request surface resolved stresses. Otherwise a fatal condition is flagged and the analysis stops.

6. This RAINFLOW flag is only necessary to use if it desired to speed up the analysis for purposes of quick critical location identification or for sanity checks. RTYPE determines how to compress the input time history data for the purposes of speeding up the calculation, possibly at the loss of accuracy of the results. RTYPE=FAST simply reduces the load time histories to a single cycle using only the minimum and maximum from the input time series data, without losing phase information across multiple load cases. RTYPE=LOAD using PCTRED performs a multi-channel peak-valley extraction on the time series data prior to processing with PCTRED value set as a percentage of the range of the input data. RTYPE=LOAD using GATE performs a multi-channel peak-valley extraction on the time series data prior to processing with GATE value applied directly in the units of the time series data.
7. Certainty of survival is based on scatter in the S-N or ε -N curves. It is used to modify the curves according to the standard error parameters (SE_n) defined in the MATFTG material entry. A higher reliability level requires a larger certainty of survival.
8. SOL 101, 103, 112: This FOS option will calculate a type of safety factor for over design analysis to be performed. This analysis is in addition to the normal fatigue life/damage output and must be requested by the presence of this FOS flag and its parameters. This analysis method can be very useful for those components which predict infinite life, providing a measure of the risk of fatigue failure. The results of this analysis are factors by which the stress would have to be scaled to attain the specified design life. A value of one suggests that the specified life will be exactly attained whereas a factor less than one means the desired life will not be attained. Factors greater than one are, therefore, most desirable. By definition the resulting life values will be the target life, thus only the scale factor and maximum/minimum stress results are of interest when FOS is defined.
SOL 108, 111: This FOS option is actually a margin of safety (MOS) calculation, which requires only the desired or required duration of entire loading sequence in time units. By default this is set to 1.0 and the time units are those defined by the TUNIT field on the FTGSEQ entry.
9. The static failure check has not had any validation in regards to spot welds and any prediction of static failure should be treated with caution. Also, setting CHECK=0 or 1 (Warn or Stop) will result in the S-N curve being adjusted in the low cycle region (< 1000 cycles) in the same way as for the standard S-N method, as illustrated in the figure below. This is not desirable for spot weld fatigue. It can be avoided, either by setting CHECK = -1 (recommended), or setting a value of the UTS such that $UTS*(1 - RR) = SRI1$ or greater.



10. For fatigue analysis of spot and seam welds, besides the “SPOTW” and “SEAMW” lines themselves, “RAINFLOW”, “CERTNTY”, and “FOS” are also applicable. “MULTI” is applicable to seam weld, but not spot welds. All others are ignored. TYPE field is ignored as an SN analysis is forced for fatigue of spot and seam welds.
11. Biaxial/Multiaxial assessment can be requested for all but fatigue analysis of spot welds and is ignored by SOL 200 optimization runs. These assessments require a 2D state of stress. For this reason only shell elements are supported with this feature. MMTHD=AUTO is not allowed for fatigue analysis of seam welds.
12. The Rupp method for spot weld life prediction has not been validated for the prediction of fatigue damage occurring at the middle sheets for spot welds joining more than two sheets. It should also be noted that such failures are relatively rare, difficult to reproduce in the laboratory, and difficult to detect in practice. For this reason, MIDDLE is set off by default to omit any fatigue calculation on the middle sheets, which might otherwise give a false positive failure prediction.
13. Setting the TORSION option enables fatigue calculations to be made based on the shear stress around the periphery of the spot weld in each sheet due to the torsional load on the spot weld. Please note the following:
 - a. This option requires that suitable S-N curves be defined for the torsion case, which may mean a separate run for torsion vs for a non-torsion run.
 - b. Do not use this option on unless your modeling strategy can generate realistic torsional loads in the spot welds. For example, this is not true of the simple CBAR modeling approach.
 - c. The mean stress correction is applied a little differently, because the sign of a mean shear stress cannot influence the fatigue damage. For this reason, the ABSOLUTE value (i.e., the magnitude) of the mean stress is used when applying any mean stress correction for the torsion case.
 - d. The calculation is a completely separate operation—there is no combination of the torsional stresses with the other stresses on the spot weld.

- e. This option has had little validation, and no suitable S-N curves are provided with the software. However, it has been successfully used in practice, using S-N curves deduced from practical experience with car body structures, and has proved a useful tool for predicting premature failure in the small number of spot welds that may experience significant torsional loads.
 - f. The results of the torsional fatigue calculation are identified in the results by being associated with an angle of -1.
14. Factor of Safety (FOS) analysis is only compatible with the MMTHD=SIMPLE if doing a multi-axial/biaxial (MULTI) assessment in the same run. MMTHD=AUTO is not allowed in fatigue analysis of seam welds (SEAMW). The stress/strain combination (COMB) is ignored if MMTHD=AUTO because the program will determine whether to use a critical plane or absolute maximum principal combination method.
15. When LOC=NODA for requesting nodal average stresses, the grid point stresses generator is used to determine the nodal averaging. A limited set of controls is provided on the NAVG line to control how the averaging method. Internally, SURFACE and/or VOLUME definitions are created for the grid point stress request, just as if the GPSTRESS case control, with its associated OUTPUT(POST) commands, were given. However, only one SURFACE and/or VOLUME is determined based on the elements defined on the FTGDEF entry for requesting fatigue analysis. The user should be aware of possible stress discontinuities that can result from this.
16. By default the response PSD at the most critical location is output using LOGLVL=0 via a <jobname>PSD.csv file. If LOGLVL=1, response PSDs for all requested locations are output to that same file. Also a <jobname>RCC.csv file is produced with limited data showing the summation of damages from the rainflow cycle count. If LOGLVL=2, both files are created and in the latter, addition the stress and damage BIN data for each event is included. Be aware that this can create a lot of data, resulting in huge output files, and can severely degrade performance. Please use this option with care and limit the output to only the critical elements of interest.

Table 9-17 Allowable Values for the COMB Field

Stress/Strain Combination	SOL 101, 103, 112* Valid for TYPE	SOL 108, 111 Valid for TYPE	Meaning
ABSMAXPR	SN (+SeamWeld) EN	SN EN	Absolute Maximum Principal (default)†
MAXPRINC	SN	SN EN	Maximum Principal Stress‡
SGVON	SN EN	SN EN	Signed von Mises Stress (recommended for SOL 108, 111 but not default)
VONMIS	SN		von Mises Stress
SGMAXSHR	SN EN		Signed Maximum Shear
MAXSHR	SN		Maximum Shear
CRITICAL	SN (+Seam Weld), EN		Critical Plane Analysis - every 10 degrees by default unless NANGLE is specified otherwise for standard SN/EN analysis only- Not valid for 3D solid or CSHEAR elements - element must have an in-plane stress state.
STNDRD	SN (Spot Weld only)		This is basically a critical plane analysis with calculations being done at NANGLEs around the circumference of the spot weld.
COMPX COMPY COMPZ COMPXY COMPYZ COMPZX		SN EN	Individual stress components

* In time domain, six multiaxial components of the stress tensor are resolved into one uniaxial, combined value for fatigue calculations at each entity per time step since damage models are based on uniaxial theories. For S-N analysis, the signed von Mises (SGVON) will be smaller than the Absolute Maximum Principal (ABS MAXPR) when there is positive biaxiality and hence this selection would be less conservative. (Note also that some BS weld classes require shear stress to be used.) The signed parameters uses the sign of the absolute maximum principal value for conservative fatigue life estimates. It is not recommended to use non-signed values (MAXPRINC, VONMIS, MAXSHR). If using these, do not set the RTYPE=FAST on the 'RAINFLOW' line as this results in no cyclic stress cycling.

† In frequency domain, this is a Maximum Principal calculation using accurate phase scanning.

‡ In frequency domain, this is a fast Maximum Principal calculation using an approximate algorithm and is less accurate than ABSMAXPR, but improves computational speed.

Table 9-18 Allowable Values for the CORR Field

Mean Stress Correction	SOL 101, 103, 112 Valid for TYPE	SOL 108, 111 Valid for TYPE	Meaning
NONE	SN (+Spot/Seam Weld) EN	SN EN	No mean stress correction (default)
GOODMAN	SN	SN EN	Goodman mean stress correction
GERBER	SN	SN EN	Gerber mean stress correction
GDMANT	SN	SN EN	Tension only Goodman mean stress correction
GRBERT	SN	SN EN	Tension only Gerber mean stress correction
FKM	SN (+Spot/Seam Weld)		FKM mean stress correction method. Uses M1, M2, M3, M4 slopes as defined in MATFTG entry. See remarks in MATFTG entry.
SIMPLE	SN (Spot Weld only)		Modified FKM mean stress correction method where M1=M2=M3=M4=-MSS as defined in the MATFTG entry.
INTERP	SN		Interpolation method used with multiple SN curves only. TYPE field of MATFTG entry must be set to MEAN, RRATIO, or LIFE. Requires that there be multiple curves defined, one at R=-1 for TYPE=RRATIO, or zero (0) mean stress for TYPE=MEAN.
SWT	EN	SN EN	Smith-Watson-Topper mean stress correction
MORROW	EN	EN	Morrow mean stress correction
WALKER		EN	Walker mean stress correction
MMPDS		SN EN	Walker mean stress correction built into the MMPDS-01 material curves

Table 9-19 Allowable S-N vs. Mean Stress Correction Methods for SOLs 101, 103, 112

Mean Stress Correction	S-N Method				
	Standard	Multi Mean Curve	Multi R-Ratio Curve	Haigh	Bastenaire
NONE	YES	YES*	YES†	NO	YES
GOODMAN	YES	YES*	YES†	NO	YES
GERBER	YES	YES*	YES†	NO	YES
GDMANT	YES	YES*	YES†	NO	YES
GRBERT	YES	YES*	YES†	NO	YES
FKM	YES	YES*	YES†	NO	YES
SIMPLE‡	YES	NO	NO	NO	NO
INTERP	NO	YES	YES	YES	NO

* Allowed but a curve at zero mean stress must be present.

†Allowed but a curve at $R = -1$ must be present.

‡Spot Welds only.

Table 9-20 Alternative NCALC options to calculate “N” (cycles to failure)

NCALC option	Meaning
AVG	Calculation of “N” is done by adding up all the values of N within the BIN and then dividing through by the number of N values used. In this context the average value of N is calculated over the BIN.
MID	“N” is extracted at the mid (stress) point of the BIN.
UPPER	“N” is extracted at the upper (stress) point of the BIN.

FTGSEQ

Fatigue Load Sequence

Defines the loading sequence for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or SOL 112 or vibration fatigue using SOL 108 or SOL 111.

Format (SOL 101, 103, 112):

1	2	3	4	5	6	7	8	9	10
FTGSEQ	ID	EVNTOUT	METHOD						
	FID1	N1	FID2	N2	FID3	N3	FID4	N4	
	FID5	N5	
	-etc.-								
	"UNITS"	EQUIV	EQNAME						

Format (SOL 108, 111):

1	2	3	4	5	6	7	8	9	10
FTGSEQ	ID	EVNTOUT		TUNIT	LDM				
	FID1	N1	FID2	N2	FID3	N3	FID4	N4	
	FID5	N5	
	-etc.-								
	"UNITS"	EQUIV	EQNAME						

Examples:

FTGSEQ	1								
	6	15							

Descriptor	Meaning
ID	Unique ID with respect to all other FTGSEQ and FTGEVNT entries called out by a FATIGUE case control. (Integer>0) See Remark 1.
EVNTOUT	Flag for requesting fatigue output for each event individually. (Integer; 0=no or 1=yes; Default = 0). See Remark 7. For SOL 108 and 111, this also controls the written output request for spectral statistics (m0, m1, m2, m4, etc.).
METHOD	SOL 101, 103, 112 only. Event processing method. 0=independent, 1=Combined Full, 2= Combined Fast (Integer; Default = 0). This entry is only honored from the controlling FTGSEQ entry when called out by the FATIGUE case control. Otherwise it is ignored. See Remark 2.
TUNIT	SOL 108, 111 only. Time units of Ni values. SECS, MINS, HRS, DAYS, WKS, MTHS, YRS for seconds, minutes, hours, days, weeks, months, or years, respectively (Character; Default=SECS).

Descriptor	Meaning
LDM	SOL 108, 111 only. FE load magnitude used to create transfer function. Used to normalize the transfer function stresses (Real, Default=1.0).
FIDi	<ul style="list-style-type: none"> ■ SOLs 101, 103: ID of a FTGEVNT or another FTGSEQ entry for pseudo-static fatigue analysis using SOL 101 or a modal analysis using SOL 103 (Integer > 0). See Remark 3. and 4. ■ SOL 112: A subcase ID that represents the loading event or another FTGSEQ entry for modal transient fatigue analysis using SOL 112. (Integer > 0). See Remark 3. and 4. ■ SOL 108, 111: ID of a FTGEVNT entry for random vibration fatigue using SOLs 108 or 111. Nested FTQSEQs are not allowed.
Ni	<ul style="list-style-type: none"> ■ SOLs 101, 103, 112: Number of repeats of this loading sequence or event (Real>0.0, Default=1.0). Ignored if only one event is defined. For METHOD equal 1 and 2, Ni must be a whole number, i.e., 1.0, 2.0, 3.0, etc. In other words, fractions of events are not allowed. Fractional events are also not allowed for any METHOD if the corresponding FID references a FTGSEQ entry. See Remark 3. and 4. ■ SOL 108, 111: Duration of each event in TUNIT units. This value is not used for random loading only. These values are also not needed and overwritten if SWRATE/NSWEEP are provided on a FTGLOAD entry of TYPE= SINE (sine sweep). (Real>0.0; Default=1.0).
"UNITS"	Flag indicating that a fatigue equivalent unit name is applied to this loading. See Remark 5. and 6.
EQUIV	Number of equivalent units. (Real>0.0; Default=1.0). See Remark 5. and 6.
EQNAME	Equivalent name of this loading event. EQNAME can span across fields 4 through 9. If not defined it will be called <i>Repeats</i> (Character). No spaces are allowed. See Remark 5. and 6.

Remarks:

1. FTGSEQ bulk data are called out by FATIGUE case control.
2. Processing of events can be done by determining the damage due to each event independently (default) and then summing the damage due to all events. Or the events can be concatenated and damage determined after rainflow cycle counting over all events. The advantage of the independent method over the combined methods is computational expense versus accuracy. The combined method will close all cycles, whereas the individual method may miss a large damaging cycle if the cycle begins in one event and ends or closes in a subsequent event. The combined fast method performs a rainflow count data reduction to speed up the analysis and determine the most critical locations first and then redoes a full analysis on the critical locations.

3. Once a FTGSEQ bulk data entry is referenced in an FIDi field, it can't be referenced again in any other FTGSEQ entries (within its own associations - the same fatigue analysis) to avoid infinite loops. And if it is referenced by the FATIGUE case control, it cannot appear in any FIDI field of the FTGSEQ bulk data.
4. Different FTGEVNTs can be set up and the user can construct each sequence by specifying how many times to repeat each event in a sequence. One sequence could then be referenced in another sequence to tell the new sequence how many times to repeat that sequence. As an example, assume there are three events an automobile is subjected to: cobble stones, pot holes, speed bumps. One sequence might be five (5) "cobble stones," six (6) "potholes" and three (3) "speed bumps." This sequence may be called "torture track." Also define two more events called "cornering left" and "cornering right." A load sequence of ten (10) "cornering left" and ten (10) "cornering right" might be called "country road." Now with a nested FTGSEQ you can put these together any way you want. So one fatigue analysis might use a sequence of only "country road," another of only "torture track" and another of a combined six (6) "torture tracks," five (5) "country roads," followed by one (1) more "torture track" and one (1) more "country road" This would result in 3 fatigue analyses as shown in [Table 9-21](#) below.
5. If the "UNITS" flag is absent, the default fatigue equivalent unit is 1.0 *Repeats* of the resulting stress time history sequence. Equivalent units specified on FTGLOAD entries are ignored when FTGSEQ entries are used
6. Example of using equivalent units: If one repeat (or total duration) of the sequence is equivalent to five (5) times around a test track, the equivalent unit name, EQNAM, might be "laps," and the equivalent unit, EQUIV, would be five (5). Fatigue life will be reported in these units if defined, otherwise they are reported as repeats of the loading sequence. Life output is reported in both *Repeats* and the fatigue equivalent units, if defined.
7. **SOL 101, 103, 112:** For duty cycle jobs, temporary time series channel files are created. Add 10 to the EVNTOUT value if you wish to keep the files (.dac or .s3t extension), otherwise they are deleted after the job terminates. For SOL 200 only EVNTOUT=0 is supported. Also some additional output file formats are not supported if EVNTOUT>0. See the [FATIGUE case control entry](#) for STROUT=4 limitations.
8. **SOL 108, 111:** For vibration fatigue a FTGSEQ cannot reference both an event with single input random loading and multiple input random loading.

Table 9-21 Example of nested FTGSEQ entries

```

SET 1=22,33,44
FATIGUE=1
BEGIN BULK
$
FTGSEQ,22      $<-- 1 country road only
,2
FTGSEQ,33      $<-- 1 torture track only
,3
$combined
FTGSEQ,44      $<-- 6 torture track, 5 country road, + 1 of each
,3,6.0,2,5.0,3,1.0,2,1.0
$country road:
FTGSEQ,2
,8,10.0,9,10.0   $<-- 10 cornering left + 10 cornering right

```

```
$torture track:  
FTGSEQ 3  
    ,5,5.0,6,6.0,7,3.0 $<-- 5 cobble stones, 6 potholes, 3 speed bumps  
$  
FTGEVNT,5...    $cobble stones  
FTGEVNT,6...    $potholes  
FTGEVNT,7...    $speed bumps  
  
FTGEVNT,8...    $cornering right  
FTGEVNT,9...    $cornering left
```

GBAG**Gas-Bag Pressure Definition**

Defines the pressure within an enclosed volume. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
GBAG	GID	BSID	TRIGGER	TRIGGERV	PORID	INFID	HTRID	INTID	
	CPGAS	PGPASV	RGAS	PENV		REVERSE	CHECK	PINIT	
	TINIT	TENV							

Examples:

GBAG	101	37			11	12			
	CONSTANT	1000.	297.0	101325.		ON	ON		
	293.0								

Descriptor	Meaning			
GID	Unique gas-bag number.	I > 0	Required	
BSID	Identification number of a BSURF, BCBOX, BCPROP, BCMATL, or BCSEG entry defining the surface of the gas-bag.	I > 0	Required	
TRIGGER	The time-dependent parameters are offset in time. TIME The offset is defined at TRIGGERV.	Character	TIME	
TRIGGERV	The value of the offset in time.	Real	Required	
PORID	Number of a set of LEAKAGE entries, that defines the porosity (permeability) and holes for the gas-bag surface and/or subsurfaces.	Integer > 0	No porosity	
INFID	Number of a set of ABINFL entries, that defines the one or more inflators on subsurface(s) of the GBAG surface.	Integer > 0	No inflators	
HTRID	Number of a set of HEATLOS entries, that defines the heat transfer definitions for the gas-bag surface and/or subsurfaces.	I > 0	No heat transfer	
INTID	ID of an INITGAS entry specifying the initial gas composition for this GBAG.	I > 0	No initial gas composition	

Descriptor	Meaning			
CPGAS	The variation of the specific heat constant at constant pressure. CONSTANT The specific heat is constant and specified in CPGASV.	Character	CONSTANT	
CPGASV	The specific heat of the gas.	Real	Required	
RGAS	Gas constant of the inflowing gas.	Real	Required	
PENV	Environmental pressure surrounding the gas bag.	Real	Required	
REVERSE	Normal auto-reverse switch. ON The normals of the surface are automatically reversed if necessary so that they point in the same direction and provide a positive volume. OFF The normals are not automatically reversed.	C	ON	
CHECK	Normal checking switch. ON The normals of the surface are checked to see if they all point in the same direction and provide a positive volume. OFF The normals are not checked. If REVERSE is set to ON, CHECK is automatically set to ON.	C	ON	
PINIT	Initial pressure inside the gas bag.	Real	PENV	
TINIT	Initial temperature inside the gas bag. See Remark 4.	Real	Required.	
TENV	Environmental Temperature.	Real > 0	TINIT	

Remarks:

1. The BSURF entry referenced by the BSID field must form a closed volume.
2. The pressure in the gas bag is applied to all the faces of the outer boundary.
3. TINIT is the temperature of the gas inside the volume at *time* = 0. At *time* = 0, the mass of the gas inside the gas bag is calculated as

$$m = \frac{P_{init}V}{RT_{init}}$$

where, P_{init} the initial pressure, V the volume, R the gas constant, and T_{init} the initial gas temperature.

4. The flow through exhaust openings, leakage areas and user-specified outflow rate is accumulated. The volumetric porosity contributes to the outflow of gas as

$$\dot{m}_{out} = \rho \cdot Q = \frac{p}{R \cdot T} \cdot Q$$

where

Q = volumetric flow rate

ρ = density inside the bag

p = pressure inside the bag

R = gas constant

T = temperature inside the bag

\dot{m}_{out} = mass outflow rate

The value of Q can be specified as a constant, as a function of the pressure difference, or as a function of time. Negative values for the volumetric flow rate are not allowed, since this would mean inflow of outside air.

5. A mixture of BSURF, BCBOX, BCPROP, BCMATL or BCSEG with the same BSID is allowed. However multiple BSID of the same type is not allowed. When using this option, special care must be taken to assure the same element is not part of multiple BSID definitions.

GBAGCOU**General Coupling to Gas-Bag Switch**

Defines a switch from full gas dynamics to uniform pressure formulation. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
GBAGCOU	ID	CID	GID	TSTART	PERCENT				

Examples:

GBAGCOU	1	100	101	0.0	5				
---------	---	-----	-----	-----	---	--	--	--	--

Descriptor	Meaning			
ID	Unique number of a GBAGCOU entry.	I > 0	Required	
CID	Number of a COUPLE entry.	I > 0	Required	
GID	Number of a GBAG entry.	I > 0	0.0	
TSTART	Time after which the coupling algorithm checks if a switch to the uniform pressure method is valid. It is valid when the following is true:	Real ≥ 0	0.0	
	$\text{Max}\left[\frac{P_{\max} + P_{\text{average}}}{P_{\text{average}}}, \frac{P_{\text{average}} - P_{\min}}{P_{\text{average}}}\right] < \frac{\text{PERCENT}}{100}$			
	where			
	Pmax = maximum Eulerian pressure exerted on the SURFACE			
	Pmin = minimum Eulerian pressure exerted on the SURFACE			
	Paverage = average Eulerian pressure exerted on the SURFACE			
PERCENT	Value used in validity check as defined previously.	R ≥ 0 .	0.0	

Remarks:

1. The BSID referenced by the COUPLE entry CID and by the GBAG entry GID must be the same.
2. All Eulerian and general coupling calculations are deactivated after transition from gas dynamics to uniform pressure.

GENEL**General Element**

Defines a general element.

Format:

1	2	3	4	5	6	7	8	9	10
GENEL	EID		UI1	CI1	UI2	CI2	UI3	CI3	
	UI4	CI4	UI5	CI5	-etc.-				

UIm -- The last item in the UI list will appear in one of fields 2, 4, 6, or 8.

	"UD"		UD1	CD1	UD2	CD2	-etc.-		
--	------	--	-----	-----	-----	-----	--------	--	--

UDn -- The last item in the UD list will appear in one of fields 2, 4, 6, or 8.

	"K" or "Z"	KZ11	KZ21	KZ31	-etc.-	KZ22	KZ32		
	-etc.-		KZ33	KZ43	-etc.-				

KZmm -- The last item in the K or Z matrix will appear in one of fields 2 through 9.

	"S"	S11	S12	-etc.-		S21	-etc.-		
--	-----	-----	-----	--------	--	-----	--------	--	--

Smn -- The last item in the S matrix will appear in one of fields 2 through 9.

Example:

GENEL	629		1	1	13	4	42	0	
	24	2							
	UD		6	2	33	0			
	Z	1.0	2.0	3.0	4.0	5.0	6.0	7.0	
	8.0	9.0	10.0						
	S	1.5	2.5	3.5	4.5	5.5	6.5	7.5	
	8.5								

Descriptor	Meaning
EID	Unique element identification number. (Integer > 0)
Uli, Cli UDj, CDj	Identification numbers of degrees-of-freedom in the UI or UD list, in sequence corresponding to the [K], [Z], and [S] matrices. UIi and UDi are grid point numbers, and CLI and CDj are the component numbers. If a scalar point is given, the component number is zero. (Integer ≥ 0)
KZij	Values of the [K] or [Z] matrix ordered by columns from the diagonal, according to the UI list. (Real)

Descriptor	Meaning
Sij	Values of the [S] matrix ordered by rows according to the UD list. (Real)
"UD", "K", "Z", and "S"	Character strings that indicate the start of data belonging to the UD list or the [K], [Z], or [S] matrices.

Remarks:

1. The stiffness approach:

$$\begin{bmatrix} f_i \\ f_d \end{bmatrix} = \begin{bmatrix} K^{-1} & -KS \\ -S^T K & S^T KS \end{bmatrix} \begin{bmatrix} u_i \\ u_d \end{bmatrix}$$

The flexibility approach:

$$\begin{bmatrix} u_i \\ f_d \end{bmatrix} = \begin{bmatrix} Z^{-1} & S \\ -S^T & O \end{bmatrix} \begin{bmatrix} f_i \\ u_d \end{bmatrix}$$

where

$$\{u_i\} = [u_{i1}, u_{i2}, \dots, u_{im}]^T$$

$$\{u_d\} = [u_{d1}, u_{d2}, \dots, u_{dn}]^T$$

$$[KZ] =$$

$$[K] \text{ or } [Z] = \begin{bmatrix} KZ_{11} & \dots & \dots & \dots \\ KZ_{21} & KZ_{22} & \dots & \dots \\ KZ_{31} & KZ_{32} & \dots & \dots \\ \vdots & \vdots & \vdots & \vdots \\ KZ_{m1} & \dots & \dots & KZ_{mm} \end{bmatrix} \text{ and } [KZ]^T = [KZ]$$

$$[S] = \begin{bmatrix} S_{11} & \dots & S_{1n} \\ S_{21} & \dots & \dots \\ S_{31} & \dots & \dots \\ \vdots & \vdots & \vdots \\ S'_{m1} & \dots & S_{mn} \end{bmatrix}$$

The required input is the $\{u_i\}$ list and the lower triangular portion of $[K]$ or $[Z]$. Additional input may include the $\{u_d\}$ list and $[S]$. If $[S]$ is input, $\{u_d\}$ must also be input. If $\{u_d\}$ is input but $[S]$ is omitted, $[S]$ is internally calculated. In this case, $\{u_d\}$ must contain six and only six degrees-of-freedom.

The forms shown above for both the stiffness and flexibility approaches assume that the element is a free body with rigid body motions that are defined by $\{u_i\} = [S]\{u_d\}$. See [General Element Capability \(GENEL\)](#) (Ch. 3) in the *MSC Nastran Reference Guide* for further discussion.

2. When the stiffness matrix K is input, the number of significant digits should be the same for all terms.
3. Double-field format may be used for input of K or Z .
4. The DMIG entry or the INPUTT4 module offer alternative methods for inputting large matrices.
5. The general element entry in the example above defines the following:

$$\{u_i\} = [1-1, 13-4, 42, 24-2]^T$$

$$\{u_d\} = [6-2, 33]^T$$

where i-j means the j-th component of grid point i. Points 42 and 33 are scalar points.

$$[Z] = \begin{bmatrix} 1.0 & 2.0 & 3.0 & 4.0 \\ 2.0 & 5.0 & 6.0 & 7.0 \\ 3.0 & 6.0 & 8.0 & 9.0 \\ 4.0 & 7.0 & 9.0 & 10.0 \end{bmatrix} \quad [S] = \begin{bmatrix} 1.5 & 2.5 \\ 3.5 & 4.5 \\ 5.5 & 6.5 \\ 7.5 & 8.5 \end{bmatrix}$$

GENUDS**User Data for Notify User Defined Service or Subroutine**

The GENUDS defines integer, real and character type data that will be passed to the notify method in runtime info interface. The notify method will be called at start of load case, start of increment, end of increment and end of load case.

Format:

1	2	3	4	5	6	7	8	9	10
GENUDS	SRV_ID								
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	...	IDATAAn				
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	...	RDATAAn				
	“CHAR”	CDATA1	CDATA2	...	CDATAAn				

Example:

GENUDS	MY_SRV								
	INT	1	2	100					

Descriptor	Meaning
SRV_ID	The service identifier used in the CONNECT SERVICE statement. (Character, no default)
“INT”	Keyword indicating that the following data is integer. (Character)
IDATAi	User supplied integer data. (Integer, no default)
“REAL”	Keyword indicating the following data is real. (Character)
RDATAi	User supplied real data. (Real, no default)
“CHAR”	Keyword indicating the following data is character. (Character)
CDATAi	User supplied character data. (Character, no default)

Remarks:

1. The SRV_ID is the service identifier of SCA service in the CONNECT SERVICE statement. The SCA service should have implemented the RuntimeInfo interface.
2. A CDATAi entry cannot be the character “INT”, “REAL” or “CHAR”.

GMBC**General Enforced Displacement Definition**

Defines enforced displacements for GRID, FEEDGE, GMCURV, FEFACE, and GMSURF entries.

Format:

1	2	3	4	5	6	7	8	9	10
GMBC	LID	SPCID	C	ENTITY	ID	METHOD	FIELD1	FIELD2	
	FIELD3	FIELD4	-etc.-						

Example:

GMBC	127	1	2	GMCURV	1	QUAD	1.	2.	
	1.0								

Descriptor	Meaning		
LID	LOAD set identification number. See Remark 2.	Integer ≥ 0	Required
SPCID	SPC set identification number. See Remark 2.	Integer > 0	Required
C	Component number in the output coordinate system (global). See Remarks 3. and 4.	$0 \leq \text{Integer} \leq 6$	Required
ENTITY	Entity that the enforced displacement is applied to (Specify GRID, FEEDGE, GMCURV, FEFACE, or GMSURF). See Remarks.	Character	Required
ID	ID of the entity selected above. See Remarks.	Integer > 0	Required
METHOD	Method used to supply the data (EQUATION or TABLE, CONSTANT, LINEAR, QUAD, CUBIC). See Remarks.	Character	Remark
FIELDi	Enforced displacement data. See Remarks.	Integer or Real	Required

Remarks:

1. GMBC is the recommended entry for specifying boundary conditions and must be selected with Case Control command SPC = SPCID.
2. LID and SPCID refer to Case Control commands for specifying loads and boundary conditions. Whenever there are several nonzero enforced motion vectors supplied, the most efficient processing of the data (single decomposition of the stiffness matrix) is accomplished by specifying both LID and SPCID.

LID	Result
> 0	Generate SPC entries with zero displacements and SPCD entries with non-zero displacements.
0	Generate SPC entries with non-zero displacements only

3. The components of motion specified by C (field 4) of all degrees-of-freedom in the output coordinate system (Global System) associated with an entity will be constrained.
4. If C = 0 is specified then the degrees-of-freedom will be taken out of the constraint set. In this case the method field is not required.
5. The component is a single integer (1 or 2 or 3 etc.). Use multiple GMBC entries to enforce constraints on multiple components.
6. If METHOD = “EQUATION”, “TABLE”, or “CONSTANT” then FIELD1 is:

METHOD	FIELD1	Type
EQUATION	ID of a DEQATN entry defining the displacement value as a function of location	Integer > 0
TABLE	ID of a TABLE3D entry defining the displacement value as a function of location	Integer > 0
CONSTANT	Value of enforced displacement	Real

7. When METHOD = CONSTANT, a constant displacement is specified for the FEEDGE, GMCURV, FEFACE, and GMSURF entities.
8. If ENTITY = “FEEDGE” the METHOD field can be used to specify, linear, quadratic, and cubic displacements. FIELD1 and FIELD2 correspond to GRID1 and GRID2 on the FEEDGE entry. The values in FIELD3 and FIELD4 are:

Applying Linear, Quadratic, and Cubic Displacements to an FEEDGE		
METHOD	FIELD3	FIELD4
LINEAR	blank	blank
QUAD	Value at 1/2 chord length	blank
CUBIC	Value at 1/3 chord length	Value at 2/3 chord length

9. If ENTITY = “FEFACE” the METHOD field specifies linear or quadratic displacements. The values of FIELDi are location specific:
 - Quadrilateral FEFACE

METHOD	FIELD1 through FIELD4	FIELD5 through FIELD8	FIELD9	Displacement Function
LINEAR	Value at GRID1, 2, 3, 4	blank	blank	Linear
QUAD	Value at GRID1, 2, 3, 4	Value at mid side of EDGE1, 2, 3, 4	Value at middle of FEFACE	Quadratic

- Triangular FEFACE

METHOD	FIELD1 through FIELD3	FIELD4 through FIELD6	Displacement Function
LINEAR	Value at GRID1, 2, 3	blank	Linear
QUAD	Value at GRID1, 2, 3	Value at mid side of EDGE1, 2, 3	Quadratic

10. In general, the hierarchy set to resolve the conflicts arising in the enforced displacement input data is: GRIDs followed by FEEDGEs followed by GMCURVs followed by FEFACEs followed by GMSURFs. This means that:
- In general the program does not allow the user to supply multiple values of enforced displacements for the same component (C).
 - Displacement values specified for each component of a given GMSURF entry are applied to the same component of all GRID, FEEDGE, and FEFACE degrees-of-freedom that lie within the GMSURF.
 - Displacement values specified for each component of different GMSURF entries are averaged and applied to the same component of all GRID, FEEDGE degrees-of-freedom that are shared by (that are common to) the multiple GMSURFs.
 - Displacement values specified for a given FEFACE entry are applied to all GRID, FEEDGE, and FEFACE degrees-of-freedom that lie within the FEFACE. This data overrides the data that is specified for all the components of the given GRID, FEEDGE and FEFACE degrees-of-freedom that lie within the FEFACE by using GMSURF entries.
 - Displacement values specified for each component of different FEFACE entries are averaged and applied to the same component of all GRID, FEEDGE degrees-of-freedom that are shared by (that are common to) the multiple FEFACES. This data overrides the data that is specified for all the components of the given FEEDGE and GRIDs by using GMSURF entries.
 - Displacement values specified for each component of a given GMCURV entry are applied to the same component of all GRID, FEEDGE, degrees-of-freedom that lie within the GMCURV. This data overrides the data for all the components that are specified for the given FEEDGE and GRIDs by using GMSURF or FEFACE entries.

- Displacement values specified for each component of different GMCURV entries are averaged and applied to the same component of all GRID degrees-of-freedom that are shared by (that are common to) the multiple GMCURVs. This data overrides the data for all the components that are specified for the given GRIDs by using GMSURF or FEFACE entries.
- Displacement values specified for each component of a given FEEDGE entry are applied to the same component of all GRID, FEEDGE, degrees-of-freedom that lie within the FEEDGE. This data overrides the data for all the components that is specified for the given FEEDGE and GRIDs by using GMCURV or FEFACE or GMSURF entries.
- Displacement values specified for each component of different FEEDGE entries are averaged and applied to the same component of all GRID degrees-of-freedom that are shared by (that are common to) the multiple FEEDGES. This data overrides the data for all the components that are specified for the given GRIDs by using GMCURV or FEFACE or GMSURF entries.
- Grids have the highest priority, i.e., any value/property specified using a GRID entry overrides all other information associated with that GRID. If multiple entries are used for a given GRID, e.g., multiple SPCs, then the existing rules govern (SPCs are combined, FORCE is added, SPCDs for the same component are not allowed).
- It is important to recall that these displacements are assumed to be in the Global Coordinate System and that the interconsistency of the output coordinate systems of the various GRIDs, FEEDGES, FEFACEs is not checked.
- If an entity is specified on both a GMBC and GMSPC entry then the GMSPC specification will be ignored.
- Coordinate transformations should not be applied to grid points on FEEDGE, GMCURVE, FEFACE or GMSURF.

11. For the example in [Figure 9-96](#),

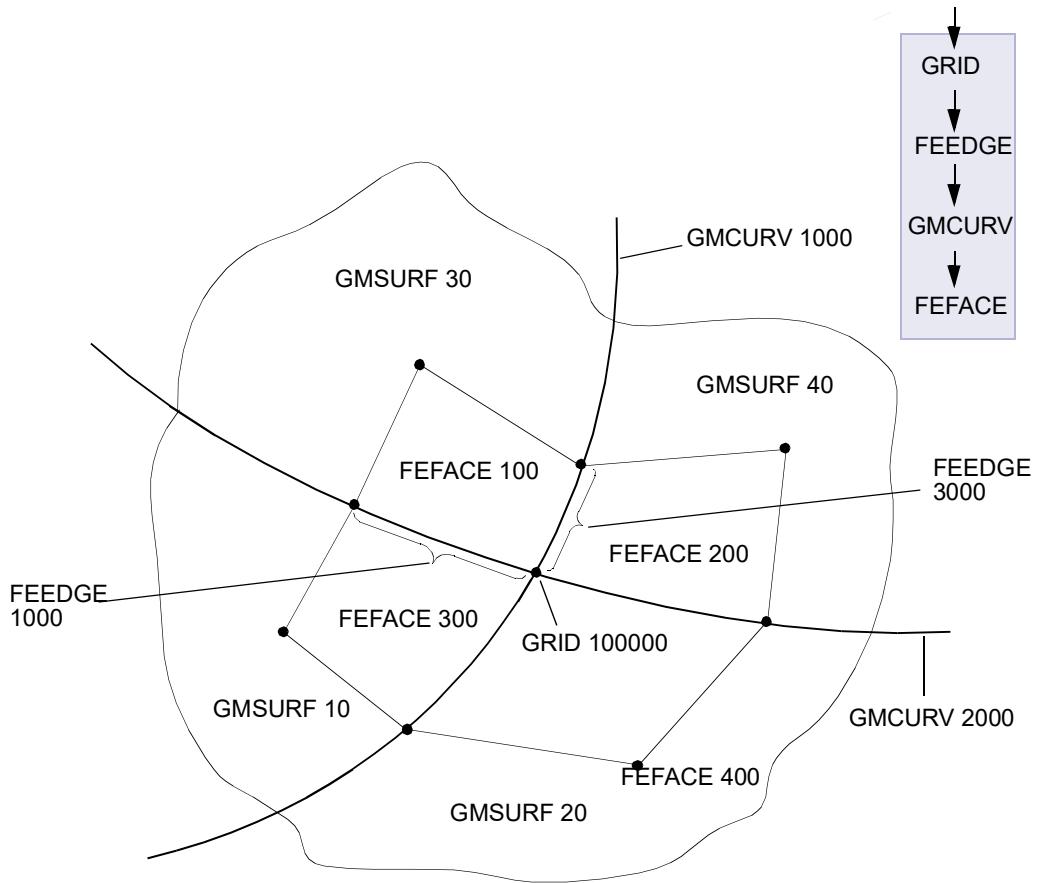


Figure 9-96 Use of Multiple Surface and Curves

- The enforced displacement for GRID 100000 can be specified using SPCD, GMBC referring to an FEEDGE, GMBC referring to a GMCURV, GMBC referring to a FEFACE, and GMBC referring to a GMSURF. [Table 18](#) describes the outcome of using these different methods:

Table 18 Enforced Displacement Used for GRID 10000

When Specified Using	Action
SPCD	Overrides all other information supplied for all components.
Single GMBC (FEEDGE)	Overrides information supplied for all components using GMBC(GMCURV) GMBC(FEFACE), GMBC(GMSURF) entries.
Multiple GMBC (FEEDGE)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMCURV) GMBC(FEFACE), GMBC(GMSURF) entries.
Single GMBC (GMCURV)	Overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.
Multiple GMBC (GMCURV)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.
Single GMBC (FEFACE)	Overrides information supplied for all components using GMBC(GMSURF) entries.
Multiple GMBC (FEFACE)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMSURF) entries.
Single GMBC (GMSURF)	Values are applied.
Multiple GMBC (GMSURF)	Values are averaged on a component basis.

- The enforced displacement for GRID DOFs and edge DOFs belonging to FEEDGE 10000 can be specified using GMBC referring to an FEEDGE, GMBC referring to a GMCURV, GMBC referring to a FEFACE, and GMBC referring to a GMSURF. [Table 19](#) describes the outcome of using these different methods:

Table 19 Enforced Displacement Used for FEEDGE 10000

When Specified Using	Action
Single GMBC (FEEDGE)	Overrides information supplied for all components using GMBC(GMCURV), GMBC(FEFACE), GMBC(GMSURF) entries.
Multiple GMBC (FEEDGE)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMCURV), GMBC(FEFACE), GMBC(GMSURF) entries.
Single GMBC (GMCURV)	Overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.
Multiple GMBC (GMCURV)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.
Single GMBC (FEFACE)	Overrides information supplied for all components using GMBC(GMSURF) entries.
Multiple GMBC (FEFACE)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMSURF) entries.
Single GMBC (GMSURF)	Values are applied.
Multiple GMBC (GMSURF)	Values are averaged on a component basis.

- The enforced displacement for the GRID DOFs and edge DOFs belonging to GMCURV 1000 can be specified using GMBC referring to a GMCURV, GMBC referring to a FEFACE, and GMBC referring to a GMSURF. [Table 20](#) describes the outcome of using these different methods:

Table 20 Enforced Displacement Used for GMCURV 10000

When Specified Using	Action
Single GMBC (GMCURV)	Overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.
Single GMBC (FEFACE)	Overrides information supplied for all components using GMBC(GMSURF) entries.
Multiple GMBC (FEFACE)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMSURF) entries.
Single GMBC (GMSURF)	Values are applied.
Multiple GMBC (GMSURF)	Values are averaged on a component basis.

- The enforced displacement for the GRID DOFs, the edge DOFs, and the face DOFs belonging to FEFACE 300 can be specified using GMBC referring to a FEFACE and GMBC referring to a GMSURF. [Table 21](#) describes the outcome of using these different methods:

Table 21 Enforced Displacement Used for FEFACE 300

When Specified Using	Action
Single GMBC (FEFACE)	Overrides information supplied for all components using GMBC(GMSURF) entries.
Single GMBC (GMSURF)	Values are applied.

GMBNDC**Geometric Boundary - Curve**

Defines a geometric boundary consisting of either h- or p-element edges along a curve interface. The boundary may consist of edges of shell, beam, or solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
GMBNDC	BID	GRIDI	GRIDF						
	ENTITY	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
		ID8	-etc.-						

Examples:

GMBNDC	1	101	106						
	GMCURV	1							

GMBNDC	1	101	106						
	FEEDGE	11	12	13	14	15			

GMBNDC	1	101	106						
	GRID	102	103	104	105				

Descriptor	Meaning
BID	Boundary identification number to be referenced by a GMINTC entry. (Integer > 0)
GRIDI	Initial grid identification number for boundary. (Integer > 0)
GRIDF	Final grid identification number for boundary. (Integer > 0)
ENTITY	Entity type for defining boundary. (Character)
IDi	Entity identification numbers for boundary of subdomain. Values in the list must be unique. (Integer > 0)

Remarks:

1. All boundary identification numbers must be unique.
2. GRIDI and GRIDF define the end points of the boundary.
3. For each boundary, one of the entity types GMCURV, FEEDGE, or GRID is required.
4. For the GMCURV entity type, if there are multiple paths on the GMCURV from the GRIDI to the GRIDF, such as two segments of a circle, the FEEDGE or GRID method must be used instead to uniquely define the path.
5. For the GRID entity type, the entities should be listed in order from the GRIDI to the GRIDF. The GRIDI and GRIDF need not be repeated in the IDi list.

6. If more than one boundary references the same GMCURV entry with the same GRIDI and GRIDF, then the FEEDGE or GRID entity type must be used instead for each to uniquely identify the boundaries.
7. Multiple continuation entries may be specified for additional entity identification numbers, IDi.

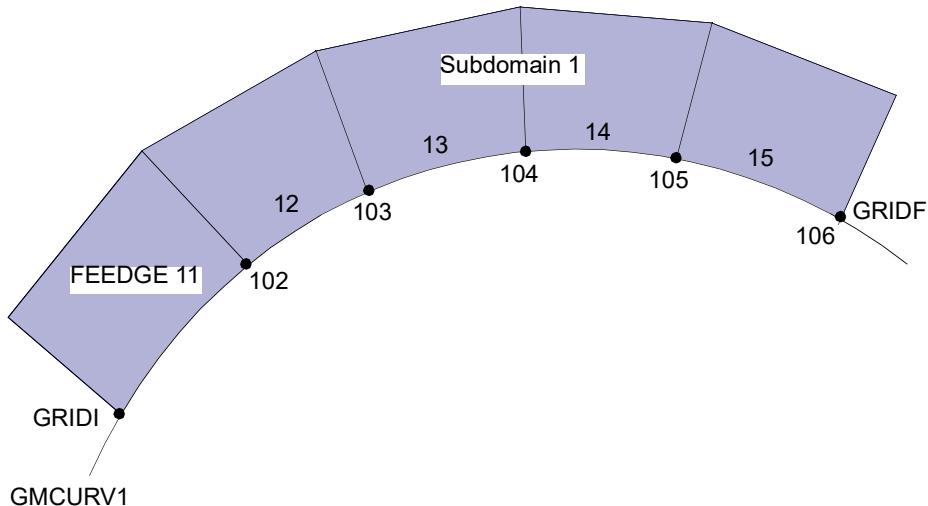


Figure 9-97 Geometric Boundary Definition

8. Interface elements may generate high or negative matrix/factor diagonal ratios. If there are no other modelling errors, these messages may be ignored and PARAM,BAILOUT,-1 may be used to continue the run.
9. When it is referenced by Bulk Data entry, CINTC, which defines an h-interface element, the value of field ENTITY must be GRID.

GMBNDS**Geometric Boundary - Surface**

Defines a geometric boundary consisting of p-element faces along a surface interface. The boundary may consist of faces of p-solid or p-shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
GMBNDS	BID								
	ENTITY	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
		ID8	-etc.-						

Examples:

GMBNDS	1								
	GMSURF	1							

GMBNDS	1								
	FEFACE	11	12	13	14	15	16		

GMBNDS	1								
	GRID	101	102	103	104	105	106	107	
		108	109	110	111	112			

Descriptor **Meaning**

BID	Boundary identification number.	Integer > 0	Required
ENTITY	Entity type for defining boundary.	Character	Required
IDi	Entity ID i for boundary	Integer > 0	Optional

Remarks:

1. All BIDs must be unique.
2. For each boundary, one of the entity types GMSURF, FEFACE, or GRID is required.
3. For the GMSURF entity type, all the faces referencing the GMSURF will be included in the boundary.
4. If more than one boundary references the same GMSURF, then the FEFACE or GRID entity type must be used instead for each to uniquely identify the boundaries.
5. Multiple continuation entries may be used without repeating the ENTITY field.

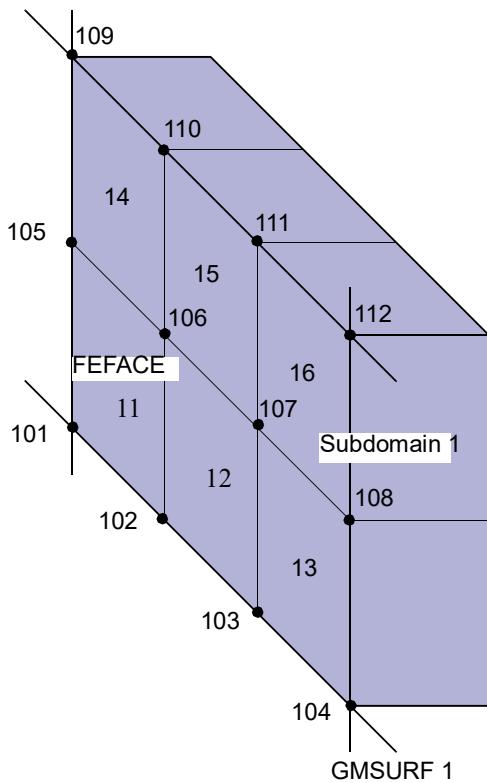


Figure 9-98 Surface Boundary Definition

6. Interface elements may generate high or negative matrix/factor diagonal ratios. If there are no other modelling errors, these messages may be ignored and PARAM,BAILOUT,-1 may be used to continue the run.

GMCONV

Define Convection Boundary Conditions

Defines free convection boundary conditions for GRID, FEEDGE, GMCURV, FEFACE, and GMSURF entries.

Format:

1	2	3	4	5	6	7	8	9	10
GMCONV	LID	ENTITY	ID	METHOD	FIELD1	FIELD2			

Example:

GMCONV	15	FEFACE	3	10	1001	20.0			
--------	----	--------	---	----	------	------	--	--	--

Descriptor	Meaning			
LID	Load set identification number.	Integer > 0	Required	
ENTITY	Entity that the convection boundary condition is applied to (specify GRID, FEEDGE, GMCURV, FEFACE, or GMSURF). See Remarks.	Character	Required	
ID	ID of the entity selected above. See Remarks.	Character	Required	
METHOD	Method used to specify the data. See Remark 2.	Integer ≥ 0	0	
FIELD1	Convection heat transfer coefficient data. See Remark 2.	Integer or Real	Required	
FIELD2	Ambient temperature data. See Remark 2.	Integer or Real	Required	

Remarks:

1. For steady-state analysis, the load set is selected in the Case Control Section (LOAD=LID).
2. METHOD specifies the data types of FIELD1 and FIELD2 to be constants, equation IDs, or table IDs. Values in FIELD1 and FIELD2 are:

METHOD	FIELD1	FIELD2
0	Value of heat transfer coefficient (Real > 0.0).	Value of ambient temperature (Real).
1	Value of heat transfer coefficient (Real > 0.0).	ID of a DEQATN entry defining the ambient temperature as a function of location (Integer > 0).
2	Value of heat transfer coefficient (Real > 0.0).	ID of a TABLE3D entry defining the ambient temperature as a function of location (Integer > 0).
10	ID of a DEQATN entry defining the heat transfer coefficient as a function of location (Integer > 0).	Value of ambient temperature (Real).
11	ID of a DEQATN entry defining the heat transfer coefficient as a function of location (Integer > 0).	ID of a DEQATN entry defining the ambient temperature as a function of location (Integer > 0).
12	ID of a DEQATN entry defining the heat transfer coefficient as a function of location (Integer > 0).	ID of a TABLE3D entry defining the ambient temperature as a function of location (Integer > 0).
20	ID of a TABLE3D entry defining the heat transfer coefficient a location (Integer > 0).	Value of ambient temperature (Real).
21	ID of a TABLED3 entry defining the heat transfer coefficient as a function of location (Integer > 0).	ID of a DEQATN entry defining the ambient temperature as a function of location (Integer > 0).
22	ID of a TABLE3D entry defining the heat transfer coefficient as a function of location (Integer > 0).	ID of a TABLE3D entry defining the ambient temperature as a function of location (Integer > 0).

3. The proper units must be specified for the value of FIELD1.

Units of FIELDi for Different ENTITY Fields	
ENTITY	Units
GRID	Power/Degree
FEEDGE	Power/Length-Degree
GMCURV	Power/Length-Degree
FEFACE	Power/Area-Degree
GMSURF	Power/Area-Degree

4. Multiple values of convection boundary conditions can be applied to the same geometry region. In general, a hierarchy is set to resolve the conflicts arising in the input data:
 - a. Information provided on multiple GMSURF and FEFACE entries are added for all FEFACE entries.
 - b. Information provided on multiple GMCURV and FEEDGE entries are added for all FEEDGE entries.

GMCORD

Convective/Follower Coordinate System Definition

Defines a convective/follower coordinate system on an FEEDGE, GMCURV, FEFACE, or GMSURF entry.

Format:

1	2	3	4	5	6	7	8	9	10
GMCORD	CID	ENTITY	ID1	ID2					

Example:

GMCORD	101	GMCURV	26	44					
--------	-----	--------	----	----	--	--	--	--	--

Descriptor	Meaning			
CID	Coordinate system identification number, unique with respect to all CORDij entries.	Integer > 0	Required	
ENTITY	Type of Bulk Data entry that is used to define the coordinate system. See Remark 3.	Character	Required	
ID1, ID2	Entity identification numbers. See Remark 3.	Integer > 0	Required	

Remarks:

1. GMCORD defines a (convective) coordinate system associated with an entity. This type of coordinate system can be used to apply loads and boundary conditions only.
2. GMCORD can only be specified for p-version elements.
3. The Bulk Data entries referenced by ID1 and ID2 depends on ENTITY.

ENTITY	ID1	ID2
FEEDGE	FEEDGE entry ID	FEFACE entry ID
GMCURV	GMCURV entry ID	GMSURF entry ID
FEFACE	FEFACE entry ID	Blank
GMSURF	GMSURF entry ID	Blank

- For ENTITY = “FEEDGE” normal is defined by the FEFACE.
- For ENTITY = “GMCURV” normal is defined by the GMSURF.

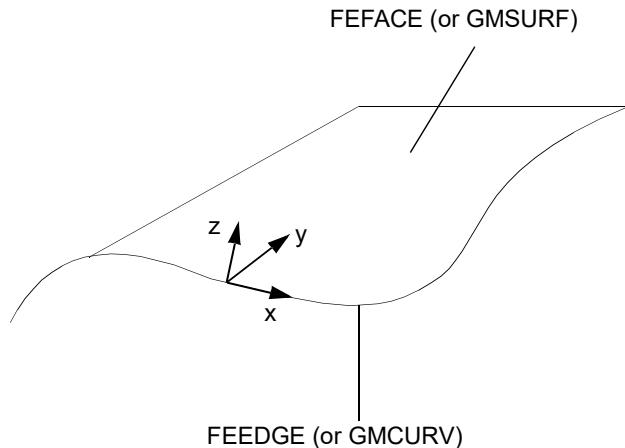


Figure 9-99

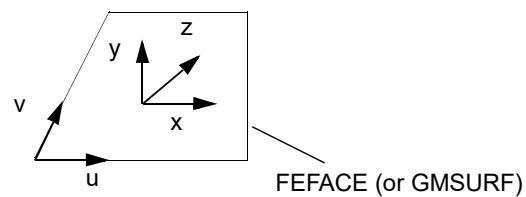


Figure 9-100

GMCURV**Curve Definition**

Defines geometric curve that will be used in element geometry, load definition, and boundary condition definition.

Format:

1	2	3	4	5	6	7	8	9	10
GMCURV	CURVID	GROUP	CIDIN	CIDBC					
	Evaluator	Specific	Data	and	Format				

Example:

GMCURV	101	FENDER							
	RPC	POINT							
	0.0, 2.0	1.0	1.0		2.0	3.0	4.0,1.0		
	0.0, 2.0	1.0	1.0	1.0	2.0	3.0	4.0		

Descriptor	Meaning			
CURVID	Unique identification number. See Remarks 1. and 2.	Integer > 0		Required
GROUP	Group of curves/surfaces that this curve belongs to. See Remarks 4. through 8.	Character		Required
CIDIN	Coordinate system identification number used in defining the geometry of the curve. The coordinate system must be rectangular.	Integer ≥ 0		0
CIDBC	Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined.	Integer ≥ 0		0

Remarks:

1. GMCURV is used to calculate geometric information only. The edges of the finite elements that are connected to the curve will be parametric cubic curves that are calculated from the more complex curve geometry.
2. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
3. The continuation entries are passed directly to the geometry evaluator indicated by the GROUP parameter.
4. The GROUP parameter is initialized by an CONNECT GEOMEVAL statement in the FMS Section. This command specifies the evaluator that will be used for this curve.
5. Two reserved names, MSCGRP0 and MSCGRP1, are provided for the GROUP parameter. These need not be explicitly initialized on the CONNECT FMS statement.

6. If the GROUP parameter is specified as MSCGRP0, the MSC rational parametric cubic (MSCRPC) geometry evaluator is used for this curve. In this case the evaluator specific data in lines 2 through 4 of this Bulk Data entry should be provided as given below. Spaces or a comma character may be used to delimit each value. However, a comma must not be specified in the first field.

1	2	3	4	5	6	7	8	9	10
	RPC	REPRES							
	XW(1)	XW(2)	XW(3)	XW(4)	YW(1)	YW(2)	YW(3)	YW(4)	
	ZW(1)	ZW(2)	ZW(3)	ZW(4)	W(1)	W(2)	W(3)	W(4)	

Descriptor	Meaning	Character	Required
RPC	Rational Parametric Cubic Curve.	Character	Required
REPRES	Representation of the curve (“ALGEBRAIC”, “POINT”, “BEZIER”).	Character	Required
XW(1) through W(4)	Data used to define the curve.	Real	Required

- A rational parametric curve (RPC) is defined as

$$x(t) = \frac{xw(t)}{w(t)}$$

$$y(t) = \frac{yw(t)}{w(t)}$$

$$z(t) = \frac{zw(t)}{w(t)}$$

$$0.0 \leq t \leq 1.0$$

- For REPRES = “ALGEBRAIC”, the parametric curve is defined by the algebraic coefficients (a, b, c, d) for a rational cubic equation.

$$P(t) = at^3 + bt^2 + ct + d$$

Expressed in matrix form:

$$P(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} a \\ b \\ c \\ d \end{bmatrix}$$

for the Bulk Data input as algebraic coefficients

$$xw(t) = XW(1)t^3 + XW(2)t^2 + XW(3)t + XW(4)$$

$$yw(t) = YW(1)t^3 + YW(2)t^2 + YW(3)t + YW(4)$$

$$zw(t) = ZW(1)t^3 + ZW(2)t^2 + ZW(3)t + ZW(4)$$

$$w(t) = W(1)t^3 + W(2)t^2 + W(3)t + W(4)$$

where $XW(i)$, $YW(i)$, $ZW(i)$, and $W(i)$ are the algebraic coefficients for each of the independent equations $xw(t)$, $yw(t)$, $zw(t)$, and $w(t)$.

and

$$x(t) = \frac{xw(t)}{w(t)}; y(t) = \frac{yw(t)}{w(t)}; z(t) = \frac{zw(t)}{w(t)}$$

- For REPRES = “BEZIER”, the curve parametric is defined by four rational Bezier control points (V_1 , V_2 , V_3 , and V_4) expressed in matrix form

$$P(t) = V_1(1-t)^3 + V_23t(1-t)^2 + V_33t^2(1-t) + V_4t^3$$

$$P(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix}$$

where Bezier constants are

$$\begin{bmatrix} -1 & 3 & -3 & 1 \\ 3 & -6 & 3 & 0 \\ -3 & 3 & 0 & 0 \\ 1 & 0 & V_0 & 0 \end{bmatrix}$$

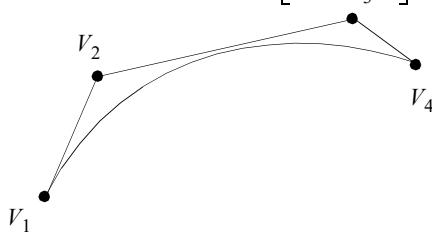


Figure 9-101 REPRES = “BEZIER”

for Bulk Data defined as Bezier control points

$$xw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} XW(1) \\ XW(2) \\ XW(3) \\ XW(4) \end{bmatrix}$$

$$yw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} YW(1) \\ YW(2) \\ YW(3) \\ YW(4) \end{bmatrix}$$

$$zw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} ZW(1) \\ ZW(2) \\ ZW(3) \\ ZW(4) \end{bmatrix}$$

$$w(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} W(1) \\ W(2) \\ W(3) \\ W(4) \end{bmatrix}$$

where $XW(i)$, $YW(i)$, $ZW(i)$, and $W(i)$ correspond to V_p
and

$$x(t) = \frac{xw(t)}{w(t)}; y(t) = \frac{yw(t)}{w(t)}; z(t) = \frac{zw(t)}{w(t)}$$

- For REPRES = “POINT”, the parametric curve is defined by four uniformly spaced rational points that are all on the curve similarly expressed in matrix form:

$$P(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix}$$

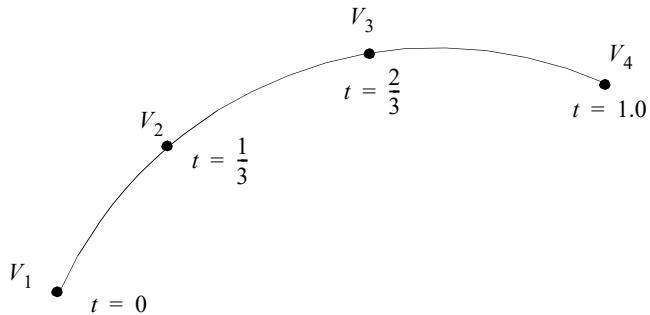


Figure 9-102 RÉPRES = “POINT”
22EQ222

for the Bulk Data input are uniformly spaced rational points

$$xw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} XW(1) \\ XW(2) \\ XW(3) \\ XW(4) \end{bmatrix}$$

$$yw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} YW(1) \\ YW(2) \\ YW(3) \\ YW(4) \end{bmatrix}$$

$$zw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} ZW(1) \\ ZW(2) \\ ZW(3) \\ ZW(4) \end{bmatrix}$$

$$w(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} W(1) \\ W(2) \\ W(3) \\ W(4) \end{bmatrix}$$

and
 $x(t) = \frac{xw(t)}{w(t)}$; $y(t) = \frac{yw(t)}{w(t)}$; $z(t) = \frac{zw(t)}{w(t)}$

7. If the GROUP parameter is specified as MSCGRP1, the MSC generic equation (MSCEQN) geometry evaluator is used for this curve. In this case the evaluator specific data in lines 2 through 3 of this Bulk Data entry should be provided as given below. Spaces or a comma character may be used to delimit each value. However, the comma character should not be used in the first “field”.

1	2	3	4	5	6	7	8	9	10
	EQUATION, MINU	MAXU	IDX	IDY	IDZ	IDDXU	IDDYU		
	IDDZU	IDDXU2	IDDYU2	IDDZU2					

Descriptor	Meaning			
EQUATION	EQUATION method is to be used	Character		
MINU, MAXU	Range of the curve parameter u . If MAXU is found less than MINU, the range is assumed to be $[-\infty, +\infty]$.	Real	0.0,1.0	
IDX, IDY, IDZ	ID of DEQATN entries providing equations for the X,Y,Z coordinates of the curve in terms of the curve parameter u .	Integer > 0	Required	
IDDXU, IDDYU, IDDZU	ID of DEQATN entries providing equations for the first derivatives of X,Y,Z functions with respect to the curve parameter u . If a value of 0 is used, the derivatives are computed numerically.	Integer ≥ 0	0	
IDDXU2, IDDYU2, IDDZU2	ID of DEQATN entries providing equations for the second derivatives of X,Y,Z functions with respect to the curve parameter u . If a value of 0 is used, the second derivatives are computed numerically.	Integer > 0	0	

8. When a user-supplied geometry evaluator is selected for this curve (through the CONNECT GEOMEVAL FMS command) the continuation entries will not be interpreted. In this case an image of this entry is passed on to the evaluator modules provided by the user. Depending on the configuration, these modules could either be linked in with Nastran or connected with Nastran during execution. If these modules are not accessible, a User Fatal Message will be issued. For example, if in the FMS Section, the following command is given:

- CONNECT GEOMEVAL FENDER,CATIA,’/u/kiz/data’, Version = 68 as of 1/3/94
- and the GMCURV Bulk Data entry is provided as follows:

1	2	3	4	5	6	7	8	9	10
GMCURV	102	FENDER							
	Sweep	/u/kiz	2.5	arc	2.7	66			

- In this case, “Sweep /u/kiz 2.5 arc 2.7 66” is passed to the geometry evaluator supplied by the user, and it is expected that the user supplied routines to interpret and use this record.

GMINTC**Geometric Interface -- Curve**

Defines an interface element along a curve interface between boundaries of multiple subdomains. Typically, the boundaries will consist of edges of p-shell subdomains but also may consist of p-beam subdomains or edges of p-solid subdomains.

Format:

1	2	3	4	5	6	7	8	9	10
GMINTC	EID	PID	ID1	ID2	ID3	ID4	ID5	ID6	

Example:

GMINTC	1001	1	1	2					
--------	------	---	---	---	--	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PINTC property entry. (Integer > 0)
IDi	Boundary identification number of a GMBNDC entry. (Integer > 0)

Remarks:

1. All element identification numbers must be unique.
2. For the curve interface it is recommended that only two boundaries be specified.
3. All of the end points for each boundary IDi should be coincident, and may not refer to the same grid point. The two end points of a particular boundary may not refer to the same grid, because there would be multiple directions. The boundaries of each of the subdomains should also be coincident, because no geometrical adjustment is performed.
4. Connecting curve boundaries of solid p-elements is not recommended because of the possibility of stress singularities.

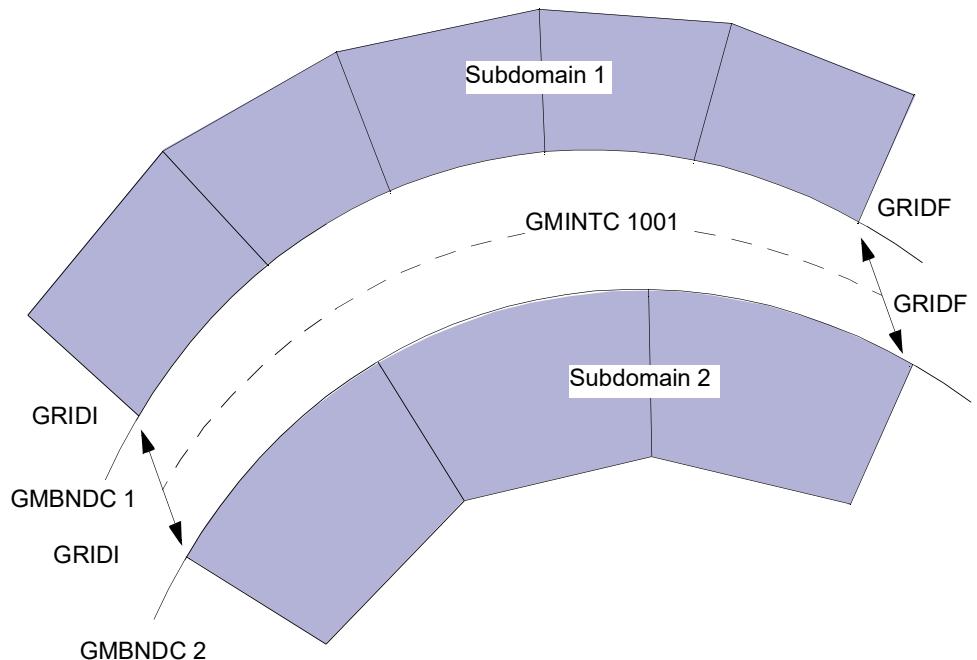


Figure 9-103 Geometric Interface Element Definition (Exploded View)

5. Because of the structure of the interface matrices, the sparse solver (default) should be used for linear statics, and the Lanczos eigensolver should be used for normal modes. In addition, for normal modes, SYSTEM(166) = 4 should be set for models where the shell normal rotations are parallel on the boundaries.

GMINTS**Geometric Interface -- Surface**

Defines an interface element along a surface interface between boundaries of multiple subdomains. Typically, the boundaries will consist of faces of p-solid subdomains, but also may consist of p-shell subdomains.

Format:

1	2	3	4	5	6	7	8	9	10
GMINTS	EID	PID	ID1	ID2	ID3	ID4			

Example:

GMINTS	1001	1	1	2					
--------	------	---	---	---	--	--	--	--	--

Descriptor	Meaning
------------	---------

EID	Element identification number.	Integer > 0	Required
PID	Property identification number.	Integer > 0	Required
IDi	Boundary IDi of subdomain	Integer > 0	Required

Remarks:

1. All EIDs must be unique.
2. The PID refers to a PINTS Bulk Data entry.
3. The boundary IDi of each subdomain must be defined on a GMBNDS Bulk Data entry.
4. For the surface interface, more than two boundaries are possible, but should be used carefully.
5. The perimeters of each boundary i should be coincident. In addition, the boundaries of each of the subdomains should also be coincident, because no geometrical adjustment is performed.

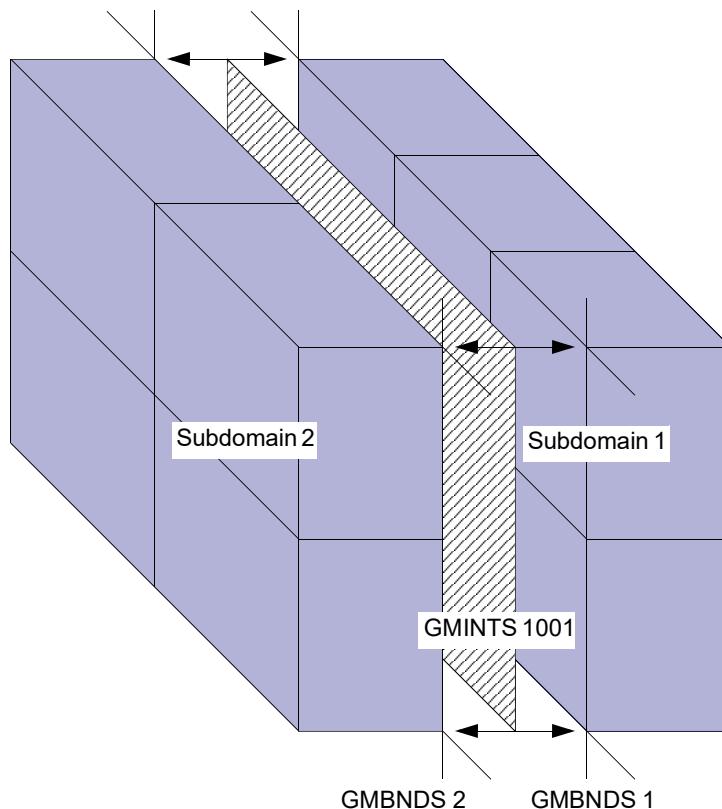


Figure 9-104 Geometric Interface Element Definition (Exploded View)

GMLOAD**General Load Definition**

Define the forces and moments to be applied to a FEEDGE, GMCURV, FEFACE, or GMSURF entry.

Format:

1	2	3	4	5	6	7	8	9	10
GMLOAD	LID	CID	N1	N2	N3	ENTITY	ID	METHOD	
	FIELD1	FIELD2	FIELD3	FIELD4	-etc.-				

Example:

GMLOAD	105	11	1.0	0.	0.	FEFACE	41	LINEAR	
	10.	-10.	10.						

Descriptor	Meaning			
LID	Load set identification number.	Integer > 0	Required	
CID	Coordinate system in which the load is supplied. See Remark 2.	Integer ≥ -1	0	
Ni	Direction of the force vector or axis of rotation of the moment. See Remark 3.	Real	0., 0., 1.	
ENTITY	Entity that is being loaded (FEEDGE, GMCURV, FEFACE, GMSURF).	Character	Required	
ID	ID of the entity selected by ENTITY.	Integer ≥ 0	Required	
METHOD	Method used to specify forces (EQUATION, TABLE, CONSTANT, LINEAR, QUAD, CUBIC) or moments (MEQUA, MTABLE, MCONST, MLINEAR, MQUAD, MCUBIC). See Remarks 4. through 6.	Character	Required	
FIELDi	Load magnitude data. See Remarks 4. through 8.	Real or Integer		

Remarks:

1. GMLOAD is the only method of applying forces and moments to any FEEDGE, FEFACE, GMCURV, or GMSURF in the model.
2. If CID=-1, the coordinate system on the edge or face is a local system based on the FEEDGE or FEFACE definition. (Note that an edge only has the tangent direction uniquely defined.)
3. If N1=N2=N3=0., the normal direction to the face is assumed, with the positive sense dependent on the FEFACE definition. No load will be applied for edges.

4. For both an FEEDGE and FEFACE, the METHOD field can be used to specify equation, table or constant load density. The value of FIELD1 is method-specific:

Applying Equation, Table or Constant Load Density	
METHOD	FIELD1
EQUATION, MQUA	ID of a DEQATN entry defining the load density as a function of location.
TABLE, MTABLE	ID of a TABLE3D entry defining the load density as a function of location.
CONSTANT, MCONST	Value of load density.

5. For an FEEDGE, the METHOD field can be used to specify linear, quadratic or cubic load density. The values of FIELDi are method-specific:

Applying Linear, Quadratic or Cubic Load Density to an FEEDGE					
METHOD	FIELD1	FIELD2	FIELD3	FIELD4	Load Density
LINEAR, MLINEAR	Value at GRID 1	Value at GRID 2	blank	blank	Linear
QUAD, MQUAD	Value at GRID 1	Value at GRID 2	Value at 1/2 edge length	blank	Quadratic
CUBIC, MCUBIC	Value at GRID 1	Value at GRID 2	Value at 1/3 edge length	Value at 2/3 edge length	Cubic

6. For an FEFACE, the METHOD field can be used to specify linear or quadratic load density. The edges of the face are defined in the order of the grids entered (e.g., edge 1 is between the first and second grid etc.). The values of FIELDi are method-specific:

Applying Linear and Quadratic Load Density to a Quadrilateral FEFACE				
METHOD	FIELD1 through FIELD4	FIELD5 through FIELD8	FIELD9	Load Density
LINEAR,MLINEAR	Value at GRID 1, 2, 3, 4	blank	blank	Linear
QUAD, MQUAD	Value at GRID 1, 2, 3, 4	Value at midside of EDGE 1,2,3,4	Value at middle of FEFACE	Quadratic

Applying Linear and Quadratic Load Density to a Triangular FEFACE			
METHOD	FIELD1 through FIELD3	FIELD4 through FIELD6	Load Density
LINEAR, MLINEAR	Value at GRID 1, 2, 3	blank	Linear
QUAD, MQUAD	Value at GRID 1, 2, 3	Value at midside of EDGE 1, 2, 3	Quadratic

7. The proper units must be specified for the value of FIELD*i*.

Units of FIELD <i>i</i> for Different ENTITY Fields	
ENTITY	Units
FEEDGE	Load/Length
GMCURV	Load/Length
FEFACE	Load/Area
GMSURF	Load/Area

8. The load density applied to the edge or face is given by the product of the specified density with the direction vector.
9. The shell p-elements do not have stiffness in the direction of the normal rotation. Any component of moment applied in that direction will be ignored.
10. In general, a hierarchy is set to resolve the conflicts arising in the input data:
- Information provided on multiple GMSURF and FEFACE entries are added for all GRID, FEEDGE, and FEFACE degrees-of-freedom.
 - Information provided on multiple GMCURV and FEEDGE entries are added for all GRID and FEEDGE degrees-of-freedom.
 - Loads are summed over all entities.

GMNURB**3D Contact Region Made Up of NURBS in SOL 600**

Defines a 3D contact region made up of NURBS using the MSC Marc style used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
GMNURB	ID	NPTU	NPTV	NORU	NORV	NSUBU	NSUBV	NTRIM	
		G1 or X1	G2 or Y1	G3 or Z1	G4 or X2	G5 or Y2	G6 or Z2	G7	
		G8 or X3	G9 or Y3	G10 or Z3	etc.		[abs(nptu)*n ptv values]	See Remark 3	
		Homo1	Homo2	Homo3	Homo4	Homo5	Homo6	Homo7	
		Homo8	Homo9	Homo10	Homo11	etc	[nptu*nptv values]		
		Knot1	Knot2	Knot3	Knot4	Knot5	Knot6	Knot7	
		Knot8	Knot9	Knot10	etc.		[(nptu+noru) +(nptv+norv) values]		
		IDtrim	NPTUtrim	NORUtrim	NSUBtrim		(repeat this and all following lines NTRIM times)		
			Xisoparam	Yisoparam			(NPTUtrim entries)		
			Homo1	Homo2	Homo3	etc	(NPTUtrim entries)		
			Knot1	Knot2	Knot3	etc	(NPTUtrim + NORUtrim entries)		

Example:

GMNURB	901	9	5	4	4	20	5		
		101	102	103	104	105	106	107	
		108	109	110	111	112	113	114	
		115	116	117	118	119	120	121	
		122	123	124	125	126	127	128	
		129	130	131	132	133	134	135	
		136	137	138	139	140	141	142	
		143	144	145					
		1	1	1	1	1	1	1	
		1	1	1	1	1	1	1	
		1	1	1	1	1	1	1	
		1	1	1	1	1	1	1	
		1	1	1	1	1	1	1	
		1	1	1					
		0	0	0	0	0.16667	0.33333	0.5	
		0.66667	0.833333	1	1	1	1	0	
		0	0	0	0.5	1	1	1	
		1							

Descriptor	Meaning
ID	Identification number of a surface defined by NURBS. ID is called out on a BCBODY entry with a NURBS2 header. (Integer > 0; Required)
NPTU	Absolute value of the number of control points. Enter NPTU as a positive number if the control points are to be input using GRID points. Enter NPTU as a negative number if the control points are to be entered using x,y,z. (Integer > 0; Required)
NPTV	Number of control points in V direction. (Integer > 0; Required)
NORU	Order along U direction. (Integer > 0; Required)
NORV	Order along V direction. (Integer > 0; Required)
NSUBU	Number of subdivisions in U direction. (Integer > 0; Required)
NSUBV	Number of subdivisions in V direction. (Integer > 0; Required)
NTRIM	Number of trimming curves. (Integer ≥ 0 or blank)
G1, G2, G3, etc.	Grid point IDs defining control points (Integer > 0; Required). There must be NPTU*NPTV entries.
X1, Y1, Z1, X2, Y2, Z2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.

Descriptor	Meaning
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) in the V direction. There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)
IDtrim	ID of trimming vector. There must NTRIM of these entries and those entries that follow. (Integer > 0)
NPUTtrim	Number of control points for this trimming vector. (Integer > 0)
NORUtrim	Order for this trimming vector. (Integer > 0)
NSUBtrim	Number of subdivisions for this trimming vector. (Integer > 0)
Xisoparam	First coordinate of point in isoparametric space. (Real)
Ysoparam	Second coordinate of point in isoparametric space (Real)
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)

Remarks:

1. GMNURB is recognized only in SOL 600.
2. WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e. in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.
3. For NURBS, enter either IDs of NPTU*NPTV grid points as G1, G2, G3 etc. (set positive NPTU) or coordinates of abs(NPTU)*NPTV grid point as (X1, Y1, Z1), (X2, Y2, Z2), etc. (set negative NPTU).

GMQVOL**Define Volumetric Heat Loads**

Defines a rate of volumetric heat generation in a conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
GMQVOL	LID	METHOD	FIELD1		EID1	EID2	EID3	EID4	
	EID5	etc.							

Example:

GMQVOL	100	TABLE	20		18	23	7		
--------	-----	-------	----	--	----	----	---	--	--

Descriptor	Meaning
------------	---------

LID	Load set identification number.	Integer > 0	Required
METHOD	Method used to specify the data (EQUATION, TABLE, CONSTANT). See Remark 2.	Character	Required
FIELD1	Volumetric heat load data. See Remark 2.	Integer or Real	Required
EIDi	A list of heat conduction elements.	Integer > 0	Required

Remarks:

- For steady-state analysis, the load set is selected in the Case Control Section (LOAD=LID).
- METHOD specifies the data type of FIELD1 to be constants, equation IDs, or table IDs. Values in FIELD1 and FIELD2 are:

METHOD	FIELD1
EQUATION	ID of a DEQATN entry defining the volumetric heat generation rate as a function of location (Integer > 0).
TABLE	ID of a TABLE3D entry defining the volumetric heat generation rate as a function of location (Integer > 0).
CONSTANT	Value of volumetric heat generation rate (Real).

Note that the fifth field will be reserved for the future development of temperature dependent functions.

GMSPC

General Constraint Definition

Defines constraints for entities.

Format:

1	2	3	4	5	6	7	8	9	10
GMSPC	SID	C	ENTITY	ID					

Example:

GMSPC	12	1	FEEDGE	109					
-------	----	---	--------	-----	--	--	--	--	--

Descriptor	Meaning		
SID	SPC set identification number.	Integer > 0	Required
C	Component number in the global coordinate system.	$0 \leq \text{Integer} \leq 6$	0
ENTITY	Entity that the enforced displacement is applied to (Specify GRID, FEEDGE, GMCURV, FEFACE, or GMSURF). See Remark 4.	Character	Required
ID	ID of the entity selected above.	Integer > 0	Required

Remarks:

1. The components of motion specified by C (field 3) of all degrees-of-freedom associated with an entity will be constrained.
2. If C = 0 is specified then the degrees-of-freedom will be taken out of the constraint set.
3. The component C has to be a single integer (1 or 2 or 3, etc.). Use multiple GMSPC entries for constraining multiple components.
4. In general, the hierarchy set to resolve the conflicts arising in the enforced displacement input data is the same as for the constraints. See Remark 10 under [GMBC, 2072](#) for a description of the hierarchy.
5. Coordinate transformations should not be applied to grid points on FEEDGE, GMCURVE, FEFACE or GMSURF.

GMSURF**Surface Definition**

Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.

Format:

1	2	3	4	5	6	7	8	9	10
GMSURF	SURFID	GROUP	CIDIN	CIDBC					
	Evaluator Specific Data and Format								

Example:

GMSURF	101	MSCGRP0							
	RPC,	POINT							
	0.0, 2.0	1.0	1.0		2.0	3.0	4.0,1.0		
	0.0, 2.0	1.0	1.0	1.0	2.0	3.0	4.0		

Descriptor	Meaning			
SURFID	Surface Identification number. See Remark 2.	Integer > 0	Required	
GROUP	Group of curves/surfaces that this surface belongs to. See Remarks 5. through 9.	Character	Required	
CIDIN	Coordinate system identification number used in defining the geometry of the curve. The coordinate system must be rectangular.	Integer ≥ 0	0	
CIDBC	Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined.	Integer ≥ 0	0	

Remarks:

1. All SURFIDs must be unique.
2. A GMSURF entry is required if:
 - the geometry of the surface defined by this entry is to be used by an element.
 - output (global) coordinate system is assigned to a GMSURF.
 - permanent constraints are specified for a GMSURF.
 - loads are applied to a GMSURF.
 - enforced boundary conditions are applied to a GMSURF.
3. GMSURF is used to calculate geometric information only. The edges of the finite elements that are connected to the surface will be parametric cubic curves that are calculated from the more complex surface geometry.

4. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
5. The continuation entries are passed directly to the geometry evaluator indicated by the GROUP parameter.
6. The GROUP value is initialized by an CONNECT GEOMEVAL command in the FMS section. This command specifies the evaluator that will be used for this surface.
7. Two reserved names, MSCGRP0 and MSCGRP1, are provided for the GROUP parameter. These need not be explicitly initialized in the FMS Section.
8. If the GROUP parameter is specified as MSCGRP0, the MSC rational parametric cubic (MSCRPC) geometry evaluator is used for this surface. In this case the evaluator specific data in lines 2 through 9 of this Bulk Data entry should be provided as given on the following page. Spaces or a comma character may be used to delimit each value. However, the comma character should not be used in the first field.

1	2	3	4	5	6	7	8	9	10
GMSURF	SURFID	MSCGRP0	CIDIN	CIDOUT					
	RPC	REPRES							
	XW(1)	XW(2)	XW(3)	XW(4)	XW(5)	XW(6)	XW(7)	XW(8)	
	XW(9)	XW(10)	XW(11)	XW(12)	XW(13)	XW(14)	XW(15)	XW(16)	
	YW(1)	YW(2)	YW(3)	YW(4)	YW(5)	YW(6)	YW(7)	YW(8)	
	YW(9)	YW(10)	YW(11)	YW(12)	YW(13)	YW(14)	YW(15)	YW(16)	
	ZW(1)	ZW(2)	ZW(3)	ZW(4)	ZW(5)	ZW(6)	ZW(7)	ZW(8)	
	ZW(9)	ZW(10)	ZW(11)	ZW(12)	ZW(13)	ZW(14)	ZW(15)	ZW(16)	
	W(1)	W(2)	W(3)	W(4)	W(5)	W(6)	W(7)	W(8)	
	W(9)	W(10)	W(11)	W(12)	W(13)	W(14)	W(15)	W(16)	

Descriptor	Meaning	Type	Default
RPC	Rational Parametric Cubic Surface.	Character	Required
REPRES	Representation of the curve, (ALGEBRAIC, POINT, BEZIER).	Character	Required
XW(1) through XW(16)	Data used to define the surface.	Real	Required

- A rational parametric surface is defined as

$$x(u, v) = \frac{xw(u, v)}{w(u, v)}$$

$$y(u, v) = \frac{yw(u, v)}{w(u, v)}$$

$$z(u, v) = \frac{zw(u, v)}{w(u, v)}$$

$$0.0 \leq u \leq 1.0$$

$$0.0 \leq v \leq 1.0$$

- For REPRES = “ALGEBRAIC”, the rational parametric surface is defined by the algebraic coefficient for rational cubic equations.
- Expressed as a tensor product

$$P_{(u, v)} = \begin{bmatrix} u^3 & u^2 & u & 1 \end{bmatrix} [\text{algebraic coefficients}] \begin{Bmatrix} v^3 \\ v^2 \\ v \\ 1 \end{Bmatrix}$$

for the Bulk Data input in algebraic form

$$xw(u, v) = \begin{bmatrix} u^3 & u^2 & u & 1 \end{bmatrix} \begin{bmatrix} XW(1) & XW(2) & XW(3) & XW(4) \\ XW(5) & XW(6) & XW(7) & XW(8) \\ XW(9) & XW(10) & XW(11) & XW(12) \\ XW(13) & XW(14) & XW(15) & XW(16) \end{bmatrix} \begin{Bmatrix} v^3 \\ v^2 \\ v \\ 1 \end{Bmatrix}$$

and similarly for $yw(u, v)$, $zw(u, v)$, and $w(u, v)$.

and

$$x(u, v) = \frac{xw(u, v)}{w(u, v)}; y(u, v) = \frac{yw(u, v)}{w(u, v)}; z(u, v) = \frac{zw(u, v)}{w(u, v)}$$

where $XW(i)$, $YW(i)$, $ZW(i)$, and $W(i)$ are the algebraic coefficients for the independent equations $xw(u, v)$, $yw(u, v)$, $zw(u, v)$, and $w(u, v)$.

- For REPRES = “BEZIER”, the surface is defined by 16 rational Bezier control points V_{11} through V_{44} .

and

$$P(u, v) = \sum_{i=1}^4 \sum_{j=1}^4 V_{ij} B_{i,4}u \cdot B_{j,4}v$$

where $B_{i,4}u$ and $B_{j,4}v$ are the Bernstein polynomials for curves of degree 3.

For Bulk Data input defined as Bezier control points

$$xw(u, v) = \sum_{i=1}^4 \sum_{j=1}^4 XWi,j B_{i,4}u \cdot B_{j,4}v$$

V_{11}	V_{12}	V_{13}	V_{14}
V_{21}	V_{22}	V_{23}	V_{24}
V_{31}	V_{32}	V_{33}	V_{34}
V_{41}	V_{42}	V_{43}	V_{44}

where $XW(1)$ through $XW(16)$ are ordered to conform to the two-dimensional array for V_{ij} ; that is, $XW(4 \cdot (i-1) + j)$ corresponds to $xw(u, v)$ for V_{ij} . For example, $XW(7)$ corresponds to V_{23} , $yw(u, v)$; $zw(u, v)$; and $w(u, v)$ are solved in a similar fashion.

and

$$x(u, v) = \frac{xw(u, v)}{w(u, v)}; y(u, v) = \frac{yw(u, v)}{w(u, v)}; z(u, v) = \frac{zw(u, v)}{w(u, v)}$$

- When the point representation is used, the surface is defined by 16 uniformly spaced rational points lying on the surface.

V_{11}	V_{12}	V_{13}	V_{14}
V_{21}	V_{22}	V_{23}	V_{24}
V_{31}	V_{32}	V_{33}	V_{34}
V_{41}	V_{42}	V_{43}	V_{44}

where $XW(1)$ through $XW(16)$ are ordered to contain to the two-dimensional mapping above for V_{ij} ; that is, $XW(4 \cdot (i - 1) + j)$ corresponds to $xw(u, v)$ for V_{ij} . For example, $XW(7)$ corresponds to V_{23} .

- If the GROUP parameter is specified as MSCGRP1, the MSC generic equation (MSCEQN) geometry evaluator is used for this surface. In this case the evaluator specific data should be on the continuation entries. Spaces or a comma character may be used to delimit each value. However, a comma must not be specified in the first field.

1	2	3	4	5	6	7	8	9	10
EQUATION, MINU		MAXU	MINV	MAXV	IDX	IDY	IDZ		
IDDXU	IDDYU	IDDZU	IDDXV	IDDYV	IDDZV	IDDXU2	IDDYU2		
IDDZU2	IDDXV2	IDDYV2	IDDZV2	IDDXUV	IDDYUV	IDDZUV			

Descriptor	Meaning	Type	Default
EQUATION	EQUATION method is to be used.	Character	
MINU, MAXU	Range of the first parameter describing the surface. If MAXU is found less than MINU, the range for U is assumed to be $[-\infty, +\infty]$.	Real	0.0,1.0
MINV, MAXV	Range of the second parameter describing the surface. If MAXV is found less than MINV, the range is assumed to be $[-\infty, +\infty]$.	Real	0.0,1.0
IDX, IDY, IDZ	ID of DEQATN entries providing equations for the X,Y,Z coordinate of the surface in terms of two parameters u and v .	Integer > 0	Required
IDDXU, IDDYU, IDDZU	ID of DEQATN entries providing equations for the first derivatives of X,Y,Z functions with respect to the first surface parameter u . If a value of 0 is used, the derivatives are computed numerically.	Integer ≥ 0	0
IDDXV, IDDYV, IDDZV	ID of an DEQATN entry describing the first derivatives of X, Y, Z functions with respect to the first surface parameter v . If a value of 0 is used, the derivatives are computed numerically.	Integer ≥ 0	0
IDDXU2, IDDYU2, IDDZU2	ID of an DEQATN entry describing the second derivatives of X,Y,Z functions with respect to the first surface parameter u . If a value of 0 is used, the derivatives are computed numerically.	Integer ≥ 0	0

Descriptor	Meaning	Type	Default
IDDXV2, IDDYV2, IDDZV2	ID of an DEQATN entry describing the second derivatives of X,Y,Z functions with respect to the second surface parameter v . If a value of 0 is used, the derivatives are computed numerically.	Integer ≥ 0	0
IDDXUV, IDDYUV, IDDZUV	ID of an DEQATN entry describing the mixed second derivatives of X,Y,Z functions with respect to the surface parameters u , and v . If a value of 0 is used, the derivatives are computed numerically.	Integer ≥ 0	0

10. When an external geometry evaluator class is selected for this group (which is the case when the CONNECT GEOMEVAL statement selects an external geometry evaluator for the specified group), the data in Fields 1 to n will not be interpreted. In this case an image of this entry is passed on to the evaluator modules provided by the user for the specific geometric package that being used. These modules are connected with Nastran during execution. If these modules are not provided, a User Fatal Message will be issued. For example, if in the FMS Section, the following command is given:

- CONNECT GEOMEVAL FENDER,CATIA,’/u/kiz/data’, Version=68 as of 1/3/94
- then the GMSURF entry could use that geometry data base as follows:

1	2	3	4	5	6	7	8	9	10
GMSURF	765	FENDER							
	Extrude	/u/kiz	2.5	arc	2.7	66			

- In this case, “Extrude u/kiz 2.5 arc 2.7 66” is passed to the geometry evaluator supplied by the user, and it is expected that the user-supplied routines interpret and use this record.

GRAV**Acceleration or Gravity Load**

Defines acceleration vectors for gravity or other acceleration loading.

Format:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	A	N1	N2	N3	MB		

Example:

GRAV	1	3	32.2	0.0	0.0	-1.0			
------	---	---	------	-----	-----	------	--	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0 ; Default = 0)
A	Acceleration vector scale factor. (Real)
Ni	Acceleration vector components measured in coordinate system CID. (Real; at least one Ni $\neq 0.0$)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 10. (Integer; Default = 0)

Remarks:

1. The acceleration vector is defined by $\vec{a} = A\vec{N}$, where \vec{N} is the vector defined by (N1, N2, N3). The magnitude of \vec{a} is equal to A times the magnitude of \vec{N} . The static loads generated by this entry are in the direction of \vec{a} .
2. A CID of zero references the basic coordinate system.
3. Acceleration or gravity loads may be combined with “simple loads” (e.g., FORCE, MOMENT) only by specification on a LOAD entry. That is, the SID on a GRAV entry may not be the same as that on a simple load entry.
4. In the static solution sequences, SID must be selected by the LOAD Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

5. At most nine GRAV entries can be selected in a given run either by Case Control or the LOAD Bulk Data entry. Multiples or reflections of a given acceleration or gravity load can be economically accomplished by use of the LOAD Bulk Data entry.

6. In cyclic symmetry solution sequences, the T3 axis of the coordinate system referenced in field 3 must be parallel to the axis of symmetry. In dihedral cyclic symmetry (where STYPE = "DIH" on the CYSYM entry), the T1 axis must, in addition, be parallel to Side 1 of segment 1R of the model.
7. For image superelements, the coordinate system must be rotated if the image is rotated relative to its primary superelement.
8. Acceleration or gravity loads do not include effects due to mass on scalar points.
9. The RFORCE entry may be used to specify rotational accelerations.
10. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.
11. If Modules are present then this entry may only be specified in the main Bulk Data section.

GRDSET**GRID Entry Defaults**

Defines default options for fields 3, 7, 8, and 9 of all GRID entries.

Format:

1	2	3	4	5	6	7	8	9	10
GRDSET		CP				CD	PS	SEID	

Example:

GRDSET		16				32	3456		
--------	--	----	--	--	--	----	------	--	--

Descriptor	Meaning
CP	Identification number of coordinate system in which the location of the grid points are defined. (Integer ≥ 0 or blank)
CD	Identification number of coordinate system in which the displacements, degrees-of-freedom, constraints, and solution vectors of the grid point are defined. (Integer ≥ -1 or blank)
PS	Permanent single-point constraints on the grid point. (Any combination of Integers 1 through 6 with no embedded blanks, or blank.)
SEID	Superelement identification number. (Integer ≥ 0 or blank)

Remarks:

1. The contents of fields 3, 7, 8, or 9 of this entry are assumed for the corresponding fields of any GRID entry whose field 3, 7, 8, and 9 are blank. If any of these fields on the GRID entry are blank, the default option defined by this entry occurs for that field. If no permanent single-point constraints are desired, one of the coordinate systems is basic, or the grid is assigned to the residual structure then the default may be overridden on the GRID entry by making one of fields 3, 7, 8, or 9 zero (rather than blank). Only one GRDSET entry may appear in the Bulk Data Section.
2. The primary purpose of this entry is to minimize the burden of preparing data for problems with a large amount of repetition (e.g., two-dimensional pinned-joint problems).
3. At least one of the fields CP, CD, PS, or SEID must be specified.

GRID**Grid Point**

Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	

Example:

GRID	2	3	1.0	-2.0	3.0		316		
------	---	---	-----	------	-----	--	-----	--	--

Free Field Large Format Example:

GRID*, 2, , 1.0, -2.0,
*, 3.0, , 136

Translates to:

1	2	3	4	5	6	7	8	9	10
GRID*	2				1.0		-2.0		
*+	3.0				136				+

The comma immediately following the -2.0 is required if NASTRAN IFPSTAR=NO is used. The comma is always recommended even if NASTRAN IFPSTAR=YES (Default) is used.

Descriptor	Meaning
ID	Grid point identification number. (0 < Integer < 100,000,000, see Remark 9.)
CP	Identification number of coordinate system in which the location of the grid point is defined. (Integer ≥ 0 or blank*)
X1, X2, X3	Location of the grid point in coordinate system CP. (Real; Default = 0.0)
CD	Identification number of coordinate system in which the displacements, degrees-of-freedom, constraints, and solution vectors are defined at the grid point. (Integer ≥ -1 or blank, see Remark 3.)*
PS	Permanent single-point constraints associated with the grid point. (Any of the Integers 1 through 6 with no embedded blanks, or blank*.)
SEID	Superelement identification number. (Integer ≥ 0 ; Default = 0)

*See the GRDSET entry for default options for the CP, CD, PS, and SEID fields.

Remarks:

1. All grid point identification numbers must be unique with respect to all other structural, scalar, fluid and extra (EPOINT) points.
2. The meaning of X1, X2, and X3 depends on the type of coordinate system CP as follows (see the CORDij entry descriptions):

Type	X1	X2	X3
Rectangular	X	Y	Z
Cylindrical	R	θ (degrees)	Z
Spherical	R	θ (degrees)	ϕ (degrees)

See [Grid Point and Coordinate System Definition](#) (Ch. 2) in the *MSC Nastran Reference Guide*, for a definition of coordinate system terminology.

3. The collection of all CD coordinate systems defined on all GRID entries is called the global coordinate system. All degrees-of-freedom, constraints, and solution vectors are expressed in the global coordinate system. It is recommended that points on the z-axis not have their displacement directions defined with cylindrical or spherical coordinates. (See further remarks on the CORD1S, CORD2S, CORD1C, and CORD2C.)
4. The SEID field can be overridden by use of the SESET entry.
5. If CD = -1, then this defines a fluid grid point in coupled fluid-structural analysis. This type of point may only connect the CAABSF, CHACBR, CHACAB, CHEXA, CPENTA, and CTETRA elements to define fluid elements.
6. A zero (or blank if the GRDSET entry is not specified) in the CP and CD fields refers to the basic coordinate system.
7. In p-version analysis, the hierarchy set to resolve the conflicts arising in the global system input data is described under Remark 10 of the GMBC entry description.
8. CID can reference GMCORD type coordinate systems only when the GRID is connected to p-version elements.
9. For SOL 600, ID may range from 1 to $2^{31} - 1$ (2147483647) if there are no OUTR options specified on the SOL 600 entry. If any OUTR option is specified the limit is 100000000.
10. For RC network solver in thermal analysis, the CD, PS and SEID are ignored.

GRIDA

Associative GRID for PAA

Defines an associative GRID point to be used in the COMBINE step of PAA.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDA	GID	ORIG_GID							
	PARTNAME								

Example:

GRIDA	1	1000							
	Left_outboard_wing_tank								

Descriptor	Meaning
GID	GRID ID to be used in the current run. (Integer > 0)
ORIG_GID	GRID ID in Part being referenced. (Integer > 0)
PATNAMEi	Name of Part ORIG_GID is in. (Character, C64, no internal blank spaces)

Remarks:

1. GRIDA entries are used only in the COMBINE step of PAA processing. They are used to allow the other entries in the current bulk data to reference GRID points or SPOINTs in Parts that are being combined to create the Assembly.
2. ORIG_GID must exist in the referenced Part. If the Part has been reduced, it must be a boundary GRID for that Part (in the A-set of that Part).
3. The GRIDA id may be treated the same as if it were defined using a GRID or SPOINT entry in the current run. The program will handle the connection to the referenced GRID/SPOINT in the Part.
4. Only dof which are in the Part A-set will be available for connection. If a GRIDA references a GRID in a Part, which has fewer than 6 dof in the Part A-set (dof available in the Part matrices), although the GRIDA will have 6 dof, only the dof which exist in the Part matrices will be connected to the Part.

GRIDB**Axisymmetric Grid Point**

Defines the location of a geometric grid point on a fluid point (RINGFL entry) for an axisymmetric fluid model and/or axisymmetric structure. Also defines the boundary of the fluid.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDB	ID			PHI		CD	PS	IDF	

Example:

GRIDB	30			30.0		3	345	20	
-------	----	--	--	------	--	---	-----	----	--

Descriptor	Meaning
ID	Grid point identification number. (0 < Integer < 1000000)
PHI	Azimuthal position of the fluid in degrees. (Real)
CD	Identification number of the coordinate system in which the displacements are defined at the grid point. (Integer ≥ 0 or blank)
PS	Permanent single-point constraints associated with grid point. (Any combination of the Integers 1 through 6 with no embedded blanks, or blank.)
IDF	Identification number of a RINGFL entry. (Integer > 0)

Remarks:

1. GRIDB is allowed only if an AXIF entry is also present. The AXIF entry must define a fluid coordinate system.
2. All GRIDB identification numbers must be unique with respect to other scalar, structural, fluid and extra (EPOINT) points.
3. The referenced RINGFL entry must be present and be included in a boundary list (BDYLIST entry).
4. If no harmonic numbers on the AXIF entry are specified, no fluid elements are necessary.
5. The collection of all CD coordinate systems defined on all GRID and GRIDB entries is called the global coordinate system.
6. Fields 3, 4, and 6 are ignored, which facilitates the conversion of GRID entries to GRIDB entries. Note that the fields are the same except for fields 1 and 9 when a cylindrical coordinate system is used.

GRIDF**Fluid Point**

Defines a scalar degree-of-freedom for harmonic analysis of a fluid.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDF	ID	R	Z						

Example:

GRIDF	23	2.5	-7.3						
-------	----	-----	------	--	--	--	--	--	--

Descriptor	Meaning
ID	Identification number of axisymmetric fluid point. (0 < Integer < 1000000)
R	Radial location of point in basic coordinate system. (Real > 0.0)
Z	Axial location of point in basic coordinate system. (Real)

Remarks:

1. This entry is allowed only if an AXSLOT entry is also present.
2. ID must be unique with respect to all other scalar, structural, fluid and extra (EPOINT) points.
3. Grid points on slot boundaries are defined on GRIDS entries. Do not also define them on GRIDF entries.
4. For plotting purposes, the R location corresponds to the basic X coordinate. The Z location corresponds to the basic Y coordinate. Pressures will be plotted as displacements in the basic Z direction.
5. Load and constraint conditions are applied as if GRIDF were a scalar point. Positive loads correspond to inward flow. A single-point constraint causes zero pressure at the point.

GRIDS**Slot Surface Point**

Defines a scalar degree-of-freedom with a two-dimensional location. Used in defining pressure in slotted acoustic cavities.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDS	ID	R	Z	W	IDF				

Example:

GRIDS	25	2.5	-7.3	0.5					
-------	----	-----	------	-----	--	--	--	--	--

Descriptor	Meaning
ID	Identification number of the slot point. (Integer > 0)
R	Radial location of point in basic coordinate system. (Real ≠ 0.0)
Z	Axial location of point in basic coordinate system. (Real)
W	Slot width or thickness at the GRIDS point. (Real ≥ 0.0 or blank)
IDF	Identification number to define a GRIDF point. (Integer > 0 or blank)

Remarks:

1. This entry is allowed only if an AXSLOT entry is also present.
2. ID (and IDF if present) must be unique with respect to all other scalar, structural, fluid and extra (EPOINT) points.
3. If W is blank, the default value on the AXSLOT entry will be used.
4. The IDF number is referenced on the CAXIFI entry for central cavity fluid elements next to the interface. The IDF number is entered only if the grid point is on an interface. In this case, the IDF should also be defined on a GRIDF entry.
5. If IDF is nonzero, then R must be greater than zero.
6. For plotting purposes, the R location corresponds to the basic X coordinate. The Z location corresponds to the basic Y coordinate. The slot width, W, corresponds to the basic Z coordinate. The pressure will be plotted in the basic Z direction.
7. Load and constraint conditions are applied as if the GRIDS is a scalar point. Positive loads correspond to inward flow. A single-point constraint causes zero pressure at the point.

GUST**Aerodynamic Gust Load Description**

Defines a stationary vertical gust for use in aeroelastic response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
GUST	SID	DLOAD	WG	X0	V				

Example:

GUST	133	61	1.0	0.	1.+4				
------	-----	----	-----	----	------	--	--	--	--

Descriptor	Meaning
SID	Gust set identification number. (Integer > 0)
DLOAD	Set identification number of a TLOADi or RLOADi entry that defines the time or frequency dependence. (Integer > 0)
WG	Scale factor (gust velocity/forward velocity) for gust velocity. (Real ≠ 0.0)
X0	Streamwise location in the aerodynamic coordinate system of the gust reference point. (Real)
V	Velocity of vehicle. See Remark 5. (Real > 0.0)

Remarks:

1. The GUST entry must be selected with the Case Control command GUST = SID.
2. The gust angle is in the +z direction of the aerodynamic coordinate system. The value is

$$WG \cdot T\left(t - \frac{X-X0}{V}\right)$$

where T is the tabular function.

3. In random analysis, a unit gust velocity (WG = 1/velocity) is suggested. The actual rms value is entered on the TABRNDG entry.
4. X0 and V may not change between subcases under one execution.
5. V must be equal to VELOCITY on the AERO Bulk Data entry.

HADACRI**Mesh Adaptivity Criterion and Corresponding Parameters - SOL 101 & 400 (Linear)**

Specifies Mesh refinement criterion for adaptive mesh refinement and corresponding parameters.

Format:

1	2	3	4	5	6	7	8	9	10
HADACRI	CRITID	TYPE	F1	F2	F3	F4	F5	F6	

Example:

HADACRI	1	1	0.9						
---------	---	---	-----	--	--	--	--	--	--

Descriptor	Meaning
CRITID	Identification number referenced by the CRITID field in the HADAPTL Bulk Data entry. (Integer > 0 ; no Default)
TYPE	Type of Mesh refinement adaptivity criteria. See Remark 1. (Integer ≥ 1 and ≤ 4 ; Default = 1)
F1 to F8	Criteria specific parameters. See Remark 2. (Real; no Default)

Remarks:

1. The mesh refinement criteria currently available (and selected in the TYPE field) are:

TYPE	Name of Mesh Refinement Criterion
1	Error indicator based
2	Element within a spatial spherical region
3	Elements within a spatial cubic region
4	Elements in contact criteria

2. The following table describes the different refinement criteria and corresponding parameters:

TYPE	Description
1	<p>In this case a scalar error indicator E_e is computed for all elements 'e' in the finite element mesh. Then, an element 'e' will be refined if</p> $E_e^2 \geq F_1 \bar{E}^2$ <p>where F_1 is a weight factor ($0 \leq F_1 \leq 1$) specified in the F1 field and \bar{E} is the quadratic mean of the error indicator defined as</p> $\bar{E}^2 = \frac{1}{N} \sum_{e=1}^N E_e^2$ <p>with N the total number of elements in the element set where element 'e' belongs. For this criteria the fields F2 to F6 are ignored.</p>
	<p>The elemental error indicator E_e is computed using the grid point stresses following the procedure utilized by the ELSDCON Case Control command and described in Mesh Stress Discontinuities at Grid Points in the <i>MSC Nastran Reference Guide</i>. This procedure can be summarized as follows:</p> <ul style="list-style-type: none"> ■ Let $\sigma_{aij} = \sum_{e=1}^{N_a} W_a^e \sigma_{aij}^e$ <p>be the weighted average over all elements 'e' concurrent to a given node 'a' of each component 'ij' of the grid point stresses σ_{aij}^e where W^e is a weighting factor assigned to element 'e' and N_a is the number of elements connected to the given node 'a'.</p> ■ An estimate of the error in a particular component of stress 'ij' at a grid point 'a' is then be computed as $E_{aij}^2 = \sum_{f=1}^N W_a (\sigma_{aij}^e - \sigma_{aij})^2$ ■ Averaging the latter over the different stress components, 'ij', over the different shell fibers (for shell elements) and over the different grid points 'a' connected by a given element 'e' we obtain the elemental, scalar error indicator E_e.

TYPE	Description
2	In this case the user specifies a spherical region in space with center given by (F1,F2,F3) and radius given by F4. Then, all elements with at least one node with basic coordinates (X,Y,Z) with the spherical region (i.e., such that $\ (X, Y, Z) - (F1,F2,F3)\ < F4$) will be refined. For this criteria the fields F5 and F6 are ignored.
3	In this case the user specifies a hexahedral region in space, aligned with the basic coordinates system, with corners given by (F1,F2,F3) and (F4,F5,F6). Then, all elements with at least one node with basic coordinates (X,Y,Z) within the specified hexahedral region (i.e., such that $F1 \leq X \leq F4$, $F2 \leq Y \leq F5$, $F3 \leq Z \leq F6$) will be refined.
4	In this case all elements with at least one node involved in contact either as touching or touched nodes in deformable contact bodies are refined. For this criteria the fields F1 to F6 are ignored.

3. Each criteria must have a unique ID (specified by the CRITID field and referenced by the CRITID field of the Bulk Data entry, [HADAPTL, 2126](#)).
4. The user might need to adjust the VARPHI parameter to ensure proper singular geometric feature detection (such as sharp edges or corners) (See the Parameter, [VARPHI, 1005](#)).

HADAPTL

Local Adaptive Mesh Refinement Control Parameters - SOL 101 & 400 (Linear)

Specifies Local Adaptive Mesh Refinement control parameters.

Format:

1	2	3	4	5	6	7	8	9	10
HADAPTL	ID			REPEAT	CRITID	WHEREME T	WHEREID	SNAPMETH	
	MAXLEVEL								

Example:

HADAPTL	1			10	1	PROP	5		
---------	---	--	--	----	---	------	---	--	--

Descriptor	Meaning
ID	Identification number referenced by the Case Control HADAPT command. (Integer > 0; no Default)
REPEAT	Maximum number of analysis performed before the adaptive mesh refinement process is stopped. See Remark 3. (Integer ≥ 0 ; Default = 6)
CRITID	Identification number of a mesh refinement criterion to be selected with the HADACRI Bulk Data entry. See Remarks 1. and 2. and the HADACRI Bulk Data entry. (Integer > 0; no Default)
WHEREMET	Method to specify the mesh refinement region subjected to the refinement criterion referenced by the field CRITID. It can take the values: "ALL" or "SUPER" or "PROP". See Remark 4. (Character; Default = ALL)
WHEREID	ID of the mesh refinement region subjected to the refinement criterion referenced by the field CRITID. Must be specified if WHEREMET is "SUPER" or "PROP". If WHEREMET=ALL, this field will be ignored. See Remark 4. (Integer ≥ 0 ; Default = 0)
SNAPMETH	Method to project, snap or relax new grid points created on mid-edge or mid-faces on the mesh boundary during the refinement process onto the analysis domain boundary:
0	No projection; New grids are placed in the mid-side of edges.
1	New grid points are projected onto a smooth approximation of the analysis domain boundary interpolated from the initial mesh boundary. (Integer ≥ 0 ; Default = 0)
MAXLEVEL	Maximum refinement level allowed for each individual element in the mesh. No elements in the mesh will be refined to a level bigger than MAXLEVEL. (Integer > 0; Default = REPEAT)

Remarks:

1. The adaptive mesh refinement occurs when a particular refinement criterion is satisfied. Data for the refinement criterion is specified by the Bulk Data entry HADACRI referenced by the CRITID field.

2. Multiple mesh refinement criteria can be selected in different subsets of the model. To this end, the user needs to define multiple HADAPTL entries with the same ID. Each entry might specify a different criteria (referenced in the CRITID field and defined on the corresponding HADACRI Bulk Data entry) on different subsets of the mesh (defined in the WHEREMET and WHEREID fields).
3. When multiple HADAPTL entries with the same ID are specified, NASTRAN will chose for the REPEAT, SNAPMETH, and MAXLEVEL field the maximum among all multiple instances.
4. The fields WHEREMET and WHEREID refer respectively to the Method to specify the mesh refinement region (subjected to the refinement criterion referenced by the field CRITID and defined with the Bulk Data entry, [HADACRI, 2123](#)) and its corresponding ID. For example, WHEREMET=SUPER, WHEREID=3 means that local adaptive mesh refinement (with the criteria referenced by the CRITID field) should be effected only in superelement 3. Likewise, WHEREMET=PROP, WHEREID=5 (see the previous Example) means that local adaptive mesh refinement (with the criteria referenced by the CRITID field) should be effected only in those element with Property ID equal to 5. Finally, WHEREMET=ALL imply mesh refinement in all elements.
5. In partitioned superelements, the HADAPT entry must be specified in the main bulk data section. Entries specified in the Bulk Data Section corresponding to individual parts (sections beginning with BEGIN SUPER) will be ignored.
6. When using regular superelements, the Bulk Data Section must begin with BEGIN SUPER as opposed to BEGIN BULK, in order for the refinement to be appropriately propagated across superelement boundaries. If BEGIN BULK is used, grid points on the superelement boundaries will be duplicated and not shared by the joining superelements.
7. The user should avoid the use of MPC sets 90000000 to 99999999 which are reserved for hanging nodes constraints generated during the adaptive mesh refinement process.
8. The user might need to adjust the VARPHI parameter to ensure proper singular geometric feature detection (such as sharp edges or corners) (See the Parameter, [VARPHI, 1005](#)).
9. When SNAPMETH=0, all mid-edge nodes belonging to straight edges are placed on the mid-side of its edge. By contrast, when SNAPMETH=1, mid-edge nodes belonging to the boundary of the mesh are projected to a smooth approximation of the analysis domain boundary interpolated from the mesh boundary.
10. Mid-face nodes belonging to bilinear quadrilateral faces are placed at the baricenter of its face.

HEATLOS

Heat Loss Through Convection or Radiation of the Airbag Surface - SOL 700

Defines the heat-transfer model to be used with GBAG or COUPLE. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
HEATLOS	HID	HTRID	SUBID	HTRTYPE	HTRTYPID	COEFF	COEFFV		

Example:

HEATLOS	101	83		HTRCONV	2	TABLE	14		
---------	-----	----	--	---------	---	-------	----	--	--

Descriptor	Meaning
HID	Unique number of a HEATLOS entry. (Integer > 0; Required)
HTRID	Number of a set of HEATLOS entries HTRID must be referenced from a GBAG or COUPLE entry. (Integer > 0; Required)
SUBID	(Integer $\geq 0, 0$) <ul style="list-style-type: none"> > 0 Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG, which must be a part of the surface as defined on the GBAG or COUPLE entries. = 0 HEATLOS definitions are used for the entire surface as defined on the GBAG or COUPLE entries.
HTRTYPE	Defines the type of heat transfer. (Character; Required)
HTRCONV	The HTRCONV logic is used to model heat transfer through convection in an air bag. The area of convection through which the energy is transported is equal to the area of the subsurface multiplied by COEFFV. A value of COEFFV = 1.0 will expose the complete subsurface area, while a value of COEFFV = 0.0 will result in no heat transfer through the subsurface.
HTRRAD	The HTRRAD logic is used to model heat transfer through radiation in an air bag. The area of convection through which the energy is transported is equal to the area of the subsurface multiplied by COEFFV. A value of COEFFV = 1.0 will expose the complete subsurface area, while a value of COEFFV = 0.0 will result in no heat transfer through the subsurface.
HTRTYPID	Heat transfer ID. References existing HTRTYPE entry. (Integer > 0; Required)
COEFF	Method of defining the area coefficient. (Character, CONSTANT)
CONSTANT	The area coefficient is constant and specified on COEFFV.

Descriptor	Meaning
TABLE	The area coefficient varies with time. COEFFV is the number of a TABLED1 entry giving the variation with time.
COEFFV	The area coefficient or the number of a TABLED1 entry depending on the COEFF entry. $(0.0 \leq R \leq 1.0 \text{ or } 1 > 0, 1.0)$

Remarks

1. A combination of multiple HEATLOS with different HTRTYPEs is allowed.
2. It allows for setting up the exact same model for either a uniform pressure model (GBAG to HEATLOS) or an Eulerian model (COUPLE to HEATLOS). It is then possible to set up the model using the switch from full gas dynamics to uniform pressure (GBAGCOU).
3. For the same BSURF multiple, different types of heat transfer may be defined.
4. A more detailed description can be found in Porosity in Air Bag for more details.

HGSUPPR

Hourglass Suppression Method - SOL700

Defines the hourglass suppression method and the corresponding hourglass damping coefficients. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
HGSUPPR	HID	PROP	PID	HGTYP	HGCMEM	HGCWRP	HGCTWS	HGCSOL	
+	RBRCOR	VALUE							

Example:

HGSUPPR	1	SHELL	100	FBV	0.1	0.1	0.1		
+	YES	10000							

Descriptor	Meaning	
HID	Hourglass suppression definition number. (Integer > 0; Required.)	
PROP	Property type. (Character; See Remark 1.)	
PID	Property number. (Integer > 0; See Remark 1.)	
HGTYP	Hourglass suppression method: (Character; See Remark 2.)	
	FBV	For shells only (default for shells)
	FBS	For shells and solids (default for solids)
	Dyna	For solids only
HGCMEM	Membrane damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)	
HGCWRP	Warping damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)	
HGCTWS	Twisting damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)	
HGCSOL	Solid damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)	
RBRCOR	Rigid body rotation correction: (Character; default=NO)	
	NO	No rigid-body rotation correction is applied to hourglass resisting forces.
	YES	Rigid-body rotation correction is applied to hourglass resisting forces.
VALUE	Number of steps. (Integer > 0; See Remark 3.)	

Remarks:

1. The property type definition and the property number are required. Since property numbers are unique within a certain class of element types, the property type and the property number uniquely define to what elements the hourglass suppression method and coefficients apply. The following property types are valid entries:

BAR	For bar elements
BEAM	For beam elements
BELT	For belt elements
COMP	For composite shell elements
DAMP	For damper elements
ELAS	For spring elements
EULER	For Eulerian elements
ROD	For rod elements
SHELL	For shell elements
SOLID	For solid Lagrangian elements

It must be noted however, that only shell CQUAD4 and Lagrangian CHEXA and CPENTA elements can suffer from undesired hourglass modes. All HGSUPPR entries referring to other types of elements are ignored.

2. There are three types of hourglass suppression methods available in Nastran. These are standard DYNA viscous (DYNA) hourglass damping, the Flanagan-Belytschko Stiffness (FBS) hourglass damping, and the Flanagan-Belytschko Viscous (FBV) hourglass damping.
Lagrangian solid elements can address DYNA and FBS suppression; shell elements can address DYNA and FBV suppression. The default for the Lagrangian solid elements is FBS. The default for the shell elements is FBV.
3. The rigid-body rotation correction on the hourglass forces is only necessary in cases where shell elements undergo a large rigid-body rotation. If the RBRCOR field is set to YES, and the VALUE field is left blank, the correction is applied during each time step. If the VALUE field is set to a number, the rotation correction is applied only when the rigid-body rotation would result in a rotation of the element over 90° in less than VALUE time steps. Usually, if the rigid-body rotation correction is necessary; 10000 is a good value. This option saves some CPU time.
The RBRCOR option applies to the Key-Hoff shell formulation only; for all other element types and formulations, the option is ignored.
4. The membrane, warping and twisting coefficients apply to shell elements only; for all other element types, the data is ignored. The solid damping coefficient applies to solid Lagrangian elements only; for all other element types, the data is ignored.

HTRCONV**Air Bag Convection**

Defines the heat transfer through convection for a COUPLE and/or GBAG surface.

Convection is heat transfer from the air bag to the environment through the air bag surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
HTRCONV	HTRID	HTRCFC	HTRCFT	TENV					

Example:

HTRCONV	8		14	293.0					
---------	---	--	----	-------	--	--	--	--	--

Descriptor	Meaning
HTRID	Unique number of a HTRCONV entry. Referenced from HEATLOS. (Integer > 0; Required)
HTRCFC	Constant heat transfer convection coefficient. See Remark 3. (Real ≥ 0)
HTRCFT	The heat transfer convection coefficient is a tabular function of time. The number given here is the number of a TABLED1 entry. See Remark 3. (Integer ≥ 0)
TENV	Environmental temperature. (Real > 0; Required)

Remarks:

1. The HTRCONV entry can be referenced from a HEATLOS entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state. Two of the four gas constants (γ , R , c_v and/or c_p) have to be defined on the EOSGAM entry.
3. Either HTRCF-C or HTRCF-T must be specified.
4. Energy will only transfer out of the air bag if the temperature inside the air bag is higher than the environmental temperature.

HTRRAD**Air Bag Radiation**

Defines the heat transfer through radiation for a COUPLE and/or GBAG surface.

Radiation is heat transfer from the air bag to the environment through the air bag surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
HTRRAD	HTRID	GASBMI-C	GASEMI-T	TENV	SBOLTZ				

Example:

HTRRAD	2	0.15		293.0	5.676E-8				
--------	---	------	--	-------	----------	--	--	--	--

Descriptor	Meaning
HTRID	Unique number of a HTRRAD entry. Referenced from HEATLOS. (Integer > 0; Required)
GASEMI-C	Constant gas emissivity. See Remark 3. (Real ≥ 0)
GASEMI-T	The gas emissivity is a tabular function of time. The number given here is the number of a TABLED1 entry. See Remark 3. (Integer ≥ 0)
TENV	Environmental temperature. (Real > 0; Required)
SBOLTZ	Stephan-Boltzman constant. (Real > 0; Required)

Remarks:

1. The HTRRAD entry can be referenced from a HEATLOS entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state. Two of the four gasconstants (γ , R , c_v and/or c_p) have to be defined on the EOSGAM entry.
3. Either GASEMI-C or GASEMI-T must be specified.
4. Energy will only transfer out of the air bag if the temperature inside the air bag is higher than the environmental temperature.

HYBDAMP

Hybrid Modal Damping for Direct Dynamic Solutions

Specifies hybrid damping parameters.

Format:

1	2	3	4	5	6	7	8	9	10
HYBDAMP	ID	METHOD	SDAMP	KDAMP	PRTEIG				

Example:

HYBDAMP	101	2000	2001	NO	YES				
---------	-----	------	------	----	-----	--	--	--	--

Descriptor	Meaning
ID	Identification number of HYBDAMP entry (Integer > 0; Required)
METHOD	Identification number of METHOD entry for modes calculation. (Integer > 0; Required)
SDAMP	Identification number of TABDMP1 entry for modal damping specification. (Integer > 0; Required)
KDAMP	Selects modal “structural” damping. See Remark 1. (Character: “Yes” or “NO”; Default = “NO”)
PRTEIG	Print eigenvalue summary from hybrid damping calculation (Character: “Yes” or “NO”; Default = “NO”)

Remarks:

- For KDAMP = “YES”, the viscous modal damping is entered into the complex stiffness matrix as structural damping.
- Hybrid damping is generated using modal damping specified by the user on TABDMP entries.

$$BH = [M] \begin{bmatrix} \phi_1 & \phi_2 & \dots & \phi_n \end{bmatrix} \begin{bmatrix} b(\omega_1) & & & \\ & b(\omega_2) & & \\ & & \ddots & \\ & & & b(\omega_n) \end{bmatrix} \begin{bmatrix} \phi_1^T \\ \phi_2^T \\ \vdots \\ \phi_n^T \end{bmatrix} [M]$$

For KDAMP = “YES”

$$KH = [M] \begin{bmatrix} \phi_1 & \phi_2 & \dots & \phi_n \end{bmatrix} \begin{bmatrix} g(\omega_1) & & & \\ & g(\omega_2) & & \\ & & \ddots & \\ & & & g(\omega_n) \end{bmatrix} \begin{bmatrix} \phi_1^T \\ \phi_2^T \\ \vdots \\ \phi_n^T \end{bmatrix} [M]$$

where

ϕ_i = modes of the structure

$[M]$ = structural mass matrix

$b(\omega_i)$ = modal damping values, $b(\omega_i) = g(\omega_i)\omega_i m_i$

$g(\omega_i)$ = twice the critical damping ratio determined from user specified TABDMP entry

ω_i = natural frequency of mode ϕ_i

m_i = generalized mass of mode ϕ_i

3. Hybrid damping is based on a modal solution and the mass matrix. Even if user requested all the modes and specified a flat value of G the result is still dependent on the Rank of the Mass matrix. Also note that Hybrid damping does not include residual vectors to compensate for high frequency contribution and other possible massless degrees of freedom effects in the Rank of the mass matrix. PARAM, COUPMASS, 1 may improve the mass rank.
4. HYBDAMP is referenced by the DAMPING and ROTHYBD Bulk Data entries.
5. When hybrid damping is used during FRF component generation, it is recommended to set the KDAMP field to YES on the HYBDAMP entry. See remarks under SDAMPING Case Control description for further details.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

HYDROC

Hydrostatic Pressure Loading on Surface and Faces of Shell or Solid Elements using fluid cavities.

Defines a hydrostatic pressure load on a face of a CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element using fluid cavity data. The HYDROC entry is NOT supported in SOL101 because SOL 101 does not support the Fluid Structure Interface.

Format:

1	2	3	4	5	6	7	8	9	10
HYDROC	SID	PSOLID	PNOM	G	PCH	PLD4	OVRD		
	CID/GRD	HGHTB	HABOV						

Example:

HYDROC	66	33	42.	980.		1			
	99	0.	10.						

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
PSOLID	PSOLID ID of the cavity. Fluid density will be obtained from the associated MAT10 entry. (Integer > 0); HYDROC requires ACMODL,DIFF (the default value).
PNOM	Nominal or atmosphere pressure. If G=0.0, PNOM only is applied. (Real; default = 0.0)
G	Acceleration (e.g. of gravity) in the direction from HABOV toward the origin defined by the CID/GRD field. (Real; Default = 1.0)
PCH	0 - default (no punch); 1 - Punch Set of structural grids. 2 - Punch Set of structural elements. 3 - Punch both structural grids and structural elements. (Integer ≥ 0)
PLD4	0 - default (no punch); 1 - Punch the resulting PLOAD4 entries. (Integer ≥ 0)
OVRD	0-default. If PARAM, INREL, is present and G not equal to 1.0, then a fatal error message will be issued. 1 - Allow non unit value of G.
CID/GRD	Coordinate ID of a reference coordinate system to measure the coordinate z-direction. If Integer ≥ 0 ; a coordinate system ID is used. If Integer < 0 ; a GRID ID is used with the z-direction of the GRID, CD field. Note: A spherical coordinate system will result in a fatal error. See Remark 1.; (Integer - default 0 -basic).

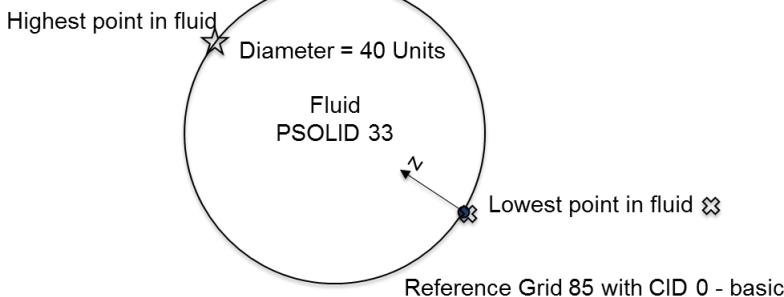
Descriptor	Meaning
HGHTB	The z-distance to the lowest fluid point measured from reference origin. See Remark 1.. (Real; Default = 0.0)
HABOV	The height of the fluid above the lowest point measured in the z-direction from reference origin. See Remark 1. (Real; Default = 0.0)

Remarks:

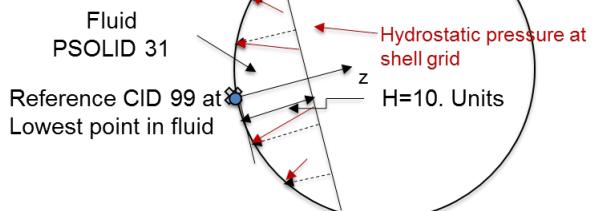
1. On the HYDROC entry, HGHTB along with the z-coordinate direction, are used to determine the location of the “bottom” of the fluid body in the case where the fluid cavity is full and there is no free surface.

In the case where a free fluid surface has been modeled in the cavity, CID/GID and HGHTB are for reference only. The lowest fluid level is determined as the lowest wetted grid in the FSI coupling measure in the direction defined by CID/GID. From the FSI coupling, free surface formed by examining wetted grids may not be perpendicular to the z-direction of CID/GID. Hence, HABOV will be considered as average free surface and hydrostatic pressure will be computed with respect to HABOV. However, a fatal message will occur if the average free surface specified by HABOV is significantly above the free surface of wetted grids. Also note that some wetted grids/elements may not get loaded if HABOV is too low.

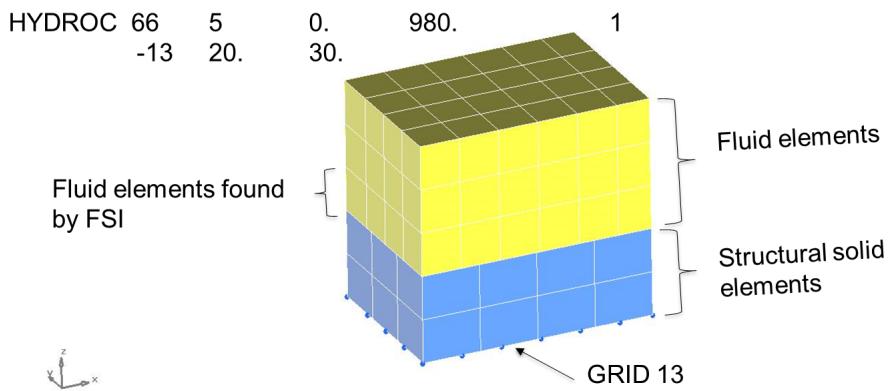
HYDROC, 95, 33, 0., 1., 0, 1, , ,
, -85, 0., 40.



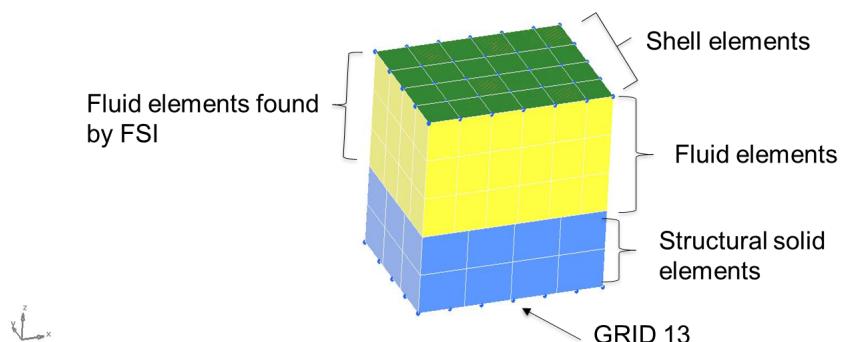
HYDROC, 95, 31, 0., 1., 0, 1, , ,
, 99, 0., 10.



2. The HYDROC entry will internally be converted to an equivalent set of PLOAD4 entries with the same load SID
3. Each fluid cavity should have a unique PSOLID entry as HYDROC uses the PSOLID to identify the cavity.
4. The PNOM is applied only from fluid surface to bottom of fluid.
5. HYDROC interaction with the Fluid Structure Interface (FSI) algorithm is shown in the two figures below. The top figure shows a column of fluid over some structural elements. The column of fluid is 30 units high, however, only its bottom surface is in contact with structural elements. Therefore, the FSI will only see a fluid that is 10 units high. A HYDROC entry for this model, with a HABOV = 50 units would fail to compute any hydrostatic loading, because 50 units is well outside the tolerance of the FSI interface, which is at 30 units.



The second figure shows the same fluid column, but a layer of shell elements has been laid on top of the fluid column.



HYDROC 66 5 0. 980. 1
-13 20. 50.

In this case, the FSI sees a column of fluid 50 units high. Thus a HABOV = 50 units will generate PLOAD4 entries using a height of 30 units for the fluid. For this model, any HABOV with values ($30 \text{ units} \leq \text{HABOV} \leq 50 \text{ units}$) will yield valid PLOAD4 entries on the fluid structure interface of the structural solid elements with the height used depending on the value of HABOV.

6. The figures below show additional HYDROC PNOM rules associated with the FSI interface. Also note how fluid cavities are defined by PSOLID ID.

HYDROS (1) applies a hydrostatic loading of $55. + 0.*h*0.$ interior to the cylinder with POINT 17
 HYDROC (2) applies a hydrostatic loading $0. + \rho * h * 980.$

HYDROC (3) applies a hydrostatic loading $42. + \rho * h * 980.$ – above the fluid there is no pressure of 42. units

1. HYDROS, 66, 55., 0., 0., 6, ,1 , ,

, -85, 0., 40., 17

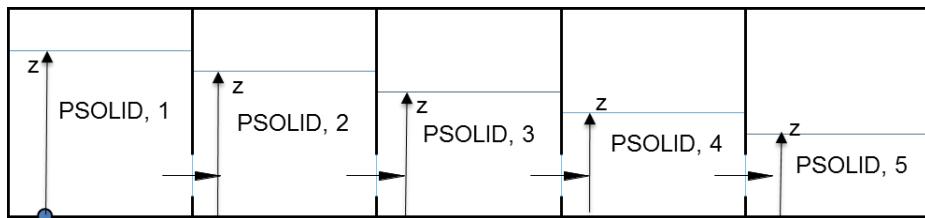
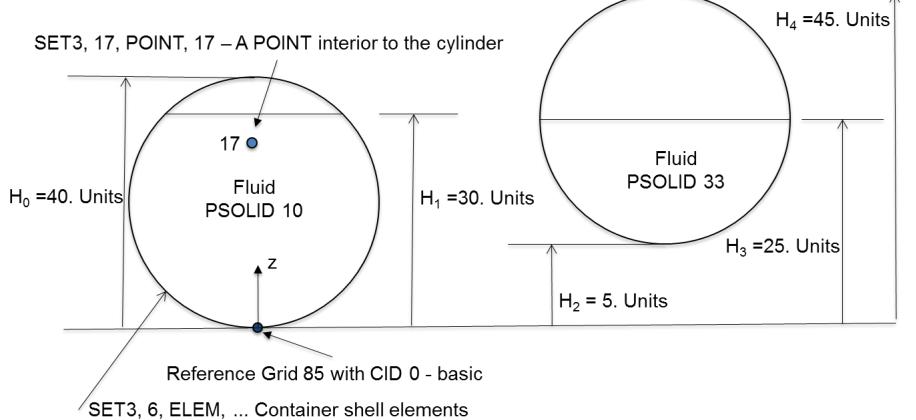
2. HYDROC, 66, 10, 0., 980., 0, 1, , ,

, -85, 0., 30.

3. HYDROC, 66, 33, 42., 980., 0, 1, , ,

, -85, 5., 25.

SET3, 17, POINT, 17 – A POINT interior to the cylinder



Reference origin with CID

Staggered grid for flooding simulation.

HYDROS

Hydrostatic Pressure Loading on Surface and Faces of Shell or Solid Elements using element or grid sets

Defines a hydrostatic pressure load on a face of a CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element using element set data.

Format:

1	2	3	4	5	6	7	8	9	10
HYDROS	SID	PNOM	RHO	G	SETE	SETG	PLD4	OVRD	
	CID/GRD	HGHTB	HABOV	CNTL					

Example:

HYDROS	77	55.	0.08	17.	300		1		
	200		97.	83					

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
PNOM	Nominal or atmosphere pressure. If G=0.0, PNOM only is applied. (Real; Default = 0.0)
RHO	Mass density. (Real > 0.0)
G	Acceleration (e.g. of gravity) in the direction from HABOV toward the origin defined by the CID/GRD field. (Real; Default = 1.0)
SETE	The ID of a SET3 entry using (DES= ELEM) or SET1 to identify the element IDs of structure elements at fluid-structure interface used for hydrostatic loading. (Integer ≥ 0)
SETG	The ID of a SET3 entry using (DES= GRID) or SET1 to identify the element grid IDs of structure elements at fluid-structure interface used for hydrostatic loading. (Integer ≥ 0)
PLD4	0 - default (no punch); 1 - Punch the resulting PLOAD4 entries. (Integer ≥ 0)
OVRD	0-default. If PARAM, INREL, is present and G not equal to 1.0, then a fatal error message will be issued. 1 - Allow non unit value of G.
CID/GRD	Coordinate ID of a reference coordinate system to measure the coordinate z-direction. If Integer ≥ 0 ; a coordinate system is used. If Integer < 0 ; a GRID ID is used with the z-direction of the GRID, CD field. Note: A spherical coordinate system will result in a fatal. See Remark 1.; (Integer - default 0 -basic). Note: A spherical coordinate system will result in a fatal error.
HGHTB	The z-distance to the lowest fluid point measured from reference origin. (Real; Default = 0.0)

Descriptor	Meaning
HABOV	The height of the fluid above the lowest point measured in the z-direction from reference origin. (Real; Default = 0.0)
CNTL	SET3 ID of control grids to select wetted surface(s). For an immersed physical body, at least one grid per quadrant. (Integer ≥ 0)

Remarks:

1. The HYDROS entry will internally be converted to an equivalent set of PLOAD4 entries with the same load SID.

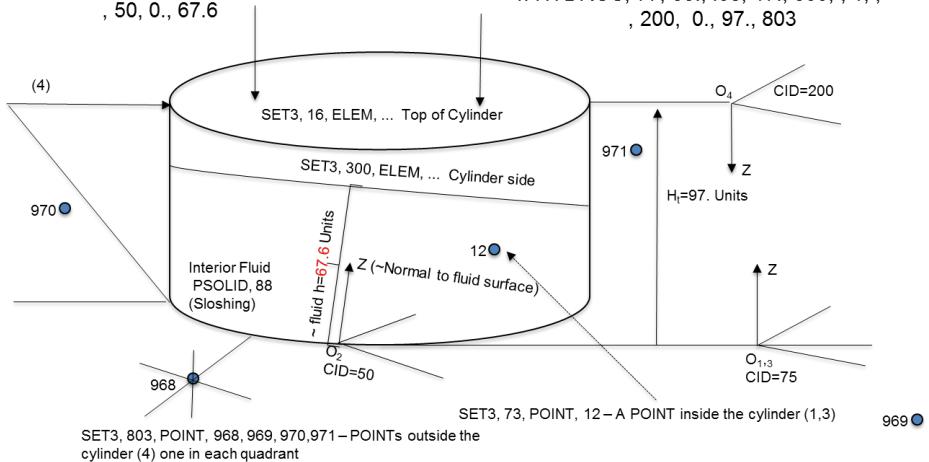
HYDROS (1) applies a hydrostatic loading of $25. + .08*h*0.$ interior to the cylinder with POINT 12
 HYDROC (2) applies a hydrostatic loading $0. + p^* h * 1.$; h – measured normal to sloshed surface
 HYDROS (3) applies a hydrostatic loading $55. + .08* h * 980.$ (h is constant 97. units for this set)
 HYDROS (4) applies a hydrostatic loading $55. + .08*h * 17.$ for all elements on exterior cylinder side.

1. HYDROS, 77, 25., .08, 0, 300, , 1, ,
 , 75, 0, 97., 73

3. HYDROS, 77, 55., .08, 980., 16, , 1, ,
 , 75, 0, 97., 73

2. HYDROC, 77, 88, 0, 1., 0, 1, ,
 , 50, 0., 67.6

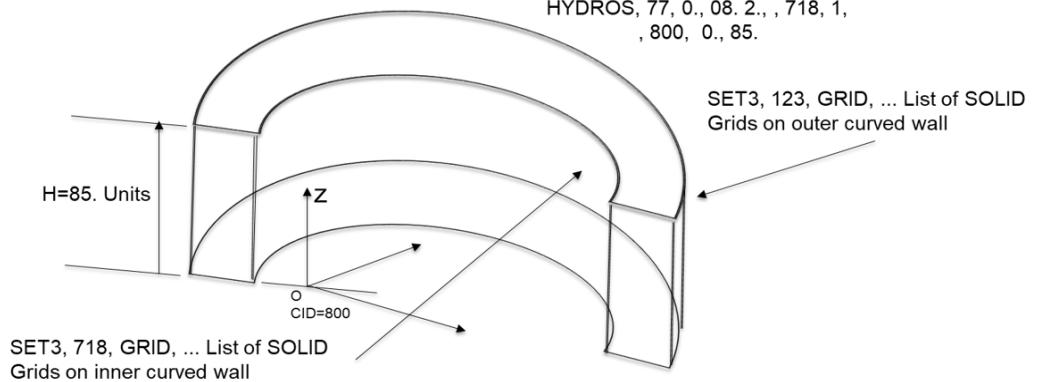
4. HYDROS, 77, 55., .08, 17., 300, , 1, ,
 , 200, 0, 97., 803



SOLID Model

HYDROS, 77, 0., 08. 2., , 123, 1,
, 800, 0., 85.

HYDROS, 77, 0., 08. 2., , 718, 1,
, 800, 0., 85.



HYDSTAT**Hydrostatic Preset of Density in Euler Elements**

Initializes the Euler element densities in accordance to a hydrostatic pressure profile. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
HYDSTAT	HID	MID	GID	CID	XCG	YCG	ZCG	PATM	

Example:

HYDSTAT	101	4			0.0	0.0	0.0	100000.	
---------	-----	---	--	--	-----	-----	-----	---------	--

Descriptor	Meaning
HID	Identification number of the HYDSTAT entry referenced from the COUPLE entry. (Integer > 0; Required)
MID	Material MATDEUL id to which the hydrodynamic pressure profile will be applied. (Integer > 0; Required)
GID	Number of a grid point at the free surface. See Remark 4. (Integer ≥ 0)
CID	Local coordinate system. See Remark 4. (Integer ≥ 0)
XCG,YCG, ZCG	Coordinates of a point at the free surface. See Remark 4. (Real)
PATM	Pressure at free surface. (Real; Required)

Remarks:

1. It is assumed that each Euler domain contains at most two Eulerian materials and includes the GRAV entry. One material has to be a fluid using EOSPOL the other a gas or void. This EOSPOL material is given by the MID entry. The interface between gas and fluid is the free surface and is assumed to be normal to the gravity vector as specified on the GRAV entry. For example if the gravity vector points in the z-direction then the interface between the gas and the fluid has to be horizontal.
2. The hydrostatic preset changes the density of the fluid like material in order to conform to the hydrostatic preset. It overrules the material densities as specified on the TICEL and TICVAL entries. Densities of the gas like material are not changed.
3. The free surface has to match with material fractions as defined in the initialization of Euler elements by the TICEL and TICEUL entries. The hydrostatic preset only changes densities, it does not change material fractions.
4. There are two options to enter the location of the free surface. The first option is to enter a grid point number. In that case the fields CID and XCG-ZCG have to be left blank. The GRIDPOINT entry already has the option of using a local coordinate system. When coordinates are used then the field GID has to be left blank.

5. If there is no structural grid point indicating the free surface then a new grid point can be defined that will only be used for determining the free surface level. If the Gravity vector points in the z-direction, only the z-coordinate of the grid point will be used. The x and y ordinate can be chosen arbitrarily. Similar remarks hold when the gravity vector is in one of the other coordinate directions. The same holds when using coordinates instead of a grid point.
6. PATM should be equal to the pressure in the air.
7. When coupling surfaces are present then the HYDSTAT ID needs to be referenced by at least one coupling surface. For each coupling surface a different HYDSTAT entry can be defined. Several COUPLE entries can refer to the same HYDSTAT ID. If no HYDSTAT ID is specified on a COUPLE entry then Euler elements associated to this coupling surface will not be initialized with a hydrostatic preset.

IMPCASE

Defines a collection of imperfection cases

Defines a collection of imperfection cases by listing of identification numbers of IMPGEOM entries. Used in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
IMPCASE	ID	IMPFID1	IMPFID2	...	IMPFIDI				

Example:

IMPCASE	5	1	2						
---------	---	---	---	--	--	--	--	--	--

Descriptor Meaning

ID Identification number (integer > 0, required)

IMPFIDI Identification number of imperfection case (integer >0, required)

Remarks:

1. When the ID is referenced by case control IMPERFECT command, the list of IMPGEOM entries are used to apply geometric imperfections.
2. "THRU" can be used to specify IMPFIDI.

IMPGEOM**Defines Geometric Imperfection**

Defines geometric imperfection by selecting subcases, steps, mode numbers or increments from imperfection input files. Used in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
IMPGEOM	IMPFID			SETID	SCALE		UNIT		
	SUBCASE1	STEP1	MODEINC1	SETID1	SCALE1		UNIT1		
	SUBCASE2	STEP2	MODEINC2	SETID2	SCALE2		UNIT2		
	etc.								

Example:

IMPGEOM	1								
	1	3	1		0.011		34		
	1	3	2		0.002		34		

Descriptor Meaning

IPMFID	Identification number of imperfection case (integer >0, required)
SETID	Default of SETID _i (integer ≥ 0 , default=0)
SCALE	Default of SCALE _i (real, default=0.0)
UNIT	Default of UNIT _i (integer ≥ 0 , default=0)
SUBCASE _i	Identification number of a subcase (integer ≥ 0 , default=0)
STEP _i	Identification number of a step (integer ≥ 0 , default=0)
MODEINC _i	Mode number if the selected step by SUBCASE _i and STEP _i or subcase by SUBCASE _i is a result of buckling analysis or modal analysis, increment number if the selected result is from a static analysis. (integer ≥ 0 , default=0)
SETID _i	Identification number of a SET1 or SET3 bulk entry (integer ≥ 0 , default = 0). When it is defined, only the grid points with the id listed in SET1 or SET3 have geometric imperfection effect. If a SET3 is used, it must be GRID type.
SCALE _i	Scale factor (real, default = 0.0)
UNIT _i	Imperfection file unit number specified by ASSIGN HDF5IN, INPUTTT2 or IMPFIN FMS command.

Remarks:

1. All IMPGEOM identification numbers must be unique. An IMPGEOM entry is referenced by IMPERFECT case control directly or by an IMPCASE bulk entry.

2. Geometric imperfection effect is applied to grid points as coordinates variation, the variation is superposed by eigenvectors of multiple modes or displacement of increments defined in multiple continuation lines.
3. An imperfection input file can be an MSC Nastran HDF5 result file, a Nastran op2 file or a text based file called IMPF. To use it, an ASSIGN HDF5IN, INPUTTT2 or IMPFIN FMS command is required.
4. The imperfection input file of HDF5 or op2 can be a result file of SOL 101, 103, 105 or a SOL 400 with ANALYSIS type of BUCK, MODES, STATIC, NLSTATIC or NLTRAN, and must contain eigenvectors or displacement results.
5. IMPF file format is described in remark 11. It can be DISP format or GEOM format. If an IMPF file is used in an IMPGEOM entry, only one continuation line of IMPGEOM is allowed, and SUBCASEi, STEPi and MODEINCi are ignored. If format is GEOM, SCALE and SCALEi are ignored too.
6. SUBCASEi must be specified if the imperfection input file is an op2 or hdf5 file.
7. STEPi is ignored when the imperfection input file is a result of SOL 101, 103 or 105. It must be specified for a SOL 400 result file.
8. MODEINCi is ignored when the file is a result of SOL 101 or a linear static step of SOL 400. It is required for other cases if the imperfection file is an hdf5 or op2 file.
9. UNITi must be specified if UNIT is 0 or not specified.
10. Geometric imperfection is applied to residual structure for superelement models. In the case of part superelement, it is advised to adjust boundary grid point searching tolerance due to geometry changing. This can be done by adding or modifying SEBULK or SECONCT entries.
11. Format of IMPF file

The IMPF file is a csv-like file, delimiters of a line can be spaces, a comma or a tab space. A line starting with a “\$” or “#” is a comment line. If the first line is a comment line and contains string “GEOM” or “GRID”, then it is a GEOM file, i.e., the grid coordinates provided here will replace the grid coordinates in the FE model; otherwise it is a DISP file, i.e., the displacements provided in the file will be superimposed to the coordinates of the corresponding grids.

For each line, the first column is grid point id, followed by x,y and z. For DISP format, x,y and z are coordinate variations from the original coordinates, and coordinate system is MSC Nastran global system, this means that x,y,z are in the coordinate system indicated by CD field of GRID entries in the original input file. For GEOM format, x,y and z are “imperfect” coordinates of the grid points, coordinate system is MSC Nastran input system, this means the values of x,y,z respect CP field of the GRID entries in the original input file.

Below is a DISP format of IMPF file.

disp_1.impf

18 ,	0.000000000000E+00 ,	0.000000000000E+00 ,	0.000000000000E+00
19 ,	0.000000000000E+00 ,	0.000000000000E+00 ,	0.000000000000E+00
20 ,	1.882586315422E-04 ,	-1.382379038267E-03 ,	-5.111620104649E-05

```

21 ,   3.080245224512E-04 ,  -1.360710847814E-03 ,  -5.111620104647E-05
22 ,   4.254461607540E-04 ,  -1.328686826189E-03 ,  -5.111620104646E-05
23 ,   5.396298968822E-04 ,  -1.286550695534E-03 ,  -5.111620104647E-05
24 ,   6.497067236577E-04 ,  -1.234623137246E-03 ,  -5.111620104650E-05
25 ,   7.548388899626E-04 ,  -1.173299351397E-03 ,  -5.111620104657E-05
26 ,   8.542262765314E-04 ,  -1.103046049026E-03 ,  -5.111620104668E-05
27 ,   9.471124853400E-04 ,  -1.024397900185E-03 ,  -5.111620104681E-05
28 ,   1.032790596252E-03 ,  -9.379534647748E-04 ,  -5.111620104691E-05
29 ,   1.110608547111E-03 ,  -8.443706371464E-04 ,  -5.111620104698E-05
30 ,   1.179974096322E-03 ,  -7.443616391243E-04 ,  -5.111620104702E-05
31 ,   1.240359330172E-03 ,  -6.386875995705E-04 ,  -5.111620104705E-05

```

Below is a GEOM format of IMPF file.

`geom_example.impf`

```

# GEOM
1,   0.000000000000E+00,   0.000000000000E+00,   0.000000000000E+00
2,   1.000000014901E-01,   4.367130542590E-24,   1.464331138939E-08
3,   2.000000029802E-01,   7.552163438029E-24,   3.787209513278E-08
4,   2.999999821186E-01,   8.873575793200E-24,   5.096678857312E-08
5,   3.999999463558E-01,   0.000000000000E+00,   0.000000000000E+00
6,   4.999999701977E-01,   0.000000000000E+00,   0.000000000000E+00
7,   5.999999642372E-01,   0.000000000000E+00,   0.000000000000E+00
8,   6.999999880791E-01,   0.000000000000E+00,   0.000000000000E+00
9,   7.9999998927116E-01,   0.000000000000E+00,   0.000000000000E+00
10,   8.999999165535E-01,   0.000000000000E+00,   0.000000000000E+00
11,   9.999999403954E-01,   0.000000000000E+00,   0.000000000000E+00
12,   1.099999904633E+00,   0.000000000000E+00,   0.000000000000E+00
13,   1.199999928474E+00,   0.000000000000E+00,   0.000000000000E+00
14,   1.299999952316E+00,   0.000000000000E+00,   0.000000000000E+00

```

INCLUDE**Insert External File**

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

Format:

INCLUDE'filename'

Describer:

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

Example:

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

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

Remarks:

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

/dir123/dir456/dir789/filename.dat

may be included with the following input:

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

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

INFLCG

Airbag Cold Gas Inflator Model

Defines the cold gas-inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLCG	INFLID	TANKVOL	INITPRES	INITTEMP	INITMAS	γ GASNAM	C_v	R	
	C_p								

Example:

INFLCG	11	0.875	131325	293	1.37	1.4		286	

Descriptor	Meaning
INFLID	Unique number of an INFLCG entry. Referenced from ABINFL. (Integer > 0; Required)
TANKVOL	Tank Volume. (Real > 0; Required)
INITPRES	Initial tank pressure. (Real > 0; Required)
INITTEMP	Initial tank temperature. (Real > 0; Required)
INITMAS	Initial gas mass of inflator. (Real > 0; Required)
γ , GASNAM	Ratio of specific heat constants if real. Name of an INFLGAS entry if character. See Remarks 4. and 5. (Real > 0 or 1)
C_v	Specific heat at constant volume. See Remark 6. (Real > 0)
R	Gas constant. See Remark 6. (Real > 0)
C_p	Specific heat at constant pressure. See Remark 6. (Real > 0)

Remarks:

1. The INFLCG entry can be referenced from an ABINFL entry.
2. When used in an Euler coupled analysis, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state.
3. Either INITPRESS or INITMASS has to be specified, but not both. The relation between INITMASS and INITPRESS is given by

$$INITPRES = R \frac{INITMAS}{TANKVOL} INITTEMP$$

4. The cold gas inflator is a reservoir filled with high pressure gas. It is assumed that the volume stays constant at TANKVOL. The mass inside the inflator will steadily decrease due to flow into the Euler domain or to a GBAG. Due to inertia it can happen that the pressure of the inflator becomes less than the outside pressure. In that case some inflow into the inflator occurs. Transport between inflator and the Euler domain or GBAG is based on the constancy of total temperature. This is equivalent to the pressure method.
5. If this field contains a real entry real or is left blank, the inflator gas constants are given on the INFLTR entry itself. Otherwise, the entry will be read as the name of an INFLGAS entry. In this case, the remaining entries must be left blank.
6. Specify only two of the four gas constants. They are related as:

$$\nu = \frac{c_p}{c_v} \quad R = c_p - c_v$$

INFLFRC**Hybrid Inflator Gas Fraction Definition**

Defines the gas fractions as a function of time for hybrid inflators. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLFRC	FRACID	TYPE							
	"TIME"	TIME1	FRAC1	FRAC2	FRAC3	etc.			
	"TIME"	TIME2	FRAC1	FRAC2	FRAC3	etc.			

Example:

INFLFRC	12								
	TIME	14.0E-3	11.342	13.391	9.626	57.019	7.31	1.312	
	0.0	0.1	0.2						
	TIME	14.1E-3	43.332	6.2817	4.5155	26.747	3.4291	0.4898	
	15.077	0.0	0.4						

Descriptor	Meaning
FRACID	Unique number of an INFLFRC entry.(Integer > 0, required.)
TYPE	Specifies whether mass fractions or molar fractions will be given (Character, default=MASS.)
	MASS The fractions on INFLFRC are mass fractions.
	MOLAR The fractions on INFLFRC are molar fractions. See Remark 5.
TIMEID	Defines a new line of data (Character, required.)
	TIME Specifies that data for a new time increment will be given. See Remark 3.
TIMEi	Time for which the gas fractions are given. (Real \geq 0.0, required.)
FRACi	Fraction of gas i at the specified time..See Remark 7. (Real \geq 0.0, required.)

Remarks:

1. The INFLFRC entry is referenced with FRACID from an AIRBAG entry with the option "INFLATOR" or from INFLHB.
2. Fraction values of the inflowing gas will be linearly interpolated between the specified time increments.
3. Use as many continuation lines as necessary to completely define the gas fractions. The data for a time step are preceded by a TIME keyword. Missing entries will be set to 0.0.

4. The order of the gases for which the fractions are specified is identical to the order in which the gases are specified on the AIRBAG entry with the option “INFLATOR”.
5. At least one line of gas fractions must be given.
6. If molar fractions (TYPE=MOLAR) are to be used, the universal gas constant must be specified through PARAM, UGASC.
7. At least one of the fractions for each time step must have a value greater than 0.0.
8. Fractions for each timestep should add up to 1.0. If this is not the case, they will be scaled so that they do.

INFLGAS**Inflator Gas Definition**

Defines a thermically ideal gas to be used with a standard or hybrid inflator. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLGAS	GASNAM	TYPE	VALUE	CPGAS	V1	V2	V3	V4	

Example:

INFLGAS	101	RSPEC	189.	CONSTANT	846.				
---------	-----	-------	------	----------	------	--	--	--	--

Descriptor	Meaning
GASNAM	Unique number of an INFLGAS entry. (Integer; Required)
TYPE	Specific gas constant or molar weight specified. (Character; RSPEC)
	RSPEC Specific gas constant
	MOLWT Molar weight, see Remark 2.
VALUE	Value of the variable TYPE. (Real > 0; Required)
CPGAS	The variation of the specific heat constant at constant pressure. (Character; CONSTANT)
	CONSTANT The specific heat is constant and specified in V1.
	TABLE The specific heat constant is temperature-dependent. V1 is the number of a TABLED1 entry giving the variation of the specific heat with the temperature.
POLY	The specific heat constant is temperature-dependent. V1 through V4 are the coefficients of a polynomial expression, see Remark 3.
V1	The specific heat constant, the number of a TABLED1 entry or the first polynomial coefficient, depending on the value of CPGAS. (Real or Integer > 0; Required)
V2, V3, V4	Coefficients of polynomial expression when CPGAS equals POLY. (Real; 0.0)

Remarks:

1. INFLGAS can be referenced by an INFLTR, INFLHB or INITGAS entry.
2. When the molar weight is given, the universal gas constant R_{uni} must be specified using PARAM, UGASC, so that:

$$R_{spec} = R_{uni}/MOLWT$$

3. A polynomial expression for c_p is given by:

$$c_p(t) = V1 + V2 \cdot T + V3 \cdot T^2 + \frac{V4}{T^2}$$

4. The specific heat constant at constant volume c_v is calculated from the specific heat constant at constant pressure c_p , the universal gas constant and the molecular weight according to:

$$c_v = c_p(T) - R_{spec}$$

5. The ratio of specific heats is given as:

$$\gamma = c_p / c_v$$

INFLHB**Hybrid Inflator Model**

Defines the hybrid-inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLHB	INFLHID	MASFLRT	TBMPT	TEMPC	FRAC				
	GASNAM1	GASNAM2	GASNAM3	-etc.-					

Example:

INFLHB	9	15		650.	12				
	22	25	3						

Descriptor	Meaning
INFLID	Unique number of an INFLHB entry. (Integer > 0; Required)
MASFLRT	Table number of a TABLED1 entry specifying the massflow-rate as a function of time. (Integer > 0; Required)
TEMPT	Table number of a TABLED1 entry specifying the static temperature of the inflowing gas as a function of time. See Remark 3. (Integer > 0)
TEMPC	Constant value of the temperature of the inflowing gas. See Remark 3. (Real > 0.0)
FRAC	Number of an INFLFRC entry specifying the fractions of the inflowing gas as a function of time. (Integer > 0; Required)
GASNAMi	ID of an INFLGAS entry. See Remark 4. (Integer)

Remarks:

1. The INFLHB entry can be referenced from a [ABINFL, 1186](#) entry.
2. When used in an Euler coupled analysis, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state.
3. Either TEMPC or TEMPT must be specified. The INFLHB entry uses the specified temperature as the static temperature of the inflowing gas. In literature the static temperature is also known as total-, rest-, or stagnation temperature and refers to the temperature of the gas when brought to rest from its moving condition as opposed to the dynamic temperature that refers to the temperature of the moving gas.
4. At least one inflator gas must be specified using an INFLGAS entry. There is no limit to the number of inflator gases per INFLHB.

INFLTNK**Airbag Tanktest Inflator Model**

Defines the Tanktest-inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLTNK	INFLID	METHOD	TPTABLE	TANKVOL	INFLMAS	INITPRES	ENDTPRES	INITTEMP	
	ENDTEMP	γ	c_v	R	c_p	IPTABLE	INFLPRES	INFLTEMP	
	INFLAREA	SFTP	SFIP						

Example:

INFLTNK	111	AVTEMP	10	0.12	0.01	0.0			
			1.4		286.				

Descriptor	Meaning
INFLID	Unique number of an INFLTNK entry. (Integer > 0; Required)
METHOD	Method of calculating the mass-flowrate: (Character; Required)
AVTEMP	Average Temperature Method
INFPRES	Inflator Pressure Method
TPTABLE	Table number of a TABLED1 entry specifying the tank pressure as a function of time. (Integer > 0; Required)
TANKVOL	Tank Volume. (Real > 0; Required)
INFLMAS	Total gas mass generated by inflator. (Real > 0; Required)
INITPRES	Initial tank pressure. See Remark 3. (Real > 0; Required)
ENDPRES	End tank pressure. See Remarks 4. and 5. (Real > 0; Required)
INITTEMP	Initial tank temperature. See Remark 5. (Real > 0; Required)
ENDTEMP	End tank temperature. See Remark 5. (Real > 0; Required)
γ	Ratio of specific heat constants. See Remark 7. (Real > 0)
c_v	Specific heat at constant volume. See Remark 7. (Real > 0)
R	Gas Constant. See Remark 7. (Real > 0; Required)
c_p	Specific heat at constant pressure. See Remark 7. (Real > 0)
IPTABLE	Table number of a TABLED1 entry specifying the inflator pressure as a function of time. See Remark 5. (Integer > 0; Required)
INFLPRES	Initial inflator pressure. See Remarks 5. and 6. (Real > 0; Required)

Descriptor	Meaning	
INFLTEMP	Temperature of inflowing gas: See Remark 5. (Real > 0 or Character; ATM)	
	ATM	Use average temperature of AVTEMP method.
	Real value	User specified temperature.
INFLAREA	Total area of inflator holes. See Remark 5. (Real > 0; Required)	
SFTP	Scale factor for tank pressure. See Remark 5. (Real > 0; 1.0)	
SFIP	Scale factor for inflator pressure. See Remark 5. (Real > 0, 1.)	

Remarks:

1. The INFLTNK entry can be referenced from an ABINFL entry.
2. When used in an Euler coupled analysis, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state.
3. The initial tank pressure entry (INITPRES) is interpreted as an absolute pressure and used to define reference pressure at t=0 in the tank. The difference between INITPRES and the pressure value at t=0 from the table will be added to the entire pressure curve of TPTABLE.
4. The end tank pressure entry (ENDPRES) is interpreted as an absolute pressure at t=tend of tank pressure table (TPTABLE). This value is used for calculation of total generated mass in the tank.
5. This field must be specified only when Inflator Pressure Method (INFPRES) is defined in the METHOD field.
6. The initial inflator pressure entry (INFLPRES) is interpreted as an absolute pressure and used to define reference pressure at t=0 in the inflator. The difference between INFLPRES and the pressure value at t=0 from the table will be added to the entire pressure curve of IPTABLE.
7. Specify only two of the four gas constants. They are related as:

$$\gamma = \frac{c_p}{c_v} \quad R = c_p - c_v$$

INFLTR**Airbag Inflator Model**

Defines the inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLTR	INFLID	MASFLRT	TEMPT	TEMPC	γ GASNAME	c_v	R	c_p	
	MID								

Example:

INFLTR	5	100		907.0	1.5	283.0			
	0	0	0						

Descriptor	Meaning
INFLID	Unique number of an INFLTR entry. (Integer > 0; Required)
MASFLRT	Table number of a TABLED1 entry specifying the massflow-rate as a function of time. (Integer > 0; Required)
TEMPT	Table number of a TABLED1 entry specifying the static temperature of the inflowing gas as a function of time. See Remark 3. (Integer > 0)
TEMPC	Constant value of the static temperature of the inflowing gas. See Remark 3. (Real > 0)
γ , GASNAME	Ratio of specific heat constants if real. Name of an INFLGAS entry if character. See Remarks 4. and 5. (Real > 0 or Character)
c_v	Specific heat at constant volume. See Remark 5. (Real > 0)
R	Gas constant. See Remark 5. (Real > 0)
	Specific heat at constant pressure. See Remark 5. (Real > 0)
MID	Material MATDEUL ID of the Inflator material. See Remark 2. Only used for MMHYDRO solver. (Integer ≥ 0)

Remarks:

1. The INFLTR entry can be referenced from an ABINFL entry.
2. When used in combination with the single material hydrodynamic Euler solver an EOSGAM (ideal gas) equation of state is required. In that case the material number MID can be left blank. When using the Multi-material solver the Material number MID has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM. The Multi-material solver does not allow the use of gas fractions.

3. Either TEMP-C or TEMP-T must be specified. The INFLTR entry uses the specified temperature as the static temperature of the inflowing gas. In literature the static temperature is also known as total-, rest-, or stagnation temperature and refers to the temperature of the gas when brought to rest from its moving condition, as opposed to the dynamic temperature which refers to the temperature of the moving gas.
4. If this field contains a real entry real or is left blank, the inflator gas constants are given on the INFLTR entry itself, see Remark 5. Otherwise, the entry will be read as the name of an INFLGAS entry. In this case, the remaining entries must be left blank.
5. Specify only two of the four gas constants. They are related as:

$$\gamma = \frac{c_p}{c_v} \quad R = c_p - c_v$$

INITGAS**Gasbag or Coupling Surface Initial Gas Fraction Definition**

Specifies the initial gas composition inside a gasbag or Euler coupling surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INITGAS	INTID		GASNAM1	FRAC1	GASNAM2	FRAC2	-etc.		

Example:

INITGAS	4		14	0.4	32	0.11			
---------	---	--	----	-----	----	------	--	--	--

Descriptor Meaning

INTID Unique number of an INITGAS entry. (Integer > 0; Required)

GASNAMi ID of an INFLGAS entry. See Remark 3. (Integer > 0)

FRACi Mass fraction of gas i. See Remark 4. (Real ≥ 0.0)

Remarks:

1. The INITGAS entry can be used to specify the initial gas composition for a gasbag or for an Eulerian coupling surface. The INTID must be referenced either from a GBAG cad or a COUPLE entry.
2. Use as many continuation lines as necessary to completely define the gas fractions.
3. At least one INFLGAS reference must be given.
4. Fractions should add up to 1.0. If this is not the case, they will be scaled so that they do.

IPSTRAIN

Initial Equivalent Plastic Strain Values in SOL 400

Defines initial equivalent plastic strain values. This is the initial plastic strain option used in SOL 400 only.
 (NOTE: This entry is SINGLE FIELD ONLY! Double field will fatal)

Format:

1	2	3	4	5	6	7	8	9	10
IPSTRAIN	EID1	EID2	INT1	INTN	LAY1	LAYN	STRAIN		

Example:

IPSTRAIN	2001	2020	1	4	1	5	0.025		
----------	------	------	---	---	---	---	-------	--	--

Descriptor **Meaning**

EID1	First Element ID to which these strains apply. (Integer > 0)
EID2	Last Element ID to which these strains apply. (Integer; Default = EID1)
INT1	First Integration point for which the strain applies. (Integer > 0; Default = 1)
INTN	Last Integration point for which the strain applies. (Integer > 0; Default = 4)
LAY1	First element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
LAYN	Last element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
STRAIN	Equivalent plastic strain value at start of analysis. (Real; Default is 0.0)

Remarks:

1. This entry only applies to SOL 400 advanced nonlinear elements (selected with PSHLN1, PSLDN1, etc.) and is ignored for other solutions.
2. This entry is normally used for metal forming and represents the amount of plastic deformation that the model was previously subjected to. It is used in work (strain) hardening models.

IPSTRN**Initial Equivalent Plastic Strain Values in SOL 600**

Defines initial equivalent plastic strain values. This is the MSC Marc's initial plastic strain option used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
IPSTRN	EID1	EID2	INT1	INTN	LAY1	LAYN	STRAIN		

Example:

IPSTRN	2001	2020	1	4	1	5	0.025		
--------	------	------	---	---	---	---	-------	--	--

Descriptor	Meaning
EID1	First Element ID to which these strains apply. (Integer > 0)
EID2	Last Element ID to which these strains apply. (Integer; Default = EID1)
INT1	First Integration point for which the strain applies. (Integer > 0; Default = 1)
INTN	Last Integration point for which the strain applies. (Integer > 0; Default = 4)
LAY1	First element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
LAYN	Last element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
STRAIN	Equivalent plastic strain value at start of analysis. (Real; Default is 0.0)

Remarks:

1. This entry only applies to SOL 600 advanced nonlinear elements and is ignored for other solutions.
2. This entry is normally used for metal forming and represents the amount of plastic deformation that the model was previously subjected to. It is used in work (strain) hardening models.
3. For SOL 600 only, if the extra precision available with large field formats is not required and small field precision is adequate, the header IPSTRAIN may be used instead of IPSTRN.

ISTRESS**Initial Stress Values in SOL 400 and SOL 600**

Defines initial stress values. This is the MSC Marc's initial stress option used in SOL 400 and SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
ISTRESS	EID1	EID2	INT1	INTN	LAY1	LAYN	STRESS1	STRESS2	
	STRESS3	STRESS4	STRESS5	STRESS6	STRESS7				

Example:

ISTRESS	2001	2020	1	4	1	5	45000.	-2000.	
	0.0	4500.	0.0	2350.					

Descriptor	Meaning
EID1	First Element ID to which these stresses apply. (Integer > 0)
EID2	Last Element ID to which these stresses apply. (Integer; Default = EID1)
INT1	First Integration point for which the stress applies. (Integer > 0; Default = 1)
INTN	Last Integration point for which the stress applies. (Integer > 0; Default = 4)
LAY1	First element layer for which the stress applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
LAYN	Last element layer for which the stress applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
STRESS(i)	Up to 7 stress components may be entered. (Real; Default = 0.0)

Remarks:

1. This entry only applies when MSC Marc is executed from Nastran using SOLs 400 and 600 and is ignored for other solutions.
2. Initial stresses must be self-equilibrating and may not produce material nonlinearity.
3. Stress components are as follows:

Definitions:

s - normal type stress

t - shear type stress

x,y,z in global system

1,2,3 in element local system

3D solid elements (for example type 7)

1 - sxx

2 - syy

3 - szz

4 - txy

5 - tyz

6 - tzx

7 - hydrostatic pressure (Herrmann elements only, otherwise 7 should be blank)

Thick shells (for example type 75)

1 - s11

2 - s22

3 - t12

4 - t23

5 - t31

Thin shells (for example type 72)

1 - s11

2 - s22

3 - t12

Beams (for example type 14 or 98)

1 - s - axial

2 - t - twist

ITER**Iterative Solver Options**

Defines options for the iterative solver in SOLs 101, 106, 108, 111, 153 and 400.

Format:

1	2	3	4	5	6	7	8	9	10
ITER	SID								
		OPTION1VALUE1	OPTION2VALUE2	-etc.-					

Example:

ITER	100		
		ITSEPS=1.0E0-7, MSGFLG=YES, PRECOND=BICWELL, IPAD=3	

Descriptor	Meaning
SID	Set identification number. (Integer > 0).
PRECOND	Preconditioner option. (Character; Default = "BIC" for real analysis, "BICCMPLX" for complex analysis and "PBDJ" for p-version analysis.) See Remarks 3. and 4.
J	Jacobi
JS	Jacobi with diagonal scaling.
C	Incomplete Cholesky.
CS	Incomplete Cholesky with diagonal scaling.
RIC	Reduced incomplete Cholesky.
RICS	Reduced incomplete Cholesky with diagonal scaling.
PBCJ	p-version block Cholesky/Jacobi.
PBDJ	p-version block Direct/Jacobi.
PBDC	p-version block Direct/Cholesky.
BIC	Block incomplete Cholesky for real problems.
BICCMPLX	Block incomplete Cholesky for complex problems.
CASI	Element-based third party iterative solver.
USER	User given preconditioning. For direct frequency response (SOL 108): a decomposition will be done for 1st frequency and the factor will be used for all subsequent frequencies as a preconditioner with the iterative solver. Other solutions require a DMAP alter. Please refer to the <i>MSC Nastran Numerical Methods User's Guide</i> description of the SOLVIT module.
CONV	Convergence criterion. (Character; Default = "AREX")

Descriptor	Meaning	
AR	$\ r\ /\ b\ $ where r is the residual vector of current iteration and b is the initial load vector; internal criterion.	
GE	Alternative convergence criterion using geometric progression and the differences between two consecutive solution updates; internal criterion.	
AREX	Same criterion as AR but with the additional consideration of the external convergence criterion. See Remark 2. (Default).	
GEXX	Same criterion as GE but with the additional consideration of the external convergence criterion. See Remark 2.	
MSGFLG	Message flag. (Character; Default = "NO")	
YES	Messages will be printed for each iteration.	
NO	Only minimal messages will be printed from the iterative solver (Default).	
ITSEPS	User-given convergence parameter epsilon. (Real > 0.0) Default= 1.E-4 for PRECOND = "CASI" Default= 1.E-8 for PRECOND = "CASI" with contact in the model Default= 1.E-6 for all other PRECOND options	
ITSMAX	Maximum number of iterations. (Integer > 0; Default = N/4 where N is the number of rows in the matrix)	
IPAD	Padding value for RIC, RICS, BIC, and BICCMPLX preconditioning. (Integer > 0) Default = 0 for PRECOND = "RIC" or "RICS" Default = 2 for PRECOND = "BIC" for purely three-dimensional models and three for two-dimensional and mixed element models. IPAD may be reset automatically by the program to the best value. Default = 5 for PRECOND = "BICCMPLX".	
IEXT	Extraction level in reduced incomplete Cholesky preconditioning. Block structuring method in block incomplete Cholesky preconditioning. (Integer = 0 thought 7; Default = 0) 0 Uses USET/SIL tables (Default). 1 - 7 The default value of 0 is recommended for all problems. The values 1 - 7 use a heuristic algorithm with a maximum block size equal to IEXT. Although setting IEXT to a value other than 0 could lead to slightly improved performance or reduced disk space use, it should be considered exploratory without the expectation of a benefit.	
PREFONLY	Specifies early termination of the iterative solver. (Integer = 0 or -1; Default = 0) 0 Runs to completion (Default). -1 Terminates after preface giving resource estimates.	

Remarks:

1. The optional ITER Bulk Data entry is selected by the SMETHOD Case Control command and is only required to override the defaults specified above.
2. The external epsilon is computed as follows:

$$\varepsilon = \frac{(r, x)}{(b, x)}$$

where r is the final residual vector, x is the final solution vector and b is the initial load vector (r, x) indicates the inner product of r and x and (b, x) indicates the inner product of b and x .

3. See the *MSC Nastran Numerical Methods User's Guide* for more information on these options.
4. The element-based iterative solver is primarily intended for the solution of very large solid element structural analysis problems. The following restrictions apply:
 - SOLs 101, 200 and 400 only. SOL 200 availability is limited to topology optimization and thus cannot be used for COMPLIANCE, FRMASS, WEIGHT, and VOLUME in DRESP1.
 - SOL 101 convergence parameter epsilon of 1.E-4 may be too large for some models and a decrease to 1.E-8 may be necessary.
 - Solver selection criteria and parameters cannot vary across subcases.
 - No GENEL elements allowed
 - x2GG/x2PP direct input matrix selection is allowed; however, the matrix size is limited to 100 grid points and must be symmetric.
 - No ASET/OMIT reduction allowed
 - Inertia relief is supported in SOL 101 for PARAM,INREL,-1. A SUPORT entry is required. PARAM,INREL,-2 is not supported.
 - No transfer functions allowed
 - No RFORCE or PLOADX follower forces allowed
 - Follower force stiffness must be symmetricized
 - No heat transfer allowed
 - No p-elements allowed
 - DMP is not supported with the CASI solver.

Only BAR, BEAM, BUSH, ROD, CONMi, CONROD, DAMPi, ELASi, HEXA, MASSi, PENTA, QUAD4, QUAD8, QUADR, SEAM, SHEAR, TRIA3, TRIA6, TRIAR, TETRA, VISC and WELD elements are allowed.

5. GPGPU devices are not supported for iterative methods.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

LBSH3DG Tables used by an NLBSH3D to find force as function of displacement, velocity and rotor speed

Defines the force tables of NLBSH3D nonlinear loads as function of relative displacement or velocity and current rotor speed.

Format:

FORM = ASYM (Asymmetric matrix representation)

1	2	3	4	5	6	7	8	9	10
LBSH3DG	LID	FORM	MIN_FREQ	MAX_FREQ					
	P11	P12	P13	P14	P15	P16			
	P21	P22	P23	P24	P25	P26			
	P31	P32	P33	P34	P35	P36			
	P41	P42	P43	P44	P45	P46			
	P51	P52	P53	P54	P55	P56			
	P61	P62	P63	P64	P65	P66			

FORM = DIAG (Only Diagonal terms representation)

1	2	3	4	5	6	7	8	9	10
LBSH3DG	LID	FORM	MIN_FREQ	MAX_FREQ					
	P11	P22	P33	P44	P55	P66			

FORM = SYM (Symmetric matrix representation)

1	2	3	4	5	6	7	8	9	10
LBSH3DG	LID	FORM	MIN_FREQ	MAX_FREQ					
	P11								
	P21	P22							
	P31	P32	P33						
	P41	P42	P43	P44					
	P51	P52	P53	P54	P55				
	P61	P62	P63	P64	P65	P66			

Descriptor	Meaning
LID	Load identification group number for a NLBSH3D. (Integer > 0; Required)
FORM	Type of Input Curves. (Character; Default SYM)
SYM	Symmetric, only lower triangular terms needed

Descriptor	Meaning
DIAG	Only diagonal terms are needed
ASYM	Asymmetric, all terms are needed
MIN_FREQ	Minimum valid rotor speed for all the table lookups provided. (Real). See remark 2 .
MAX_FREQ	Max valid rotor speed for all the table lookups provided. (Real > 0.0 Required). Refer remark 2 .
Pij	TABLED5 Id that provides TABLED1 ids as a function of rotor speed. The TABLED1 will have force as a function of relative displacement and/or velocity.

Remarks:

1. Values for the nonlinear forces will be determined using interpolation based on rotor speed and relative deflection/velocity of grid pair. If the rotor speed is beyond the range specified by MIN_FREQ and MAX_FREQ, a FATAL message will be produced. (Extrapolation will NOT be used beyond this range).
2. For SYNC analysis, rotor speed is obtained from NLFREQ / NLFREQ1 bulk data card. For ASYNC analysis, rotor speed is obtained from RGYRO bulk data card.
3. All the tables (TABLED5 Pij) should provide lookup tables (TABLED1) for the range of rotor speeds between Min_FREQ and Max_Freq.

LEAKAGE

Mass Loss Through Holes or Permeability of the GBAG or COUPLE Surface

Defines the porosity model to be used with GBAG or COUPLE. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
LEAKAGE	LID	PORID	SUBID	PORTYPE	PORTYPID	COEFF	COEFFV		

Example:

LEAKAGE	7	100	365	PERMEAB	63		0.99		
---------	---	-----	-----	---------	----	--	------	--	--

Descriptor	Meaning	
LID	Unique number of a LEAKAGE entry. (Integer > 0; Required)	
PORID	Number of a set of LEAKAGE entries PORID must be referenced from a GBAG or COUPLE entry. (Integer > 0; Required)	
SUBID	> 0 Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG, which must be a part of the surface, as defined on the GBAG or COUPLE entry. = 0 LEAKAGE definitions are used for the entire surface as defined on the GBAG or COUPLE entry.	
PORTYPE	Defines the type of porosity. (Character; Required)	
PORFLOW	The PORFLOW logic is used to model a constant flow boundary in the coupling surface. The flow boundary acts on the open area of the coupling surface. The open area is equal to the area of the (sub) surface multiplied by COEFFV. A hole can be modeled when COEFFV is set to 1.0. A closed area results for COEFFV = 0.0. The characteristics of the flow are defined on a PORFLOW entry, with ID as defined on the PORTYPID.	
PORFLWT	The PORFLWT logic is used to model a time dependent flow boundary in the coupling surface. The flow boundary acts on the open area of the coupling surface. The open area is equal to the area of the (sub) surface multiplied by COEFFV. A hole can be modeled when COEFFV is set to 1.0. A closed area results for COEFFV = 0.0. The characteristics of the flow are defined on a PORFLWT entry, with ID as defined on the PORTYPID.	

Descriptor	Meaning
PORHOLE	The PORHOLE logic can be used to model small holes in an air bag. A BSURF defines the hole. The open area of the hole is equal to the area of the (sub) surface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete area of the hole, while a value of COEFFV = 0.0 will result in a closed hole. The characteristics for the flow are defined on a PORHOLE entry, with ID as defined on the PORTYPID.
PERMEAB	The PERMEAB logic is used to model permeable air-bag material. The permeable area can be defined for a BSURF or for the entire coupling surface. The velocity of the gas flow through the (sub) surface is defined as a linear or tabular function of the pressure difference between the gas inside the air bag and the environmental pressure. The function is specified on a PERMEAB entry, with ID as defined on the PORYPID. The area actually used for outflow is the subsurface area multiplied by the value of COEFFV
PORFGBG	The PORFGBG logic can be used to model gas flow through a hole in the coupling surface connected to a GBAG. A BSURF defines the hole. The open area of the hole is equal to the area of the surface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete area of the hole, while a value of COEFFV = 0.0 will result in a closed hole. The characteristics for the flow are defined on a PORFGBG entry, with ID as defined on the PORTYPID.
PERMGBG	The PERMGBG logic is used to model gas flow through a permeable area in the coupling surface connected to a GBAG. The permeable area can be defined for a BSURF or for the entire coupling surface. The velocity of the gas flow through the (sub) surface is defined as a linear or tabular function of the pressure difference. This function is specified on a PERMGBG entry, with ID as defined on the PORYPID. The area actually used for outflow is the subsurface area multiplied by the value of COEFFV.
PORFCPL	The PORFCPL logic can be used to model gas flow through a hole in the coupling surface connected to another coupling surface. A BSURF defines the hole. The open area of the hole is equal to the area of the (sub) surface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete area of the hole, while a value of COEFFV = 0.0 will result in a closed hole. The characteristics for the flow are defined on a PORFCPL entry, with ID as defined on the PORTYPID.
PORHYDS	Prescribes a hydrostatic pressure profile.
PORTYPID	Porosity ID. References existing PORTYPE entry. (Integer > 0; Required)
COEFF	Method of defining the porosity coefficient. (Character; CONSTANT)
CONSTANT	The porosity coefficient is constant and specified on COEFFV.

Descriptor	Meaning
TABLE	The porosity coefficient varies with time. COEFV is the number of a TABLED1 entry giving the variation with time.
COEFFV	The porosity coefficient or the number of a TABLED1 entry depending on the COEFF entry. (0.0 < Real < 1.0 or Integer < 0, 1.0)

Remarks:

1. The combination of multiple LEAKAGEs with different PORTYPEs is allowed.
2. It allows for setting up the exact same model for either a uniform pressure model (GBAG to LEAKAGE) or an Eulerian model (COUPLE to LEAKAGE). It is then possible to set up the model using the switch from full gas dynamics to uniform pressure (GBAGCOU).

The options PORFGBG and PERMGBG can be used to model air bags with multiple compartments.

LOAD**Static Load Combination (Superposition)**

Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, DAREA (if these entries have been converted), PLOAD, PLOAD1, PLOAD2, PLOADB3, PLOAD4, PLOADX1, SLOAD, RFORCE, and GRAV, ACCEL and ACCEL1 entries (as well as SPCD and SPCR for SOL 600 only). In addition, all the thermal loads are also supported in the NASTRAN thermal analysis (SOL 153 and SOL 400 with analysis=hstat) such as QBDY1, QBDY2, QBDY3, QVECT, QVOL, SLOAD, QHBDY Bulk Data entries.

Format:

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-						

Example:

LOAD	101	-0.5	1.0	3	6.2	4			
------	-----	------	-----	---	-----	---	--	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
S	Overall scale factor. (Real)
Si	Scale factor on Li. (Real)
Li	Load set identification numbers defined on entry types listed above. (Integer > 0)

Remarks:

1. The load vector $\{P\}$ is defined by

$$\{P\} = S \sum_i Si \{P_{Li}\}$$

2. Load set IDs (Li) must be unique.
3. This entry must be used if acceleration loads (GRAV entry) are to be used with any of the other types.
4. In the static solution sequences, SID must be selected by the LOAD Case Control command. In dynamic solution sequences the EXCITEID of a RLOADi or TLOADi may refer to a LOAD entry.
5. A LOAD may be referenced by another LOAD entry. Nested LOAD references are permitted. A LOAD entry may not reference itself.
6. Note that the LOAD entry is scaled differently when using the RFORCE with rotor dynamics. See Remark 21. of the RFORCE entry.
7. LOAD Bulk Data entry will not combine an SPCD load entry except for SOL 600. In the static solution sequences, the set ID of the SPCD entry (SID) is selected by the LOAD Case Control command. In dynamic analysis refer to Remark 2. of the SPCD, 3127 entry.

8. If Modules are present then this entry may only be specified in the main Bulk Data section.

LOADCLID**Define Solution Load Combinations Using Part Load ID**

Define a loading combination in PAA using the Load ID from SUBCASEs of the Parts.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCLID	LID	S							
	S1	LID1							
PARTNAME1									
	S2	LID2							
PARTNAME2									
	Etc.								

Example:

LOADCLID	202	1.00							
	1.00	101							
first_part_name									
	1.0	301							
second_part_name									

Descriptor	Meaning
LID	Load set ID. Selected by the LOAD Case Control command. (Integer > 0)
S	Overall scale factor. (Similar to LOAD Bulk Data entry) (Real < > 0)
Si	Scale factor applied to Load ID loads. (Real < > 0)
LIDi	Load ID specified during GENERATE by LOAD or CLOAD Case Control directive. (Integer > 0)
PARTNAMEi	Name of Part. (64 -Characters maximum)

Remarks:

1. LOADCLID entries are used only in the COMBINE and SOLVE steps of PAA processing.
2. The LOADCLID allows a loading combination to be created using the Load id (LIDi) from case control. In each PAA run, each SUBCASE may have a LIDi provided. These are used by the LOADCLID to define a loading combination.
3. As with other loading entries, all loads with the same LID are combined. Therefore, care must be exercised when using loading combinations.

LOADCNAM**Define Solution Load Combination by Load Name**

Used only in PAA to define a loading combination using the LOADNAMEs used in Case Control for Parts.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCNAM	LID	S							
	S1								
LOADNAME1									
PARTNAME1									
	S2								
LOADNAME2									
PARTNAME2									
	Etc.								

Example:

LOADCNAM	101	1.00							
	2.5								
gravity_x									
outboard_right_side_engine_nacelle									
	1.1								
gravity_x_on_another_part									
second_part_name									

Descriptor	Meaning
LID	Load set ID. Selected by the LOAD Case Control command. (Integer > 0)
S	Overall scale factor. (Similar to LOAD Bulk Data entry) (Real < 0)
Si	Scale factor applied to LOADNAME loads. (Real < 0)
LOADNAMEi	Name of Load for that Part. (64 -Characters maximum)
PARTNAMEi	Name of Part. (64 -Characters maximum)

Remarks:

1. LOADCNAM entries are used only in the COMBINE and SOLVE steps of PAA processing.
2. The LOADCNAM allows a loading combination to be created using the LOADNAME from case control. In each PAA run, each SUBCASE may have a LOADNAME provided. These LOADNAMEs are used by the LOADCNAM to define a loading combination.
3. All loads with the same LID are combined. Therefore, care must be exercised when using loading combinations.

LOADCSUB**Define Solution Load Combinations Using Subcase Number**

Used only in PAA to define a loading combination using the LOADNAMEs used in Case Control for Parts.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCSUB	LID	S							
	S1	SUB1							
PARTNAME1									
	S2	SUB2							
PARTNAME2									
	Etc.								

Example:

LOADCSUB	1001	1.0							
	1.0	1							
part_we_are_using_subcase_1_from									
	1.0	3							
part_we_are_using_subcase_3_from									

Descriptor	Meaning
LID	Load set ID. Selected by the LOAD Case Control command. (Integer > 0)
S	Overall scale factor. (Similar to LOAD Bulk Data entry) (Real < > 0)
Si	Scale factor applied to Load ID loads. (Real < > 0)
SUBi	Subcase Number specified during GENERATE or COMBINE run. (Integer > 0)
PARTNAMEi	Name of Part. (Character, C64)

Remarks:

1. LOADCSUB entries are used only in the COMBINE and SOLVE steps of PAA processing.
2. The LOADCSUB allows a loading combination to be created using the SUBCASE ids from case control. In each PAA run, each SUBCASE has an id. These ids are used by the LOADCSUB to define a loading combination.
3. All loads with the same LID are combined. Therefore, care must be exercised when using loading combinations.

LOADCYH**Harmonic Load Input for Cyclic Symmetry**

Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCYH	SID	S	HID	HTYPE	S1	L1	S2	L2	

Example:

LOADCYH	10	1.0	7	C	0.5	15			
---------	----	-----	---	---	-----	----	--	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
S	Scale Factor. (Real)
HID	Harmonic. See Remark 6. (Integer or blank)
HTYPE	Harmonic type. (Character: "C", "S", "CSTAR" "SSTAR", "GRAV", "RFORCE", or blank).
Si	Scale factor on Li. (Real)
Li	Load set identification number. See Remark 10. (Integer > 0)

Remarks:

1. The LOADCYH entry is selected with the Case Control command LOAD = SID.
2. If HTYPE is blank, the load will be applied to all applicable types in the problem.
3. If HTYPE is "GRAV" or "RFORCE", GRAV or RFORCE entry loading will be used. Harmonic loads for appropriate available harmonics will be generated automatically in these cases.
4. L1 and L2 may reference LOAD entries. However, the LOAD entry in such a case must not reference load sets defined via RFORCE and/or GRAV entries.
5. If L1 refers to a set ID defined by an SPCD entry, the same ID must not reference any nonzero loading specified by the other Bulk Data loading entries given via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, DAREA (if these entries have been converted), MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX, and SLOAD or converted DAREA entries. In other words, an enforced deformation loading via the SPCD entry is specified by a load set ID L1 or L2 and a zero magnitude loading via a load (such as FORCE) with the same ID.
6. If HTYPE is "GRAV" or "RFORCE", the entry in HID will be ignored and therefore may be blank. S2 and L2 must be blank for this case.
7. Load set IDs L1 or L2 may not be referenced by load set ID L1 or L2 of LOADCYN entries.
8. If HTYPE = "C", "S", "CSTAR", or "SSTAR", the load on component (HTYPE) of harmonic (HID) is $L = S(S1 \cdot L1 + S2 \cdot L2)$.

9. S must be nonzero. In addition, either S1 or S2 must be nonzero.
10. L1 and L2 may reference any of the static or dynamic loading entries including GRAV and RFORCE.

LOADCYN**Physical Load Input for Cyclic Symmetry**

Defines a physical static or dynamic load for use in cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCYN	SID	S	SEGID	SEGTYP	S1	L1	S2	L2	

Example:

LOADCYN	10	1.0	1	R	0.5	17			
---------	----	-----	---	---	-----	----	--	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
S	Scale Factor. (Real)
SEGID	Segment identification number. (Integer)
SEGTYP	Segment type. (Character: "R", "L", or blank)
Si	Scale Factors. (Real)
Li	Load set ID numbers. See Remark 8. (Integer > 0)

Remarks:

1. The LOADCYN entry is selected by the LOAD Case Control command.
2. If SEGTYP is blank, both R and L segments will be used in DIH-type symmetry.
3. L1 and L2 may reference LOAD entries. However, the LOAD entry in such a case must not reference load sets defined via RFORCE and/or GRAV entries.
4. If L1 refers to a set ID defined by SPCD loading entry, the same ID must not reference any nonzero loading specified by the other Bulk Data loading entries given via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, DAREA (if these entries have been converted), MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, and SLOAD entries. In other words, an enforced deformation loading via the SPCD entry is specified by a load set ID L1 or L2 and a zero magnitude loading via a load (such as FORCE) with the same ID.
5. Load set IDs L1 or L2 may not be referenced by load set ID L1 or L2 of LOADCYH entries.
6. The load on the segment (or half-segment) is $L = S(S1 \cdot L1 + S2 \cdot L2)$.
7. S must be nonzero. In addition, either S1 or S2 must be nonzero.
8. L1 and L2 may reference any of the static or dynamic loading entries except GRAV and RFORCE.
9. For cyclic buckling loads may only be applied to the first segment and only zero harmonic loads may be applied so the LOADCYH entry should be used.

LOADCYT

Table Load Input for Cyclic Symmetry

Specifies loads as a function of azimuth angle by references to tables that define scale factors of loads versus azimuth angles. This entry is used only when STYPE = "AXI" on the CYSYM entry.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCYT	SID	TABLEID1	LOADSET1	METHOD1	TABLEID2	LOADSET2	METHOD2		

Example:

LOADCYT	10	19	27		21	26	1		
---------	----	----	----	--	----	----	---	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
TABLEID _i	Table ID for table load input for load set L _i . See Remark 3. (Integer > 0)
LOADSET _i	Load set L _i . (Integer > 0)
METHOD _i	Method of interpolation. See Remark 5. (Integer: 0 or 1; Default = 0)
0	Interpolate the load with the Fourier coefficients specified in the table up to the specified number of harmonics. (Default)
1	Interpolate the magnitude of the load at corresponding grid points in all segments.

Remarks:

1. The LOADCYT entry is selected by the LOAD Case Control command.
2. The load set ID given in fields 4 or 7 of this entry may refer to FORCE, MOMENT, PLOAD, PLOAD2, PLOAD4, SPCD, TEMP, or TEMPP1 Bulk Data entries.
3. Either TABLED1 or TABLED2 type tabular data of azimuth angle (X_i) versus scale factors (Y_i) may be used. The azimuth angle values must be in degrees.
4. The scale factors given in the tables referenced by TABLED_i entries will be applied only to the magnitudes of the loads defined by LOADSET IDs given in fields 4 or 7.
5. For grid point loading entries, (like FORCE, MOMENT, SPCD, and TEMP Bulk Data entries) METHOD_i = 1 option should be used. For element loading entries (like PLOAD, PLOAD2, PLOAD4, and TEMPP1 Bulk Data entries) either METHOD_i = 0 or METHOD_i = 1 option can be used. In particular, if harmonic output of element stresses under temperature loading via TEMPP1 Bulk Data entry, METHOD_i = 0 option should be used to specify TEMPP1 load set.

LOADOF

Specifies Table IDs for Individual Degrees of Freedom for the Static Loads with Tables Described using LOADT Entries

Specifies table IDs for individual degrees of freedom for the static loads with tables described using LOADT entries for SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
LOADOF	T0	T1	T2	T3	T4	T5	T6		

Example:

LOADOF	100	101	102	103	104	105	106		
--------	-----	-----	-----	-----	-----	-----	-----	--	--

Descriptor	Meaning
T0	Table ID specified as a negative Ti value on a LOADT entry (specified as a positive value on this entry. (Integer > 0; no Default)
Ti (i=1 to 6)	ID of a TABLEL1 entry that specifies the time history of each degree of freedom for the loads specified by Li, -Ti on LOADT entries. Up to 6 degrees of freedom may be specified. See Remark 2. (Integer; no Default)

Remarks:

1. LOADOF can only be used in SOL 600
2. This entry must not be used unless there is a negative value of Ti on one or more LOADT entries.

LOADT

Specifies Static Loads that use a Table to Describe Their Variation with Pseudo-Time

Specifies static loads that will use a table to describe their variation with pseudo-time. Available static entries are FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, SPCD, SPCR, RFORCE and GRAV for SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
LOADT	SID	L1	T1	L2	T2	L3	T3	L4	
	T4	L5	T5	etc.					

Example:

LOADT	101	1	51	2	52	3	53		
-------	-----	---	----	---	----	---	----	--	--

Descriptor	Meaning
SID	Load set identification number which must have a matching Bulk Data entry LOAD with the same SID. (Integer > 0; no Default)
Li	Load set identification numbers defined on entry types listed above. (Integer > 0)
Ti	ID of a TABLEL1 or TABL3D0 entry that specifies the time history of the entry type. See Remarks 4. through 6. (Integer or Blank; no Default)

Remarks:

1. LOADT can only be used in SOL 600.
2. In SOL 600 applied displacements must use SPCD (or SPCD) rather than SPC.
3. In SOL 600, if any LOADT entries are found in the model, the following parameters are automatically set:
 - PARAM,MARCVERR,11 (Allows table-driven inputs)
 - PARAM,MRCONVER,11 (Uses contact compatible with Version 11)
 - PARAM,MARCTOTT,1 (Allows loads with tables)
4. If Ti is positive, the table will be used for all degrees of freedom for the loads specified by Li. If Ti is negative, the absolute value of Ti will be used as the ID of a LOADOF entry providing the table IDs for each degree of freedom for which Li applies.
5. If more than one Li references the same force, moment pload4, etc, the first Ti encountered for that entity will be used for all occurrences of that entity for that subcase. Therefore, the 2nd and subsequent occurrences of Ti for these common entities may be left blank. An example can be found in ETL example mtloadt03.

6. It is strongly suggested that for a particular dof of a FORCE,MOMENT, etc. that only one table describe the variation for all subcases in the model. For example, if there are 3 subcases the time should normally cover the entire range of 0.0 to 3.0 (rather than having three tables, one for the range 0.0 to 1.0, a second for the range of 1.0 to 2.0 and a third from 2.0 to 3.0).

LORENZI

Fracture Mechanics J-Integral in SOL 600

This option gives an estimation of the J-Integral for a crack configuration using the domain integration method. The domain integration method has the advantage that it can also be used for problems with thermal behavior and for dynamic analysis. This procedure is only available for continuum elements. Only the nodes defining the crack front (crack tip in two dimensions) need to be defined. The program automatically finds integrations paths according to the format below. The complete J-Integral is evaluated and output. For the case of linear elastic material with no external loads on the crack faces, the program automatically separates mode I, mode II, and mode III (3-D only) stress intensity factors from the J-Integral.

Format:

1	2	3	4	5	6	7	8	9	10
LORENZI	IDCR		MSEP	ITYPE	G2D	ISET0			
	DIRECT	ISETD	SV1	SV2	SV3				
	TOP	NREG	JTYPE	TOL	SV1	SV2	SV3		
	GEOM	NREG	JTYPE	TOL	SV1	SV2	SV3	RAD	
		CYLIN							

Example:

LORENZI	1		0	1	1				
	DIRECT	101	0.0	-2.0E-3	0.0				
	DIRECT	102	0.0	-2.0E-3	0.0				
LORENZI	1		0	1	1				
	TOP	2	1						

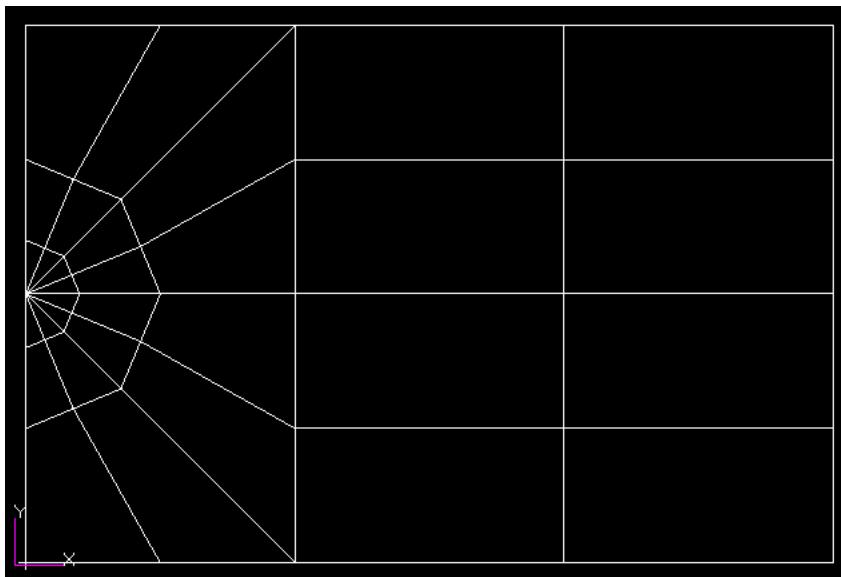
Descriptor	Meaning
IDCR (4-2)	Identification of this particular crack. (Integer > 0; Required field) IDCR must be unique among all LORENZI cracks.
MSEP (3-1)	Enter 0 for no mode separation or 1 for mode separation. (Integer; Default = 1)
ITYPE (4-1)	Type of crack propagation (Integer; Default = 2)
	<ul style="list-style-type: none"> 1 The “rigid region” will be described using direct inputs of nodes or elements. 2 The “rigid region” will be described using an automatic search based on topology. 3 The “rigid region” will be described using an automatic search based on geometry.

Descriptor	Meaning
G2D (5 for 2D)	If the crack is 2D, enter the grid ID of the node at the crack tip. If the crack is 3D, leave this field blank. (Integer; no Default)
ISET0 (5 for 3D)	If the crack is 3D, enter the ID of a SET3 entry that defines an ordered set of grids or elements along the crack tip (for example in the thickness direction of the crack). If the crack is 2D, enter the grid ID of the node at the crack tip. (Integer; no Default)
DIRECT (Start of 5a)	Enter the string DIRECT if ITYPE=1, otherwise skip this line. See Remark 4. (Character,)
ISETD (6a)	Enter the ID of a SET3 entry defining an ordered set of grids or elements within the rigid region. See Remark 4. (Integer; no Default)
SV1 (7a-1 or 6b-1 or 6c-1)	First component of shift vector. (Real; Default = .0.)
SV2 (7a-2 or 6b-2 or 6c-2)	Second component of shift vector. (Real; Default = 0.0)
SV3 (7a-3 or 6b-3 or 6c-3)	Third component of shift vector, enter only for 3D. (Real; Default = 0.0)
TOP (Start of 5b)	Enter the string TOP if ITYPE=2, otherwise skip this line. (Character)
NREG (5b-1 or 5c-1)	Number of regions for which crack growth is to be estimated. See Remark 5. (Integer; Default = 1)
JTYPE (5b-2 or 6b-2)	Enter 0 if SV1, SV2, SV3 on this line are to be used or enter 1 for automatic determination of SV1, SV2 and SV3. (Integer; Default = 1)
TOL (5b-3 or 5c-3)	Enter tolerance for multiple grids at the crack tip. (Real; Default = 0.0)
GEOM (Start of 5c)	Enter the string GEOM if ITYPE=3, otherwise skip this line (Character)
RAD (7c-1)	Radius of the rigid region to be found. (Real; no Default) Enter only if ITYPE=3.
CYLIN (7c-2)	Relative length of cylinder for the path search. (Real; no Default) Enter only if ITYPE=3.

Remarks:

1. This entry corresponds to Marc's LORENZI model definition option.
2. (i,j) corresponds to Marc Vol C LORENZI entry ith datablock jth field.
3. Repeat the LORENZI as many times as necessary to define all cracks for which J-Integrals should be evaluated.

4. If the DIRECT input is used, repeat the DIRECT line as many times as necessary to define different regions from which crack growth should be estimated. In some cases it is informative to calculate crack growth for several regions and compare the results of these calculations. For example, in the model shown below, the crack tip is at the center on the left side. It might be informative to compare crack growth using a region including all nodes in the first ring (of node) around the crack with that produced by two rings.
5. For the TOP and GEOM methods, only one line per crack should be entered.
6. The continuation line for GEOM is not required except for 3D cracks.
7. For the previous examples, the crack tip is at grid 1 in the following figure. For the direct input, SET3 with ID 101 would reference all grids on and within the ring closest to the crack tip. SET3 with ID 102 would reference all grids on and within the two rings closest to the crack tip. Both examples provide identical results. The topology input is considerably simpler and therefore recommended.
8. The definition of shift vector is the function q_1 in the equation and text described below.



Background

The J-integral evaluation is based upon the domain integration method. A direct evaluation is not very practical in a finite element analysis due to the difficulties in defining the integration path. In the domain integration method for two dimensions, the line integral is converted into an area integration over the area inside the path. This conversion is exact for the linear elastic case and also for the nonlinear case if the loading is proportional, that is, if no unloading occurs. By choosing this area as a set of elements, the integration is straightforward using the finite element solution. In two dimensions, the converted expression is

$$\bar{J} = \int_A \left(\sigma_{ij} \frac{\partial u_j}{\partial x_1} - W \delta_{1i} \right) \frac{\delta q_1}{\delta x_i} dA$$

for the simplified case of no thermal strains, body forces or pressure on the crack faces. A is the area inside the integration path and q_1 is a function introduced in the conversion into an area integral. The function can be chosen fairly generally, as long it is equal to one at the crack tip and zero on. The form of the function chosen is that it has the constant value of one at all nodes inside, and decreases to zero over the outermost ring of elements. It can be interpreted as a rigid translation of the nodes inside while the nodes on remain fixed. Thus, the contribution to the above equation comes only from the elements in a ring away from the crack tip. This interpretation is that of virtual crack extension and this method can be seen as a variant of such a technique, although it is extended with the effects of thermal strains, body forces, and pressure on the crack faces. The set of nodes moved rigidly is referred to as the *rigid region* and the function q_1 in the above equation as the **shift vector**. For the evaluation of the J-integral the direction of the shift vector is simply the x axis in the local crack tip system.

In three dimensions, the line integral becomes an area integral where the area is surrounding a part of the crack front. In this case, the selection of the area is even more cumbersome than in two dimension. The converted integral becomes a volume integral which is evaluated over a set of elements. The rigid region is a set of nodes which contains a part of the crack front, and the contribution to the integral comes from the elements which have at least one but not all its nodes in the rigid region.

LSEQ**Static Load Set Definition**

Defines a sequence of static load sets.

Format:

1	2	3	4	5	6	7	8	9	10
LSEQ	SID	EXCITEID	LID	TID					

Example:

LSEQ	100	200	1000	1001					
------	-----	-----	------	------	--	--	--	--	--

Descriptor	Meaning
SID	Set identification of the set of LSEQ entries. See Remark 5. (Integer > 0)
EXCITEID	The EXCITEID set identification assigned to this static load vector. See Remark 5. (Integer > 0)
LID	Load set identification number of a set of static load entries such as those referenced by the LOAD Case Control command. (Integer > 0 or blank)
TID	Temperature set identification of a set of thermal load entries such as those referenced by the TEMP(LOAD) Case Control command. (Integer > 0 or blank)

Remarks:

1. LSEQ will not be used unless selected in the Case Control Section with the LOADSET command.
2. The number of static load vectors created for each superelement depends upon the type of analysis. In static analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all LSEQ entries in the Bulk Data; in dynamic analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data.
3. EXCITEID may be referenced by CLOAD, RLOAD1, RLOAD2, TLOAD1, and TLOAD2 entries in order to apply the static load in nonlinear, static and dynamic analysis.
4. Element data recovery for thermal loads is not currently implemented in dynamics.
5. The SID-EXCITEID number pair must be unique with respect to similar pairs on all other LSEQ entries in the Bulk Data.
6. In a nonsuperelement analysis, LID and TID cannot both be blank. In superelement analysis, they may both be blank as long as static loads are prescribed in the upstream superelements.
7. It is no longer necessary to employ LOADSET/LSEQ data to specify static loading data for use in dynamic analysis. In the absence of LSEQ Bulk Data entries, all static loads whose load set IDs match the EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data are automatically processed.

Entries M - O

MACREEP

AUTO CREEP Iteration Control in SOL 600

Controls a transient creep analysis. This entry or the MTCREEP entry is required if ITYPE is not zero on the MPCREEP entry in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
MACREEP	ID	Tinc	Ttot	Nmax	NIM	NIK	Tstab	VV1	
	VV2	VV3	IABS						

Example:

MACREEP	1	1.0	1000.	99999	5	1		.01	
	100.	.05	0						

Descriptor	Meaning
ID	Identification number of a matching Case Control NLPARM command (for statics) or command (for dynamics). (Integer; Required Field)
Tinc	Suggested time increment for creep analysis - The specified value will automatically be adjusted. (Real; Required Value) (2,1)
Ttot	Total time (final time) of the creep analysis. (Real; Required Value) (2,2)
Nmax	Maximum number of time increments allowed in the creep analysis (Integer; Default = 50) (2,3)
NIM	Maximum number of iterations allowed to modify a time step (Integer; Default = 5) (2,4)
NIK	Number of increments between stiffness matrix updates (Integer; Default = 1) (2,5)
Tstab	Stable time step, if known, must be entered for viscoplasticity (Real; Required field for viscoplasticity) (2,7)
VVT	Tolerance value #1. (Real; see below for Defaults) (3,1)
	IABS=0 Enter the tolerance on the creep strain increment divided by the elastic strain (Default = 0.5).
	IABS=1 Enter the maximum creep strain increment allowed. (Default = 0.1)
VV2	Tolerance value #2. (Real, see below for defaults) (3,2)
	IABS=0 Enter the tolerance on the stress change divide by the total stress. (Default = 0.1)
	IABS=1 Enter the maximum stress increment. (Default = 100.0)

Descriptor	Meaning
VV3	Tolerance on low stress point cutoff. Points with a stress lower than this ratio relative to the maximum stress in the structure are not used in the creep tolerance checking (Real; Default = 0.05) (3,3)
IABS	Flag controlling relative or absolute convergence testing. (Integer; Default = 0) (3,5)
0	Relative checking is used
1	Absolute checking is used

Remarks:

1. This entry maps to Marc's AUTO CREEP entry.
2. This entry will be used instead of AUTO STEP or AUTO INCREMENT entries in the Marc file for creep analysis. It is suggested that if this entry is used, NLAUTO and NLSTRAT should not be specified (and will be ignored if entered).
3. Bulk Data entry MPCREEP must also be entered in addition to this entry.
4. (i,j) refer to Marc's AUTO CREEP (data block, field).
5. Bulk Data entries MACREEP and MTCREEP should not be entered in the same input file.

MARCIN**Inserts a Text String in Marc from SOL 600**

Inserts a text string directly in the Marc input file used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MARCIN	ILOC	ICONT			String				

Example:

MARCIN	-1	0	feature,2301	
--------	----	---	--------------	--

Descriptor	Meaning
ILOC	Identification of the location in the Marc file where the string will be placed. (Required; Integer)
	-1 String is placed at end of Marc's Parameter Section
	0 String is placed at the end of Marc's Model Definition Section
	N (N > 0) String is placed in load case N after the AUTO STEP, AUTO INCREMENT, etc. entry.
ICONT	(Required; Integer)
	-1 String is used as the name of an include file which contains all of the direct Marc input data for that portion of Marc (parameter, model definition, etc.). Only one include file per Marc Section is allowed. The string must be entirely in lower case and the include file must be in lower case, except for PC systems where the case does not matter.
	0 Entry is not a continuation of previous MARCIN entry.
	1 Entry is a continuation of previous MARCIN entry and the strings will be placed one after the other on the same Marc line
String	(Required; Character)
	0 The desired text string. The string is limited to 48 characters per entry. Multiple entries will be placed in the order entered within each Marc location.
	-1 String is an include file name limited to 48 characters.

Remarks:

1. Standard Nastran fields 4-9 are ignored for this entry. The string may be entered anywhere within fields 4-9 and will be translated directly to Marc.
2. If a long line for the Marc data is required, enter as many MARCIN entries as necessary to describe the entire Marc string using ICONT=1 for each except the first.

3. The total string length including continuation lines is limited to 160 characters.
4. As many MARCIN entries as necessary may be entered to define all desired input.
5. Each entry in the Nastran data file must start with the MARCIN header. Each line in an include file will be translated directly to Marc (there should be no MARCIN, ILOC or ICONT information in the include file(s)).
6. If the direct Marc input is placed in include file(s), separate files are necessary for each portion of Marc (parameter, model definition, etc.) requiring direct input.
7. As part of the Nastran input process, all strings are converted to uppercase. The internal Marc translator will convert them to lower case. For input entered without include files, this will normally make any difference. For include file names, file names must be entirely lower case for computer systems that are case sensitive.
8. MARCIN entries are not always recognized in restart runs and are not recommended.
9. ILOC greater than zero is not available for SOL 600 heat transfer analysis or structural analysis if table-driven loads are requested (for example, by specifying param,marctott,1).
10. For ILOC > 0 if STRING starts with "\$", "comment", or "COMMENT", the string will usually be skipped and no continuation lines should be entered.

MARCOUT

Selects Data Recovery Output in SOL 600

Selects output to be saved on the Marc t16 end/or t19 file(s) used in SOL 600 only. This entry is available using the small field format only and should normally be used only when post-processing using the T16 file is to be done (in other words, it should not normally be used if OUTR on the SOL 600 entry is selected - see PARAM,MARROUTT).

Format:

1	2	3	4	5	6	7	8	9	10
MARCOUT	LAYCODE	IO1	IO2	IO3	IO4	IO5	IO6	IO7	
	IO8	IO9	etc.						

Example:

MARCOUT	125	E11	E21	E41	N1	N2	N35		
---------	-----	-----	-----	-----	----	----	-----	--	--

Descriptor	Meaning
LAYCODE	Specifies which shell or beam layers are to be output. (Integer; Default is all layers from 1 to the value of PARAM,MARCSLHT) This parameter also applies to composites whether made of shells or solids. For options less than 100, output will be all integration points. For options 100-200, output will only be at the element center. See Remark 13.
0	All shell and beam layer results from 1 to MARCSLHT are output, solid output is available only at the centroid. This option also requires PARAM,MROUTLAY to be set to the maximum numbers of layers desired (MARCSLHT and MROUTLAY should be set to the same values).
1	Top and Bottom layer shell, beam and solid bottom layer results only are output (Marc layer codes 1 and 1000).
2	Top, Bottom and Middle layer results for shell beams and centroid results for solids only are output (Marc layer codes 1, 5000 and 10000)
3	Top and Bottom layer shell, beam and solid bottom layer results are output. In addition, composite layer output listed in the Bulk Data entry, MLAYOUT will be included.
100	Same as option 0 except output is only at the center of each layer
101	Same as option 1 except output is only at the center of each layer
102	Same as option 2 except output is only at the center of each layer
103	Same as option 3 except output is only at the center of each layer.
IO(i)	Indicates the type of Marc output requested according to the following table:

Element Output	Description
-9999	If -9999 is entered in field 3, the defaults for the post file of the Marc version being run will be used. Fields 4-9 must be blank and no continuation lines should be entered. No OUTR options on the SOL 600 entry are available with this option.
"E-USER"*	First user-defined element post code(s) are generated by user subroutine plotv.f
"E-USER1"*	2 nd user-defined element post code(s)
"E-USER2"*	3 rd user-defined element post code(s)
"E-USER3"*	4 th user-defined element post code(s)
"E-USER99"*	100 th user-defined element post code(s)

*These outputs are only available in the t16 file, not in op2, xdb, f06, punch.

A maximum number of 100 user-defined element post codes may be entered for SOL 600).

E1 to E7	strain components, VM strain
E8	Equivalent plastic strain
E9	Total Temperature (combined heat/structural analysis)
E10	Incremental Temperature (combined heat/structural analysis)
E11 to E17	stress components, VM stress, Mean stress
E18	Mean normal stress
E20	Element Thickness (thickness can change vs time)
E21 - E27	plastic strain components, VM plastic strain
E28	Plastic strain rate
E29	Value of second state variable if applicable
E31 to E37	Creep strain components and equivalent creep strain
E38	Total swelling strain
E39	Value of third state variable if applicable
E41 to E47	Cauchy stress components, VM Cauchy stress
E49	Thickness strain for plane stress, Mooney or Ogden material
E58	Elastic strain energy density
E59	Equivalent stress/yield stress
E60	Equivalent stress/yield stress at current temperatures
E48	Strain Energy Density
E68	Plastic strain energy density
E69	Current Volume
E71 to E76	Components of thermal strain (separated from total strain)
E78	Original Volume

Element Output	Description
E79	Grain size if applicable
E80	Damage indicator for Cockcroft-Latham, Oyane and principal stress criteria
E81 to E86	Components of cracking strain if applicable
E91 to E107	Failure indices associated with failure criteria from MATF
E108 to E109	Interlaminar shear for thick composite shells
E110	Interlaminar shear bond index for thick composite shells
E111 to E116	Components of stress in "preferred coordinate system"
E121 to E126	Elastic strain components
E127	Equivalent elastic strain
E128	Major engineering strain
E129	Minor engineering strain
E171	Porosity
E172	Void ratio
E173	Pore pressure
E174	Preconsolidation pressure
E175	Equivalent viscoplastic strain rate (powder material)
E176	Relative density (powder material)
E177	Void volume fraction (damage model)
E178	Lemaitre damage factor
E179	Lemaitre relative damage
E180	Total temperature (same as E9)
E181 to E183	Components of temperature gradient
E184 to E186	Components of flux
E241	Gasket pressure if applicable
E242	Gasket closure if applicable
E243	Plastic Gasket closure if applicable
E244	Exponential powder parameter kappa
E245	Exponential powder parameter x
E251 to E253	Components of interlaminar normal stress in basic coordinate system
E254 to E256	Components of interlaminar shear stress in basic coordinate system
E257	Interlaminar shear bond index for composite solids
E216	Beam axis or Orientation of CBUSH/CFAST elements
E261	Beam Axis

Element Output	Description
E264-E269	Beam/Bar or Bush Element Forces
	The next 6 lines are for CBUSH Force output (CBUSH stress/strain output and can be obtained with standard stress/strain codes if the stress/strain recovery. Coefficients are defined on the PBUSH entries.)
E264	CBUSH Axial Force (local bush coord sys)
E265	CBUSH Moment Mxx (local bush coord sys)
E266	CBUSH Moment Myy (local bush coord sys)
E267	CBUSH Shear Force Vxy (local bush coord sys)
E268	CBUSH Shear Force Vyx (local bush coord sys)
E269	CBUSH Torque (local bush coord sys)
E270	Bimoment (for beam element)
E301	Total strains tensor
E311	Stress tensor
E321	Plastic strain tensor
E331	Creep strain tensor
E341	Cauchy stress tensor
E371	Thermal strain tensor
E381	Cracking strain tensor
E391	Stress in composite ply ("preferred") direction
E401	Elastic strain tensor
E411	Stress tensor in basic coordinate system
E421	Elastic strain tensor in basic coordinate system
E431	Plastic strain tensor in basic coordinate system
E441	Creep strain tensor in basic coordinate system
E451	Velocity strains (for fluids)
E461	Elastic strain in preferred direction tensor
E471	Rebar stress components in basic coordinate system in undeformed config if applicable
E481	Rebar stress components in basic coordinate system in deformed config if applicable
E487	Rebar angle
E501	Interlaminar normal stress
E511	Interlaminar shear stress
E531	Volume fraction of Martensite

Element Output	Description
E541	Phase transformation strain tensor
E547	Equivalent phase transformation strain
E548	Equivalent TWIN strain
E549	Equivalent TRIP strain
E551	Equivalent plastic strain in a multiphase aggregate
E552	Equivalent plastic strain in Austenite
E553	Equivalent plastic strain in Martensite
E557	Yield stress of multiphase aggregate
E610 to E617	Strength ratios based on MATF failure modes
E641	Generalized strain curvatures tensor
E661	Generalized stress moments tensor
E681	True strain tensor for continuum elements
E691	Element orientation vector 1
E694	Element orientation vector 2
E697	Layer orientation angle
E704	Real harmonic axial force
E705	Real harmonic Moment Mxx
E706	Real harmonic Moment Myy
E707	Real harmonic Shear force Vxy
E708	Real harmonic Shear force Vy়
E709	Real harmonic Torque
E710	Real harmonic Bimoment
E714	Imaginary harmonic axial force
E715	Imaginary harmonic Moment Mxx
E716	Imaginary harmonic Moment Myy
E717	Imaginary harmonic Shear force Vxy
E718	Imaginary harmonic Shear force Vy়
E719	Imaginary harmonic Torque
E720	Imaginary harmonic Bimoment
E721	Cauchy Stress Tensor in Preferred (Material) Coordinate System
E731	Curvature Tensor (for shell elements)
E741	Moment Tensor (for shell elements)

Nodal Output	Description
"N-USER"*	First user-defined nodal post cod are generated by user subroutine upstnd.f
"N-USER1"*	2 nd user-defined nodal post cod are generated by user subroutine upstnd.f
"N-USER2"*	3 rd user-defined nodal post cod are generated by user subroutine upstnd.f
"N-USER3"*	4 th user-defined nodal post cod are generated by user subroutine upstnd.f
"N-USER4"*	5 th user-defined nodal post cod are generated by user subroutine upstnd.f
"N-USER99"*	100 th user-defined nodal post cod are generated by user subroutine upstnd.f, etc.
*User-defined outputs are only available in the t16 file, not in op2, xdb, f06, punch. A maximum of 100 user-defined nodal post codes may be entered for SOL 600.	
N1,N2	displacements, rotations
N3,N4	Applied forces & moments
N5,N6	reaction forces & moments
N7	Fluid velocity if applicable
N8	Fluid pressure if applicable
N9	External fluid force if applicable
N10	Reaction fluid force if applicable
N11	Sound pressure if applicable
N12	External sound pressure if applicable
N13	Reaction sound pressure if applicable
N14	Temperature
N15	External heat flux
N16	Reaction heat flux
N23	Pore pressure if applicable (soil analysis)
N24	External mass flux if applicable
N25	Reaction mass flux if applicable
N26	Bearing pressure if applicable
N27	Bearing force if applicable
N28,N29	Velocity
N30,N31	Acceleration
N32,N33	Modal mass and rotational modal mass
N34,N35	Contact normal stress/force
N36,N37	Contact friction stress/force
N38,N39	Contact status, Contact touched body

Nodal Output	Description
N40	Herrmann variable
N46,N48	Tying force and moments
N47	Coulomb Force
N49	Generalized nodal stress
N50	Generalized nodal strain
N51,N52	Inertia relief force and moment
N53	J-Integral
N54	Stress Intensity, Mode I
N55	Stress Intensity, Mode II
N56	Stress Intensity, Mode III
N57	Energy release
N58	Energy release rate I
N59	Energy release rate II
N60	Energy release rate III
N62	Crack system local X
N63	Crack system local Y
N64	Crack system local Z
N65	Contact Separation Distance
N66	Normal Breaking Index
N67	Tangential Breaking Index
N68	Total Breaking Index
N69	Normal Delamination Index
N70	Tangential Delamination Index
N71	Total Delamination Index
N73	Glue deactivation status
N74	VCCT Failure Index
N76	Lorentz Force
N77	Wear Index
N78	Wear Rate

Remarks:

1. MARCOUT is only available when Marc is executed from within Nastran and controls what results are available in the Marc t16 file. All elements or nodes of each type selected will be placed on the t16 file (in other words, it is not possible to control this output by selecting various sets). Some, but not all, of the results in the t16 file may be used to obtain .op2, .xdb, punch or .f06 results output by specifying OUTR options on the SOL 600 Executive Control statement. The type of results which may be placed in the .op2, .xdb, punch or .f06 files are the same as those available in SOL 106 or 129. Other types of results are only available for postprocessing using the t16 file. Whenever possible, i.e., if your gui allows it, the t16 file should be used instead of .op2, .xdb, punch, or .f06 files for postprocessing.
2. Values such as E1, E21 correspond to Marc's postcodes 1 and 21, respectively.
3. Outputs produced by MARCOUT are the same for all subcases, load steps, iterations, etc.
4. The MARCOUT entry may be repeated as many times as desired, or all entries may be placed on continuation lines.
5. For entries E1, E11 and E21 corresponding entries E2-E7, E12-E17 and E22-E27 will be generated automatically. These terms correspond to 3 normal stress (or strain) and 3 shear stress/strain values plus the equivalent von Mises value. See Marc volume C POST description for more details.
6. If this entry is not used, the following defaults are entered automatically: E301, E341, E47, N1, N2, N35, N37, N39. When creep or heat transfer is present, additional items are added appropriately. If the MARCOUT entry is entered, only those items specified will be output.
7. Only displacements, rotations, Cauchy stresses and one type of strains (total, plastic or elastic) may be transferred to the Nastran database.
8. Displacements, at least one stress tensor and one strain tensor must be selected if any OUTR options are to be used.
9. For SOL 600, Nastran Case Control commands such as SET ID=, DISP=, STRESS=, and STRAIN= only control the output in the .OP2, .XDB, punch, .F06 and jid.marc.out files. The Case Control requests do not affect the t16 output.
10. Default MARCOUT options are sufficient for most needs and it is recommended that the MARCOUT entry only be employed by advanced users.
11. If some (but not all) forces (E264-E269) are specified, the missing ones will be added automatically since the t16op2 conversion requires all be present (this capability is available starting in MD Nastran R2 and the MSC Nastran 2006 release, prior to that, all needed to be specified if any OUTR options were requested.)
12. LAYCODE values 100-103 will automatically set PARAM,MARCCENT,1
13. For LAYCODE=1, if op2.xdb,f06 or punch output is requested element stress/strain results which normally are output at the center of the element will be output on the bottom surface of the element. For composite solids, if output at all layers is desired, postprocessing using the t16 file is required.
14. Additional outputs described in Marc Vol C (POST Section) may be available for certain types of SOL 600 analyses.

MARPRN**Defines "Print" Options for SOL 600**

This option corresponds to Marc's PRINT parameter which controls a variety of output and other information. for the most used options, PARAM,MARCPRN should be used.

Format:

1	2	3	4	5	6	7	8	9	10
MARPRN	IP1	IP2	IP3	IP4	IP5	IP6	IP7	IP8	
	IP9	-etc.-							

Example:

MARPRN	2	5	39						
--------	---	---	----	--	--	--	--	--	--

Descriptor	Meaning
IPi	"Print" code from list shown below (Integer; no Default). Include as many options as desired
1	Output element stiffness matrices (this also prints out the shell surface metric for doubly curved shells 4, 8, and 24), consistent mass matrix, and equivalent nodal loads. CAUTION: This produces significant output.
2	Output of the matrices used in tying. (TYING, SERVO LINK, UFORMS.)
3	Forces the solution of a nonpositive definite matrix. This is only recommended for the AUTO INCREMENT option to pass collapse points in the collapse analysis. This can be entered on the CONTROL option.
5	To obtain additional information concerning gap convergence. In contact analysis, set to 5 to obtain information concerning nodes touching or separating from surfaces and also to print out the maximum residual and reaction forces.
6	To obtain output of nodal value array during rezoning.
8	To obtain incremental displacements in local system in contact problems.
9	To obtain latent heat output.
10	To obtain the stress-strain relation in the local coordinate system.
11	To obtain additional information on the interlaminar stress calculation.
12	To output the right-hand side and solution vector. CAUTION: This produces significant output.
13	To obtain additional information regarding CPU resources used.
15	To obtain additional information regarding surface energy balances.
20	To obtain information regarding the evaluation of tables.

Descriptor	Meaning
21	To obtain information about application of kinematic boundary conditions when table input is used.
22	To obtain information about distributed loads, point loads, films, foundations, and initial conditions when table input is used.
26	To print additional information regarding sink points.
27	To obtain reaction forces at tied nodes.
28	To obtain additional information about convective terms in heat transfer and fluid analysis.
34	To print a description of what independent variables may be used with a physical quantity.
36	To obtain CASI solver debug information (has the least details).
37	To obtain CASI solver debug information (has more details).
38	To obtain CASI solver debug information (has the most details).
39	To obtain detailed information about memory allocation.
43	To obtain information about VCCT
44	To obtain information during progressive failure calculations

Remark:

1. See PARAM,MARCPRN for a simpler way to enter the most used print options.

MASSET**Mass Combination Definition**

The MASSSET bulk data entry defines a linear combination of mass cases to form the subcase-dependent mass which is selected by associated MASSSET case control. The subcase mass is defined as follows by the MASSSET bulk:

$$M_{ID} = S_0 \sum_i S_i M_i$$

Where, i refers to the MASSID qualifier: $i = 0$ is the base mass case, $i > 0$ are the incremental mass cases. ID's of mass combinations defined by MASSSET bulk data entry and mass increments defined by MASSID bulk data sections should be unique.

Format:

1	2	3	4	5	6	7	8	9	10
MASSET	ID	S0	S1	ID1	S2	ID2	S3	ID3	
	S4	ID4							

Example:

MASSET	11	1.0	1.0	0	1.0	101			
--------	----	-----	-----	---	-----	-----	--	--	--

Descriptor	Meaning
ID	Set identification number referenced by MASSSET case control. (Integer > 0; Required)
S0	The overall scale factor for the linear combination (Real >= 0.0; Default = 0.0)
Si	The scale factor for the ith mass increment (Real >= 0.0; Default = 0.0)
IDi	The MASSID for the ith mass increment (Integer > 0; No Default)

MATBV**Material Bulk Viscosity - SOL700**

Defines the bulk viscosity for materials. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATBV	MID	BULKTYP	BULKQ	BULKL	CSCALE	DMPFAC	IIMMREL		

Example:

MATBV	7	1.2							
-------	---	-----	--	--	--	--	--	--	--

Descriptor	Meaning
MID	Material identification. See Remark 1. (Integer > 0, Required.)
BULKTYP	Bulk viscosity type. (Character: default=DYNA) DYNA Standard DYNA3D model. DYTRAN Enhanced DYNA model.
BULKL	Linear bulk-viscosity coefficient. (Real \geq 0.0; default=1.0)
BULKQ	Quadratic bulk-viscosity coefficient. (Real \geq 0.0; default=0.0)
CSCALE	When this material model is used with MEMB shell elements (See PSHELL1), the compressive stresses in the principal directions will be scaled with this factor. CSCALE=0.0 results in a tension only material. See Remark 2. (Real \geq 0.0; default=1.0)
DMPFAC	When this material model is used with MEMB shell elements, damping is applied to the stresses. DMPFAC = 0.05 results in 5% damping applied to membrane stresses. See Remark 2. (Real \geq 0.0; default=0.0)
IIMMREL	Relaxation factor used with the Initial Metric Method. This option is only used when this material model is used with MEMB shell elements and the IMM method is activated. See Remark 3. (0.0 < Real < 1.0; default=1.0e-3)

Remarks:

1. The material number must refer to a basic material definition such as MAT1/MAT2/MAT8/MATORT/MATHE.
2. For air bag modeling the following values of CSCALE and DMPFAC are suggested:
 $CSCALE = 0.1$
 $DMPFAC = 0.05 \text{ to } 0.20$
3. The Initial Metric Method relaxation factor is used to slow down the expansion of the membrane elements during the calculation. The default is sufficient in most simulations.

MAT1**Isotropic Material Property Definition**

Defines the material properties for linear isotropic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
	ST	SC	SS	MCSID					

Example:

MAT1	17	3.+7		0.33	4.28	6.5-6	5.37+2	0.23	
	20.+4	15.+4	12.+4	1003					

Descriptor	Meaning
MID	Material identification number. (Integer > 0)
E	Young's modulus. (Real ≥ 0.0 or blank)
G	Shear modulus. (Real ≥ 0.0 or blank)
NU	Poisson's ratio. (-1.0 < Real ≤ 0.5 or blank)
RHO	Mass density. See Remark 5. (Real)
A	Thermal expansion coefficient. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 9. and 10. (Real; Default = 0.0 if A is specified.)
GE	Structural element damping coefficient. See Remarks 8., 9., and 4. (Real)
ST, SC, SS	Stress limits for tension, compression, and shear are optionally supplied, used only to compute margins of safety in certain elements; and have no effect on the computational procedures. (Real ≥ 0.0 or blank). See Remark 14.
MCSID	Material coordinate system identification number. Used only for PARAM,CURV processing. See Parameters, 783. (Integer ≥ 0 or blank)

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.
2. The following rules apply when E, G, or NU are blank:
 - E and G may not both be blank.
 - If NU and E, or NU and G, are both blank, then both are set to 0.0.

- If only one E, G, or NU is blank, then it will be computed from the equation:

$$E = 2 \cdot (1 + NU) \cdot G$$
. If this is not desired, then the MAT2 entry is recommended. If E, G, or NU are made temperature dependent by the MATT1 entry, then the equation is applied to the initial values only. Caution: Inconsistent E and G can result in NU >.5 which for three dimensional elasticity results in negative stress for positive strain. This can cause large max ratios.
3. If values are specified for all of the properties E, G, and NU, then it is recommended that the following relationship be satisfied:

$$\left| 1 - \frac{E}{2 \cdot (1 + NU) \cdot G} \right| < 0.01$$

If this relationship is not desired, then the MAT2 entry is recommended.

It should also be noted that some of the properties are not applied in the stiffness formulation of certain elements as indicated in [Table 22](#). Therefore, it is recommended that only the applicable properties be specified for a given element.

Table 22 Material Property Usage Versus Element Types

Element Entry	E	NU	G
CROD CBEAM CBAR	Extension and Bending	Not Used	Torsion Transverse Shear
CQUADi CTRIAi CCONEAX	Membrane, including In-plane Shear, and Bending		Transverse Shear
CSHEAR	Not Used		Shear
CRAC2D	All Terms		Not Used
CHEXA CPENTA CTETRA CRAC3D	All Terms		Not Used
CTRIAX6	Radial, Axial, Circumferential	All Coupled Ratios	Shear

- MAT1 materials may be made temperature-dependent by use of the MATT1 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
- The mass density RHO will be used to compute mass for all structural elements automatically.
- Weight density may be used in field 6 if the value 1/g is entered on the PARAM,WTMASS entry, where g is the acceleration of gravity (see [Parameters, 783](#)).
- MCSID must be nonzero if PARAM,CURV is specified to calculate stresses or strains at grid points on plate and shell elements only.

8. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 , by 2.0.
9. TREF and GE are ignored if the MAT1 entry is referenced by a PCOMP/PCOMPG entry.
10. TREF is used in two different ways:
 - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.
 - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection.

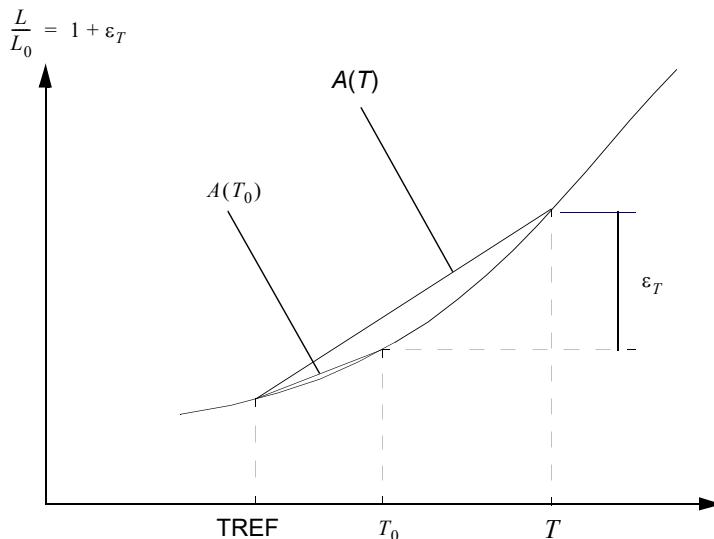


Figure 9-105 Use of TREF in Calculation of Thermal Loads

$$\varepsilon_T = A(T) \cdot (T - TREF) - A(T_0) \cdot (T_0 - TREF)$$

where T is requested by the TEMPERATURE(LOAD) command and T_0 is requested by the TEMPERATURE(INITIAL) command.

Notes:

- A is a secant quantity.
- TREF is obtained from the same source as the other material properties; e.g., ASTM, etc.
- If $A(T)$ constant, then $\varepsilon_T = A \cdot (T - T_0)$
- If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).

11. In nonlinear static analysis (SOL 106) the QUAD4 and TRIA3 thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPISILONT,INTEGRAL' in bulk data. See [Parameters, 783](#).
12. For SOL 600, E must not be blank or zero.
13. Negative values for ST, SC, and SS lead to no margins of safety being computed.
14. ST, SC, and SS are not used in SOL 400 with advanced elements or SOL 600. Use MATF instead.

MAT1A**Additional Material Properties for MAT1- SOL700**

Specifies additional the material properties of MAT1 for solid elements. Use SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1A	MID	EID							

Example:

MAT1A	101	201							
-------	-----	-----	--	--	--	--	--	--	--

Descriptor	Meaning
MID	Identification number of MAT1. (Integer > 0; required.)
EID	EOSxxx ID. (Integer > 0; required.)

Remarks:

1. MAT1A always required for solid elements when MATEP is used. However, when RYIELD option of MATEP is set to VMISES, ORTHOCR or SOIL, MATEOS is not required.

MAT1F**Isotropic Material Frequency Dependence**

Specifies frequency-dependent material properties on MAT1 entry fields via TABLEDi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1F	MID	E	G	NU	RHO			GE	

Example:

MAT1F	33	15							
-------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)
E	Identification number of a TABLEDi entry for the Young's modulus. (Integer > 0 or blank)
G	Identification number of a TABLEDi entry for the shear modulus. (Integer > 0 or blank)
NU	Identification number of a TABLEDi entry for the Poisson's ratio. (Integer > 0 or blank)
GE	Identification number of a TABLEDi entry for the structural damping coefficient. (Integer > 0 or blank)
RHO	Identification number of a TABLEDi entry for the mass density. (Integer > 0 or blank). This table is only used for isotropic poroelastic material with BEGIN BULK TRMC and is applicable to that trim component only.

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT1 entry referenced in field 2. The value in a particular field of the MAT1 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT1 entry.
- The user is reminded that on the MAT1 entry, if any one of the entries E, G, or, NU is left blank, they are connected by the relationship $G=E/(2(1+NU))$. Therefore, this entry requires that if any E, G, or NU is to be frequency dependent then Table references must be present for all three of the E, G, NU even if this requires a table of constant values.
- Table references must be present for each item that is frequency dependent.
- IF GE=0.0 on corresponding MAT1 then GE table must be blank or 0.

MAT2**Shell Element Anisotropic Material Property Definition**

Defines the material properties for linear anisotropic materials for two-dimensional elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT2	MID	G11	G12	G13	G22	G23	G33	RHO	
	A1	A2	A3	TREF	GE	ST	SC	SS	
	MCSID	GE11	GE12	GE13	GE22	GE23	GE33		

Example:

MAT2	13	6.2+3		6.2+3		5.1+3	0.056		
	6.5-6	6.5-6		-500.0	0.002	20.+5			
	1003								

Descriptor	Meaning
MID	Material identification number. See Remark 1. (Integer > 0)
Gij	The material property matrix. (Real) See Remarks 4., 5., 6., and 9.
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient vector. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 10., 11. and 14. (Real or blank)
GE	Structural element damping coefficient. See Remarks 8., 10., and 13. (Real)
ST, SC, SS	Stress limits for tension, compression, and shear are optionally supplied (these are used only to compute margins of safety in certain elements) and have no effect on the computational procedures. (Real or blank). See Remark 17.
MCSID	Material coordinate system identification number. Used only for PARAM, CURV processing. See Parameters, 783. (Integer \geq 0 or blank)
GEij	Structural damping matrix. See Remark 17. (Real)

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.
2. MAT2 materials may be made temperature dependent by use of the MATT2 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
3. The mass density, RHO, will be used to automatically compute mass for all structural elements.

4. The convention for the Gij in fields 3 through 8 are represented by the matrix relationship

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} - (T - T_0) \begin{Bmatrix} A1 \\ A2 \\ A3 \end{Bmatrix}$$

The G matrix, including G13 and G23 is an extension to conventional orthotropic plans stress behavior that allows membrane-shear coupling. If the MAT2 is used for plane stress elements or referenced as MID1 or MID2 in a PSHELL, the G matrix must be invertible and positive definite.

5. When MAT2 is referenced as a MID3 in a PSHELL, then it is used as:

$$\begin{aligned} \tau_{13} &= \begin{bmatrix} G_{11} & G_{12} \\ G_{12} & G_{22} \end{bmatrix} \begin{Bmatrix} \gamma_{13} \\ \gamma_{23} \end{Bmatrix} \\ \tau_{23} & \end{aligned}$$

This may lead to user warning message 6134 which may be ignored. See *The NASTRAN Theoretical Manual*, Section 4.2.

6. When MAT2 is referenced as a MID4 in a PSHELL, then only G₁₃ and G₂₃ are required. One will observe a user warning message 9994, The Material Property Matrix is not positive definite which may be ignored.
7. MCSID must be nonzero if PARAM,CURV is specified to extrapolate element centroid stresses or strains to grid points on plate and shell elements only. CQUAD4 element corner stresses are not supported by PARAM,CURV.
8. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.
9. Unlike the MAT1 entry, data from the MAT2 entry is used directly without adjustment of equivalent E, G, or NU values.
10. TREF and GE are ignored if this entry is referenced by a PCOMP/PCOMPG entry.
11. TREF is used in two different ways:
 - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Remark 10. in the MAT1 description.
 - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.
12. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).
13. PCOMP/PCOMPG entries generate MAT2 entries starting from 10,000,001 onwards. Explicitly specified MAT2 IDs must not conflict with internally generated MAT2 IDs. Furthermore, if MID is greater than 400,000,000 then A1, A2, and A3 are a special format. They are [G4] · [α4] not [α4]. If MIDs larger than 99999999 are used, PARAM,NOCOMPS,-1 must be specified to obtain stress output.

14. In nonlinear static analysis (SOL 106) the QUAD4, TRIA3, QUADR, and TRIAR thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPISILONT,INTEGRAL' in bulk data. See [Parameters, 783](#).
15. Negative values for ST, SC, and SS lead to no margins of safety being computed.
16. If the MAT2 is referenced by the PCOMP/PCOMPG entry, the transverse shear flexibility for the referenced lamina is zero.
17. Defines structural damping matrix data for advanced composites. If the GE_{ij} values are present, then Nastran will ignore the GE value given on the first continuation entry field (6) and the GE entry given in field (8) of the PCOMP/PCOMPG entry and use the given GE_{ij} values. The rational of these entries is that with composite materials made of a viscoelastic resin system, the damping matrix does not scale with the stiffness matrix. For example, in the directions where you have more fibers, the stiffness is higher because you have more fibers but the damping coefficient is smaller because you have less resin matrix which is the main source of damping. To use this feature in composites, the PCOMP/PCOMPG MIDI entries must refer to MAT2 entries.

For any ply that does not point to a MAT2 with extended GE_{ij}, the GE value will be taken from the GE field of the corresponding MAT_i entry (MAT1 field 9, MAT2 field 6 of first continuation, MAT8 field 2 of second continuation.).

The damping when extended GE_{ij} is in effect is applied in element material routines by forming the product GE_{ij} * G_{ij} when forming the K4 damping matrix. For any ply not using an extended GE_{ij} of MAT2 the product GE*G_{ij} is used.

MAT2F**Shell Element Anisotropic Frequency Dependence**

Specifies frequency-dependent material properties on MAT2 entry fields via TABLEDi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT2F	MID	G11	G12	G13	G22	G23	G33		
					GE				
		GE11	GE12	GE13	GE22	GE23	GE33		

Example:

MAT2F	34								
		47	48	51	47	48	51		

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT2 entry. (Integer > 0)
Gij	Identification number of a TABLEDi entry for the material property matrix. (Integer > 0 or blank)
GE	Identification number of a TABLEDi entry for the element structural damping coefficient. (Integer > 0 or blank)
GEij	Identification number of a TABLEDi entry for the element structural damping coefficients. (Integer > 0 or blank)

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT2 entry referenced in field 2. The value in a particular field of the MAT2 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT2 entry.
- Table references must be present for each item that is frequency dependent.
- If the GEij values are present on the MAT2 entry for any ply, then Nastran will ignore the GE value given on the first continuation entry field (6) and the GE entry given in field (8) of the PCOMP/PCOMPG entry and use the given GEij values. The rational of these entries is that with composite materials made of a viscoelastic resin system, the damping matrix does not scale with the stiffness matrix. For example, in the directions where you have more fibers, the stiffness is higher because you have more fibers but the damping coefficient is smaller because you have less resin matrix

which is the main source of damping. To use this feature in composites, the PCOMP/PCOMPG MIDI entries must refer to MAT2 entries. See Remarks 17. and of MAT2 entry. The damping is applied in element material routines by forming the product $GE_{ij} * G_{ij}$ when forming the K4 damping matrix.

4. IF $GE=0.0$ on corresponding MAT2 then GE table must be blank or 0. For any $GE_{ij}=0.0$ on corresponding MAT2 the corresponding GE_{ij} table must be blank or 0.

MAT3

CTRIAX6 or PSHLN2 or PLCOMP Material Property Definition

Defines the material properties for linear orthotropic materials used by the CTRIA6 element entry. It also is allowed with orthotropic materials on the PSHLN2 and PLCOMP entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT3	MID	EX	ETH	EZ	NUXTH	NUTHZ	NUZX	RHO	
			GZX	AX	ATH	AZ	TREF	GE	

Example:

MAT3	23	1.0+7	1.1+7	1.2+7	.3	.25	.27	1.0-5	
			2.5+6	1.0-4	1.0-4	1.1-4	68.5	.23	

Descriptor	Meaning
MID	Material identification number. (Integer > 0)
EX, ETH, EZ	Young's moduli in the x , θ , and z directions, respectively. (Real > 0.0)
NUXTH, NUTHZ	Poisson's ratios (coupled strain ratios in the $x\theta$, θz , and zx directions, respectively). (Real)
NUZX	Mass density. (Real)
RHO	Shear modulus. (Real > 0.0)
AX, ATH, AZ	Thermal expansion coefficients. (Real)
TREF	Reference temperature for the calculation of thermal loads or a temperature-dependent thermal expansion coefficient. See Remark 10. (Real or blank)
GE	Structural element damping coefficient. See Remarks 9. and 11. (Real)

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.
2. MAT3 materials may be made temperature dependent by use of the MATT3 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE (INIT) Case Control command.
3. The few numbers EX, ETH, EZ, and GZX must be present.
4. A warning message will be issued if any value of NUXTH or NUTHZ has an absolute value greater than 1.0.

5. MAT3 materials may only be referenced by the CTRIA6 entry or the PSHLN2 or PLCOMP entries.
6. The mass density RHO will be used to automatically compute mass for the CTRIA6 element.
7. The x-axis lies along the material axis (see [Figure 9-80](#) in the CTRIA6 entry). The θ-axis lies in the azimuthal direction. The z-axis is normal to both.
8. The strain-stress relationship is

$$\begin{bmatrix} \epsilon_x \\ \epsilon_\theta \\ \epsilon_z \\ \gamma_{zx} \end{bmatrix} = \begin{bmatrix} 1 & \frac{\text{NUTHX}}{\text{ETH}} & \frac{\text{NUZX}}{\text{EZ}} & 0 \\ \frac{\text{NUXTH}}{\text{EX}} & 1 & \frac{\text{NUZTH}}{\text{EZ}} & 0 \\ \frac{\text{NUZX}}{\text{EX}} & \frac{\text{NUTHZ}}{\text{ETH}} & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_\theta \\ \sigma_z \\ \sigma_{zx} \end{bmatrix} + (T - \text{TREF}) \begin{bmatrix} \text{AX} \\ \text{ATH} \\ \text{AZ} \\ 0 \end{bmatrix}$$

Note that:

$$\frac{\text{NUXTH}}{\text{EX}} = \frac{\text{NUTHX}}{\text{ETH}}$$

$$\frac{\text{NUZX}}{\text{EZ}} = \frac{\text{NUXZ}}{\text{EX}}$$

$$\frac{\text{NUTHZ}}{\text{ETH}} = \frac{\text{NUZTH}}{\text{EZ}}$$

The compliance matrix as shown must be positive definite. This requires that EX, ETH, EZ, and GZX be positive. There are additional restrictions as well, see a standard text on elasticity.

9. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.
10. TREF is used for two different purposes:
 - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Remark 10. under the MAT1 description.
 - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.
11. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).

MAT4**Heat Transfer Material Properties, Isotropic**

Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.

Format:

1	2	3	4	5	6	7	8	9	10
MAT4	MID	K	CP	ρ	H	μ	HGEN	REFENTH	
	TCH	TDELTA	QLAT						

Example:

MAT4	17	6.66-4	5.01-5	456.2		1.03-6			
------	----	--------	--------	-------	--	--------	--	--	--

Descriptor	Meaning
MID	Material identification number. (Integer > 0)
K	Thermal conductivity. (Blank or Real > 0.0)
CP	Heat capacity per unit mass at constant pressure (specific heat). (Blank or Real ≥ 0.0)
ρ	Density. (Real ≥ 0.0 ; Default = 1.0)
H	Free convection heat transfer coefficient. (Real or blank)
μ	Dynamic viscosity. See Remark 2. (Real > 0.0 or blank)
HGEN	Heat generation capability used with QVOL entries. (Real ≥ 0.0 ; Default = 1.0)
REFENTH	Reference enthalpy. (Real or blank)
TCH	Lower temperature limit at which phase change region is to occur. See Remark 6. (Real or blank)
TDELTA	Total temperature change range within which a phase change is to occur. (Real ≥ 0.0 or blank)
QLAT	Latent heat of fusion per unit mass associated with the phase change. (Real > 0.0 or blank)

Remarks:

1. The MID must be unique with respect to all other MAT4 and MAT5 entries. MAT4 may specify material properties for any conduction elements as well as properties for a forced convection fluid (see CONVM). MAT4 also provides the heat transfer coefficient for free convection (see CONV).
2. For a forced convection fluid, μ must be specified.
3. REFENTH is the enthalpy corresponding to zero temperature if the heat capacity CP is a constant. If CP is obtained through a TABLEM lookup, REFENTH is the enthalpy at the first temperature in the table.

4. Properties specified on the MAT4 entry may be defined as temperature dependent by use of the entry.
5. For RC network solver in thermal analysis, the REFENTH, TCH, TDELTA and QLAT are ignored.
6. The advanced nonlinear elements should be used for phase change to achieve accurate results.

MAT5

Thermal Material Property Definition

Defines the thermal material properties for anisotropic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MAT5	MID	KXX	KXY	KXZ	KYY	KYZ	KZZ	CP	
	RHO	HGEN							

Example:

MAT5	24	.092			.083		0.20	0.2	
	2.00								

Descriptor	Meaning
MID	Material identification number. (Integer > 0)
Kij	Thermal conductivity. (Real)
CP	Heat capacity per unit mass. (Real ≥ 0.0 or blank)
RHO	Density. (Real > 0.0 ; Default = 1.0)
HGEN	Heat generation capability used with QVOL entries. (Real ≥ 0.0 ; Default = 1.0)

Remarks:

1. The thermal conductivity matrix has the following form:

$$K = \begin{bmatrix} KXX & KXY & KXZ \\ KXY & KYY & KYZ \\ KXZ & KYZ & KZZ \end{bmatrix}$$

2. The material identification number may be the same as a MAT1, MAT2, or MAT3 entry but must be unique with respect to other MAT4 or MAT5 entries.
3. MAT5 materials may be made temperature-dependent by use of the MATT5 entry.
4. When used for axisymmetric analysis (CTRIAX6), material properties are represented where:
 KXX = radial conductivity component
 KYY = axial conductivity component
5. The KZZ is required if using the advanced nonlinear Shell element with temperature across thickness called out by NLMOPTS,TEMPP,LINE or QUAD option.

MAT8**Shell Element Orthotropic Material Property Definition**

Defines the material property for an orthotropic material for isoparametric shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
	GE	F12	STRN						
	"HFAIL"	HF1	HF2	HF3	HF4	HF10	HF11		
	"HTAPE"	HT1	HT2	HT3	HT4	HT5	HT6	HT10	
		H11	HT12						
	"HFABR"	HFB1	HFB2	HFB3	HFB4	HFB5	HFB6	HFB10	
		HFB11	HFBT12						

Example:

MAT8	171	30.+6	1.+6	0.3	2.+6	3.+6	1.5+6	0.056	
	28.-6	1.5-6	155.0	1.+4	1.5+4	2.+2	8.+2	1.+3	
	1.-4		1.0						
	"HFAIL"	1010.	1020.	33.	34.	1011.	1012.		
	"HTAPE"	2001.	2002.	35.	36.	2003.	1.0	1004.	
		1007.	1008.						
	"HFABR"	3001.	3002.	3003.	3004.	1005.	1005.	1006.	
		1007.	1008.						

Descriptor	Meaning
MID	Material identification number. Referenced on a PSHELL, PCOMP or PCOMPG entry only. (0 < Integer < 100,000,000)
E1	Modulus of elasticity in X material coordinate direction, also defined as the fiber direction or 1-direction. (Real > 0.0)
E2	Modulus of elasticity in Y material coordinate direction, also defined as the matrix direction or 2-direction. (Real > 0.0)
NU12	Poisson's ratio ($\nu_{21} = \varepsilon_2 / \varepsilon_1$ for uniaxial loading in 1-direction). Note that $\nu_{21} = \varepsilon_1 / \varepsilon_2$ for uniaxial loading in 2-direction is related to ν_{12} , E_1 , and E_2 by the relation $\nu_{12}E_2 = \nu_{21}E_1$. (Real)
G12	In-plane shear modulus. (Real > 0.0; Default = 0.0)
G1Z	Transverse shear modulus for shear in 1-Z plane. (Real ≥ 0.0 or blank; Default implies infinite shear modulus.)

Descriptor	Meaning
G2Z	Transverse shear modulus for shear in 2-Z plane. (Real ≥ 0.0 or blank; Default implies infinite shear modulus.)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient in i-direction. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 4. and 5. (Real or blank)
Xt, Xc	Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction. Required if failure index is desired. See the FT field on the PCOMP/PCOMPG entry. (Real > 0.0 ; Default value for Xc is Xt.)
Yt, Yc	Allowable stresses or strains in tension and compression, respectively, in the lateral direction. Required if failure index is desired. (Real > 0.0 ; Default value for Yc is Yt.)
S	Allowable stress or strain for in-plane shear. See the FT field on the PCOMP/PCOMPG entry. (Real > 0.0)
GE	Structural damping coefficient. See Remarks 4. and 6. (Real)
F12	Interaction term in the tensor polynomial theory of Tsai-Wu. Required if failure index by Tsai-Wu theory is desired and if value of F12 is different from 0.0. See the FT field on the PCOMP/PCOMPG entry. (Real)
STRN	For the maximum strain theory only (see STRN in PCOMP/PCOMPG entry). Indicates whether Xt, Xc, Yt, Yc, and S are stress or strain allowables. (Real = 1.0 for strain allowables; blank (Default) for stress allowables.)
“HFFAIL”	Keyword indicating that Hashin failure criterion, to calculate its four failure indices.
HF1	Maximum fiber tensile stress, no default
HF2	Maximum fiber compressive stress, default=HF1
HF3	Maximum matrix tensile stress, no default
HF4	Maximum matrix compressive stress, default=HF3
HF10	Maximum in-plane shear stress, no default
HF11	Maximum transverse shear stress, default=HF10
“HTAPE”	Keyword indicating that Hashin-Tape criterion, a variant of the Hashin criterion, adapted for tape type of materials are calculated.
HT1	Maximum tape fiber tensile stress, no default
HT2	Maximum tape fiber compressive stress, default=HT1
HT3	Maximum tape cross-fiber tensile stress, no default
HT4	Maximum tape cross-fiber compressive stress, default=HT3
HT5	Maximum fiber tensile stress for matrix compression; Required if HT6=1.0, otherwise not used.

Descriptor	Meaning
HT6	Contribution factor for HT5 (Real, 0.0 or 1.0, default = 0.0)
HT10	Maximum in-plane shear stress, no default
HT11	Maximum transverse shear stress, default=HT10
HT12	Maximum z-x transverse shear stress, default=HT11
"HFABR"	Keyword indicating that Hashin-Fabric criterion, a variant of the Hashin criterion, adapted for fabric type of materials are calculated.
HFB1	Maximum first fiber tensile stress, no default
HFB2	Maximum first fiber compressive stress, default=HFB1
HFB3	Maximum second cross-fiber tensile stress, no default
HFB4	Maximum second cross-fiber compressive stress, default=HFB3
HFB5	Maximum thickness tensile stress, no default
HFB6	Maximum thickness compressive stress, default=HFB5
HFB10	Maximum in-plane shear stress, no default
HFB11	Maximum transverse shear stress, default=HFB10
HFB12	Maximum z-x transverse shear stress, default=HFB11

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.
2. If G1Z and G2Z values are specified as zero or blank, then transverse shear flexibility calculations will not be performed, which is equivalent to zero shear flexibility (i.e., infinite shear stiffness). If MAT8 is referenced by Advanced Nonlinear Element, G1Z and G2Z must be positive.
3. An approximate value for G1Z and G2Z is the in-plane shear modulus G12. If test data are not available to accurately determine G1Z and G2Z for the material and transverse shear calculations are deemed essential; the value of G12 may be supplied for G1Z and G2Z. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed in the TEMPERATURE Case Control command.
4. Xt, Yt, and S are required for composite element failure calculations when requested in the FT field of the PCOMP/PCOMPG entry. Xc and Yc are also used but not required.
5. TREF and GE are ignored if this entry is referenced by a PCOMP/PCOMPG entry.
6. TREF is used in two different ways:
 - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See [Figure 9-105](#) in Remark 10. in the MAT1 description.

- In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must then be blank.
7. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).
 8. In nonlinear static analysis (SOL 106) the QUAD4 and TRIA3 thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPSILONT,INTEGRAL' in bulk data. See [Parameters, 783](#).
 9. If MAT8 with Hashin is used in SOL 400 with PSLDN1 then Hashin FT will be ignored with an appropriate warning message to include MATF instead.

MAT8A**Orthotropic Failure Material Properties - SOL700**

Defines the failure properties for an orthotropic material for shell elements. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MAT8A	MID	FT	NV	S	ALPHA	TRSFAIL	F12		
	XT	XC	YT	YC	PFD	VALUE	PFDS		
	FBTEN	FBCOM	MXTEN	MXCOM	MXSHR				
	PRDFT	PRDFC	PRDMT	PRDMC	PRDSH				

Example:

MAT8A	7	COMBINAT		100.					
+	200.0	150.0	100.0	110.0	STEPS	200.0			
+	CHANG	STRSS	MODSAI	MODTSAI	STRSS				
+									
+					0011				

Descriptor	Meaning
MID	Material identification. See Remark 1. (Integer > 0, Required.)
FT	Failure theory to be used to test whether the element layer fails. (Character; default=blank)
	Blank No failure
	HILL Tsai-Hill theory
	TSAI Tsai-Wu theory
	MODTSAI Modified Tsai-Wu theory
	STRSS Maximum stress
	CHANG Chang-Chang theory
	COMBINAT Combination (See Remark 8.)
	HASHIN Hashin theory (See Remark 2.)
NV	Number of additional history variables for a user model. See Remark 8. Currently it is not available. (0 < Integer < 10; default=0)
S	Failure stress for in-plane shear. See Remark 4. (Real > 0.0; required.)
ALPHA	Nonlinear shear coefficient. See Remark 5. (Real \geq 0.0; default=0.0)
TRSFAIL	Transverse shear failure. (Character; default=SUBL)

Descriptor	Meaning	
	ELEM	Failure if element fails
	SUBL	Failure if sublayer fails
F12		Interaction term in Tsai-Wu theory. (Real, default=0.0)
XT		Tensile failure stress in the large structural direction. See Remark 4. (Real ≥ 0.0 ; default=0.0)
XC		Compressive failure stress in the large structural direction. See Remark 4. (Real ≥ 0.0 ; default=0.0)
YT		Tensile failure stress in the lateral direction. See Remark 4. (Real ≥ 0.0 ; default=0.0)
YC		Compressive failure stress in the lateral direction. See Remark 4. (Real ≥ 0.0 ; default=0.0)
PFD		Post-failure degradation model. See Remark 9. (Character; default=STEPS)
	STEPS	Degrade stresses by time steps
	TIME	Degrade stresses by time
	VELOC	Degrade stresses by velocity
VALUE		Depending on PFD, VALUE gives the number of time steps, time interval, or propagation velocity. (Real > 0 ; default=100.0)
PFDST		Post-failure degradation start. See Remark 9. (Character; default=INDV)
	INDV	Stresses are degraded per distinct failure mode.
	ALL	Stresses are degraded if all elastic constants are zero.
FBTEN		Tensile failure modes in fiber direction for individual failure definition. See Remark 6. (Character; required.)
FBCON		Compressive failure modes in fiber direction for individual failure definition. See Remark 6. (Character; required.)
MXTEN		Tensile failure modes in matrix direction for individual failure definition. See Remark 6. (Character; required.)
MXCOM		Compressive failure modes in matrix direction for individual failure definition. See Remark 6. (Character; required.)
MXSHR		Shear failure modes for individual failure definition. See Remark 6. (Character; required.)
PRDFT		Property degradation due to fiber-tension failure. See Remark 7. (Integer; default=1111)
PRDFC		Property degradation due to fiber-compression failure. See Remark 7. (Integer; default=1010)
PRDMT		Property degradation due to matrix-tension failure. See Remark 7. (Integer; default=0110)

Descriptor	Meaning
PRDMC	Property degradation due to matrix-compression failure. See Remark 7. (Integer; default=0110)
PRDSH	Property degradation due to in-plane shear failure. See Remark 7. (Integer; default=0001)

Remarks:

1. The material number must refer to a MAT8 material definition.
2. If a failure theory is selected other than COMBINAT, the theory defines the following failure modes:

CHANG	Fiber tension, matrix tension/compression
HILL	All modes
TSAI	All modes
MODTSAI	Matrix tension/compression
STRSS	All modes
HASHIN	Fiber tension/compression Matrix tension/compression

For an element to fail completely, both fiber and matrix in all sublayers must fail.

3. This material model can only be referenced from a PCOMP entry.
4. Failure stresses are required if a failure theory is selected.
5. ALPHA is used for all failure theories to define a nonlinear stress-strain relation.
6. The individual failure modes are defined according to the corresponding mode in the theory as listed under FT. To be relevant, the theory must define the failure mode (see Remark 2.). You must enter data if FT is set to COMBINAT.
7. The property degradation rules due to the various failure modes are listed below:

Material Constant	Failure Mode				
	Fiber Tens	Fiber Comp	Matrix Tens	Matrix Comp	Shear
E1	X	X			
E2	X		X	X	
V12	X	X	X	X	
G12	X				X

The Poisson's ratio Nu21 is treated the same as Nu12.

To override the default model, an integer value is defined as a packed word in the following order:

- 1 Denotes property degradation.
- 0 Denotes no degradation.

The last five fields of the MAT8A Bulk Data entry are input for the user to specify the degradation behavior for each mode of failure.

8. NV is required input and NV new user variables are automatically created. User variables for sublayers are used on restart and archive output. Refer to them as USRnLxx where n is the user ID and xx is the sublayer number. The user variables are available in the EXCOMP user subroutine. The values S, XT, XC, YT, and YC are also required input when FT is set to USER. For the model USER1, the routine EXCOMP1 has to be used. Currently, user routine is not available.
9. The PFD entry indicates how the stresses are degraded to zero. The PFDST indicates when the stresses start to degrade.
Using ALL means that degradation starts when all material constants (E1, E2, V12, G12) are degraded to zero as specified by the FT entry and the property degradation rules. Note that property degradation means that the stress increments are zero but that the stresses degrade according to PFD.
INDV means that stress degradation starts for the fiber stress if E1=0.0, for matrix stress if E2=0.0, and for shear stress if G12=0.0.
10. Any failure theory introduces five additional sublayer variables. The PFDST entry introduces three additional variables. The number of user variables is defined by NV.

MAT8F**Shell Element Orthotropic Material Frequency Dependence**

Specifies frequency-dependent material properties on MAT8 entry fields via TABLEDi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT8F	MID	E1	E2	NU12	G12	G1Z	G2Z		
	GE								

Example:

MAR8F	76								
	97								

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT8 entry. (Integer > 0)
E1	Identification number of a TABLEDi entry for the Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction. (Integer > 0 or blank)
E2	Identification number of a TABLEDi entry for the Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction. (Integer > 0 or blank)
NU12	Identification number of a TABLEDi entry for Poisson's ratio. (Integer > 0 or blank)
G12	Identification number of a TABLEDi entry for the in-plane shear modulus. (Integer > 0 or blank)
G1Z	Identification number of a TABLEDi entry for the transverse shear modulus for shear in 1-Z plane. (Integer > 0 or blank)
G2Z	Identification number of a TABLEDi entry for the transverse shear modulus for shear in 2-Z plane. (Integer > 0 or blank)
GE	Identification number of a TABLEDi entry for the element structural damping coefficient. (Integer > 0 or blank)

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT8 entry referenced in field 2. The value in a particular field of the MAT8 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT8 entry.
- Table references must be present for each item that is frequency dependent.
- IF GE=0.0 on corresponding MAT8 then GE table must be blank or 0.

MAT9 Solid Element (and Shell Element for SOL 600 only) Anisotropic Material Property Definition

Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements (see PSOLID entry description).

Format:

1	2	3	4	5	6	7	8	9	10
MAT9	MID	G11	G12	G13	G14	G15	G16	G22	
	G23	G24	G25	G26	G33	G34	G35	G36	
	G44	G45	G46	G55	G56	G66	RHO	A1	
	A2	A3	A4	A5	A6	TREF	GE		
	GE11	GE12	GE13	GE14	GE15	GE16	GE22	GE23	
	GE24	GE25	GE26	GE33	GE34	GE35	GE36	GE44	
	GE45	GE46	GE55	GE56	GE66				

Example:

MAT9	17	6.2+3						6.2+3	
					6.2+3				
	5.1+3			5.1+3		5.1+3	3.2	6.5-6	
	6.5-6					125.	.003		

Descriptor	Meaning
MID	Material identification number. (Integer > 0)
Gij	Elements of the 6 x 6 symmetric material property matrix in the material coordinate system. (Real)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient. (Real)
TREF	Reference temperature for the calculation thermal loads, or a temperature-dependent thermal expansion coefficient. See Remark 7. (Real or blank)
GE	Structural element damping coefficient. See Remarks 6. and 8. (Real)
GEij	Structural damping matrix. See Remark 9. (Real)

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.

2. MAT9 materials may be made temperature-dependent by use of the MATT9 entry. In nonlinear static analysis (e.g., SOL 106), linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
3. The mass density RHO will be used to compute mass in a structural dynamics problem automatically.
4. The third continuation entry is optional.
5. The subscripts 1 through 6 refer to x, y, z, xy, yz, and zx of the material coordinate system (see the CORDM field on the PSOLID entry description). The stress-strain relationship is

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{Bmatrix} G_{11} & G_{12} & G_{13} & G_{14} & G_{15} & G_{16} \\ & G_{22} & G_{23} & G_{24} & G_{25} & G_{26} \\ & & G_{33} & G_{34} & G_{35} & G_{36} \\ & & & G_{44} & G_{45} & G_{46} \\ & & & & G_{55} & G_{56} \\ & & & & & G_{66} \end{Bmatrix}_{\text{symmetric}} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \\ A_4 \\ A_5 \\ A_6 \end{Bmatrix} (T - TREF)$$

6. The damping coefficient GE is given by
$$GE = \frac{2.0 \cdot C}{C_0}$$
 7. TREF is used in two different ways:
 - In nonlinear static analysis (e.g., SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See [Figure 9-105](#) in Remark 10. in the MAT1 description.
 - In all solutions except nonlinear static analysis, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must then be blank.
 8. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).
 9. Defines structural damping matrix data for advanced composites. If the GEij values are present, the Nastran will ignore the GE value given on the third continuation entry field (8) and use the given GEij values. The rational of these entries is that with composite materials made of a viscoelastic resin system, the damping matrix does not scale with the stiffness matrix. For example, in the directions where you have more fibers, the stiffness is higher because you have more fibers but the damping coefficient is smaller because you have less resin matrix which is the main source of damping. To use this feature in composites, the PCOMPLS MIDi entries must refer to MAT9 entries.
- The damping is applied in element material routines by forming the product GEij * Gij when forming the K4 damping matrix in EMG.

MAT9F Solid Element Anisotropic Frequency Dependence

Specifies frequency-dependent material properties on MAT9 entry fields via TABLEDi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT9F	MID	G11	G12	G13	G14	G15	G16	G22	
	G23	G24	G25	G26	G33	G34	G35	G36	
	G44	G45	G46	G55	G56	G66			
							GE		
	GE11	GE12	GE13	GE14	GE15	GE16	GE22	GE23	
	GE24	GE25	GE26	GE33	GE34	GE35	GE36	GE44	
	GE45	GE46	GE55	GE56	GE66				

Example:

MAT9F	101	5						5	
					5				
				5		5			

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT9 entry. (Integer > 0)
Gij	Identification number of a TABLEDi entry for the material property matrix. (Integer > 0 or blank)
GE	Identification number of a TABLEDi entry for the element structural damping coefficient. (Integer > 0 or blank)
GEij	Identification number of a TABLEDi entry for the element structural damping coefficients. (Integer > 0 or blank)

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT9 entry referenced in field 2. The value in a particular field of the MAT9 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT9 entry.
- Table references must be present for each item that is frequency dependent.
- IF GE=0.0 on corresponding MAT9 then GE table must be blank or 0. For any GEij=0.0 on corresponding MAT9, the corresponding GEij table must be blank or 0.

MAT10**Fluid Material Property Definition**

Defines material properties for fluid elements in coupled fluid-structural analysis.

Format:

1	2	3	4	5	6	7	8	9	10
MAT10	MID	BULK	RHO	C	GE	ALPHA			

Example:

MAT10	103	0.656	0.011						
-------	-----	-------	-------	--	--	--	--	--	--

Descriptor	Meaning
MID	Material identification number. (Integer > 0)
BULK	Bulk modulus. (Real > 0.0)
RHO	Mass density. (Real > 0.0)
C	Speed of sound. (Real > 0.0)
GE	Fluid element damping coefficient. (Real)
ALPHA	Normalized admittance coefficient for porous material. See Remark 7. (Real or blank)

Remarks:

1. MAT10 is referenced, with MID, by the PSOLID entry only.
2. The material identification numbers must be unique for all MAT1, MAT2, MAT3, MAT9, and MAT10 entries.
3. The mass density RHO will be used to compute the mass automatically.
4. BULK, RHO, and C are related by

$$\text{BULK} = C^2 \cdot \text{RHO}$$

Two out of the three must be specified, and the other will be calculated according to this equation. If all three are specified and are inconsistent in values, the supplied values of BULK and RHO are used in the computations.

5. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 , by 2.0.
6. If PARAM,W4FL is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).
7. If a value of ALPHA is entered, BULK RHO and GE may have negative values.
8. The value defined in the ALPHA field always defines the normalized admittance coefficient for porous material but it is differently interpreted depending on the value defined in the FCNT field of the referencing PSOLID entry.

- a. If the MAT10 entry is referenced in a PSOLID entry where FFLUID option is selected, the value defined for ALPHA is considered as the normalized admittance coefficient calculated at unit circular excitation frequency ($\omega = 1$). Its value will be automatically calculated by the program, at each excitation frequency, considering the current circular excitation frequency as scaling factor.
 - b. If the MAT10 entry is referenced in a PSOLID entry where PFLUID option is selected, the value defined for ALPHA has no special meaning but it is only the normalized admittance coefficient calculated by the user at the most appropriate excitation frequency (defined in order to have good results in the frequency range of interest).
9. For Poro-Elastic Material, PEM, user is expected to provide adiabatic BULK modulus for fluid-phase of PEM which is equal to GAMMA, fluid ratio of specific heats (see MATPE1 for detailed), times isothermal BULK modulus. Isothermal BULK modulus is BULK modulus at constant temperature for fluid itself.
 10. When used in conjunction with PSLDN1, only pyramid elements can be associated with the MAT10 fluid property. CHEXA, CPENTA, CTETRA elements using MAT10 are not supported with PSLDN1.
 11. The terms of the fluid mass matrix are not multiplied by the value of PARAM and WTMASS.

MAT10F**Fluid Frequency Dependence**

Specifies frequency-dependent material properties on MAT10 entry fields via TABLED_i entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT10F	MID	BULK	RHO	C	GE	ALPHA			

Example:

MAT10	103	666							
-------	-----	-----	--	--	--	--	--	--	--

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT10 entry. (Integer > 0)
BULK	Identification number of a TABLED _i entry for the Bulk modulus. (Integer > 0 or blank)
RHO	Identification number of a TABLED _i entry for the mass density. Used only in SOL108. (Integer > 0 or blank)
C	Identification number of a TABLED _i entry for the speed of sound. Used only in SOL108. (Integer > 0 or blank)
GE	Identification number of a TABLED _i entry for the element structural damping coefficient. (Integer > 0 or blank)
ALPHA	Identification number of a TABLED _i entry for the normalized admittance coefficient for porous material. See Remark 4.

Remarks:

1. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT10 entry referenced in field 2. The value in a particular field of the MAT10 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT10 entry.
2. Table references must be present for each item that is frequency dependent.
3. When ALPHA on MAT10 is specified it is multiplied by the excitation frequency in radians/unit time. If F(ALPHA) is specified then the value supplied by F(ALPHA) will take precedence and ALPHA WILL NOT BE MULTIPLIED by the excitation frequency.

MATDEUL

General Constitutive Model to be Used for the Eulerian Materials - SOL 700

Defines a complete constitutive model as a combination of an equation of state, a shear model, a yield model, a failure model, a spall model (PMIN), and corotational frame. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATDEUL	MID	RHO	EID	SID	YID	FID	PID		
	BULKL	BULKQ			BULKTYP				

Example:

MATDEUL	22	3000.	100	109	307	308	402		
---------	----	-------	-----	-----	-----	-----	-----	--	--

Descriptor	Meaning
MID	Unique material number. (Integer > 0; Required)
RHO	Density. (Real > 0.0; Required)
EID	Number of an EOSxxx entry defining the pressure/density characteristic of the material. (Integer > 0; Required)
	EOSGAM Gamma Law Gas Equation of State
	EOSIG Ignition and Growth Equation of State
	EOSJWL JWL Explosive Equation of State
	EOSMG Mie-Gruneisen Equation of State
	EOSPOL Polynomial Equation of State
	EOSTAIT Tait Equation of State
SID	Number of a SHRxxy entry defining the shear properties of the material. (Integer ≥ 0 ; Hydrodynamic shear model)
	SHREL Elastic Shear Model
	SHRPOL Polynomial Shear Mode
YID	Number of a YLDxxx entry defining the yield model for the material. (Integer ≥ 0 ; Hydrodynamic yield model)
	YLDHY Hydrodynamic Yield Model
	YLDJC Johnson-Cook Yield Model
	YLDMC Mohr-Coulomb Yield Model
	YLDMSS Multi-surface yield model for Snow
	YLDPOL Polynomial Yield Model
	YLDRPL Rate Power Law Yield Model

Descriptor	Meaning
YLDHG	Steinberg-Guinan Yield
YLDHM	Tanimura-Mimura Yield Model
YLDVM	von Mises Yield Model
YLDZA	Zerilli-Armstrong Yield Model
FID	Number of a FAILMPS entry defining the failure model for the material. (Integer ≥ 0 ; no failure)
	FAILMPS Maximum Plastic Strain Failure Model
PID	Number of a PMINC entry defining the spallation characteristics of the material. See Remark 3. (Integer > 0)
BULKL	Linear bulk-viscosity coefficient. (Real ≥ 0.0 ; 0.0)
BULKQ	Quadratic bulk-viscosity coefficient. (Real > 0.0 ; 1.0)
BULKTYP	Bulk viscosity type. (Character, DYNA)
	DYNA Standard DYNA3D model.
	DYTRAN Enhanced DYNA model.

Remarks:

1. If YID is blank or zero, a hydrodynamic yield model is used.
2. If the TYPE field on the PEULER entry is set to HYDRO, then YID is either blank or references a YLDHY entry, and SID is blank. Conversely, if the TYPE field is set to STRENGTH, a nonhydrodynamic yield model is specified.
3. If no PMINC entry is referenced, a minimum pressure of zero is assumed for the Eulerian elements. The PMINC entry will be ignored when a cavitation model through the EOSTAIT entry has been given.

MATDIGI**Material Digimat**

Defines material data for the advanced composites with Digimat from e-Xstream engineering (SOL 400 only). For more information about e-Xstream engineering and Digimat, please contact support@e-xstream.com or consult <http://www.e-xstream.com/>

Format:

1	2	3	4	5	6	7	8	9	10
MATDIGI	MID	UDID					RHO		

Example:

MATDIGI	5	10					3000.		
---------	---	----	--	--	--	--	-------	--	--

Descriptor	Meaning
MID	Material identification number. (Integer > 0)
UDID	References UDNAME entry (Required, Integer>0)
RHO	Density (Real>0.0 or blank)

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.
2. SOL400 uses only the MID, UDID and RHO entries. All other entries are ignored by SOL400.
3. The entry UDSESV is required input to define the number of state variables which will be used by MATDIGI. The number of state variables (NSTATS) must be larger than 1.
4. License feature NASTRAN_DIGIMAT must be present to use this feature. For parallel computations, NASTRAN_DIGIMAT_DMP must be present.
5. The use of MATDIGI is limited to shell and solid elements.
6. MATDIGI should not be used in SOL 400 perturbation steps.

MATEP**Elasto-Plastic Material - SOL 400**

Elasto-plastic material properties for SOL 400 only.

Format: Note that the primary entry is required. All other continuation lines are required only for certain options and only one such option may be entered.

1	2	3	4	5	6	7	8	9	10
MATEP	MID	FORM	Y0	FID	RYIELD	WKHARD		H	
	“Reffect”	OPTION	RTID	C	P				
	“Aniso”	N/A	R11 or M	R22 or C1	R33 or C2	R12 or C3	R23 or C6	R31	
	“Press”	OPTION	ALPHS	BETA					
	“Chaboche”	R0	Rinf	B	C	Gam	Kap	N	
		Qm	μ	η					
	“JhCook”	A	B	N	C	M	Tmelt	Troom	
		ε_0 DOT							
	“YldOpt”	OPTION							
	“VParam”	Nvp	Vp1	Vp2					
	“IMPCREEP”	VMISES							

Example:

MATEP	100	TABLE		20		HILL			
	ANISO		1.1	0.9		1.02			

Descriptor	Meaning
MID	Identification number of MAT1, MAT2, MATORT, MAT8 or MAT9 entry. (Integer > 0).
FORM	Selects a form of stress-plastic strain function to be specified (Character): SLOPE Defines nonzero H. (Default) TABLE Defines the function using TABLES1, TABLEST or TABL3D entries. Units of table are unspecified PERFECT Defines perfectly plastic material (H is zero or blank and FID blank). See Remarks 2.
Y0	Initial yield stress Y_0 or hydrostatic stress for Mohr-Coulomb materials. See Remark 2. (Real > 0 or blank)
FID	Identification number of TABLES1 or TABL3D entry. See Remarks 2., 3., and 4. (Integer > 0 or blank)

Descriptor	Meaning
RYIELD	Enter one of the following yield criteria rules. (Character; Default = VMISES) See Remarks 12. VMISES von Mises yield criteria (Default) HILL Hill's 1948 yield criteria BARLAT Barlat's 1991 yield criteria LINMOHR Linear Mohr-Coulomb yield criteria PBLMOHR Parabolic Mohr-Coulomb yield criteria IMPCREEP Implicit creep model combining both plasticity and creep, von Mises yield criteria.
WKHARD	Selects a hardening rule defined by various work-hardening rules (Character). See Remarks 12. ISOTROP Isotropic hardening. (Default) KINEM Kinematic hardening. COMBINE Combination between kinematic and isotropic hardening. (Default kinematic fraction=0.5). CHABOCHE Caboche formulation - like combined (see MSC Nonlinear User Guide (SOL 400)) for details of this model). JHCOOK For Johnson-Cook formulation (see MSC Nonlinear User Guide (SOL 400)) for details of this model). See Remark 5.
H	Workhardening slope. See Remarks 2., 3., and 4. (Real ≥ 0 ; Default = 0)
"Reffect"	A keyword signifying that the following data pertains to the rate-dependent material properties (do not enter rate-dependent effects if they are not to be included). If TABL3D is entered and one of the independent variables is strain rate, this field and its options should not be specified.
OPTION	Selects an option for strain-rate dependent yield stress (Character): TABLE Specifies that TABLES1 input used to define rate-dependency. (Default) COWPER Specifies the Cowper and Symonds rate-dependency model. See Remark 6.
RTID	TABLES1 ID for strain-rate effects on yield stress, i.e., the table defines yield stress as a function of strain-rate starting from zero strain-rate. The yield stresses in this table should comprise the initial yield stress specified on Y0 or FID field at zero strain-rate.
C	Specifies the constant C for Cowper and Symonds model. (Real; Default = 1)
P	Specifies the constant P for Cowper and Symonds model. (Real; Default = 1)

Descriptor	Meaning
"Aniso"	A keyword signifying that the following data (R_{ij}) pertain to the Hill's (R_{ij}) or Barlat's (M , C_i) yield criteria. an isotropic material option. See Remark 7. and MSC Nonlinear User Guide (SOL 400) (specify only if RYIELD = HILL or BARLAT).
R_{ij}	Stress ratios of initial yield stresses in various material directions to the reference yield stress from FID/Y0 field. (Real > 0; Default = 1.0 for R11, R22, R33, R12, R23, R31, respectively for Hill) or
M	Barlat M coefficient
C1	Barlat C1 coefficient
C2	Barlat C2 coefficient
C3	Barlat C3 coefficient
C6	Barlat C6 coefficient
"Press"	A keyword signifying that the following data pertain to the pressure-dependent yield criteria. Enter only if Ryield = LinMohr or PblMohr cracking is to be simulated. See Remarks 8. and 9.
OPTION	Selects an option for pressure-dependent yield criteria. (Character):
LIN	Linear Mohr-Coulomb model. (Default)
PBL	Parabolic Mohr-Coulomb model.
ALPHA	Specifies the parameter alpha for linear Mohr-Coulomb model. (Real > 0 or blank)
BETA	Specifies the parameter beta for parabolic Mohr-Coulomb model. (Real > 0 or blank)
"Chaboche"	A keyword specifying the following data pertains to the Chaboche model. (Enter only if WKHARD=CHABOCHE.)
R0	R0 for isotropic hardening (Real)
Rinf	Rinfinity for isotropic hardening (Q0 in case of using plastic-strain-range memorization) (Real)
B	Coefficient B for isotropic hardening (Real)
C	C coefficient C for kinematic hardening (Real)
Gam	Gamma coefficient for kinematic hardening (Real)
Kap	Kappa value for viscosity model (Real)
N	Coefficient N for viscosity model (Real)
Qm	Coefficient Qm for isotropic hardening (Real)
μ	Coefficient μ for isotropic hardening (Real)
η	Coefficient η to introduce progressive memory (Real)
"JhCook"	A keyword specifying the following data pertains to the Johnson-Cook model (see the MSC Nonlinear User Guide (SOL 400) for more details. Enter only if WKHARD = JHCOOK).

Descriptor	Meaning
A	Coefficient A. (Real)
B	Coefficient B. (Real)
N	Exponent n. (Real)
C	Coefficient C. (Real)
M	Exponent m. (Real)
Tmelt	Melting temperature. (Real)
Troom	Ambient temperature. (Real)
$\dot{\varepsilon}_0$ Dot	Reference strain rate. (Real)
"YldOpt"	A keyword specifying one of the following yield models will be used. Select a yield option from the list below:
OPTION	Specifies a yield option. (Character):
	IMCREEP Implicit creep model, both plasticity and creep with von Mises yield criteria. -user subroutine UVSCPL. MATUDS is required.
"VParam"	A keyword specifying that the number of viscoplastic parameters is to follow. Up to two viscoelastic parameters are to be entered (used only when RYIELD = IMPCREEP).
Nvp	Number of viscoplastic parameters. (Integer must be 0, 1, or 2)
Vp1, Vp2	A list of viscoplastic parameters. (Real, Default = 0.0)
"IMPCREEP"	A keyword specifying that the equivalent (von Mises) tensile yield stress will be entered in the next field (used only when RYIELD = IMPCREEP).
VMISES	Equivalent (von Mises) tensile yield stress. Overrides any similar entry made for this material elsewhere. (Real, no Default)

Remarks:

- Unless continuation entry is present specifying various material models, von Mises yield criterion is used as default.
- If FORM = SLOPE, the one defines the initial yield stress with Y_0 and a single workhardening slope is entered as $H \cdot H = \frac{\partial \bar{\sigma}}{\partial \dot{\varepsilon}^p}$. In a uniaxial test, , the $\dot{\varepsilon}_0$ is the strain at which yielding first occurs is often considered the 0.2% offset strength.

If FORM = TABLE, then the table ID is entered using FID field, Y_0 is not used. The table can be entered as TABLES1 or TABL3D.

When TYPE = 1 on the TABLES1 option, one is defining $\bar{\sigma} = \bar{\sigma}(\dot{\varepsilon})$ as shown in [Figure 9-107](#).

When TYPE = 2 on the TABLES1 or TABL3D option, one is defining $\bar{\sigma} = \bar{\sigma}(\dot{\varepsilon}^p)$ as shown in [Figure 9-108](#).

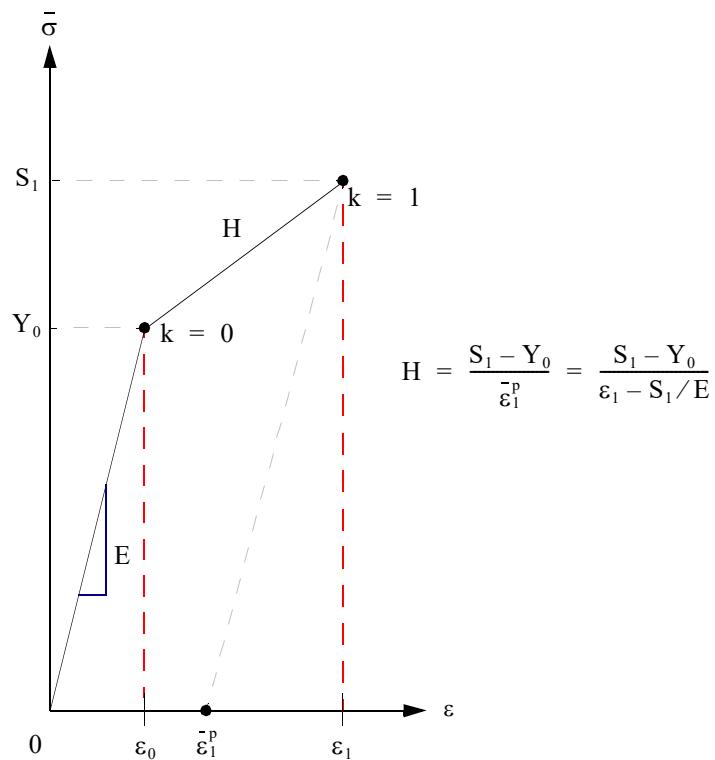


Figure 9-106 Single Workhardening Slope

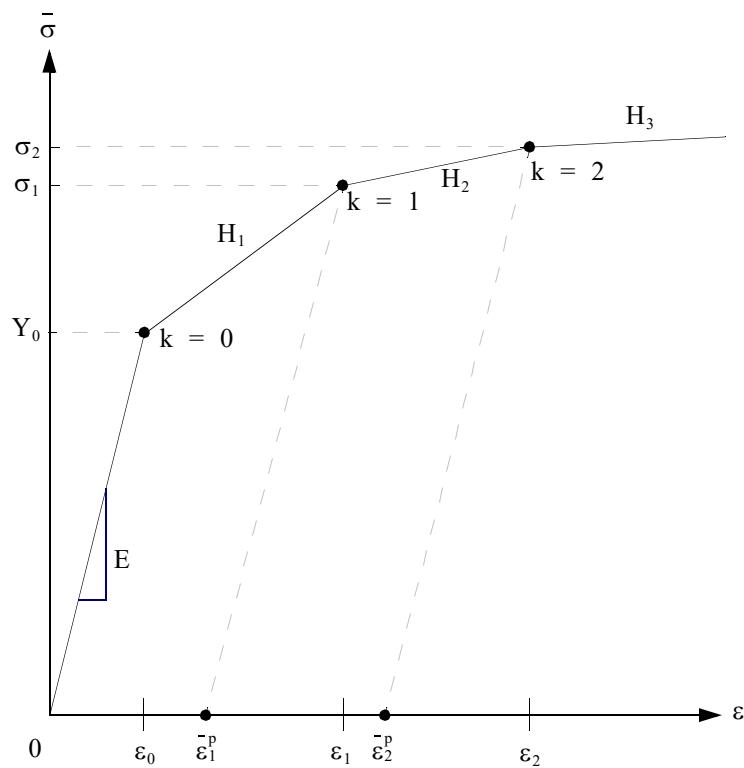


Figure 9-107 Stress vs Total Strain TABLES1(TYPE1)

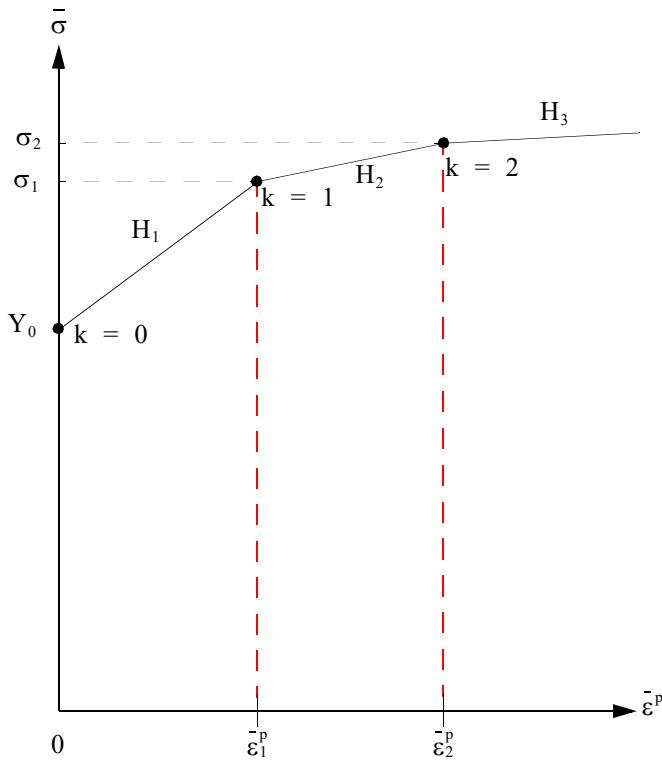


Figure 9-108 Stress vs Plastic Strain TABLES1(TYPE2) or TABL3D

If FORM = PERFECT, Y_0 is the yield stress and H and FID should be blank. This is known as elastic-perfectly plastic behavior. If temperature-dependent yield is present, then FORM = TABLE should be used and the temperature-dependent yield entered through TABL3D.

3. For temperature-dependent materials, one can either define the temperature variation through the TABL3D option or include the MATTEP option. The first procedure is recommended.
4. In a large displacement analysis (LGDISP = 1, 2, 11, 12), the Cauchy stress and logarithmic strain are used.
5. The plastic deformation starts when the effective stress ($\bar{\sigma}$) exceeds the yield stress.

The yield stress is initially defined by the initial yield point, which is subsequently modified by the hardening rule to account for strain hardening. Under the isotropic hardening rule, the size of the yield surface expands as a function of effective plastic strain ($\bar{\epsilon}^p$). Under the kinematic hardening rule, the center of the yield surface moves in stress space while keeping the same size and shape. Ziegler's law is used to define the translation of the yield surface. Under the combined hardening, the initial hardening is assumed to be entirely isotropic, but the elastic range attains a constant value (i.e., behaving like kinematic hardening) after some plastic straining. The effective stress for von Mises is expressed as

$$\bar{\sigma} = \sqrt{\frac{1}{2}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

where the stress components are measured from the center of yield surface.

6. The Cowper and Symonds model scales the initial yield stress as a function of strain-rate, i.e.,

$$Y(\dot{\varepsilon}) = Y_0 \left[1 + \left(\frac{\dot{\varepsilon}}{C} \right)^{1/p} \right]$$

7. Hill's anisotropic model introduces orthotropic plastic material. The plastic anisotropy proposed by Hill introduces six parameters to the von Mises yield function, from which an effective stress may be derived as

$$\bar{\sigma} = \sqrt{F(\sigma_y - \sigma_z)^2 + G(\sigma_z - \sigma_x)^2 + H(\sigma_x - \sigma_y)^2 + 2L\tau_{yz}^2 + 2M\tau_{zx}^2 + 2N\tau_{xy}^2}$$

in which the material parameters can be related to the yield stress ratios by

$$F = \frac{1}{2} \left(\frac{1}{R_{22}^2} + \frac{1}{R_{33}^2} - \frac{1}{R_{11}^2} \right)$$

$$G = \frac{1}{2} \left(\frac{1}{R_{33}^2} + \frac{1}{R_{11}^2} - \frac{1}{R_{22}^2} \right)$$

$$H = \frac{1}{2} \left(\frac{1}{R_{11}^2} + \frac{1}{R_{22}^2} - \frac{1}{R_{33}^2} \right)$$

$$L = \frac{3}{2R_{23}^2}, M = \frac{3}{2R_{31}^2}, N = \frac{3}{2R_{12}^2}$$

with

$$R_{11} = \frac{Y_1}{Y_a}, R_{22} = \frac{Y_2}{Y_a}, R_{33} = \frac{Y_3}{Y_a}$$

$$R_{12} = \frac{\sqrt{3}T_{12}}{Y_a}, R_{23} = \frac{\sqrt{3}T_{23}}{Y_a}, R_{31} = \frac{\sqrt{3}T_{31}}{Y_a}$$

where Y_1 , Y_2 , and Y_3 are the tensile yield stresses measured in material directions x, y and z, respectively; T_{12} , T_{23} , and T_{31} are the shear yield stresses in pure shear; and Y_a is the reference yield stress which should be an average yield stress in all directions.

In practical applications, however, the initial yield stress cannot be measured in all directions. The plastic anisotropy is pronounced in the sheet metal due to prior rolling process, for which the plastic anisotropy is customarily characterized by r-values defined by strain ratio measured in the uniaxial tension, i.e.,

$$r_a = \frac{\varepsilon_w}{\varepsilon_t} = \frac{\ln(w_0/w)}{\ln(t_0/t)} = \frac{H + (2N - F - G - 4H)\sin^2\theta \cos^2\theta}{F \sin^2\theta + G \cos^2\theta}$$

where t and w denote thickness and width, respectively; and θ denotes the angle of orientation (usually measured from the rolling direction). Assuming that the anisotropy parameters stay constant throughout the deformation, F , G , H and N can be determined by r-values from tensile specimen cut at 0, 45 and 90 degrees to the rolling direction:

$$\frac{H}{G} = r_0 \quad , \quad \frac{H}{F} = r_{90}$$

$$\frac{N}{G} = \left(r_{45} + \frac{1}{2} \right) \left(1 + \frac{r_0}{r_{90}} \right)$$

The orthotropic plasticity parameters should be calculated from the r-values and the initial yield stress either in 0 or 90 degree direction (Y_0 or Y_{90}) from the experiment. The yield stress in the thickness direction can be written as

$$Y_{th} = Y_0 \sqrt{\frac{r_{90}(1+r_0)}{r_0+r_{90}}} = Y_{90} \sqrt{\frac{r_0(1+r_{90})}{r_0+r_{90}}}$$

Similarly, yield stresses in shear may be evaluated by

$$T_{12} = Y_{th} \sqrt{\frac{1}{2r_{45}+1}}$$

and

$$T_{23} = T_{31} = \frac{Y_a}{\sqrt{3}}$$

in which the transverse direction is assumed isotropic.

8. The pressure-dependent yielding, based on Drucker-Prager yield criterion, contains three options for frictional materials such as rocks and concrete. The generalized Mohr-Coulomb criterion introduces linear and parabolic models, developed by Drucker and Prager. The linear Mohr-Coulomb model assumes a linear function of hydrostatic stress for a yield function, i.e.,

$$aI_1 + \sqrt{J_2} - \frac{\bar{\sigma}}{\sqrt{3}} = 0 \quad \text{or} \quad \bar{\sigma} = \sqrt{3}\alpha I_1 + \sqrt{3J_2}$$

where

$$I_1 = \sigma_x + \sigma_y + \sigma_z$$

and

$$J_2 = \frac{1}{6}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]$$

The parameters α and $\bar{\sigma}$ (effective stress coinciding with the yield stress) can be related to material constants c (cohesion) and ϕ (frictional angle) by

$$c = \frac{\bar{\sigma}}{3\sqrt{(1 - 12\alpha^2)}}$$

and

$$\sin \phi = \frac{3\alpha}{\sqrt{1 - 3\alpha^2}}$$

The parabolic Mohr-Coulomb model allows a yield envelope to be parabolic in the plane strain case, for which the yield function is expressed as

$$\sqrt{3J_2 + \sqrt{3}\beta\bar{\sigma}I_1} - \bar{\sigma} = 0$$

in which parameters are related to the material constants by

$$\bar{\sigma}^2 = 3\left(c^2 - \frac{\alpha^2}{3}\right)$$

and

$$\beta = \frac{\alpha}{\sqrt{3(3c^2 - \alpha^2)}}$$

9. The keywords may appear in any order. However, aniso and press, are mutually exclusive, and cannot coexist.
10. All the alphanumeric fields are recognizable by the first four letters.
11. In SOL 400, MATEP is only supported for nonlinear elements with property extensions. This implies that for such elements, PBARL, PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Also, linear 2D triangular elements specified on PSHLN2 and 3D tetrahedral elements specified on PSLDN1 should be associated with an incompressible formulation (IPS for 2D plane strain, IAX for 2D axisymmetric and ISOL for 3D tetrahedral). If the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS, SPROPMAP,-1 in the bulk data. See remark 9. under the NLMOPTS
12. RYIELD = LINMOHR, PBLMOHR and WKHARD = CHABOCHE and JHCOOK are only applicable to isotropic materials (MAT1).

MATEP**Elasto-Plastic Material Properties - SOL 600**

Specifies elasto-plastic material properties to be used for large deformation analysis. Used in SOL 600 only.

Format (Note that the primary entry is required. All other continuation lines are required only for certain options and only one such option may be entered):

1	2	3	4	5	6	7	8	9	10
MATEP	MID	FORM	Y0	FID	RYIELD	WKHARD	METHOD	H	
	“Reffect”	OPTION	RTID	C	P				
	“Aniso”	N/A	R11 or M	R22 or C1	R33 or C2	R12 or C3	R23 or C6	R31	(Hill/Barlat)
	“ORNL”	OPTION	Yc10	TID	N/A				
	“Press”	OPTION	ALPHA	BETA	CRACKS	SOFTEN	CRUSHES	SRFAC	
	“Gurson”	Q1	Q2	INITIAL	CRITICAL	FAILURE	NUCL	MEAN	
		SDEV	NFRAC						
	“Chaboche”	R0	Rinf	B	C	Gam	Kap	N	
		Qm	μ	η					
	“PwrLaw”	A	M	B	N	$\sigma_0 \epsilon_0$			
	“Kumar”	B0	A	B1	B2	B3	T(B0)	T(A)	
		N	B4	B5	B6	T(N)			
	“JhCook”	A	B	N	C	M	Tmelt	Troom	
		$\dot{\epsilon}_0$ DOT							
	“YldOpt”	OPTION							
	“Units”	Uopt							
	“VParam”	Nvp	Vp1-1	Vp2-1	Vp1-2	Vp2-2			
	“ImpCreep”	VMISES							

Example:

MATEP	100	table		20		HILL	ADDRAD		
	ANISO		1.1	0.9		1.02			

Descriptor	Meaning
MID	Identification number of MAT1, MAT2, MATORT or MAT9 entry. (Integer > 0).
FORM	Selects a form of stress-plastic strain function to be specified (Character):
	SLOPE for defining non-zero H. (Default)
	TABLE for defining the function in TABLES1 or TABL3Di units of table are unspecified
	PERFECT for defining perfectly plastic material. (H is zero or blank)

Descriptor	Meaning
Y0	Initial yield stress Y_0 or hydrostatic stress for Mohr-Coulomb materials. See Remark 2. (Real > 0 or blank)
FID	Identification number of TABLES1 or TABL3Di entry. (Integer > 0 or blank)
RYIELD	Enter one of the following yield criteria rules. (Character, Default = vMises if there are no continuation lines. Otherwise, the default will be adjusted to agree with information on the continuation lines. For example, if the second line has PRESS in field 2 and LIN in field 3, the default for RYIELD will be LinMohr.)
ELASTIC	for purely elastic material (implies von Mises for multi-directional stresses). Vmises for von Mises yield criteria.
VMISES	von Mises yield criteria
HILL	for Hill's 1948 yield criteria.
BARLAT	for Barlat's 1991 yield criteria
LINMOHR	for Linear Mohr-Coulomb yield criteria. (See SOL 600 Notes/ Remarks regarding this entry in combination with MATS1. This entry overrides.)
PBLMOHR	for parabolic Mohr-Coulomb yield criteria. (See SOL 600 Notes/Remarks regarding this entry in combination with MATS1. This entry overrides.)
BUYUK	for the Buyukozturk concrete model.
ORNL	for one of the ORNL types. (see "Ornl" below)
GENPLAS	for Generalized Plasticity model
VISCPLAS	for Viscoplastic model using user subroutine UVSCPL
RIGID	for Rigid-Plastic material (no elasticity, von Mises yield)
IMPCREEP	for implicit creep model combining both plasticity and creep, von Mises yield criteria.
WKHARD	Selects a hardening rule defined by various work-hardening rules (Character):
ISOTROP	for isotropic hardening. (Default)
KINEM	for kinematic hardening.
COMBINE	for combination between kinematic and isotropic hardening
CHABOCHE	for Chaboche formulation - like combined (see Marc Vol C documentation for details for this model)
PWRLAW	for Power Law formulation (see Marc Vol C documentation for details of this model, for superplastic forming, this is the only available option)

Descriptor	Meaning
RPWRLAW	for Rate Power Law formulation (see Marc Vol C documentation for details of this model)
JHCOOK	for Johnson-Cook formulation (see Marc Vol C documentation for details of this model)
KUMAR	for Kumar formulation - (see Marc Vol C documentation for details of this model). See Remark 4.
METHOD	Selects a material processing method. See Remark 14. (Character):
ADDMEAN	for large strain additive decomposition using the mean normal process. (Default)
ADDRAD	for large strain additive decomposition using the radial return process. (SOL 600 only)
MULTRAD	for multiplicative decomposition (FeFp) using the radial return process
MULTPLN	for plane stress or strain FeFp multiplicative decomposition. (SOL 600)
SMLMEAN	for small strain additive decomposition using the mean normal process. (SOL 600 only)
SMLRAD	for small strain additive decomposition using the radial return process. (SOL 600 only)
H	plasticity modulus, ignored if FID field is specified. H has the same meaning as H in Remark 2 of the MATS1 entry. (Real ≥ 0 ; Default = 0.0)
“Reffect”	A keyword signifying that the following data pertains to the rate-dependent material properties (do not enter if rate-dependent effects are not to be included).
OPTION	Selects an option for strain-rate dependent yield stress (Character):
TABLE	for TABLES1 input. (Default)
COWPER	for Cowper and Symonds model. See Remark 6.
RTID	TABLES1 ID for strain-rate effects on yield stress, i.e., the table defines yield stress as a function of strain-rate starting from zero strain-rate. The yield stresses in this table should comprise the initial yield stress specified on Y0 or FID field at zero strain-rate.
C	Specifies the constant C for Cowper and Symonds model. (Real; Default = 1)
P	Specifies the constant P for Cowper and Symonds model. (Real; Default = 1)
“Aniso”	A keyword signifying that the following data (R_{ij}) pertain to the Hill’s (R_{ij}) or Barlat’s (M, C_i) yield criteria. See Remark 9. and Marc Vol A and C documentation (enter only RYIELD = HILL or BARLAT).
Rij	Stress ratios of initial yield stresses in various material directions to the reference yield stress from FID/Y0 field. (Real > 0 ; Default = 1.0 for R11, R22, R33, R12, R23, R31, respectively for Hill), or

Descriptor	Meaning
M	Barlat M coefficient (see Marc Vol A and C documentation)
C1	Barlat C1 coefficient (see Marc Vol A and C documentation)
C2	Barlat C2 coefficient (see Marc Vol A and C documentation)
C3	Barlat C3 coefficient (see Marc Vol A and C documentation)
C6	Barlat C6 coefficient (see Marc Vol A and C documentation)
"ORNL"	A keyword signifying that the following data pertains to the ORNL's yield criteria. Only enter if RYIELD = ORNL. See Remark 7.
OPTION	Selects an option for ORNL yield criteria (Character): NORM for normal ORNL model for stainless steel. (Default) CRMO for ORNL 2-1/4 Cr-Mo steel model. REVP for ORNL reversed plasticity model. ARST for ORNL full alpha reset model.
Yc10	Equivalent 10th cycle tensile yield stress. (Real > 0)
TID	Identification number of TABLES1 entry for normalized 10th cycle stress-plastic strain curve. (Integer; Default = 0)
"Press"	A keyword signifying that the following data pertain to the pressure-dependent yield criteria. Enter only if Ryield=LinMohr, PblMohr or concrete cracking is to be simulated. See Remarks 8. and 10.
OPTION	Selects an option for pressure-dependent yield criteria. (Character): LIN for linear Mohr-Coulomb model. (Default) PBL for parabolic Mohr-Coulomb model. CONC for Buyukozturk concrete model.
ALPHA	Specifies a parameter alpha for linear Mohr-Coulomb model or concrete model. (Real > 0 or blank)
BETA	Specifies a parameter beta for parabolic Mohr-Coulomb model or Buyukozturk concrete model (not used for linear Mohr-Coulomb model). (Real > 0 or blank)
CRACKS	Critical cracking stress for Buyukozturk concrete model. (Real > 0 or blank)
SOFTEN	Tension-softening modulus for Buyukozturk concrete model. (Real \geq 0; Default = 0)
CRUSHS	Strain at which material crushes. (Real > 0; Default = 1.E10)
SRFAC	Shear retention factor defining shear stress carrying capacity when crack closes for Buyukozturk concrete model. ($0 \leq$ Real < 1; Default = 0)
"Gurson"	A keyword signifying that the following data pertain to the modified Gurson model for porous metal plasticity with damage effects. Enter only if the Gurson damage model is to be used. See Remark 9.
Q1	First coefficient for the Gurson yield function. (Real \geq 0; Default = 1.5)

Descriptor	Meaning						
Q2	Second coefficient for the Gurson yield function. (Real ≥ 0 ; Default = 1)						
INITIAL	Initial void volume fraction. ($0 \leq \text{Real} < 1$; Default = 0)						
CRITICAL	Critical void volume fraction at which void coalescence starts. ($0 < \text{Real} < 1$; Default = 0.2)						
FAILURE	Failure void volume fraction at which the material loses strength. ($1./\text{q1} < \text{Real} < 1$; Default = 0.733)						
NUEL	Select a method of void nucleation (character, Default = strain): <table> <tr> <td>NONE</td><td>for no nucleation.</td></tr> <tr> <td>STRAIN</td><td>for plastic strain controlled nucleation.</td></tr> <tr> <td>STRESS</td><td>for stress controlled nucleation.</td></tr> </table>	NONE	for no nucleation.	STRAIN	for plastic strain controlled nucleation.	STRESS	for stress controlled nucleation.
NONE	for no nucleation.						
STRAIN	for plastic strain controlled nucleation.						
STRESS	for stress controlled nucleation.						
	See Remark 9.						
MEAN	Mean strain or stress for void nucleation. (Real > 0 ; Default = 0.3)						
SDEV	Standard deviation in the assumed normal distribution of nucleation strain or stress. (Real > 0 ; Default = 0.01)						
NFRAC	Volume fraction of nucleating particles for void nucleation. ($0 < \text{Real} < 0.5$; Default = 0.04)						
“Chaboche”	A keyword specifying the following data pertains to the Chaboche model (Enter only if WKHARD=Chaboche)						
R0	R0 for isotropic hardening. (Real)						
Rinf	Rinfinity for isotropic hardening (Q0 in case of using plastic-strain-range memorization). (Real)						
B	b coefficient for isotropic hardening. (Real)						
C	C coefficient for kinematic hardening (Real)						
Gam	Gamma coefficient for kinematic hardening (Real)						
Kap	Kappa value for viscosity model. (Real)						
N	n coefficient for viscosity model. (Real)						
Qm	Qm coefficient for isotropic hardening. (Real)						
μ	coefficient for isotropic hardening. (Real)						
η	coefficient to introduce progressive memory. (Real)						
“PwrLaw”	A keyword specifying the following data pertains to the Power Law or Rate Power Law model (see Marc Vol C, ISOTROPIC option for more details. Enter only if WKHARD = PWRLAW or RPWRLAW).						
A	Coefficient A. (Real)						
M	Coefficient m. (Real)						
B	Coefficient B. (Real)						

Descriptor	Meaning
N	Exponent n. (Real)
$\sigma_0 \varepsilon_0$	Enter σ_0 for rate power law or ε_0 for the power law model (Real) See Marc Vol C, ISOTROPIC option if this field is left blank or set to 0.0
"Kumar"	A keyword specifying the following data pertains to the Kumar model (see Marc Vol C , ISOTROPIC option for more details. Enter only if Wkhard =Kumar).
B0	Coefficient B0. (Real)
A	Coefficient A. (Real) If A=0.0, B1,B2, and B3 are used
B1	Coefficient B1. (Real)
B2	Coefficient B2. (Real)
B3	Coefficient B3. (Real)
T(B0)	TABLEM1 ID for coefficient B0. (Integer)
T(A)	TABLEM1 ID for coefficient A. (Integer)
N	Exponent n. (Real) If N=0.0, B4, B5 and B6 are used
B4	Coefficient B4. (Real)
B5	Coefficient B5. (Real)
B6	Coefficient B6. (Real)
T(N)	TABLEM1 ID for exponent n
"JhCook"	A keyword specifying the following data pertains to the Johnson-Cook model (see Marc Vol C , ISOTROPIC option for more details. Enter only if WKHARD =JHCOOK).
A	Coefficient A. (Real)
B	Coefficient B. (Real)
N	Exponent n. (Real)
C	Coefficient C. (Real)
M	Exponent m. (Real)
Tmelt	Melting temperature. (Real)
Troom	Ambient temperature. (Real)
$\dot{\varepsilon}_0$	Reference strain rate. (Real)
"YldOpt"	A keyword specifying one of the following yield models will be used.
OPTION	Select a yield option from the list below:
GENPL	General plasticity viscoplastic model using user subroutine UVSCPL
RIGID	Rigid-plastic material, no elasticity, vonMises yield
IMCREEP	Implicit creep model, both plasticity and creep with vonMises yield criteria

Descriptor	Meaning
"Units"	A keyword specifying if data is read from a database and if so, the units to be used if data is read from a database. (Enter only if necessary).
4	Read from Marc database for flow stress (default units)
5	Use MATILDA database (default units)
6	Data read from input file in SI-mm units
7	Data read from input file in SI-m units
8	Data read from input file in US units
"VParam"	A keyword specifying that the number of viscoplastic parameters. Up to two viscoelastic parameters are to be entered (used when RYIELD=ViscPlas)
Nvp	Number of viscoplastic parameters. (Integer must be 0, 1, or 2)
Vp1-1	First Viscoplastic parameter for Nvp=1. (Real; Default = 0.0)
Vp2-1	Second viscoplastic parameter for Nvp=1. (Real; Default = 0.0)
Vp1-2	First Viscoplastic parameter for Nvp=2. (Real; Default = 0.0, leave blank if Nvp=0 or Nvp=1)
Vp2-2	Second viscoplastic parameter for Nvp=2. (Real; Default = 0.0, leave blank if Nvp=0 or Nvp=1)
"ImpCreep"	A keyword specifying that the equivalent (von Mises) tensile yield stress will be entered in the next field (used when RYIELD=ImpCreep).
VMISES	Equivalent (von Mises) tensile yield stress. Overrides any similar entry made for this material elsewhere. (Real, no Default)

Remarks:

- Unless continuation entry is present specifying various material models, von Mises yield criterion is used as default.
- If Y_0 is not specified, FID field referring to a stress-plastic strain curve must be provided. If Y_0 is specified without FID field, the material is assumed perfectly plastic or has one slope, H. If both Y_0 and FID fields are specified, FID supersedes the Y_0 field and the first data point in TABLES1 represents Y_0 .

The initial yield point corresponds to the first data point in the function specified on the FID field.

$$Y(\bar{\varepsilon}^p) = Y_0(1 + b \bar{\varepsilon}^p)^n$$

where Y_0 is an initial yield stress, b and n are parameters characterizing the stress-strain relationship.

In case of an anisotropic material, the initial yield point corresponds to the reference yield stress (Y_a in Remark 7.).

- For SOL 600, the form of the stress-strain curve is determined by parameters MRTABLS1 and MRTABLS2.

4. The plastic deformation starts when the effective stress ($\bar{\sigma}$) exceeds the yield stress.

The yield stress is initially defined by the initial yield point, which is subsequently modified by the hardening rule to account for strain hardening. Under the isotropic hardening rule, the size of the yield surface expands as a function of effective plastic strain ($\dot{\varepsilon}^p$). Under the kinematic hardening rule, the center of the yield surface moves in stress space while keeping the same size and shape. Ziegler's law is used to define the translation of the yield surface. Under the combined hardening, the initial hardening is assumed to be entirely isotropic, but the elastic range attains a constant value (i.e., behaving like kinematic hardening) after some plastic straining. The effective stress for von Mises is expressed as

with

$$\bar{\sigma} = \sqrt{\frac{1}{2} [(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

where the stress components are measured from the center of yield surface.

$$\begin{aligned} R_{11} &= \frac{Y_1}{Y_a}, R_{22} = \frac{Y_2}{Y_a}, R_{33} = \frac{Y_3}{Y_a} \\ R_{12} &= \frac{\sqrt{3}T_{12}}{Y_a}, R_{23} = \frac{\sqrt{3}T_{23}}{Y_a}, R_{31} = \frac{\sqrt{3}T_{31}}{Y_a} \end{aligned}$$

where Y_1 , Y_2 , and Y_3 are the initial tensile yield stresses measured in material directions 1, 2 and 3, respectively; T_{12} , T_{23} , and T_{31} are the shear yield stresses in pure shear; and Y_a is the reference yield stress which should be an average yield stress in all directions.

In practical applications, however, the initial yield stress cannot be measured in all directions. The plastic anisotropy is pronounced in the sheet metal due to prior rolling process, for which the plastic anisotropy is customarily characterized by r-values defined by strain ratio measured in the uniaxial tension, i.e.,

$$r_a = \frac{\varepsilon_w}{\varepsilon_t} = \frac{l_n(w_o/w)}{l_n(t_o/t)} = \frac{H + (2N - F - G - 4H)\sin^2\alpha \cos^2\alpha}{F \sin^2\alpha + G \cos^2\alpha}$$

where t and w denote thickness and width, respectively; and α denotes the angle of orientation (usually measured from the rolling direction). Assuming that the anisotropy parameters stay constant throughout the deformation, F , G , H and N can be determined by r-values from tensile specimen cut at 0, 45 and 90 degrees to the rolling direction:

$$\frac{H}{G} = r_0 \quad , \quad \frac{H}{F} = r_{90}$$

$$\frac{N}{G} = \left(r_{45} + \frac{1}{2}\right) \left(1 + \frac{r_0}{r_{90}}\right)$$

The orthotropic plasticity parameters should be calculated from the r-values and the initial yield stress either in 0 or 90 degree direction (Y_0 or Y_{90}) from the experiment. The yield stress in the thickness direction can be written as

$$Y_{th} = Y_0 \sqrt{\frac{r_{90}(1+r_0)}{r_0+r_{90}}} = Y_{90} \sqrt{\frac{r_0(1+r_{90})}{r_0+r_{90}}}$$

Similarly, yield stresses in shear may be evaluated by

$$T_{12} = Y_{th} \sqrt{\frac{1}{2r_{45}+1}}$$

and

$$T_{23} = T_{31} = \frac{Y_a}{\sqrt{3}}$$

in which the transverse direction is assumed isotropic.

5. The Cowper and Symonds model scales the initial yield stress as a function of strain-rate, i.e.,

$$Y(\dot{\varepsilon}) = Y_0 \left[1 + \left(\frac{\dot{\varepsilon}}{C} \right)^{1/p} \right]$$

6. Hill's anisotropic model introduces orthotropic plastic material. This option can only be combined with orthotropic or anisotropic elastic material (i.e., with MAT2, MATORT or MAT9). The plastic anisotropy proposed by Hill introduces six parameters to the von Mises yield function, from which an effective stress may be derived as

$$\bar{\sigma} = \sqrt{F(\sigma_2 - \sigma_3)^2 + G(\sigma_3 - \sigma_1)^2 + H(\sigma_1 - \sigma_2)^2 + 2L\tau_{23}^2 + 2M\tau_{31}^2 + 2N\tau_{12}^2}$$

in which the material parameters can be related to the yield stress ratios by

$$F = \frac{1}{2} \left(\frac{1}{R_{22}^2} + \frac{1}{R_{33}^2} - \frac{1}{R_{11}^2} \right)$$

$$G = \frac{1}{2} \left(\frac{1}{R_{33}^2} + \frac{1}{R_{11}^2} - \frac{1}{R_{22}^2} \right)$$

$$H = \frac{1}{2} \left(\frac{1}{R_{11}^2} + \frac{1}{R_{22}^2} - \frac{1}{R_{33}^2} \right)$$

$$L = \frac{3}{2R_{23}^2}, M = \frac{3}{2R_{31}^2}, N = \frac{3}{2R_{12}^2}$$

with

$$\begin{aligned} R_{11} &= \frac{Y_1}{Y_a}, R_{22} = \frac{Y_2}{Y_a}, R_{33} = \frac{Y_3}{Y_a} \\ R_{12} &= \frac{\sqrt{3}T_{12}}{Y_a}, R_{23} = \frac{\sqrt{3}T_{23}}{Y_a}, R_{31} = \frac{\sqrt{3}T_{31}}{Y_a} \end{aligned}$$

where Y_1 , Y_2 , and Y_3 are the initial tensile yield stresses measured in material directions 1, 2 and 3, respectively; T_{12} , T_{23} , and T_{31} are the shear yield stresses in pure shear; and Y_a is the reference yield stress which should be an average yield stress in all directions.

In practical applications, however, the initial yield stress cannot be measured in all directions. The plastic anisotropy is pronounced in the sheet metal due to prior rolling process, for which the plastic anisotropy is customarily characterized by r-values defined by strain ratio measured in the uniaxial tension, i.e.,

$$r_a = \frac{\varepsilon_w}{\varepsilon_t} = \frac{l_n(w_o/w)}{l_n(t_o/t)} = \frac{H + (2N - F - G - 4H)\sin^2\alpha \cos^2\alpha}{F \sin^2\alpha + G \cos^2\alpha}$$

where t and w denote thickness and width, respectively; and α denotes the angle of orientation (usually measured from the rolling direction). Assuming that the anisotropy parameters stay constant throughout the deformation, F , G , H and N can be determined by r-values from tensile specimen cut at 0, 45 and 90 degrees to the rolling direction:

$$\frac{H}{G} = r_0, \quad \frac{H}{F} = r_{90}$$

$$\frac{N}{G} = \left(r_{45} + \frac{1}{2}\right) \left(1 + \frac{r_0}{r_{90}}\right)$$

The orthotropic plasticity parameters should be calculated from the r-values and the initial yield stress either in 0 or 90 degree direction (Y_0 or Y_{90}) from the experiment. The yield stress in the thickness direction can be written as

$$Y_{th} = Y_0 \sqrt{\frac{r_{90}(1+r_0)}{r_0+r_{90}}} = Y_{90} \sqrt{\frac{r_0(1+r_{90})}{r_0+r_{90}}}$$

Similarly, yield stresses in shear may be evaluated by

$$T_{12} = Y_{th} \sqrt{\frac{1}{2r_{45}+1}}$$

and

$$T_{23} = T_{31} = \frac{Y_a}{\sqrt{3}}$$

in which the transverse direction is assumed isotropic.

7. The elasticity constants must be isotropic for ORNL plasticity except for normal ORNL model for stainless steel. The 10th cycle stress-plastic strain curve in TID field should be a normalized function so that the yield stress at zero plastic strain is unity.
8. The pressure-dependent yielding, based on Drucker-Prager yield criterion, contains three options for frictional materials such as rocks and concrete. The generalized Mohr-Coulomb criterion introduces linear and parabolic models, developed by Drucker and Prager. The linear Mohr-Coulomb model assumes a linear function of hydrostatic stress for a yield function, i.e.,

$$\alpha I_1 + \sqrt{J_2} - \frac{\bar{\sigma}}{\sqrt{3}} = 0 \quad \text{or} \quad \bar{\sigma} = \sqrt{3}\alpha I_1 + \sqrt{3J_2}$$

where

$$I_1 = \sigma_x + \sigma_y + \sigma_z$$

and

$$J_2 = \frac{1}{6}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]$$

The parameters α and $\bar{\sigma}$ (effective stress coinciding with the yield stress) can be related to material constants c (cohesion) and ϕ (frictional angle) by

$$c = \frac{\bar{\sigma}}{3\sqrt{(1 - 12\alpha^2)}}$$

and

$$\sin\phi = \frac{3\alpha}{\sqrt{1 - 3\alpha^2}}$$

The parabolic Mohr-Coulomb model allows a yield envelope to be parabolic in the plane strain case, for which the yield function is expressed as

$$\sqrt{3J_2 + \sqrt{3}\beta\bar{\sigma}I_1} - \bar{\sigma} = 0$$

in which parameters are related to the material constants by

$$\bar{\sigma}^2 = 3\left(c^2 - \frac{\alpha^2}{3}\right)$$

and

$$\beta = \frac{\alpha}{\sqrt{3(3c^2 - \alpha^2)}}$$

9. The Drucker-Prager plasticity models can only be combined with isotropic elasticity. The Buyukozturk concrete plasticity model is a particular form of the generalized Drucker-Prager plasticity model, which is developed specifically for plane stress cases by Buyukozturk. The Buyukozturk yield function is expressed as

$$\beta\sqrt{3}Y I_1 + \gamma I_1^2 + 3J_2 - Y^2 = 0$$

where β is a user-specified constant, γ is an internal parameter (set to 0.2) with no user's access, and Y is the yield stress.

The Buyukozturk concrete plasticity model is coupled with crack and crush capability, which is designed for a low-tension material. The low-tension material develops a crack in the perpendicular direction to the maximum principal stress when it exceeds a critical value. The tension softening modulus can be specified (in absolute value) by the user to process the cracking process gradually. The default value (0.) is intended for a sudden cracking with a complete loss of the stiffness upon cracking. After the initial crack, a second crack can initiate in the perpendicular direction to the first crack. Likewise, the third crack can be formed in 3D solid elements. The loading may reverse the direction after the crack is formed. In this case, the crack will close and some load carrying capacity is resumed. The compression capability is fully resumed and the shear stresses may be transmitted over the crack surface with a reduced stiffness by a factor specified as shear retention factor. The material may fail in compression by crushing. The input value for the crush strain is positive, which implies an absolute value of a compressive strain. The material loses its integrity for good upon crushing. The reinforcement bars may be simulated by adding REBAR elements.

10. The Gurson model for porous metal plasticity may be used only with isotropic hardening rule. All other hardening rules will be ignored if Gurson model is selected. The Gurson model modified by Tvergaard and Needleman is designed for porous metal plasticity with damage effects in the ductile material. The material is assumed to form voids under loading, which grow, coalesce, then leads to crack formation and eventually failure. This process is a function of hydrostatic stress and the void volume fraction f_v . The yield function is established as follows:

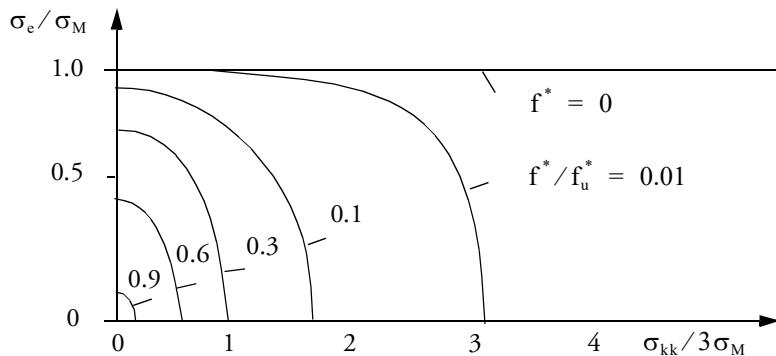
$$\frac{3J_2}{Y^2} + 2q_1 f_v^* \cosh\left(\frac{q_2 I_1}{2Y}\right) - [1 - (q_1 f_v^*)^2] = 0$$

in which Y denotes a yield stress of the fully dense matrix material, I_1 denotes the first invariant of stresses, and the modified void volume fraction f_v^* is computed by

$$f_v^* = f_v \quad \text{if } f_v \leq f_v^c$$

$$f_v^* = f_v^c + \left(\frac{f_v^u - f_v^c}{f_v^f - f_v^c} \right) (f_v - f_v^c) \quad \text{if } f_v > f_v^c$$

where f_v^c is the critical void volume fraction, f_v^f is the void volume at failure and $f_v^u = 1/q_1$. The solid loses all stress carrying capability when the void volume fraction reaches f_v^f .



11. The evolution of damage as measured by void volume fraction is due to void nucleation and void growth. Void nucleation occurs by debonding of the second phase particles. The strain for nucleation depends on the particle sizes. Assuming a normal distribution of particle sizes, the void nucleation itself is modeled as a normal distribution in strains if nucleation is strain-controlled. If the void nucleation is assumed to be stress controlled in the matrix, a normal distribution is assumed in stresses. The void volume fraction changes due to the growth of existing voids and nucleation of new voids, i.e.,

$$\dot{f}_v = \dot{f}_{\text{growth}} + \dot{f}_{\text{nucleation}}$$

in which the void growth can be determined based on the compressibility of the material

$$\dot{f}_{\text{growth}} = (1 - f_v) \dot{\epsilon}_{kk}^p$$

and the nucleation can be defined either as strain or stress-controlled with a normal distribution about the mean value. In case of strain-controlled nucleation, the rate is expressed as

$$\dot{f}_{\text{nucleation}} = \frac{f_v^n}{S \sqrt{2\pi}} \text{Exp} \left[-\frac{1}{2} \left(\frac{\bar{\epsilon}_m^p - \epsilon_n}{S} \right)^2 \right] \dot{\epsilon}_m^p$$

where f_v^n is the volume fraction of void forming particles, $\bar{\epsilon}_m^p$ denotes the effective plastic strain in the matrix material, and the void nucleation strain is assumed normally distributed with a mean value of ϵ_n and a standard deviation of S . In case of stress-controlled nucleation, the rate is expressed as

$$\dot{f}_{\text{nucleation}} = \frac{f_v^n}{S \sqrt{2\pi}} \text{Exp} \left[-\frac{1}{2} \left(\frac{\sigma + \frac{1}{3}\sigma_{kk} - \sigma_n}{S} \right)^2 \right] \left(\frac{\dot{\sigma}}{\sigma} + \frac{1}{3} \frac{\dot{\sigma}_{kk}}{\sigma} \right)$$

If the size of the second phase particles are widely dispersed, the standard deviation would be larger than more uniform cases. A typical values for an engineering alloy as suggested by numerical experiments are set as default values for ϵ_n , S , and f_v^n .

12. The keywords may appear in any order. However, aniso, ORNL, press, and Gurson are mutually exclusive, and cannot coexist.
13. All the alphanumeric fields are recognizable by the first four letters.
14. For SOL 600, METHOD (first line, field 8) corresponds to Marc's PLASTICITY options as follows. METHOD must be the same for all MATEP entries in the SOL 600 input but this is not enforced. The value of METHOD on MATEP with the smallest MID governs who the PLASTICITY option is set. If METHOD is left blank, it may alternatively be set by PARAM,MARCPPLAS.

Name	Marc PLASTICITY parameter value
Addmean	3
Addrad	4
Multrad	5
Multpln	6
Smlmean	1
Smlrad	2

15. The Bulk Data fields denoted by N/A are different from blank fields. Those fields with N/A are not used currently, but the space is reserved in case of future additions. On the other hand, the blank fields which have no specifications are neither used nor reserved (any memory space) for future use. The Method field determines the options under Marc parameter PLASTICITY.

The initial yield stress should be extracted from the table provided in the FID field. The yield stress-plastic strain function specified in TABLES1 under FID field is not a normalized function.

The anisotropic material parameters Rij are equivalent to Marc input data as follows:

R11 = YRDIR1
 R22 = YRDIR2
 R33 = YRDIR3
 R12 = YRSHR1
 R23 = YRSHR2
 R31 = YRSHR3

The crack/crush capability in Marc may be combined with other isotropic material options.

16. The strain effect on the yield stress (Reffect) is specified under the model definition option STRAIN RATE in Marc.
17. This Bulk Data entry accommodates Marc's input data under the model definition options ISOTROPIC, ORTHOTROPIC, ANISOTROPIC, RATE EFFECTS, WORK HARD, DAMAGE and CRACK DATA as well as the parameter PLASTICITY.
18. The fields without default values can be left blank only if they are specifically permitted to be blank.
19. RYIELD equates to Marc's ISOTROPIC (3,2) field.
20. WKHARD equates to Marc's ISOTROPIC (3,3) field.

21. RYIELD = LINMOHR, PBLMOHR and WKHARD = CHABOCHE, PWRLAW, RPWRLAW, JHCOOK, and KUMAR are only applicable to isotropic materials (MAT1).

MATEP**Elasto-Plastic Material - SOL 700**

Elasto-plastic material properties for SOL 700 only.

Format:

Note that the primary entry is required. All other continuation lines are required only for certain options and only one such option may be entered.

1	2	3	4	5	6	7	8	9	10
MATEP	MID	FORM	Y0	FID	RYIELD		TYPE	H	
	“Reffect”	OPTION1	RTID	C	P				
	“JhCook”	A	B	N	C	M	Tmelt	Troom	
		ε0DOT	CP						
	“PwrLaw”	A	M	B	N	C			
	“Poly”	A	B	C	D	E	F	YMAX	
	“TMmodel”	A	B	C	D	M	εSM	Tmelt	
		Troom	YCR	E	K	ε0DOT	CP		
	“ZAmodel”	A	B	N	C	M	ε0DOT	D	
		CP							
	“OrthoCr”	E	NU	RELV	TYPE	TIDXX	TIDYY	TIDZZ	
		TIDXY	TIDYZ	TIDZX	TIDSР				
	“Soil”	TABLE	TYPE	VALUE	CUTOFF	A0	A1	A2	
		YIELD	YSTYP						

Example:

MATEP	7	SLOPE	1.2E6		VMISES			1.0E3	
	“Reffect”	COWPER		2.0	4.0				

Descriptor	Meaning
MID	Identification number of MAT1 or MATORT entry. MATORT is only acceptable for RYIELD=ORTHOCR and MAT1 is acceptable for all other yield criteria except ORTHOCR. (Integer > 0; required.)
FORM	Selects a form of stress-plastic strain function to be specified for RYIELD=VMISES. (Character; default=SLOPE):
SLOPE	Defines non-zero H.
TABLE	Defines the function using TABLED1 entry. Units of table are unspecified.
PERFECT	Defines perfectly plastic material (H is zero or blank and FID is blank).

Descriptor	Meaning
Y0	Initial yield stress for RYIELD=VMISES and ORTHOCR. (Real > 0.0; default=blank)
FID	Identification number of TABLED1 for RYIELD=VMISES. (Integer > 0; default=blank)
RYIELD	Enter one of the following yield criteria rules. (Character; Default = VMISES)
VMISES	von Mises yield criteria. It supports “Reffect” option. See Remarks 1.-8 .
JHCOOK	For Johnson-Cook formulation. It supports “JhCook” option. See Remarks 9. and 10 .
RPWRLAW	Rate power law. It supports “PwrLaw” option. See Remark 11 .
POLY	Polynomial Yield Model. It supports “Poly” option. See Remark 12 .
TMMODEL	Tanimura-Mimura Yield Model. It supports “TMmodel” option. See Remark 13 .
ZAMODEL	Zerilli-Armstrong Yield Model. It supports “ZAmode” option. See Remarks 14 .
ORTHOCR	Orthotropic Crushable Model. It supports “OrthoCr” option. It is only available for solid elements with MATORT. See Remarks 15.-18 .
SOIL	Elastoplastic Crushable Model. It supports “Soil” option. It is only available for solid elements. See Remarks 19.-22 .
TYPE	The type of stress and strain defined in FID. (Character; default=TRUE)
ENG	Engineering stress and strain.
TRUE	True stress and strain.
PLAST	True stress and plastic strain.
PMOD	Plastic modulus and true stress.
H	Plasticity modulus. (Real > 0; Default = 0).
“Reffect”	A keyword signifying that the following data pertains to the rate-dependent material properties (do not enter rate-dependent effects if they are not to be included). If TABLED1 is entered and one of the independent variables is strain rate, this field and its options should not be specified.
OPTION1	Selects an option for strain-rate dependent yield stress (Character; default=TABLE):
TABLE	Specifies that RTID input used to define rate dependency.
COWPER	Specifies the Cowper and Symonds rate-dependency model.
RTID	TABLED1 ID for strain-rate effects on yield stress, i.e., the table defines yield stress as a function of strain-rate starting from zero strain-rate. The yield stresses in this table should comprise the initial yield stress specified on Y0 or FID field at zero strain-rate.

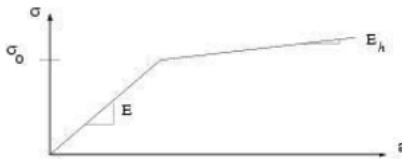
Descriptor	Meaning
C	Specifies the constant C for Cowper and Symonds model. (Real; Default = 1.0)
P	Specifies the constant P for Cowper and Symonds model. (Real; Default = 1.0)
"Jhcook"	Johnson-Cook yield model where the yield stress is a function of effective plastic strain, strain rate, and temperature.
A	Static yield stress. (Real > 0.0; required.)
B	Hardening parameter. (Real; default=0.0)
N	Hardening exponent. (Real; default=1.0)
C	Strain-rate parameter. (Real; default=0.0)
M	Temperature exponent. (Real; default=1.0)
TMELT	Melt temperature. (Real; default=1.E20)
TROOM	Room temperature. (Real; default=293.0)
$\dot{\epsilon}$ 0DOT	Reference strain rate. (Real > 0.0; default=1.0)
CP	Specific heat. (Real; default=1.E20)
"PwrLaw"	Rate power law
A	Hardening parameter. (Real; default=0.0)
M	Hardening exponent. (Real; default=1.0)
B	Initial yield parameter. (Real > 0.0; required)
N	Strain rate exponent. (Real; default=1.0)
C	Minimum yield stress. (Real; default=1.E20)
"Poly"	Polynomial Yield Model
A	Initial yield parameter. (Real > 0.0; required)
B	Coefficient B. (Real; default=0.0)
C	Coefficient C. (Real; default=0.0)
D	Coefficient D. (Real; default=0.0)
E	Coefficient E. (Real; default=0.0)
F	Coefficient F. (Real; default=0.0)
YMAX	Maximum yield stress. (Real; default=1.E20)
"SGmodel"	Steinberg-Guinan Yield Model
A1~A4	Yield parameters. (Real > 0.0; required.)
H, B	Yield parameters. (Real; default=0.0)
TMELT	Melt temperature. (Real; default=1.E20)
TROOM	Room temperature. (Real; default=293.0)
CP	Specific heat. (Real; default=1.E20)

Descriptor	Meaning					
"TMmodel"	Tanimura-Mimura Yield Model					
A	Static yield parameter. (Real > 0.0; Required)					
B	Hardening parameter. (Real; default=0.0)					
C	Strain rate parameter. (Real; default=0.0)					
D	Strain rate parameter. (Real; default=0.0)					
M	Temperature exponent. (Real; default=1.0)					
ϵ_{SM}	Quasi-static strain rate. (Real > 0.0; default=1.0)					
TMELT	Melt temperature. (Real; default=1.E20)					
TROOM	Room temperature. (Real; default=293.0)					
YCR	Critical yield stress. (Real > 0.0; default=1.E20)					
E	Strain rate parameter. (Real; default=0.0)					
K	Strain rate exponent. (Real; default=1.0)					
ϵ_0 DOT	Reference strain rate. (Real > 0.0; default=1.0)					
CP	Specific heat. (Real; default=1.E20)					
"ZAmodel"	Zerilli-Armstrong Yield Model.					
A	Static yield parameter. (Real > 0.0; required)					
B	Hardening parameter. (Real > 0.0; default=0.0)					
N	Hardening exponent. (Real > 0.0; default=1.0)					
C	Strain rate parameter. (Real; default=0.0)					
M	Temperature exponent. (Real; default=1.0)					
ϵ_0 DOT	Reference strain rate. (Real > 0.0; default=1.0)					
D	Bcc parameter. (Real; default=0.0) <table style="margin-left: 20px;"> <tr> <td>= 0</td> <td>Fcc metal</td> </tr> <tr> <td>$\neq 0$</td> <td>Bcc metal</td> </tr> </table>		= 0	Fcc metal	$\neq 0$	Bcc metal
= 0	Fcc metal					
$\neq 0$	Bcc metal					
CP	Specific heat. (Real; default=1.E20)					
"OrthoCr"	Orthotropic Crushable Model.					
E	Young's modulus for the fully compacted material. (Real > 0.0; required)					
NU	Poisson's ratio for the fully compacted material. (-1.0 < Real < 0.5; required)					
RELV	Relative volume at which the material is fully compacted. (0.0 < Real < 1.0; required)					
TYPE	The type of data defined as the x-value in TIDXX-TIDS. (Character; default=CRUSH) <table style="margin-left: 20px;"> <tr> <td>CRUSH</td> <td>Crush factor (1-relative volume)</td> </tr> <tr> <td>RELVOL</td> <td>Relative volume V/V0</td> </tr> </table>		CRUSH	Crush factor (1-relative volume)	RELVOL	Relative volume V/V0
CRUSH	Crush factor (1-relative volume)					
RELVOL	Relative volume V/V0					

Descriptor	Meaning						
TIDXX	TABLED1 ID defining the variation of the (local) xx-stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)						
TIDYY	TABLED1 ID defining the variation of the (local) yy-stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)						
TIDZZ	TABLED1 ID defining the variation of the (local) zz-stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)						
TIDXY	TABLED1 ID defining the variation of the (local) xy-shear stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)						
TIDYZ	TABLED1 ID defining the variation of the (local) yz-shear stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)						
TIDZX	TABLED1 ID defining the variation of the (local) zx-shear stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)						
TIDSR	TABLED1 ID defining the variation of a yield factor (y-value) with the deviatoric strain rate (x-value). (Integer > 0; default=not used)						
“Soil”	Elastoplastic Crushable Model.						
TABLE	TABLED1 ID defining the variation of pressure (y-value) with crush factor or volumetric strain (x- value). (Integer > 0; required)						
TYPE	The type of data defined as the x-value in the table. (Real > 0.0; default=1.0) <table> <tr> <td>1.0</td><td>Crush factor. (= 1-relative volume)</td></tr> <tr> <td>2.0</td><td>Volumetric (true) strain</td></tr> </table>	1.0	Crush factor. (= 1-relative volume)	2.0	Volumetric (true) strain		
1.0	Crush factor. (= 1-relative volume)						
2.0	Volumetric (true) strain						
VALUE	The value for cut-off stress. (Real < 0.0; See Remark 20.)						
CUTOFF	Cut-off stress. See Remark 21. (Real > 0.0; default=2.0) <table> <tr> <td>1.0</td><td>Pressure for total tensile failure</td></tr> <tr> <td>2.0</td><td>Pressure for tensile failure</td></tr> <tr> <td>3.0</td><td>Minimum pressure</td></tr> </table>	1.0	Pressure for total tensile failure	2.0	Pressure for tensile failure	3.0	Minimum pressure
1.0	Pressure for total tensile failure						
2.0	Pressure for tensile failure						
3.0	Minimum pressure						
A0, A1, A2	Yield function constants. (Real; default=0.0)						
YIELD	Surface description. See Remark 22. (Real > 0.0; default=1.0) <table> <tr> <td>1.0</td><td>The yield surface is defined as a function of p and J2.</td></tr> <tr> <td>2.0</td><td>The yield surface is defined as a function of p and sy.</td></tr> </table>	1.0	The yield surface is defined as a function of p and J2.	2.0	The yield surface is defined as a function of p and sy.		
1.0	The yield surface is defined as a function of p and J2.						
2.0	The yield surface is defined as a function of p and sy.						
YSTYP	Type of YSURF Yield Surface description. See Remark 22. (Real > 0.0; default=1.0) <table> <tr> <td>1.0</td><td>DYNA definition</td></tr> <tr> <td>2.0</td><td>Dytran additional definition</td></tr> </table>	1.0	DYNA definition	2.0	Dytran additional definition		
1.0	DYNA definition						
2.0	Dytran additional definition						

Remarks:

1. A bilinear stress-strain characteristic is used by specifying FORM=SLOPE, Y0 and H:



where the equivalent stress is given by $\sigma = \sigma_0 + \frac{E \cdot E_h}{E - E_h} \varepsilon_p$

where

σ_0 = yield stress specified in the Y0 field

E = Young's modulus

E_h = hardening modulus specified in the H field

ε_p = equivalent plastic strain

σ = equivalent stress

2. A piecewise linear, stress-strain characteristic is used by specifying FID and TYPE (beams and shells only)

$$\sigma = [(\sigma_i - \sigma_{i-1})(\varepsilon - \varepsilon_{i-1}) / (\varepsilon_i - \varepsilon_{i-1})] + \sigma_{i-1}$$

The stress-strain characteristic used internally in Nastran SOL700 is in terms of true stress and equivalent plastic strain. However, for convenience, the stress-strain characteristic can be input in any of the following ways:

True stress/true strain	(TYPE = TRUE)
-------------------------	---------------

Engineering stress/engineering strain	(TYPE = ENG)
---------------------------------------	--------------

True stress/plastic strain	(TYPE = PLAST)
----------------------------	----------------

Plastic modulus/true stress	(TYPE = PMOD)
-----------------------------	---------------

3. With solid elements, only an elastic perfectly plastic yield model is currently used. Only the Y0 field is used.
4. If FID is blank or zero, a bilinear stress-strain curve is assumed. If FID has a value, it refers to a TABLED1 entry giving the stress-strain curve for the material.
5. If FID is defined, the value of YIELD is left blank, since it is determined from the stress-strain curve.
6. If RTID is blank or zero and C and P are blank or zero, the yield stress does not vary with strain rate. If RTID has a value, then it references a TABLED1 entry, which gives the variation of the scale factor applied to the yield stress with strain rate. (C and P must be blank or zero.)

If RTID is blank or zero and C and P are defined, the enhancement of the yield stress with strain rate is calculated as

$$\frac{\sigma_d}{\sigma_y} = I + \left(\frac{\dot{\varepsilon}_p}{C} \right)^{1/p}$$

where σ_d is the dynamic stress, σ_y is the static yield stress, and $\dot{\varepsilon}_p$ is the equivalent plastic strain rate.

7. If TYPE is PLAST or PMOD, Young's modulus must be defined. If TYPE is ENG or TRUE and Young's modulus is defined, it overrides the value calculated from the stress-strain curve.
8. Note that for values exceeding the maximum x-value of either of the TABLED1 entries (see FID and RTID fields), linear extrapolation is used based upon the last two points specified in the TABLED1.
9. The Johnson Cook yield stress is computed from

$$\sigma_y = (A + B\varepsilon_p^N) \left[1 + C \ln \left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \right) \right] (1 - T^{*M})$$

where

ε_p	= effective plastic strain
-----------------	----------------------------

T^*	= $\frac{T - T_{ROOM}}{T_{MELT} - T_{ROOM}}$
-------	--

$\dot{\varepsilon}$	= reference strain rate
---------------------	-------------------------

$\dot{\varepsilon}_0$	= reference strain rate
-----------------------	-------------------------

T	= temperature
---	---------------

TROOM	= room temperature
-------	--------------------

TMELT	= melt temperature
-------	--------------------

A, B, N, C, and M are input constants.

10. The reference strain rate for Johnson Cook yield model is per unit time.

11. The rate power law yield stress is computed from

$$\sigma_y = \text{MAX}(C, B + A\varepsilon_p^M \dot{\varepsilon}^N)$$

where

ε_p	= effective plastic strain
-----------------	----------------------------

$\dot{\varepsilon}$	= reference strain rate
---------------------	-------------------------

A, B, M,N and C are constants.

12. The polynomial yield stress is computed from

$$\sigma_y = \text{MIN}(\text{YMAX}, A + B\epsilon_p + C\epsilon_p^2 + D\epsilon_p^3 + E\epsilon_p^4 + F\epsilon_p^5)$$

where

ϵ_p	= effective plastic strain
YMAX	= maximum yield stress

A, B, C,D,E and F are constants.

13. The Tanimura-Mimura yield stress is computed from

$$\sigma_y = \left[A + B\epsilon_p + (C + D\epsilon_p) \left(1 - \frac{A + B\epsilon_p}{\sigma_{cr}} \right) \ln \left(\frac{\dot{\epsilon}}{\dot{\epsilon}_s} \right) \right] (1 - T^{*M}) + E \left(\frac{\dot{\epsilon}}{\dot{\epsilon}_0} \right)^K$$

where

ϵ_p	= effective plastic strain
σ_{cr}	= critical yield stress
$\dot{\epsilon}$	= effective strain rate
$\dot{\epsilon}_s$	= quasi-static strain rate
$\dot{\epsilon}_0$	= reference strain rate
T^*	$= \frac{T - TROOM}{TMELT - TROOM}$
T	= temperature
$TROOM$	= room temperature
$TMELT$	= melt temperature

A, B, D, M, E and K are constants

14. The Zerilli-Armstrong yield stress is computed from

$$\sigma_y = \begin{cases} (A + B\epsilon_p^N) e^{\left[-MT + CT \ln \left(\frac{\dot{\epsilon}}{\dot{\epsilon}_0} \right) \right]} & \text{for Fcc metals} \\ (A + B\epsilon_p^N) + De^{\left[-MT + CT \ln \left(\frac{\dot{\epsilon}}{\dot{\epsilon}_0} \right) \right]} & \text{for Bcc metals} \end{cases}$$

where

ε_p	= effective plastic strain
-----------------	----------------------------

$\dot{\varepsilon}$	= effective strain rate
---------------------	-------------------------

$\dot{\varepsilon}_0$	= reference strain rate
-----------------------	-------------------------

T	= temperature
-----	---------------

A, B, N,C,M and D are constants.

15. If the initial Poisson's ratios in MATORT are not supplied, the default is set to zero. Therefore, the behavior of the material during compaction is uncoupled. This means that straining in the (local) x-direction produces stresses only in the (local) x-direction, and not in the (local) y- or z-direction. The tables define the variation of the stress in a particular direction with the relative volume or the crush. The relative volume is defined as (current volume)/(initial volume) and varies from 1.0 (uncompressed) to 0.0 (zero volume). Crush is defined as one minus the relative volume and varies from 0.0 to 1.0. Since the tables should be defined with increasing x-values, it is convenient to use the default value for type, which is CRUSH. When defining the curves, care should be taken that the extrapolated values do not lead to negative yield stresses.
16. The elastic moduli (and the initial Poisson's ratios only if they are supplied) in MATORT vary linearly with the relative volume from their initial uncompacted values to the fully compacted ones.
17. When the material is fully compacted, its behavior becomes isotropic with an elastic perfectly plastic material characteristic.
18. If the TIDSR option is used, you can supply a table including strain-rate effects. Strain rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor; i.e.,

$$\dot{\varepsilon} = \sqrt{\varepsilon_{ij}^{dev} \cdot \varepsilon_{ij}^{dev}}$$

The y values in this table are factors with which the stresses in the other tables are multiplied to incorporate strain-rate effects.

19. The pressure-volume characteristic can either be defined in terms of the amount of crush, which is minus the engineering strain and is defined as $(1 - \frac{V}{V_0})$, with $\frac{V}{V_0}$ as the relative volume; or in terms of the volumetric (true) strain which is defined as:

$$\int_{V_0}^V \frac{dV}{V}$$

or

$$\ln\left(\frac{V}{V_0}\right)$$

The crush factor must be between 0 and 1. The volumetric strain must always be negative.

20. If the field for the value of CUTOFF is left blank, then this value is calculated from the yield function defined by the constants A0, A1, and A2. In case of a Mohr-Coulomb yield model, the cut-off pressure is calculated as the root of the pressure-yield stress curve. If the YIELD=1.0 is used, the cut-off pressure is calculated as the intersection point of the yield surface with the hydrostat (if only A0 is nonzero, then the cut-off pressure is set to -100K, where K is the bulk modulus). The cut-off pressure must be negative.
21. Either a minimum pressure (CUTOFF=3.0) or a failure pressure (CUTOFF=1.0 or 2.0) can be specified. In the first case, this corresponds to a tensile cutoff, where the pressure cannot fall below the minimum value. In the second case, if the pressure falls below the failure pressure the element fails and cannot carry tensile loading for the remainder of the analysis. Thus, the pressure can never become negative again. If CUTOFF=1.0 is used, the elements can physically fail, which means that the stresses are set to zero, but also the failure flag is used as in normal MATF models. If CUTOFF=2.0 is used, only the stresses are set to zero.
22. If YIELD=1.0 is used, the yield stress is determined by a Mohr-Coulomb model:

$$\sigma_y = \text{MIN}(A_0 + A_1 p, A_2)$$

If YIELD=2.0 is used, the yield surface in three-dimensional space is defined by $s_s=0$ here

$$\Phi_s = \frac{1}{2} S_{ij} S_{ij} - (B_0 + B_1 p + B_2 p^2) = J_2 - (B_0 + B_1 p + B_2 p^2)$$

where S_{ij} are the deviatoric stresses and J_2 is the second invariant of the stress deviation. The coefficients B_0 , B_1 and B_2 can be related to the coefficients A_0 , A_1 and A_2 . The relation between the coefficients depends on the YSTYP field as shown below.

If YSTYP=2.0, then

$$B_0 = A_0$$

$$B_1 = A_1$$

$$B_2 = A_2$$

Thus, the yield stress is defined as

$$\sigma_y = \sqrt{3(A_0 + A_1 p + A_2 p^2)}$$

If YSTYP=1.0, then

$$B_0 = \frac{1}{3}A_0^2$$

$$B_1 = \frac{2}{3}A_0 A_1$$

$$B_2 = \frac{1}{3}A_1^2$$

and A2 is ignored.

Thus, the yield stress is defined as

$$\sigma_y = A_0 + A_1 p$$

MAT1F**Isotropic Material Frequency Dependence**

Specifies frequency-dependent material properties on MAT1 entry fields via TABLEDi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1F	MID	E	G	NU	RHO			GE	

Example:

MAT1F	33	15							
-------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)
E	Identification number of a TABLEDi; i=1 to 4 entry for the Young's modulus. (Integer > 0 or blank)
G	Identification number of a TABLEDi; i=1 to 4 entry for the shear modulus. (Integer > 0 or blank)
NU	Identification number of a TABLEDi; i=1 to 4 entry for the Poisson's ratio. (Integer > 0 or blank)
GE	Identification number of a TABLEDi; i=1 to 4 entry for the structural damping coefficient. (Integer > 0 or blank)
RHO	Identification number of a TABLEDi; i=1 to 4 entry for the mass density. (Integer > 0 or blank). This table is only used for isotropic poroelastic material with BEGIN BULK TRMC and is applicable to that trim component only.

- | | |
| --- | --- |
| MID | Material property identification number that matches the identification number on MAT1 entry. (Integer > 0) |
- | | |
| --- | --- |
| E | Identification number of a TABLEDi; i=1 to 4 entry for the Young's modulus. (Integer > 0 or blank) |
- | | |
| --- | --- |
| G | Identification number of a TABLEDi; i=1 to 4 entry for the shear modulus. (Integer > 0 or blank) |
- | | |
| --- | --- |
| NU | Identification number of a TABLEDi; i=1 to 4 entry for the Poisson's ratio. (Integer > 0 or blank) |
- | | |
| --- | --- |
| GE | Identification number of a TABLEDi; i=1 to 4 entry for the structural damping coefficient. (Integer > 0 or blank) |
- | | |
| --- | --- |
| RHO | Identification number of a TABLEDi; i=1 to 4 entry for the mass density. (Integer > 0 or blank). This table is only used for isotropic poroelastic material with BEGIN BULK TRMC and is applicable to that trim component only. |

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT1 entry referenced in field 2. The value in a particular field of the MAT1 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT1 entry.
- The user is reminded that on the MAT1 entry, if any one of the entries E, G, or, NU is left blank, they are connected by the relationship $G=E/(2(1+NU))$. Therefore, the user is cautioned that leaving one of these entries blank or only making one of these entries spatially, frequency, or temperature dependent may lead to unexpected results.
- Table references must be present for each item that is frequency dependent.
- Where applicable, secondary material entries such as the MATVE are supported through their MID field being associated with a primary material entry such as a MAT1 entry.

MATF**Material Failure Model**

Specifies failure model properties for linear elastic materials to be used for static, quasi static or transient dynamic analysis. Up to three criteria may be specified for each material. For progressive failure (ITYPE=2) only the first criterion will contribute to failure. The other two, if specified, will only provide output of failure index. SOL 400 (only with extended property entries), and all linear solution sequences between SOL101 and SOL112 (only with CPYRAM or CHEXA/PCOMPLS) are supported.

Format:

1	2	3	4	5	6	7	8	9	10
MATF	MID	ITYPE							
	"CRI"	Criterion	V ₁ ¹	V ₂ ¹	V ₃ ¹	V ₄ ¹	V ₅ ¹	V ₆ ¹	1st
	V ₇ ¹	V ₈ ¹	V ₉ ¹	Find ¹	V ₁₀ ¹	V ₁₁ ¹	V ₁₂ ¹	W ₁ ¹	
	W ₂ ¹	W ₃ ¹	W ₄ ¹	W ₅ ¹	W ₆ ¹	W ₇ ¹	W ₈ ¹	W ₉ ¹	
	"PF"	A1	A2	A3	A4	A5			
	"CRI"	Criterion	V ₁ ²	V ₂ ²	V ₃ ²	V ₄ ²	V ₅ ²	V ₆ ²	2nd
	V ₇ ²	V ₈ ²	V ₉ ²	Find ²	V ₁₀ ²	V ₁₁ ²	V ₁₂ ²	W ₁ ²	
	W ₂ ²	W ₃ ²	W ₄ ²	W ₅ ²	W ₆ ²	W ₇ ²	W ₈ ²	W ₉ ²	
	"CRI"	Criterion	V ₁ ³	V ₂ ³	V ₃ ³	V ₄ ³	V ₅ ³	V ₆ ³	3rd
	V ₇ ³	V ₈ ³	V ₉ ³	Find ³	V ₁₀ ³	V ₁₁ ³	V ₁₂ ³	W ₁ ³	
	W ₂ ³	W ₃ ³	W ₄ ³	W ₅ ³	W ₆ ³	W ₇ ³	W ₈ ³	W ₉ ³	

Example 1 (3 Criteria, no progressive failure):

MATF	100	0							
	CRI	1	2500.	4000.	2500.	4000.	2000.	3000.	1st
	4500.	4500.	4500.						
+									
	CRI	7	2500.	3000.	1500.	2000.			2nd
+	1000.	1000.							
	.11	.06	.1	.05	.075	.03	.03	.03	
	CRI	4	2500.	4000.	2500.	4000.	2000.	3000.	3rd
	4500.	4500.	4500.						

(Note: The 4th and 6th lines cannot be entirely blank and the last line of the 3rd criteria has been omitted.)

Descriptor	Meaning
MID	Identification number of a MAT1, MAT2, MAT8, MATORT or MAT9 entry. See Remarks 1. and 2. (Integer > 0; no Default)
ITYPE	Flag to invoke progressive failure. (Integer; Default = 0)

Descriptor	Meaning
0	No progressive failure, compute failure indices only.
2	Gradual selective stiffness degradation.
3	Immediate selective stiffness degradation.
“CRI”	Enter the character string “CRI” to start input data for a failure criterion.
CRI67	Used with the alternate format only. (Integer; no Default; Required) It is highly recommended that only one criterion be used. However, up to three criteria from the list under Criteria below can be specified in a packed list as follows: 1000000*ITYPE+10000*C3+100*C2+C1 where C1, C2, C3 are integer values for the various Criteria listed.
Criterion	Select an integer corresponding to the failure criterion to be applied. (Integer; no Default) Up to three failure criteria may be specified for each MID. <ul style="list-style-type: none"> 1 For maximum stress criterion. (See Remark 3.) 2 For maximum strain criterion. (See Remark 4.) 3 For Hill failure criterion. (See Remark 5.) 4 For Hoffman failure criterion. (See Remark 6.) 5 Tsai-Wu failure criterion. (See Remark 7.) 7 Hashin failure criterion. (See Remark 8.) 8 Puck failure criterion. (See Remark 9.) 10 Hashin-Tape (See Remark 10.) 11 Hashin-Fabric (See Remark 11.) 13 User-defined criterion with user subroutine UFAIL See Remark 13.
V_i^j	See the respective remark for the meaning of material data.
W_i^j	Stress limit i for criterion j . See Remarks for meaning and default.
Find ^j	Strain limit i for criterion j . See Remarks for meaning and default.
“PF”	Failure index scale factor for criterion j . Used for criteria 3, 4, and 5. (Real > 0.0, or blank, Default = 1.0)
A1	Enter the character string “PF” to start progressive failure input data if ITYPE is 2 or 3. If the defaults are to be taken, the PF line may be omitted. Only the first criterion uses the PF line. (Character; no Default)
A2	Residual stiffness fraction. For ITYPE=2, the stiffness is not reduced beyond this fraction of the initial stiffness. For ITYPE=3, this is the fraction of initial stiffness upon failure. (Real > 0.0, Default = 0.01)
	Leave blank except for Criterion values listed below.

Descriptor	Meaning
	Criterion=7 (Hashin) A2 is the factor for E2 reduction due to matrix compression failure. Takes values between 0.0 and 1.0 and defaults to 0.0 where E2 is reduced in the same way as for matrix tension. A value of 1.0 leads to no E2 reduction due to matrix compression failure.
	Criterion=8 (Puck) - Same as for Criterion=7
	Criterion=10 (Hashin Tape) - Same as for Criterion=7
A3	Leave blank except for the Criterion values listed below. Criterion=7 (Hashin) A3 is the factor for G12 reduction relative to E2 reduction. It takes values between 0.0 and 1.0 and defaults to 0.0 where G12 is reduced in the same way as E2. A value of 1.0 leads to no G12 reduction. Criterion=8 (Puck) - Same as for Criterion=7 Criterion=10 (Hashin Tape) - Same as for Criterion=7
A4	Leave blank except for the Criterion values listed below Criterion=7 (Hashin) A4 is the factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure. Criterion=8 (Puck) - Same as for Criterion=7 Criterion=10 (Hashin Tape) - Factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.
A5	Leave blank except for the Criterion values listed below Criterion=7 (Hashin) A5 is the factor for G12 reduction from fiber failure and takes values between 0.0 and 1.0. It defaults to 0.0 where G12 reduces to matrix failure. A value of 1.0 leads to G12 reduction due to only fiber failure. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure. Criterion=8 (Puck) - Same as for Criterion=7 Criterion=10 (Hashin Tape) - Same as for Criterion=7

Remarks:

1. The MATF Bulk Data entry contains supplementary data for failure index calculation and failure prediction of the elastic materials with the same MID. If this capability is used in nonlinear analysis, MATF can activate a progressive failure process.

2. Progressive failure behavior for various materials can be simulated using the MATF Bulk Data entry. Failure occurs when any one of the specified failure criterion is satisfied; that is, the calculated failure index exceeds 1.0. Upon failure, the elastic moduli are reduced. This is done differently depending on failure criterion. The behavior up to the failure point is linear elastic. After failure the behavior is still elastic but with a different stiffness. The option should not be combined with other material nonlinearities like plasticity.
3. According to the Maximum Stress Criterion, six failure indices F_i are calculated as follows:

$$F_1 = \begin{cases} \frac{\sigma_x}{V_1} & \text{for } \sigma_x > 0 \\ \frac{-\sigma_x}{V_2} & \text{for } \sigma_x < 0 \end{cases}$$

$$F_2 = \begin{cases} \frac{\sigma_y}{V_3} & \text{for } \sigma_y > 0 \\ \frac{-\sigma_y}{V_4} & \text{for } \sigma_y < 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{\sigma_z}{V_5} & \text{for } \sigma_z > 0 \\ \frac{-\sigma_z}{V_6} & \text{for } \sigma_z < 0 \end{cases}$$

$$F_4 = \frac{\tau_{xy}}{V_7}$$

$$F_5 = \frac{\tau_{yz}}{V_8}$$

$$F_6 = \frac{\tau_{zx}}{V_9}$$

where the indices (x, y, z) denote material coordinate direction.

V_1 : maximum allowable stress in the x-direction in tension. Real > 0.0, no Default.

V_2 : maximum allowable stress in the x-direction in compression. Real > 0.0 or blank; Default V_1

V_3 : maximum allowable stress in the y-direction in tension. Real > 0.0, no Default.

V_4 : maximum allowable stress in the y-direction in compression. Real > 0.0 or blank; Default V_3

V_5 : maximum allowable stress in the z-direction in tension. Real > 0.0; no Default

V_6 : maximum allowable stress in the z-direction in compression. Real > 0.0 or blank; Default V_5

V_7 : maximum allowable in-plane shear stress. Real > 0.0 no Default

V_8 : maximum allowable yz shear stress. Real > 0.0 or blank; Default V_7

V_9 : maximum allowable zx shear stress. Real > 0.0 or blank; Default V_7

- According to the Maximum Strain Criterion, 6 failure indices F_i are calculated as follows:

$$F_1 = \begin{cases} \frac{\varepsilon_x}{W_1} & \text{for } \varepsilon_x > 0 \\ \frac{-\varepsilon_x}{W_2} & \text{for } \varepsilon_x < 0 \end{cases}$$

$$F_2 = \begin{cases} \frac{\varepsilon_y}{W_3} & \text{for } \varepsilon_y > 0 \\ \frac{-\varepsilon_y}{W_4} & \text{for } \varepsilon_y < 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{\varepsilon_z}{W_5} & \text{for } \varepsilon_z > 0 \\ \frac{-\varepsilon_z}{W_6} & \text{for } \varepsilon_z < 0 \end{cases}$$

$$F_4 = \frac{\varepsilon_{xy}}{W_7}$$

$$F_5 = \frac{\varepsilon_{yz}}{W_8}$$

$$F_6 = \frac{\varepsilon_{zx}}{W_9}$$

where the indices (x, y, z) denote material coordinate direction.

W_1 : maximum allowable strain in the x-direction in tension. Real > 0.0, no Default.

W_2 : maximum allowable strain in the x-direction in compression. Real > 0.0 or blank; Default W_1

W_3 : maximum allowable strain in the y-direction in tension. Real > 0.0, no Default.

W_4 : maximum allowable strain in the y-direction in compression. Real > 0.0 or blank; Default W_3

W_5 : maximum allowable strain in the z-direction in tension. Real > 0.0; no Default

W_6 : maximum allowable strain in the z-direction in compression. Real > 0.0 or blank; Default W_5

W_7 : maximum allowable shear strain in the xy plane. Real > 0.0 no Default

W_8 : maximum allowable shear strain in the yz plane. Real > 0.0 or blank; Default W_7

W_9 : maximum allowable shear strain in the zx plane. Real > 0.0 or blank; Default W_7

5. According to the Hill Failure Criterion, there is no distinction between tensile and compressive behavior. A single failure index is calculated as:

$$F = \left[\frac{\sigma_x^2}{V_1^2} + \frac{\sigma_y^2}{V_3^2} + \frac{\sigma_z^2}{V_5^2} - \left(\frac{1}{V_1^2} + \frac{1}{V_3^2} - \frac{1}{V_5^2} \right) \sigma_x \sigma_y - \left(\frac{1}{V_1^2} + \frac{1}{V_5^2} - \frac{1}{V_3^2} \right) \sigma_x \sigma_z \right. \\ \left. - \left(\frac{1}{V_3^2} + \frac{1}{V_5^2} - \frac{1}{V_1^2} \right) \sigma_y \sigma_z + \frac{\tau_{xy}^2}{V_7^2} + \frac{\tau_{yz}^2}{V_8^2} + \frac{\tau_{zx}^2}{V_9^2} \right] / \text{Find}$$

where the indices (x, y, z) denote material coordinate direction.

V_1 : maximum allowable stress in the x-direction in tension. Real > 0.0, no Default.

V_3 : maximum allowable stress in the y-direction in tension. Real > 0.0, no Default.

V_5 : maximum allowable stress in the z-direction in tension. Real > 0.0; no Default

V_7 : maximum allowable in-plane shear stress. Real > 0.0 no Default

V_8 : maximum allowable yz shear stress. Real > 0.0 or blank; Default V_7

V_9 : maximum allowable zx shear stress. Real > 0.0 or blank; Default V_7

6. The Hoffman Failure Criterion introduces distinction between tensile and compressive stresses to generalize the Hill Failure Criterion. A single failure index F is calculated as:

$$F = (C_x(\sigma_x - \sigma_y)^2 + C_y(\sigma_y - \sigma_z)^2 + C_z(\sigma_z - \sigma_x)^2 + \left(\frac{1}{V_1} - \frac{1}{V_2}\right)\sigma_x + \left(\frac{1}{V_3} - \frac{1}{V_4}\right)\sigma_y + \left(\frac{1}{V_5} - \frac{1}{V_6}\right)\sigma_z + \frac{\tau_{xy}^2}{V_7^2} + \frac{\tau_{yz}^2}{V_8^2} + \frac{\tau_{zx}^2}{V_9^2}) / \text{Find}$$

with

$$C_x = \frac{1}{2} \left(\frac{1}{V_1 V_2} + \frac{1}{V_3 V_4} - \frac{1}{V_5 V_6} \right)$$

$$C_y = \frac{1}{2} \left(\frac{1}{V_3 V_4} + \frac{1}{V_5 V_6} - \frac{1}{V_1 V_2} \right)$$

$$C_z = \frac{1}{2} \left(\frac{1}{V_5 V_6} + \frac{1}{V_1 V_2} - \frac{1}{V_3 V_4} \right)$$

where the indices (x, y, z) denote material coordinate direction.

- V₁: maximum allowable stress in the x-direction in tension (Real > 0.0, no Default).
- V₂: maximum allowable stress in the x-direction in compression (Real > 0.0 or blank; Default V₁)
- V₃: maximum allowable stress in the y-direction in tension (Real > 0.0, no Default).
- V₄: maximum allowable stress in the y-direction in compression (Real > 0.0 or blank; Default V₃)
- V₅: maximum allowable stress in the z-direction in tension (Real > 0.0, no Default).
- V₆: maximum allowable stress in the z-direction in compression (Real > 0.0 or blank; Default V₅)
- V₇: maximum allowable in-plane shear stress (Real > 0.0, no Default).
- V₈: maximum allowable yz shear stress (Real > 0.0, no Default V₇).
- V₉: maximum allowable zx shear stress (Real > 0.0, no Default V₇).

7. The Tsai-Wu Failure Criterion is another generalization of the Hill Failure Criterion. A single failure index F is calculated as:

$$F = \left[\left(\frac{1}{V_1} - \frac{1}{V_2} \right) \sigma_x + \left(\frac{1}{V_3} - \frac{1}{V_4} \right) \sigma_y + \left(\frac{1}{V_5} - \frac{1}{V_6} \right) \sigma_z + \frac{\sigma_x^2}{V_1 V_2} + \frac{\sigma_y^2}{V_3 V_4} + \frac{\sigma_z^2}{V_5 V_6} + \frac{\tau_{xy}^2}{V_7^2} + \frac{\tau_{yz}^2}{V_8^2} + \frac{\tau_{zx}^2}{V_9^2} + 2V_{10}\sigma_x\sigma_y + 2V_{11}\sigma_y\sigma_z + 2V_{12}\sigma_x\sigma_z \right] / \text{Find}^j$$

where the indices (x, y, z) denote material coordinate direction.

V_1 : maximum allowable stress in the x-direction in tension. Real > 0.0, no Default.

V_2 : maximum allowable stress in the x-direction in compression. Real > 0.0 or blank; Default V_1

V_3 : maximum allowable stress in the y-direction in tension. Real > 0.0, no Default.

V_4 : maximum allowable stress in the y-direction in compression. Real > 0.0 or blank; Default V_3

V_5 : maximum allowable stress in the z-direction in tension. Real > 0.0; no Default

V_6 : maximum allowable stress in the z-direction in compression. Real > 0.0 or blank; Default V_5

V_7 : maximum allowable in-plane shear stress. Real > 0.0 no Default

V_8 : maximum allowable yz shear stress. Real > 0.0 or blank; Default V_7

V_9 : maximum allowable zx shear stress. Real > 0.0 or blank; Default V_7

V_{10} : interactive strength constant for the xy plane. Real; no Default

V_{11} : interactive strength constant for the yz plane. Real; no Default

V_{12} : interactive strength constant for the zx plane. Real; no Default

The following restrictions apply to V_{10} , V_{11} and V_{12} :

$$V_{10}^2 < \frac{1}{V_1 V_2 V_3 V_4} \quad V_{11}^2 < \frac{1}{V_3 V_4 V_5 V_6} \quad V_{12}^2 < \frac{1}{V_1 V_2 V_5 V_6}$$

8. For the Hashin criterion four failure indices are calculated:

$$F_1 = \begin{cases} \left(\frac{\sigma_x}{V_1}\right)^2 + \frac{1}{V_{10}^2}(\tau_{xy}^2 + \tau_{xz}^2) & \sigma_x > 0 \\ 0 & \sigma_x \leq 0 \end{cases}$$

$$F_2 = \begin{cases} \frac{|\sigma_x|}{V_2} & \sigma_x < 0 \\ 0 & \sigma_x \geq 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{1}{V_3^2}(\sigma_y + \sigma_z)^2 + \frac{1}{V_{11}^2}(\sigma_{yz}^2 - \sigma_y \sigma_z) + \frac{1}{V_{10}^2}(\sigma_{xy}^2 + \sigma_{xz}^2) & \sigma_y + \sigma_z > 0 \\ 0 & \sigma_y + \sigma_z \leq 0 \end{cases}$$

$$\begin{aligned}
 & \left(\frac{1}{V_4} \left(\left(\frac{V_4}{2V_{11}} \right)^2 - 1 \right) (\sigma_y + \sigma_z) + \frac{1}{4V_{11}^2} (\sigma_y + \sigma_z)^2 + \right. \\
 F_4 = & \left. \frac{1}{V_{11}^2} (\sigma_{yz}^2 - \sigma_y \sigma_z) + \frac{1}{V_{10}^2} (\sigma_{xy}^2 + \sigma_{xz}^2) \right) & (\sigma_y + \sigma_z > 0) \\
 & 0 & \sigma_y + \sigma_z \geq 0
 \end{aligned}$$

where the indices (x, y, z) denote material coordinate direction.

V_1 : maximum fiber tensile stress. Real > 0.0, no Default.

V_2 : maximum fiber compressive stress. Real > 0.0 or blank; Default V_1

V_3 : maximum matrix tensile stress. Real > 0.0, no Default.

V_4 : maximum matrix compressive stress. Real > 0.0 or blank; Default V_3 .

V_{10} : maximum in plane shear stress. Real > 0.0 no Default

V_{11} : maximum transverse shear stress; Default V_{10}

9. For the Puck failure criterion five failure indices are calculated for the plane stress case and four for the general 3-D case.

The first two failure indices are related to fiber failure and apply to both plane stress and 3-D case:

Tensile fiber mode, for $\sigma_x > 0$

$$F_1 = \frac{\sigma_x}{V_1}$$

Compressive fiber mode, for $\sigma_x < 0$

$$F_2 = \frac{|\sigma_x|}{V_1}$$

For plane stress case, the next three failure indices are related to the matrix failure and are given as follows:

$$F_3 = \sqrt{\left(\frac{\sigma_{xy}}{V_{10}} \right)^2 + \left(1 - W_2 \frac{V_3}{V_{10}} \right)^2 \left(\frac{\sigma_y}{V_3} \right)^2} + W_2 \frac{\sigma_y}{V_{10}} \quad \text{for } \sigma_y > 0$$

$$F_4 = \frac{1}{V_{10}} \left(\sqrt{\sigma_{xy}^2 + (W_1 \sigma_y)^2} + W_1 \sigma_y \right) \quad \text{for } \sigma_y < 0 \text{ and } \left(0 \leq \left| \frac{\sigma_y}{\sigma_{xy}} \right| \leq \frac{R_1}{R_2} \right)$$

$$F_5 = \left(\frac{\sigma_{xy}}{2(1 + W_3 V_{10})} \right)^2 + \left(\frac{\sigma_y}{V_4} \right)^2 \frac{V_4}{|\sigma_y|} \quad \text{for } \sigma_y < 0 \text{ and } \left(0 \leq \left| \frac{\sigma_{xy}}{\sigma_y} \right| \leq \frac{R_2}{R_1} \right)$$

where the parameters, R_1 and R_2 are as follows:

$$R_1 = \frac{V_{10}}{2(1 + W_1)} \left(\sqrt{1 + 2W_1 \frac{V_4}{V_{10}}} - 1 \right)$$

$$R_2 = V_{10} \sqrt{1 + 2W_3}$$

Failure index for 3-D case is calculated based on stress, $\sigma_n(\theta)$, $\sigma_{nt}(\theta)$ and $\sigma_{n1}(\theta)$ on an arbitrary plane with inclination angle θ . The stresses are given by:

$$\sigma_n = \sigma_y \cos^2 \theta + \sigma_z \sin^2 \theta + 2\sigma_{yz} \sin \theta \cos \theta$$

$$\sigma_{nt} = (\sigma_z - \sigma_y) \sin \theta \cos \theta + \sigma_{yz} (\cos^2 \theta - \sin^2 \theta)$$

$$\sigma_{n1} = \sigma_{zx} \sin \theta + \sigma_{xy} \cos \theta$$

For 3-D case, the next two failure indices related to matrix failure are given by:

$$F_3 = \sqrt{\left(\frac{1}{V_3} - P_1\right)^2 \sigma_n^2 + \left(\frac{\sigma_{nt}}{R_1}\right)^2 + \left(\frac{\sigma_{n1}}{V_{10}}\right)^2} + P_1 \sigma_n \quad \text{for } \sigma_n \geq 0$$

$$F_4 = \sqrt{\left(\frac{\sigma_{nt}}{R_1}\right)^2 + \left(\frac{\sigma_{n1}}{V_{10}}\right)^2 + (P_2 \sigma_n)^2} + P_2 \sigma_n \quad \text{for } \sigma_n \geq 0$$

where P_1 and P_2 are given as:

$$P_1 = \left(\frac{W_4}{R_1}\right) \frac{\sigma_{nt}^2}{\sigma_{nt}^2 + \sigma_{n1}^2} + \left(\frac{W_2}{V_{10}}\right) \frac{\sigma_{n1}^2}{\sigma_{nt}^2 + \sigma_{n1}^2}$$

$$P_2 = \left(\frac{W_3}{R_1}\right) \frac{\sigma_{nt}^2}{\sigma_{nt}^2 + \sigma_{n1}^2} + \left(\frac{W_1}{V_{10}}\right) \frac{\sigma_{n1}^2}{\sigma_{nt}^2 + \sigma_{n1}^2}$$

The following material parameters are used:

V_1 = Maximum fiber tensile stress (Real > 0.0; no Default)

V_2 = Maximum fiber compressive stress (Real > 0.0 or blank; Default V_1)

V_3 = Maximum matrix tensile stress (Real > 0.0; no Default)

V_4 = Maximum matrix compressive stress (Real > 0.0 or blank; Default V_3)

V_{10} = Maximum in-plane shear stress. (Real > 0.0; no Default)

$W_1 = p12c$, slope 1 of failure envelope (Real > 0.0; no Default)

$W_2 = p12t$, slope 2 of failure envelope (Real > 0.0; Default W_1)

$W_3 = p23c$, slope 3 of failure envelope (Real > 0.0 or blank; Default; calculated internally; this is recommended for plane stress)

$W_4 = p23t$, slope 4 of failure envelope (Real > 0.0; Default W_3 , not used for plane stress)

- The Hashin-Tape criterion is a variant of the Hashin criterion adapted for tape type of materials. Four failure indices are calculated:

$$F_1 = \begin{cases} \left(\frac{\sigma_x}{V_1}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x > 0 \\ 0 & \sigma_x \leq 0 \end{cases}$$

$$F_2 = \begin{cases} \left(\frac{\sigma_x}{V_2}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x < 0 \\ 0 & \sigma_x \geq 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{1}{V_3^2}(\sigma_y + \sigma_z)^2 - \frac{\sigma_y \sigma_z}{V_{11}^2} + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_y + \sigma_z > 0 \\ 0 & \sigma_y + \sigma_z \geq 0 \end{cases}$$

$$F_4 = \begin{cases} \left(\left(\frac{V_4}{2V_{11}}\right)^2 - 1\right) \frac{(\sigma_y + \sigma_z)}{V_4} + \frac{1}{4V_{11}^2}(\sigma_y + \sigma_z)^2 - \frac{\sigma_y \sigma_z}{V_{11}^2} + \left(\left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 + V_5 \left(\frac{\sigma_x}{V_6}\right)^2\right) & (\sigma_y + \sigma_z < 0) \\ 0 & \sigma_y + \sigma_z \geq 0 \end{cases}$$

V_1 = Maximum tape fiber tensile stress (Real > 0.0; no Default)

V_2 = Maximum tape fiber compressive stress (Real > 0.0 or blank; Default V_1)

V_3 = Maximum tape cross-fiber tensile stress (Real > 0.0; no Default)

V_4 = Maximum tape cross-fiber compressive stress (Real > 0.0 or blank; Default V_3)

V_{10} = Maximum in-plane shear stress. (Real > 0.0; no Default)

V_{11} = Maximum transverse shear stress. (Real > 0.0 or blank; Default V_{10})

V_{12} = Maximum z-x transverse shear stress. (Real > 0.0 or blank; Default V_{11})

V_5 = Maximum fiber tensile stress for matrix compression (Real > 0.0 or blank; required if

$V_6 = 1.0$, otherwise not used)

V_6 = Contribution factor for V_5 (Real 0.0 or 1.0; Default = 0.0)

11. The Hashin-Fabric criterion is a variant of the Hashin criterion adapted for fabric type of materials. Six failure indices are calculated:

$$F_1 = \begin{cases} \left(\frac{\sigma_x}{V_1}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x > 0 \\ 0 & \sigma_x \leq 0 \end{cases}$$

$$F_2 = \begin{cases} \left(\frac{\sigma_x}{V_2}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x < 0 \\ 0 & \sigma_x \geq 0 \end{cases}$$

$$F_3 = \begin{cases} \left(\frac{\sigma_y}{V_3}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_y < 0 \\ 0 & \sigma_y \geq 0 \end{cases}$$

$$F_4 = \begin{cases} \left(\frac{\sigma_y}{V_4}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{yz}}{V_{12}}\right)^2 & \sigma_y < 0 \\ 0 & \sigma_y \geq 0 \end{cases}$$

$$F_5 = \begin{cases} \left(\frac{\sigma_z}{V_5}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_z < 0 \\ 0 & \sigma_z \geq 0 \end{cases}$$

$$F_6 = \begin{cases} \left(\frac{\sigma_z}{V_6}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_z < 0 \\ 0 & \sigma_z \geq 0 \end{cases}$$

V_1 = Maximum first fiber tensile stress (Real > 0.0; no Default)

V_2 = Maximum first fiber compressive stress (Real > 0.0 or blank; Default V_1)

V_3 = Maximum second cross-fiber tensile stress (Real > 0.0; no Default)

V_4 = Maximum second cross-fiber compressive stress (Real > 0.0 or blank; Default V_3)

V_5 = Maximum thickness tensile stress (Real > 0.0; no Default)

V_6 = Maximum thickness compressive stress ((Real > 0.0 or blank; Default = V_5)

V_{10} = Maximum in-plane shear stress. (Real > 0.0; no Default)

V_{11} = Maximum transverse shear stress. (Real > 0.0 or blank; Default V_{10})

V_{12} = Maximum z-x transverse shear stress. (Real > 0.0 or blank; Default V_{11})

12. A MATTF entry with the same MID as MATF may be used to specify the temperature variation of the failure criterion values.

13. User subroutine UFAIL is used along with option MATUDS.
14. In SOL 400, MATF is only supported for elements with property extensions. This implies that for such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATF data was not considered in the element's formulation. If the property extensions are missing, they are automatically added with default settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
For non-composite material with MATF, only the first failure criteria is calculated and its failure index is outputted in NLOUT.
15. For the primary format, if only one criterion is needed and no data is required on the 3rd line, it can be omitted. If more than one criterion is needed, all 3 lines are required except for the last one. If the third line of the last one is not required, it may be omitted.
16. Stress limits such as ST, SC, SS, X_t, X_c, Y_t, Y_c in the MAT1, MAT2 and MAT8 entries are not used in SOL 400.
17. Stress and/or strain allowables in all directions should be defined if the default is not appropriate. Be sure not to set some of these values to low numbers.
18. If MAT8 with Hashin is used in SOL 400 run with PSLDN1 then Hashin FT will be ignored with an appropriate warning message to include MATF instead.

MATF**Material Failure Model - SOL 600**

Specifies failure model properties for linear elastic materials to be used for static, quasi static or transient dynamic analysis in SOL 600. Up to three criterion may be specified for each material if failure indices are desired (ITYPE=0). For progressive failure (ITYPE=2) only one criterion should normally be specified. Even if failure indices (no progressive failure) is the objective of the analysis, it is recommended that only one criterion per material be specified to simplify post processing.

Format:

1	2	3	4	5	6	7	8	9	10
MATF	MID	ITYPE	SB	UFAIL					
"CRI"	Criteria	V ₁	V ₂	V ₃	V ₄	V ₅	V ₆	1st	
V ₇	V ₈	V ₉	Find	V ₁₀	V ₁₁	V ₁₂	Ext		
Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx		
"PF"	A1	A2	A3	A4	A5	IC1	IC2		
IC3	IC4	IC5	IC6	CI7	IC8	IC9			
"CRI"	Criteria	Xt	Xc	Yt	Yc	Zt	Zc	2nd	
Sxy	Syz	Szx	Find	Fxy	Fyz	Fzx	Ext		
Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx		
"CRI"	Criteria	Xt	Xc	Yt	Yc	Zt	Zc	3rd	
Sxy	Syz	Szx	Find	Fxy	Fyz	Fzx	Ext		
Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx		

Example 1 (3 Criteria, no progressive failure):

MATF	100	0							
	CRI	1	2500.	4000.	2500.	4000.	2000.	3000.	1st
	4500.	4500.	4500.						
+									
	CRI	2							2nd
+									
	.11	.06	.1	.05	.075	.03	.03	.03	
	CRI	4	2500.	4000.	2500.	4000.	2000.	3000.	3rd
	4500.	4500.	4500.	0.90					

(Note: The 4th and 6th lines cannot be entirely blank and the last line of the 3rd criterion has been omitted.)

Example 2 (with progressive failure):

MATF	100	2							
	CRI	1	2500.	4000.	2500.	4000.	2000.	3000.	1st
	4500.	4500.	4500.						
+									
	PF	.001					1	1	
+	1	1							

Alternate Format:

1	2	3	4	5	6	7	8	9	10
MATF	MID	CRI67	V ₁	V ₂	V ₃	V ₄	V ₅	V ₆	
	V ₇	V ₈	V ₈	Find	V ₁₀	V ₁₁	V ₁₂	Ext	
	Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx	

Alternate Example Format:

MATF	100	1	2500.	4000.	2500.	4000.	2000.	3000.	
	4500.	4500.	4500.						

Descriptor	Meaning
MID	Identification number of a MAT1, MAT2, MAT8, MATORT or MAT9 entry. See Remarks 1. and 2. (Integer > 0; no Default)
ITYPE	Flag to invoke progressive failure. (Integer; Default = 0)
0	No progressive failure compute failure indices only. (Default)
1	Standard (original) Marc method.
2	Gradual selective stiffness degradation. (MSC Nastran 2005 and subsequent releases)
3	Immediate selective stiffness degradation. (MSC Nastran 2005 and subsequent releases)
SB	Allowable shear stress of bonding material between layers (composites only). To use SB in the stiffness formulation of composites, include PARAM,MINCLDSB,1 in the bulk data. (Real, no Default)
UFAIL	Enter the string "UFAIL" if the ufail.f user subroutine is to be used to define your own failure conditions. If UFAIL is entered, leave fields 3 and 4 blank and do not enter any continuation lines. UFAIL is not available using the alternate format. (Character; Default is blank,). The file ufail.f must be in the same directory as the Nastran input file and must be in lower case.
"CRI"	For the primary failure model, warning may lead to slow convergence.

Descriptor	Meaning
CRI67	Used with the alternate format only. (Integer; no Default; Required) It is highly recommended that only one criterion be used. However, up to three criteria from the list under Criteria below can be specified in a packed list as follows: $1000000*\text{ITYPE}+10000*\text{C3}+100*\text{C2}+\text{C1}$ where C1, C2, C3 are integer values for the various Criteria are listed.
Criteria	Select an integer corresponding to the failure criterion to be applied. (Integer; no Default) Up to three failure criterion may be specified for each MID for SOL 600. <ul style="list-style-type: none"> 1 for maximum stress criterion. (See Remark 3.) 2 for maximum strain criterion. (See Remark 4.) 3 for Hill failure criterion. (See Remark 5.) 4 for Hoffman failure criterion. (See Remark a.) 5 Tsai-Wu failure criterion. (See Remark b.) 7 Hashin failure criterion. (See Remark 8.) 8 Puck failure criterion (See Remark 9.) 9 User-defined failure criterion. (See Remark 8.) 10 Hashin-Tape (See Remark 10.) 11 Hashin-Fabric (See Remark 11.) For Hashin models or Puck model, see Remarks 9., 8., 10., 11. for meaning of material data.
$V_1 = X_t$	Maximum tensile stress in x-direction. (Real, 0.0, or blank, no Default)
$V_2 = X_c$	Maximum compressive stress (absolute value) in x-direction (Real, 0.0, or blank, Default = X_t)
$V_3 = Y_t$	Maximum tensile stress in y-direction. (Real, 0.0, or blank, no Default)
$V_4 = Y_c$	Maximum compressive stress (absolute value) in y-direction. (Real, 0.0, or blank, Default = Y_t)
$V_5 = Z_t$	Maximum tensile stress in z-direction. (Real, 0.0, or blank, no Default)
$V_6 = Z_c$	Maximum compressive stress (absolute value) in z-direction. (Real, 0.0, or blank, Default = Z_t)
$V_7 = S_{xy}$	Maximum shear stress in xy-plane. (Real, 0.0, or blank, no Default)
$V_8 = S_{yz}$	Maximum shear stress in yz-plane. (Real, 0.0, or blank, Default = S_{xy} for criterion 5)
$V_9 = S_{zx}$	Maximum shear stress in zx-plane (Real, 0.0, or blank, Default = S_{xy} for criterion 5)
Find	Failure index. See Remarks 5.-a. (Real, 0.0, or blank, Default = 1.0)
$V_{10} = F_{xy}$	Interactive strength constant for xy-plane. (Real, 0.0, or blank, Default = $\frac{1}{2}\sqrt{\frac{1}{X_t X_c} \frac{1}{Y_t Y_c}}$ for criterion 5)

Descriptor	Meaning
$V_{11} = F_{yz}$	Interactive strength constant for yz-plane. (Real, 0.0, or blank, Default = $-\frac{1}{2} \sqrt{\frac{1}{Y_t Y_c} \frac{1}{Z_t Z_c}}$ for criteria 5)
$V_{12} = F_{zx}$	Interactive strength constant for zx-plane. (Real, 0.0, or blank, Default = $-\frac{1}{2} \sqrt{\frac{1}{Z_t Z_c} \frac{1}{X_t X_c}}$ for criterion 5).
Ext	Maximum tensile strain in x-direction. (Real, 0.0, or blank, no Default)
Exc	Maximum compressive strain (absolute value) in x-direction. (Real, 0.0, or blank, Default = Ext for criterion 2)
Eyt	Maximum tensile strain in y-direction. (Real, 0.0, or blank, Default = Ext for criterion 2)
Eyc	Maximum compressive strain (absolute value) in y-direction. (Real, 0.0, or blank, Default = Eyt for criterion 2)
Ezt	Maximum tensile strain in z-direction. (Real, 0.0, or blank, Default = Ext for criterion 2)
Ezc	Maximum compressive strain (absolute value) in z-direction. (Real, 0.0, or blank, Default = Ezt for criterion 2)
Gxy	Maximum shear strain in xy-plane. (Real, 0.0, or blank, no Default)
Gyz	Maximum shear strain in yz-plane. (Real, 0.0, or blank, Default = Gxy for criterion 2)
Gzx	Maximum shear strain in zx-plane. (Real, 0.0, or blank, Default = Gxy for criterion 2)
"PF"	Enter the character string "PF" to start progressive failure input data if ITYPE is 2 or 3. If the defaults are to be taken, the PF line and the line following the PF line may be omitted (Character, no Default)
A1	Residual stiffness fraction. For Criteria=3, this is the fraction of initial stiffness upon failure. For Criteria=2, the stiffness is not reduced more than this fraction (Real, Default = 0.01)
A2	Must be 0.0 or blank except for the Criteria values listed below
Criteria=7 (Hashin)	A2 is the factor for E2 reduction due to matrix comporession failure. Takes values between 0.0 and 1.0 and defaults to 0.0 where E2 is reduced in the same way as for matrix tension. A value of 1.0 leads to no E2 reduction due to matrix comporession failure.
Criteria=10 (Hashin Tape)	Same as for Criteria=7
Criteria=8 (Puck)	Same as for Criteria=7

Descriptor	Meaning
A3	<p>Must be 0.0 or blank except for the Criteria values listed below</p> <p>Criteria=7 (Hashin) A3 is the factor for G12 reduction relative to E2 reduction. It takes values between 0.0 and 1.0 and defaults to 0.0 where G12 is reduced in the same way as E2. A value of 1.0 leads to no G12 reduction</p> <p>Criteria=10 (Hashin Tape) Same as for Criteria=7</p> <p>Criteria=8 (Puck) Same as for Criteria=7</p>
A4	<p>Must be 0.0 or blank except for the Criteria values listed below</p> <p>Criteria=7 (Hashin) A4 is the factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.</p> <p>Criteria=10 (Hashin Tape) Factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.</p> <p>Criteria=8 (Puck) Same as for Criteria=7</p>
A5	<p>Must be 0.0 or blank except for the Criteria values listed below</p> <p>Criteria=7 (Hashin) A5 is the factor for G12 reduction from fiber failure and takes values between 0.0 and 1.0. It defaults to 0.0 where G12 reduces to matrix failure. A value of 1.0 leads to G12 reduction due to only fiber failure. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure</p> <p>Criteria=10 (Hashin Tape) Same as for Criteria=7</p> <p>Criteria=8 (Puck) Same as for Criteria=7</p>
IC1	<p>For all Criteria values except those listed below, IC1 is set to 1 if failure in the positive x-direction is critical (leads to element deactivation).</p> <p>Criteria=7 (Hashin) Set IC1=1 if fiber tension is critical (leads to element deactivation).</p> <p>Criteria=10 (Hashin Tape) Same as for Criteria=7</p> <p>Criteria=8 (Puck) Same as for Criteria=7</p> <p>(Integer, Default = 0)</p>

Descriptor	Meaning
IC2	<p>For all Criteria values except those listed below, IC2 is set to 1 if failure in the negative x-direction is critical</p> <p>Criteria=7 (Hashin) Set IC2=1 if fiber compression is critical</p> <p>Criteria=10 (Hashin Tape) Same as for Criteria=7</p> <p>Criteria=8 (Puck) Same as for Criteria=7</p> <p>(Integer, Default = 0)</p>
IC3	<p>For all Criteria values except those listed below, IC3 is set to 1 if failure in the positive y-direction is critical</p> <p>Criteria=7 (Hashin) Set IC3=1 if matrix tension is critical</p> <p>Criteria=10 (Hashin Tape) Same as for Criteria=7</p> <p>Criteria=8 (Puck) Same as for Criteria=7</p> <p>(Integer, Default = 0)</p>
IC4	<p>For all Criteria values except those listed below, IC4 is set to 1 if failure in the negative y-direction is critical</p> <p>Criteria=7 (Hashin) Set IC4=1 if matrix compression is critical</p> <p>Criteria=10 (Hashin Tape) Same as for Criteria=7</p> <p>Criteria=8 (Puck) Set IC4=1 if matrix compression mode B is critical</p> <p>(Integer, Default = 0)</p>
IC5	<p>For all Criteria values except those listed below, IC5 is set to 1 if failure in the positive z-direction is critical</p> <p>Criteria=7 (Hashin) Not used, leave blank</p> <p>Criteria=10 (Hashin Tape) Not used leave blank</p> <p>Criteria=8 (Puck) Set IC5=1 if matrix compression mode C is critical</p> <p>(Integer, Default = 0)</p>
IC6	<p>For all Criteria values except those listed below, IC6 is set to 1 if failure in the negative z--direction is critical</p> <p>Criteria=7 (Hashin) Not used, leave blank</p> <p>Criteria=10 (Hashin Tape) Not used leave blank</p> <p>Criteria=8 (Puck) Not used leave blank</p> <p>(Integer, Default = 0)</p>

Descriptor	Meaning
IC7	For all Criteria values except those listed below, IC7 is set to 1 if failure in the xy plane is critical Criteria=7 (Hashin) Not used, leave blank Criteria=10 (Hashin Tape) Not used leave blank Criteria=8 (Puck) Not used leave blank (Integer, Default = 0)
IC8	For all Criteria values except those listed below, IC8 is set to 1 if failure in the yz plane is critical Criteria=7 (Hashin) Not used, leave blank Criteria=10 (Hashin Tape) Not used leave blank Criteria=8 (Puck) Not used leave blank (Integer, Default = 0)
IC9	For all Criteria values except those listed below, IC9 is set to 1 if failure in the zx plane is critical Criteria=7 (Hashin) Not used, leave blank Criteria=10 (Hashin Tape) Not used leave blank Criteria=8 (Puck) Not used leave blank (Integer, Default = 0)

Remarks:

1. The MATF Bulk Data entry contains supplementary data for failure prediction of the elastic materials with the same MID. If this capability is used in nonlinear analysis, MATF can activate progressive failure process.
2. Progressive failure behavior for various materials can be simulated using the MATF Bulk Data entry. Failure occurs when any one of the specified failure criterion is satisfied. Upon failure, the elastic modulus reduces to 10% of the original modulus if there is only one value of modulus as in isotropic material or in a beam or truss element. If it pertains to an orthotropic material, all of the material moduli at the integration point are reduced to the lowest modulus specified. The behavior up to the failure point is linear elastic even if an elasto-plastic material is specified, which is followed by a nonlinear behavior for the post-failure analysis. If the initial yield stress is less than the allowable maximum stress, the failure criterion will be ignored.
3. According to the Maximum Stress Criterion, the material fails when any of the stress components (9 components including 6 normal stress components in tension and compression, and three shear stress components) exceeds the maximum allowable stress:

$$|\sigma_{ij}| > X_t, X_c, Y_t, \dots, S_{xy}, \dots \text{etc.}$$

where the indices (x, y, z or i, j) denote material coordinate direction.

4. According to the Maximum Strain Criterion, the material fails when any of the strain components (9 components including 6 normal strain components in tension and compression, and three shear strain components) exceeds the maximum allowable strain:

$$|\epsilon_{ij}| > \text{Ext, } \text{Exc, } \text{Ext, } \dots, \text{Gxy, } \dots \text{etc.}$$

where the indices (x, y, z or i, j) denote material coordinate direction.

5. According to the Hill Failure Criterion, there is no distinction between tensile and compressive behavior. The failure is determined based on

$$\frac{\sigma_x^2}{X^2} + \frac{\sigma_y^2}{Y^2} + \frac{\sigma_z^2}{Z^2} - \left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2} \right) \sigma_x \sigma_y - \left(\frac{1}{Y^2} + \frac{1}{Z^2} - \frac{1}{X^2} \right) \sigma_y \sigma_z$$

$$- \left(\frac{1}{Z^2} + \frac{1}{X^2} - \frac{1}{Y^2} \right) \sigma_z \sigma_x + \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} > F_{\text{ind}}$$

in which X, Y, Z, S_{xy}, S_{yz}, S_{zx} are maximum allowable stresses and F_{ind} is the failure index prescribed by the user.

- a. The Hoffman Failure Criterion introduces distinction between tensile and compressive stresses to generalize the Hill Failure Criterion, i.e.,

$$C_x(\sigma_x - \sigma_y)^2 + C_y(\sigma_y - \sigma_z)^2 + C_z(\sigma_z - \sigma_x)^2 + \left(\frac{1}{X_t} - \frac{1}{X_c} \right) \sigma_x$$

$$+ \left(\frac{1}{Y_t} - \frac{1}{Y_c} \right) \sigma_y + \left(\frac{1}{Z_t} - \frac{1}{Z_c} \right) \sigma_z + \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} > F_{\text{ind}}$$

with

$$C_x = \frac{1}{2} \left(\frac{1}{X_t X_c} + \frac{1}{Y_t Y_c} - \frac{1}{Z_t Z_c} \right)$$

$$C_y = \frac{1}{2} \left(\frac{1}{Y_t Y_c} + \frac{1}{Z_t Z_c} - \frac{1}{X_t X_c} \right)$$

$$C_z = \frac{1}{2} \left(\frac{1}{Z_t Z_c} + \frac{1}{X_t X_c} - \frac{1}{Y_t Y_c} \right)$$

in which $X_t, X_c, Y_t, Y_c, Z_t, Z_c, S_{xy}, S_{yz}, S_{zx}$ are maximum allowable stresses and F_{ind} is the failure index, prescribed by the user.

- b. The Tsai-Wu Failure Criterion is another generalization of the Hill Failure Criterion:

$$\left(\frac{1}{X_t} - \frac{1}{X_c} \right) \sigma_x + \left(\frac{1}{Y_t} - \frac{1}{Y_c} \right) \sigma_y + \left(\frac{1}{Z_t} - \frac{1}{Z_c} \right) \sigma_z + \frac{\sigma_x^2}{X_t X_c} + \frac{\sigma_y^2}{Y_t Y_c} + \frac{\sigma_z^2}{Z_t Z_c}$$

$$+ \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} + 2F_{xy}\sigma_x\sigma_y + 2F_{yz}\sigma_y\sigma_z + 2F_{zx}\sigma_x\sigma_z > F_{ind}$$

in which $X_t, X_c, Y_t, Y_c, Z_t, Z_c, S_{xy}, S_{yz}, S_{zx}$ are maximum allowable stresses, F_{xy}, F_{yz}, F_{zx} are interactive strength constants, and F_{ind} is the failure index, prescribed by the user.

- 6. For the Hashin criterion the inputs are used:

V_1 = Maximum fiber tensile stress

V_2 = Maximum fiber compressive stress

V_3 = Maximum matrix tensile stress

V_4 = Maximum matrix compressive stress

V_7 = Maximum in-plane shear stress

V_8 = Maximum transverse shear stress

All other variables should be set to zero

- 7. For the Puck failure criterion the following inputs are used:

V_1 = Maximum fiber tensile stress

V_2 = Maximum fiber compressive stress

V_3 = Maximum matrix tensile stress

V_4 = Maximum matrix compressive stress

V_7 = Maximum in-plane shear stress

Ext = p12c, slope 1 of failure envelope
Exc = p12t, slope 2 of failure envelope
Eyt = p23c, slope 3 of failure envelope
Eyc = p23t, slope 4 of failure envelope

8. For the user-defined failure criterion, leave all fields blank except fields 1-3 of the primary entry (continuation entries are not required).
9. For the Hashin-Tape criterion the following inputs are used:

V_1 = Maximum tape fiber tensile stress
 V_2 = Maximum tape fiber compressive stress
 V_3 = Maximum tape cross-fiber tensile stress
 V_4 = Maximum tape cross-fiber compressive stress
 V_7 = Maximum in-plane shear stress
 V_8 = Maximum transverse shear stress
 V_9 = Maximum z-x transverse shear stress
 V_5 = Maximum fiber tensile stress for matrix compression
 V_6 = Contribution factor for Zt (Real 0.0 or 1.0; Default = 0.0)
All other variables should be set to zero

For the Hashin-Fabric criterion the following inputs are used:

V_1 = Maximum first fiber tensile stress
 V_2 = Maximum first fiber compressive stress
 V_3 = Maximum second cross-fiber tensile stress
 V_4 = Maximum second cross-fiber compressive stress
 V_5 = Maximum thickness tensile stress
 V_6 = Maximum thickness compressive stress
 V_7 = Maximum in-plane shear stress
 V_8 = Maximum transverse shear stress
 V_9 = Maximum z-x transverse shear stress
All other variables should be set to zero

10. A MATTF entry with the same MID as MATF may be used to specify the temperature variation of the failure criterion values.

Notes:

1. This Bulk Data entry accommodates Marc's input data under the model definition option FAIL DATA.
2. For the primary format, if only one criterion is needed and no data is required on the 3rd line, it can be omitted. If more than one criterion is needed, all 3 lines are required except for the last one. If the third line of the last one is not required, it may be omitted.

3. For the alternate format the third line may be omitted if no data is needed.
4. For Nastran versions prior to MSC 2007, the alternate format was the only available option. These older versions used ITYPE=1 and a packed list for CRI67 with the format 100*C3+10*C2+C1. ITYPE=1 was an older progressive failure method that is no longer recommended. If these jobs are re-run using MSC 2007, the CRI67 field should be changed as described above or the new primary format used.
5. Stress limits such as ST, SC, SS, X_t, X_c, Y_t, Y_c in the MAT1, MAT2 and MAT8 entries are not used in SOL 600.

MATF**Material Failure Model - SOL 700**

Specifies failure model properties. Use SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATF	MID								
	“CRI”	Criterion (=101)	MES	DT					
	“CRI”	Criterion (=102)	D1	D2	D3	D4	D5	ε0DOT	
			TROOM	TMELT	CP	MTH			
	“CRI”	Criterion (=103)	MPS	MPS-C	VOLF	DT			
	“CRI”	Criterion (=104)	PRS						
	“CRI”	Criterion (=105)	ALPHA	THETA	GAMMA	BETA	R	D	
			W	X0	CBAR	N	TYPE	ITER	
			TOFF						

Example:

MATF	7								
	CRI	101	2.0	1.0E20					

Descriptor	Meaning
MID	Identification number of MAT1 or MATORT entry. (Integer > 0; required.)
“CRI”	Enter the character string “CRI” to start input data for a failure criterion.
Criterion	Select an integer corresponding to the failure criterion to be applied. Only one failure criterion is allowed. (Integer; no Default)
101	Maximum Equivalent Stress Failure Model. It is only available for solid elements.
102	Johnson-Cook Failure Model. It is only available for solid elements. See Remarks 1.
103	Maximum Plastic Strain Failure Model. It is only available for Hugh-Liu beams, shells and solid elements.
104	Maximum Pressure Failure Model. It is only available for solid elements.
105	CAP Failure Model. It is only acceptable for solid elements. See Remark 2.-4.

Descriptor	Meaning				
MES	Maximum equivalent stress that causes failure on the deviatoric part of the stress tensor. (Real > 0.0; required)				
DT	Minimum time step that causes total failure. (Real > 0.0; default=not used.)				
D1~D5	Damage coefficients. (Real; default=0.0)				
$\dot{\epsilon}$ 0DOT	Reference plastic strain rate. (Real; default=1.0)				
TROOM	Room temperature. (Real; default=0.0)				
TMELT	Melt temperature. (Real; default=1.0E20)				
CP	Heat capacity. (Real; default=1.0E20)				
MTH	Specifies how failure is applied. (Character; default=CONT) <table> <tr> <td>CONT</td><td>Continuous failure</td></tr> <tr> <td>DISC</td><td>Discrete failure</td></tr> </table>	CONT	Continuous failure	DISC	Discrete failure
CONT	Continuous failure				
DISC	Discrete failure				
MPS	Maximum plastic strain that causes failure. (Real > 0.0; required)				
MPS-C	Maximum plastic strain when material is under compression that causes failure. (Real > 0.0; default=MPS-C)				
VOLF	Minimum volumetric failure criteria. It is only available for solid elements. (Real > 0.0; default=1.0E-12)				
DT	Minimum time step that causes total failure. This option is only acceptable in solid elements. (Real > 0.0; default=not used)				
PRS	Maximum pressure that causes failure. (Real > 0.0; required)				
ALPHA	Failure envelope parameters. (Real \geq 0.0; required)				
THETA	Failure envelope linear coefficient. (Real \geq 0.0; required)				
GAMMA	Failure envelope exponential coefficient. See Remark 2. (Real \geq 0.0; required)				
BETA	Failure envelope exponent. (Real \geq 0.0; required)				
R	Cap surface axis ratio. (Real \geq 0.0; required)				
D	Hardening law exponent. (Real \geq 0.0; required)				
W	Hardening law coefficient. (Real \geq 0.0; required)				
X0	Hardening law exponent. (Real \geq 0.0; default=0.0)				
CBAR	Kinematic hardening coefficient. (Real \geq 0.0; default=0.0)				
N	Kinematic hardening parameter. (Real \geq 0.0; default=0.0)				
TYPE	Formulation type. (Character; default=SOIL) <table> <tr> <td>SOIL</td><td>Soil or concrete (cap surface may contract)</td></tr> <tr> <td>ROCK</td><td>Rock (cap surface does not contract)</td></tr> </table>	SOIL	Soil or concrete (cap surface may contract)	ROCK	Rock (cap surface does not contract)
SOIL	Soil or concrete (cap surface may contract)				
ROCK	Rock (cap surface does not contract)				
ITER	Iteration scheme. (Character; default=VEC) <table> <tr> <td>VEC</td><td>Fixed number of iterations (vectorized)</td></tr> </table>	VEC	Fixed number of iterations (vectorized)		
VEC	Fixed number of iterations (vectorized)				

Descriptor	Meaning
FULL	Fully iterative
TOFF	Tension cut off (positive in compression). See Remark 3. (Real < 0.0; default=blank)

Remarks:

- Defines the properties of a failure model where failure is determined by a damage model. The damage model is given by:

$$D = \sum_{\text{time}} \frac{\Delta \varepsilon_p}{\dot{\varepsilon}^{\text{frac}}}$$

$$\dot{\varepsilon}^{\text{frac}} = (D_1 + D_2 \exp(D_3 \sigma^*)) \left(1 + D_4 \ln \frac{\dot{\varepsilon}_{\text{pl}}}{\dot{\varepsilon}_{\text{pl}}^0} \right) (1 + D_5 T^*)$$

$$T = \text{SIE/CP}$$

$$T^* = \frac{T - T_{\text{room}}}{T_{\text{melt}} - T_{\text{room}}}$$

$$\sigma^* = \frac{\sigma_m}{\sigma}$$

The summation is performed over all past time increments. The variable D measures the damage; T is the temperature, σ_m is the mean stress, σ is the von Mises equivalent stress, and $\dot{\varepsilon}^{\text{frac}}$ is the fracture strain.

The fracture strain depends on a nondimensional plastic strain rate, $\dot{\varepsilon}_{\text{pl}} / \dot{\varepsilon}_{\text{pl}}^0$. If D exceeds one it set equal to one.

There are two methods to determine when elements fail:

- Continuous failure: The yield stress is reduced by a factor (1-D). When D exceeds 1 the yield stress equals zero and the element fails.
- Discrete failure: the element fails when D equals one.

This failure model applies to high-strain rate deformation of metals. It is less suitable for quasi-static problems.

- For a physically meaningful model, the value of the failure envelope exponential coefficient should be less than the failure envelope parameter ($\text{ALPHA} < \text{GAMMA}$).

3. The tension cut off value (TOFF) can be defined on the entry and must be less than zero. If the tension cut off is left blank, Nastran SOL700 calculates the tension cut off as the intersection point of the failure envelope surface with the J1-axis as described in Remark 4.
4. The cap material model can be used for geomechanical problems with materials like soil, concrete and rock.

The cap model is characterized by the following constitutive equations:

$$\varepsilon = \varepsilon^e + \varepsilon^p$$

$$\sigma = C(\varepsilon - \varepsilon^p)$$

where ε , ε^e and ε^p are the total, elastic and plastic strain tensor, C is the elasticity matrix and σ is the stress tensor. The flow rule is given by:

$$\dot{\varepsilon}^p = \sum_{k=1}^e \dot{\lambda}_k \frac{\partial f_k}{\partial \sigma}$$

where the sum is over the active yield surfaces f^k , i.e., the failure envelope ($k=1$), the hardening cap surface ($k=2$), and the fixed tension cutoff surface ($k=3$). The yield conditions are defined by:

$$f_1(\sigma) \leq 0$$

$$f_2(\sigma, \kappa) \leq 0$$

$$f_3(\sigma) \leq 0$$

The hardening parameter, κ , for the cap model is related to the plastic volume change by a hardening law.

The cap model is a plasticity model described by a yield surface that is defined by means of a failure envelope, a hardening cap and a tension cut off. Figure 1 shows the typical yield surfaces in a cap model.

The failure envelope surface is denoted by

$$f_1 = \sqrt{J_{2D}} - \text{MIN}(F_e(J_1), T_{mises})$$

and the cap by

$$f_2 = \sqrt{J_{2D}} - F_c(J_1, \kappa) \text{ for } L(\kappa) \geq J_1 \geq X(\kappa)$$

where J_1 is the first invariant (trace) of the stress tensor, J_{2D} is the second invariant of the stress deviator, κ is an internal state variable that measures hardening as a functional of the history of plastic volumetric strain and $L(\kappa)$ and $X(\kappa)$ define the range of the cap. Note that J_1 is chosen as negative in tension.

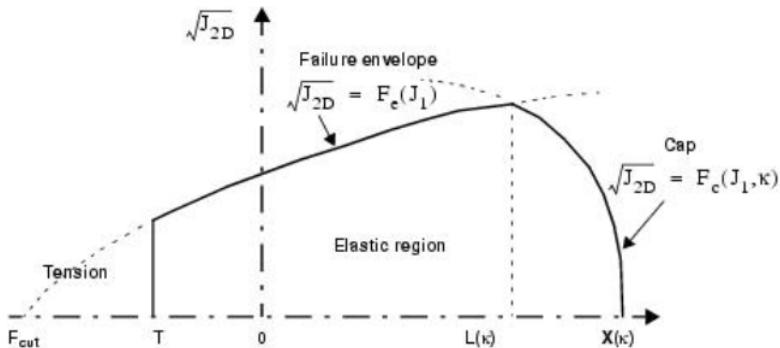


Figure 9-109 Typical Yield Surfaces in a Cap Model

The functions F_e and F_c are given by (see References 1. and 2.):

$$F_e(J_1) = a - \gamma \exp(-\beta J_1) + \theta J_1$$

$$F_c(J_1, \kappa) = \frac{1}{R} \sqrt{[X(\kappa) - L(\kappa)]^2 - [J_1 - L(\kappa)]^2}$$

The von Mises type transition failure surface is defined by the following:

$$T_{\text{mises}} = \frac{1}{R} |X(\kappa) - L(\kappa)|$$

The intersection of the cap with the (hydrostatic) J_1 axis is given by:

$$X(\kappa) = \kappa + R F_e(\kappa)$$

and $L(\cdot)$ is defined by:

$$L(\kappa) = \kappa = \begin{cases} \kappa & \text{if } \kappa > 0 \\ 0 & \text{if } \kappa \leq 0 \end{cases}$$

The hardening parameter, κ , is related to the actual plastic volume change:

$$\varepsilon_v^p(X) = W \{ 1 - \exp[-D(X(\kappa) - X_0)] \}$$

The tension cut off surface is given by the function:

$$f_3 = T - J_1$$

where T is the maximum hydrostatic tension sustainable by the material.

Kinematic work hardening for the failure envelope surface is based on the approach of Isenberg et. Al. [1978]. It is switched on by specifying N . The failure envelope surface is replaced by a family of envelope surfaces that are bounded by an initial yield surface and by a limiting failure envelope surface. Which member of the family

is taken, is implemented by replacing in all yield relations the stress tensor σ by $\sigma - \zeta$ where ζ is a deviatoric tensor that accumulates in time. This tensor ζ is called the “back stress tensor” and is defined by

$$\frac{d\zeta}{dt} = \bar{c} \bar{F}(\sigma, \zeta) \frac{d\varepsilon^p}{dt}$$

$$F = \text{MAX}\left(0, 1 - \frac{(\sigma - \zeta) \cdot \zeta}{2NF_e(J_1)}\right)$$

Here ε^p is the deviatoric plastic strain tensor, N denotes the size of the yield surface and represents the radial distance between the outside of the initial yield surface and the inside of the limit surface. After each increment of ε^p , it is checked whether its second invariant exceeds N. In that case, ε^p is scaled by a scalar such that its second invariant equals N. For consistency between the limit surface of the kinematic hardening cap model and the failure envelope of the standard cap model, the parameter \bar{c} is placed by $\bar{c} = N$.

References:

1. Sandler, I. S. and Rubin, D., “An Algorithm and a Modular Subroutine for the Cap Model,” International Journal for Numerical and Analytical Methods in Geomechanics, Vol. 3, 173-186 (1979)
2. Simo, J. C., Ju, J.W., Pister, K. S., and Taylor, R.L., “Assessment of Cap Model: Consistent Return Algorithms and Rate-dependent Extension,” Journal of Engineering Mechanics, Vol. 114, No. 2, February 1988.
3. Isenberg, Jeremy, D K. Vaughan, and I Sandler. Nonlinear Soil-Structure Interaction. Palo Alto, Calif: The Institute, 1978. Print.

MATF1**Frequency Dependent Isotropic Material Definition**

Defines the frequency dependent properties for an isotropic poroelastic material.

Format:

1	2	3	4	5	6	7	8	9	10
MATF1	MID	T(E)	T(G)	T(NU)	T(RHO)	T(A)		T(GE)	
	T(ST)	T(SC)	T(SS)						

Example:

MATF1	1	30			130				
-------	---	----	--	--	-----	--	--	--	--

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT1 entry. (Integer ≥ 0 ; Required)
T(E)	Identification number of a TABLEMi entry for the Young's modulus. (Integer ≥ 0 or Blank)
T(G)	Identification number of a TABLEMi entry for the shear modulus. (Integer ≥ 0 or Blank)
T(NU)	Identification number of a TABLEMi entry for the Poisson's ratio. (Integer ≥ 0 or Blank)
T(RHO)	Identification number of a TABLEMi entry for the mass density. (Integer ≥ 0 or Blank)
T(A)	Identification number of a TABLEMi entry for the thermal expansion coefficient. (Integer ≥ 0 or Blank)
T(GE)	Identification number of a TABLEMi entry for the damping coefficient. (Integer ≥ 0 or Blank)
T(ST)	Identification number of a TABLEMi entry for the tension stress limit. (Integer ≥ 0 or Blank)
T(SC)	Identification number of a TABLEMi entry for the compression stress limit. (Integer ≥ 0 or Blank)
T(SS)	Identification number of a TABLEMi entry for the shear stress limit. (Integer ≥ 0 or Blank)

Remarks:

1. Remarks 1 to 4 under [MATT1, 2375](#) are applicable with MATF1..
2. MATF1 must reside under Case Control 'BEGIN BULK TRMC' and is applicable to that trim component only. It may only be referenced by MATPE1 entry for a poroelastic element.

MATFAB**Woven Fabric Material- SOL700**

Defines the properties of a bi-directional woven fabric material for shell elements. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATFAB	MID	RHO	ECOAT	NUCOAT	GCOAT	DAMPCOAT	COMPRESS	PERC	
+	E1L	E1Q		THETA1	XWARP	YWARP	ZWARP		
+	E2L	E2Q		THETA2	XWEFT	YWEFT	ZWEFT		
+	SCOF	G12	DAMPFIB	COMPFIB	LOCKANG1	LOCKANG2			

Example:

MATFAB	3	850	5.52E6	0.33					
+	21.6E7				1.0	0.0	0.0		
+	21.6E7				0.0	1.0	0.0		

Descriptor	Meaning
MID	Unique material number. (Integer > 0; required)
RHO	Density. (Real > 0.0; required)
ECOAT	Young's modulus of coating material. See Remark 1. (Real > 0.0; default=blank)
NUCOAT	Poisson's ratio of coating material. See Remark 1. (Real > 0.0; default=blank)
GCOAT	Shear modulus of coating material. See Remark 1. (Real > 0.0; default=blank)
DAMPCOAT	Damping is applied to the coating stresses. See Remark 2. (Real >0.0; default=0.1)
	$d\sigma_{ij} = DAMPCOAT \cdot ECOAT \cdot \epsilon_{ij} dt_{elm}$
COMPRESS	Scale factor for coating compression stresses. See Remark 3. (0.0? Real?1.0; default=1.0)
PERC	Thickness percentage of coating material. (0.0? Real?100.0; default=0.0=no coating)
E1L	Young's modulus of fabric in warp direction, linear coefficient. (Real > 0.0; required)
E1Q	Young's modulus of fabric in warp direction, quadratic coefficient. (Real >0.0; default=0.0)
THETA1	Orientation angle between the element coordinate system and the warp ends. See Remark 4. (Real; default=blank)
XWARP	X component of vector indicating the warp direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=1.0)
YWARP	Y component of vector indicating the warp direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0)

Descriptor	Meaning
ZWARP	Z component of vector indicating the warp direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0)
E2L	Young's modulus of fabric in weft direction, linear coefficient. (Real > 0.0; required)
E2Q	Young's modulus of fabric in weft direction, quadratic coefficient. (Real ? 0.0; default=0.0)
THETA2	Orientation angle between the element coordinate system and the weft ends. See Remark 4. (Real; default=blank)
XWEFT	X component of vector indicating the weft direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0)
YWEFT	Y component of vector indicating the weft direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=1.0)
ZWEFT	Z component of vector indicating the weft direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0)
SCOF	Shear coefficient of friction. See Remark 6. (Real; default=0.0)
G12	Shear modulus of fabric material. See Remark 7. (Real; default=blank)
DAMPFIB	Damping is applied to the fiber stresses. See Remark 2. (Real \geq 0.0; default=0.1)
	$d\sigma_{ij} = DAMPCOA \bullet E\varepsilon_{ij} dt_{elm}$
COMPFB	Scale factor for fiber compression stresses. See Remark 3. ($0.0 \leq$ Real \leq 1.0; default=1.0)
LOCKANG1	Locking angle 1 for change in fiber cross-over angle. See Remark 10. (Real \geq 0.0; default=10.0)
LOCKANG2	Locking angle 2 for change in fiber cross-over angle. See Remark 10. (Real \geq 0.0; default=15.0)

Remarks:

1. When a coating is defined (PERC>0), two out of three values need to be specified for ECOAT, NUCOAT, and GCOAT.
2. For air bag modeling the following values of DAMPCOAT and DAMPFIB are suggested:
 $DAMPCOAT = 0.05$
 $DAMPFIB = 0.05$
3. The compressive stresses in the fibers are scaled with the value of COMPFB. Putting COMPFB = 0.0 results in a tension only fiber model.

The compressive stresses in the coating are scaled with the value of COMPCOAT. Putting COMPCOAT = 0.0 results in a tension only coating model.

The compressive stresses are scaled in the direction of the principal stresses.

When PERC = 100%, and the coating of this fabric model is used to simulate an isotropic air bag material, it is best to scale down the compressive stresses of the coating. A suggested value is COMPCOAT = 0.1

4. Since this is a model which tracks warp and weft directions and uses total warp/weft strain as a state variable, the initial warp and weft directions must be specified. There are two ways to indicate the initial warp and weft directions:

- a. THETA1 and THETA2

Orientation angles between the element coordinate system and the warp/weft ends. If no orientation angle is specified, vectors will be used to indicate the warp/weft directions of the fabric material with respect to the basic coordinate system.

- b. XWARP, YWARP, ZWARP and XWEFT, YWEFT, ZWEFT

Vectors indicating the warp/weft directions of the fabric material with respect to the basic coordinate system. The projection of these vectors on the surface of each element is used to determine the angle between the element and the material coordinate system. If the orientation angles are defined, these vectors are ignored.

5. For shell element properties (PSHELL1), when the material is MATFAB, the material angle THETA is ignored. The orientation of the fabric fibers is defined completely on the MATFAB entry.

For layered composite element properties (PCOMP), when the material of a ply is MATFAB, the angle THETAi is ignored. The orientation of the fabric fibers is defined completely on the MATFAB entry.

6. The maximum shear stress is given by a friction coefficient of the fabric (SCOF) times the RMS value of the direct fiber stresses.

7. If the field G12 is left blank, the shear modulus is computed from the RMS value of the two linear stiffness coefficients.

8. When MATFAB material is referenced by shell elements, the Spin Rate method (SPIN) is applied automatically when no stress-rotation correction is specified on SPINCOR option. See PSHELL1 entry for the details on SPINCOR option.

9. There are a number of specific output sublayer variables useful for this material:

Q1AF Direction cosines/sines between the element coordinate

Q2AFIB System and the warp ends

Q1BFIB Direction cosines/sines between the element coordinate

Q2BFIB System and the weft picks

SGMA Direct stress in fabric parallel to the warp ends

SGMB Direct stress in fabric parallel to the weft picks

SGFRIC Stress due only to shear in the weave of the fabric

EPSFA Strain in fabric parallel to the warp ends

EPSFB Strain in fabric parallel to the weft picks

ANGLE Crossover angle between warp ends and weft picks

10. When a fabric is being sheared, the angle between the fibers changes. At a certain moment, the fibers will reach a locking angle, after which a further change in the fiber angle is no longer possible.

The simulation models this behavior as follows:

- a. Change in Fiber Crossover Angle < LockAng1

The shear stress between the fibers is cut off based on the friction coefficient SCOF

- b. LockAng1 < Change in Fiber Crossover Angle < LockAng2

The shear stress between the fibers is linearly increased.

- c. Change in Fiber Crossover Angle > LockAng2

The shear stress between the fibers is no longer cut off.

This situation is equal to an infinite friction coefficient SCOF.

MATFTG**Fatigue Material Properties**

Defines fatigue material properties for time domain based SOLs 101, 103, 112 and frequency domain based SOLs 108 and 111.

Format (SOL 101, 103, 112):

1	2	3	4	5	6	7	8	9	10
MATFTG	MID	CNVRT							
	"STATIC"	YS	UTS	CODE	TYPE	RR	SE	mp	
		RA							
	"SN"	SRI1	b1	Nc1	b2	Nfc	SE	BTHRESH	
	"SNS1"	M1	M2	M3	M4	MSS	RTHICK	nTHICK	
	"SNS2"	SF-FXY	DE-FXY	TE-FXY	SF-MXY	DE-MXY	TE-MXY		
	"SNBR1"	SF-FZ	DE-FZ	TE-FZ	SF-MZ	DE-MZ	TE-MZ		
	"SNBR0"								
	"TABLE"	VALUE1	TID1	VALUE2	TID2	VALUE3	TID3		
		VALUE4	TID4	-etc.-			
	"BASTEN"	A	B	c	Eb	Sd			
	"EN"	Sf	b	c	Ef	n'	K'	Nc	
		SEe	SEp	SEc	Ne	FSN	S		
	"MATID"	E	NU						
		MID1	MID2	MID3	MID4	MID5	MID6	MID7	
		-etc.-							

Format (SOL 108, 111):

1	2	3	4	5	6	7	8	9	10
MATFTG	MID	CNVRT	MATOUT						
	"STATIC"	YS	UTS		TYPE				
	"SN"	SRI1	b1	Nc1	b2	Nfc			
	"TABLE"	VALUE1	TID1						
	"MMPDS"	A1	A2	A3	A4				
	"EN"	Sf	b	c	Ef	n'	K'	GAMMA	
		SEe	SEp	SEc	Ne			TOLER	
	"MATID"	E	NU						
		MID1	MID2	MID3	MID4	MID5	MID6	MID7	
		-etc.-							

Examples:

MATFTG	9								
	STATIC	430	682	99					

MATFTG	9								
	STATIC	430	682					0.1	
	SN	3095	-0.1339	1.e8	0.0	1.e8			

MATFTG	9								
	STATIC	430	682					0.1	
	MATID	2.1e5	0.3						
		9	THRU	15	18				

Descriptor	Meaning
MID	Unique material ID that matches the identification of a MAT1 entry unless the MATID line is supplied (Integer>0). See Remark 1.
CNVRT	Conversion factor. See Remark 2. regarding units.
MATOU T	SOL 108, 111 only. This is a debug option for writing the fully derived material curve to a file (Integer 0 or 1; Default = 0).
STATIC	Required flag indicating that yield and/or ultimate tensile strengths and other common parameters are supplied (Character=STATIC). See Remark 3. and 4.
YS	Yield strength (Valid range equivalent in MPa: $50.0 \leq \text{Real} \leq 3000$). See Remark 2. regarding units.
UTS	Ultimate tensile strength (Valid range equivalent in MPa: $100.0 \leq \text{Real} \leq 4000$). See Remark 2. regarding units.
CODE	SOL 101, 103, 112 only. Material code used in automatically generating S-N or ϵ -N data curves and for surface finish correction. See Remark 3.
TYPE	SOL 101, 103, 112: Specification of the type of S-N curves defined using the TABLE or BASTEN flag (no Default). See Remark 4.
	SOL 108, 111: Specification as to whether material definitions are specified as RANGE or AMPLitude. See Remark 4.
RR	SOL 101, 103, 112 only. R-ratio of test. ($-1.0e30 \leq \text{Real} \leq 1.0$, Default=-1.0).
SE	SOL 101, 103, 112 only. Standard Error of Log(N). ($0.0 \leq \text{Real} \leq 10.0$, Default=0.1).
mp	SOL 101, 103, 112 only. Plastic Poisson's Ratio. ($0.45 \leq \text{Real} \leq 0.55$, Default=0.5).

Descriptor	Meaning
RA	SOL 101, 103, 112 only. Percentage reduction in area used for deriving ε -N data for metals of type "other" (TYPE field = 200-207 or >300). (0.0 < Real ≤ 100.0; no Default).
SN	Flag indicating the definition of an S-N curve(s) follow (Character = SN, optional). See Remark 5. and 7. and 8.
SNS1/SNS2	
SNBR1/SNBR0	
SRI1	Stress range intercept. (Valid range equivalent in MPa: 1.0 ≤ Real ≤ 2.5e4, no Default). See Remark 2 regarding units.
b1	First fatigue strength exponent. (-1.0 < Real < -0.02, no Default)
Nc1	In 1-segment S-N curve, the cycles limit of endurance. In 2-segment S-N curve, this is the fatigue transition point. Both are defined in cycles. (1.0 ≤ Real ≤ 1.0E25; no Default).
b2	Second fatigue strength exponent. It is zero when defining 1-segment S-N curve; (-0.5 < Real ≤ 0.0, Default=0.0).
Nfc	Fatigue cutoff. (1.e-9 ≤ Real ≤ 1.0e30; Default=1.0e30).
SE	SOL 101, 103, 112 only. Standard Error of Log(N). (0.0 ≤ Real ≤ 10.0, Default=0.1). SE from "STATIC" line will be used if left blank.
BTHRESH	SOL 101, 103, 112 only. Threshold value of the bending (r) ratio used in interpolation between "stiff" and "flexible" SN curves for fatigue analysis of seam welds. (0.0 ≤ Real ≤ 0.999; Default=0.5)
M1 - M4	SOL 101, 103, 112 only. Mean stress slope parameters M1 through M4 representing sensitivity to mean stress in four (4) regimes of R-ratio as plotted on a constant life Haigh diagram and used in FKM mean stress correction. (-0.99 ≤ Real ≤ 0.0; Default = blank). See Remark 6.
MSS	SOL 101, 103, 112 only. Mean stress sensitivity factor, used only for fatigue analysis of spot welds with CORR=SIMPLE on the FTGPARM entry. (0.0 ≤ Real ≤ 1.0, Default=blank). This mean stress correction method is a simplified version of the FKM method where M1=M2=M3=M4=-MSS.
RTHICK	SOL 101, 103, 112 only. Reference thickness/threshold (in consistent model length units) for sheet thickness correction used in fatigue analysis of seam welds. Ignored if THICK=0 on the FTGPARM entry for "SEAMW". Must be supplied if THICK=1. (Real ≥ 1.0e-9; Default=1.0)
nTHICK	SOL 101, 103, 112 only. Sheet thickness correction exponent used in the fatigue analysis of seam welds. Ignored if THICK=0 on the FTGPARM entry for "SEAMW". Must be supplied if THICK=1. (Real; Default=0.16667)
SF-FXY	SOL 101, 103, 112 only. Scale factor for stress due to FX or FY (shear forces),
SF-MXY	MX or MY (bending moments), FZ (axial force), and MZ (torsion), respectively (Real, Default = blank) used in fatigue analysis of spot welds only. See Remark 9.
SF-FZ	
SF-MZ	

Descriptor	Meaning
DE-FXY	SOL 101, 103, 112 only. Diameter exponent for stress due to FX or FY (shear forces), MX or MY (bending moments), FZ (axial force), and MZ (torsion), respectively (Real, Default = blank) used in fatigue analysis of spot welds only. See Remark 9.
DE-MXY	
DE-FZ	
DE-MZ	
TE-FXY	SOL 101, 103, 112 only. Thickness exponent for stress due to FX or FY (shear forces), MX or MY (bending moments), FZ (axial force), and MZ (torsion), respectively (Real, Default = blank) used in fatigue analysis of spot welds only. See Remark 9.
TE-MXY	
TE-FZ	
TE-MZ	
TABLE	Flag indicating the definition of S-N curves as a number of tables follows (Character=TABLE; optional). See Remark 5.
VALUEi	SOL 101, 103, 112 : The constant mean stress, R-ratio, or life (in cycles) of this particular S-N curve. (Real; no Default). See Remark 2 regarding units. SOL 108, 111 : This should be specified as 1.0 as only one curve is supported and should be defined as zero mean.
TIDi	SOL 101, 103, 112 : A TABLEM1 ID defining the S-N curve of stress range (y) vs. life (x) (in cycles) for this particular S-N curve, or mean stress (x) vs. stress amplitude (y) for constant life (Haigh diagram) curves (Integer>0). See Remark 2. regarding units. SOL 108, 111 : A TABLEM1 (may cause unintended recalculation of eigenvalues if used in RESTART job) or TABRND1 ID defining the S-N curve of stress (y) vs. life (x) (in cycles). Only one curve is allowed.and should be defined at zero mean. See Remark 2. regarding units.
MMPDS	SOL 108, 111 only. Flag indicating the definition of an S-N curve based on the MMPDS S-N approach (Character-MMPDS; optional). See Remark 5.
A1	Coefficients or exponent of the standard MMPDS-01 material equation (must be defined in units-KSI based on amplitude). (Real; typical ranges
A2	$5 \leq A1 \leq 110$; $-47 \leq A2 \leq -1.0$, $0.1 \leq A3 \leq 1.0$; $0.0 \leq A4 \leq 114$; no Default).
A3	
A4	CNVRT is not applied to these values at all - it is ignored!
BASTEN	SOL 101, 103, 112 only. Flag indicating the definition of an S-N curve based on the Bastenaire S-N approach (Character-BASTEN; optional). See Remark 5.
A	Bastenaire coefficient - a parameter positioning the curve along the life axis. (Valid range equivalent in MPa: $1.0e4 \leq \text{Real} \leq 1.0e10$). See Remark 2. regarding units.
B	Scale Factor parameter. (Valid range equivalent in MPa: $1.0 \leq \text{Real} \leq 1.0e10$). See Remark 2. regarding units.
c	Bastenaire exponent. ($0.01 \leq \text{Real} \leq 1.0e10$)
Eb	Bastenaire fatigue limit (Valid range equivalent in MPa: $1.0 \leq \text{Real} \leq 1.0e10$). See Remark 2 regarding units.

Descriptor	Meaning
Sd	Bastenaire scatter factor. (Valid range equivalent in MPa: $0.0 \leq \text{Real} \leq 1.0e25$). See Remark 2 regarding units.
EN	Flag indicating the definition of an ε -N curve is to follow. (Character=EN; optional). See Remark 5..
Sf	Fatigue strength coefficient (Valid range equivalent in MPa: $50.0 \leq \text{Real} \leq 4000.0$; no Default). See Remark 2. regarding units.
b	Fatigue strength exponent ($-0.5 \leq \text{Real} \leq -2.0e-3$; no Default).
c	Fatigue ductility exponent. ($-1.2 \leq \text{Real} \leq -0.15$; no Default).
Ef	Fatigue ductility coefficient, ($0.001 \leq \text{Real} \leq 10.0$; no Default).
n'	Cyclic strain-hardening exponent, ($5.0e-3 \leq \text{Real} \leq 0.5$; no Default).
K'	Cyclic strength coefficient, (Valid range equivalent in MPa: $50 \leq \text{Real} \leq 1.2e4$); no Default). See Remark 5. regarding units.
Nc	SOL 101, 103, 112 only. Cut-off (reversals), ($1.0e5 \leq \text{Real} \leq 1.0e25$; Default= $2.0e8$).
GAMMA	SOL 108, 111 only. Additional coefficient needed for the Walker equation, ($0.0 \leq \text{Real} \leq 1.0$; Default=0.0).
SEe	SOL 101, 103, 112 only. Standard Error of Log(e) (elastic), ($0.0 \leq \text{Real} \leq 10.0$; Default=0.0).
SEp	SOL 101, 103, 112 only. Standard Error of Log(e) (plastic), ($0.0 \leq \text{Real} \leq 10.0$; Default=0.0).
SEC	SOL 101, 103, 112 only. Standard Error of Log(e) (cyclic), ($0.0 \leq \text{Real} \leq 10.0$; Default=0.0).
Ne	SOL 101, 103, 112 only. Endurance Limit (Reversals), ($1.0 e-9 \leq \text{Real} \leq 1.0e25$; Default= $2.0e8$).
FSN	SOL 101, 103, 112 only. Fatemi-Socie Parameter, ($0.0 \leq \text{Real} \leq 10.0$; Default=0.6)
S	SOL 101, 103, 112 only. Wang Brown Parameter, ($-10.0 \leq \text{Real} \leq 10.0$; Default=1.0)
TOLER	SOL 108, 111 only. Parameter used for frequency based strain life analysis with Neuber notch correction in vibration fatigue only. ($\text{Real} >= 1e-10$, Default = $1e-5$)
MATID	Flag indicating that a list of MAT1 IDs is to follow. See Remark 1.
E	Young's modulus, (Valid range equivalent in MPa: $\text{Real} > 0.0$, no Default).
NU	Elastic Poisson's ratio. ($0.25 \leq \text{Real} \leq 0.35$, Default = 0.3)
MIDI	IDs of existing MAT1 entries to which this MATFTG entry will be linked. At least one ID must be supplied. The "THRU" keyword is supported.

Remarks:

1. Element properties must reference MAT1 entries in order to be linked to a MATFTG entry as only metal fatigue analysis of isotropic materials is supported. The MID must match that of an existing MAT1 entry called out by the property entry (e.g. PSHELL) unless the "MATID" line is provided, in which case all the MIDI referenced on the "MATID" line are then linked to the MATFTG entry. When the "MATID" line is used, E (required) and NU (optional) must be provided on this line as the code has no way of knowing from which referenced MAT1 this data should be extracted. Young's Modulus (E) and elastic Poisson's ratio (NU) are extracted from the corresponding MAT1 entry of the same ID when no "MATID" line is present. Usage of E and NU in the fatigue analysis itself is dependent on the type of fatigue analysis requested and fatigue material properties provided. Young's Modulus (E) is mainly used for deriving S-N or ϵ -N data when none is provided. If the "MATID" line is used, and a MAT1 of the same ID also exists, that MAT1 ID must be in the MIDI list or it will be ignored in the analysis. In other words, when the "MATID" line is used, only the supplied MIDI IDs are used in the fatigue analysis.
2. The CNVRT field is only used if fatigue material stress based parameters are directly input using the "STATIC," "SN," "EN," "TABLE," or "BASTEN" methods. It is used to allow the user to input the fatigue material stress related parameters, (YS, UTS, SRI1, VALUEi, TIDI, A, B, Eb, K) in different units other than the model's consistent units. Example: model is producing stresses in PSI units, fatigue material parameters input in MPa, the CNVRT factor should be 145.0377 to convert MPa to PSI. Note that the y-values of any referenced TABLEM1 entries are also converted for the S-N method or both the x- and y-values when TYPE=LIFE (Haigh curves). It is also necessary to use the DTI,UNITS for defining the model's stress units. See DTI,UNITS. (Real; Default=1.0)
3. For SOLs 101, 103, and 112, if only STATIC is supplied, then for standard S-N or ϵ -N analysis, the S-N or ϵ -N curve is derived using the UTS and a material CODE. For fatigue analysis of spot or seam welds, a standard material set is used based on CODE, and UTS can be set to any valid number as the UTS from the standard material set will be used instead. Valid codes are listed in [Table 9-22](#) below. At a minimum UTS must be supplied along with the material CODE and E on MAT1, or an error will be issued. If either flag (SN or EN) is present, then the automatic generation is suppressed if all the data necessary to define S-N or ϵ -N curves are given. For SN, these fields are SRI1, b1, Nc1. For EN, these fields are Sf, b, c, Ef, n', K'. Either all of them are present, in which case they are used directly, or all of them are omitted, in which case the parameters are derived as mentioned above. The determination as to whether S-N or ϵ -N curves are generated is determined by the TYPE field set on the FTGPARM entry. If surface finish corrections are to be applied, CODE is also required (see PFTG entry). When curves are derived, the specified CODE gets internally converted to a generic code *ferrous*=99 for CODE<100, *aluminium*=100 for 100 ≤ CODE < 200, *other*=0 for 200 ≤ CODE < 300, or *titanium*=300 for 300 ≤ CODE < 400. When SN or EN flags exist, the generic code is set to *other*, but the CODE is used as given.
- For SOLs 108, and 111, if only STATIC is supplied, see Remark 5.
4. For SOLs 101, 103, and 111: TYPE can be set to AMPL, RANGE, or MAX for amplitude, range, or maximum stress, respectively, when using the "BASTEN" flag. Or TYPE can be set to MEAN (or AMEAN), RRATIO (or ARRATIO), or LIFE (or ALIFE) for constant mean, R-ratio, or life (Haigh diagrams), respectively, when using the "TABLE" flag. When using multiple S-N curves for mean

stress correction, CORR=INTERP on FTGPARM must be used. In this case, for RRATIO, a curve at R=-1 is required and for MEAN, a curve at zero (0) mean is required. AMPL, RANGE, and MAX are used to define stress types of Bastenaire models. MEAN vs. AMEAN, or RRATIO vs. ARRATIO, or LIFE vs. ALIFE indicates the stress type of the curves is range or amplitude, respectively.

For SOLs 108 and 111: TYPE can be set to AMPL or RANGE to indicate that the S-N curve is defined as amplitude or range and is applicable for the SN and TABLE lines.

5. **For SOLs 101, 103, and 111:** If an S-N analysis is specified (TYPE field on FTGPARM entry) but no S-N curve is supplied, an error will be issued. If an ε -N analysis is specified but no ε -N curve is supplied, an error will be issued. The STATIC data is required if SN or EN flags are specified on the MATFTG entry. The SN, BASTEN, and TABLE flags are generally mutually exclusive. When both SN and TABLE lines are defined, if TYPE is set, the SN parameters (SRI1, b1, Nc1, b2) will be ignored. EN flag may be present with SN, BASTEN, or TABLE flags, but one or the other will be ignored based on the type of analysis as set using the TYPE field on the FTGPARM entry. See Remark 7. and 8. for other “SNS1/SNS2/SNBR1/SNBR2” definitions.

For SOLs 108 and 111: The STATIC line is required and at least one of SN, TABLE, MMPDS, or EN lines. If no SN, TABLE, MMPDS, or EN lines are present, then the fatigue analysis is skipped and only the random vibration analysis is performed¹. The use of the TYPE field in the FTGPARM entry has limited affect. Please note: If only SN data is present, it is used by default. If only EN data is present, it is used by default. The SN, TABLE, and MMPDS entries are mutually exclusive and if more are present for any particular MID, then the following hierarchical rule is implement to choose which to use: MMPDS, TABLE, SN. If both SN and EN are defined for a particular MID, EN is selected by default unless TYPE on the FTGPARM is set, in which case the corresponding SN or EN definition is used. However, TYPE is ignored if the corresponding material definition is not present, and the material definition used is then based on the hierarchy just defined.

6. **For SOLs 101, 103, and 111:** Mean stress sensitivity: M1 for R>1; M2 for $-\infty < R < 0$; M3 for $0 < R < 0.5$; M4 for $0.5 < R < 1$. If M₁₋₄ are undefined, and the material type (CODE) is given , all the parameters will be estimated using empirically defined rules for the FKM mean stress correction method. If only M₂ is defined, then M₁ and M₄ will be set to zero and M₃ to M₂/3.
7. **For SOLs 101, 103, and 111:** The “SN” line is used for the definition of standard SN parameters for any SN analysis. In the case of fatigue analysis of spot welds, “SNS1” and “SNS2” are used to define the SN curves of the top and bottom sheets to which the spot weld is connected and “SN” is used to define the SN curve of the weld nugget. If no “SN”, “SNS1”, or “SNS2” flags are present, generic SN parameters are used for steel or aluminum per the material CODE supplied. See [Table 9-23](#) for default values used in this case. If CODE is blank, steel is assumed. If only one of the flags is provided, then “SN” = “SNS1” = “SNS2”. If “SNS1” or “SNS2” is missing but the other is present, then “SNS1” = “SNS2”. If “SNS1” and “SNS2” are defined, but not “SN”, then “SN” = “SNS1”.
8. **For SOLs 101, 103, and 111:** The “SN” line is used for the definition of standard SN parameters for any SN analysis. In the case of fatigue analysis of seam welds, “SNBR1” and “SNBR0” are used to define the flexible (bending ratio=1.0) and stiff (bending ratio=0.0) SN curves. If no “SN”, “SNBR1”, or “SNBR0” flags are present, generic SN parameters are used for steel or aluminum per the material

¹If the EN line is supplied with only n' and K' defined, then the Neuber notch correction is also applied in the random only analysis affecting the resultant output stress PSD.

CODE supplied. See [Table 9-23](#) for default values used in this case. If CODE is blank, steel is assumed. If only "SN" is supplied, then the "SN" parameters are used for both "SNBR1" and SNBR0". If "SNBR1" or "SNBR0" is missing but the other is present, then "SNBR1" = "SNBR0". The 3rd and 4th lines of data for these keywords should not be entered as they are applicable to fatigue analysis of spot welds only, and not for seam welds.

9. For SOLs 101, 103, and 111: If SF, DE, & TE values are not supplied, default values are used based on material CODE for steel or aluminum. See [Table 9-24](#) and . These should only be entered for fatigue analysis of spot welds and not for anything else.

Table 9-22 Table of Material CODEs

CODE	Description
1	Flake cast iron (FCI)
2	Ferritic cast iron with compacted graphite (FCICG)
3	Pearlitic cast iron with compacted graphite (PCICG)
4	Bainitic cast iron with compacted graphite (BCICG)
5	Ferritic cast iron with spheroidal graphite (FCISG)
6	Ferrite/pearlite cast iron with spheroidal graphite (FPCISG)
7	Pearlitic cast iron with spheroidal graphite (PCISG)
8	Bainitic cast iron with spheroidal graphite (BCISG)
9	Cast steel with less than 0.2% carbon (CSL2C)
10	Normalized cast steel with 0.2-0.4% carbon (NCS24C)
11	Quenched & tempered cast steel with 0.2-0.4% carbon (QTCS24)
12	Normalized cast steel with 0.4-0.7% carbon (NCS47)
13	Plain carbon wrought steel with < 0.2% carbon (PCWS)
14	Hot rolled/normalized plain carbon wrought steel, 0.2-0.4% carbon (HNPCWS24)
15	Quenched & tempered cast steel with 0.4-0.7% carbon (QTCS47)
16	Quenched & tempered plain carbon wrought steel, 0.2-0.4% carbon (QTPCWS24)
17	Hot rolled/normalized plain carbon wrought steel, 0.4-0.7% carbon (HNPCWS47)
18	Quenched & tempered plain carbon wrought steel, 0.4-0.7% carbon (QTPCWS47)
19	Normalized low alloy wrought steel (NLAWS)
20	Quenched & tempered low alloy wrought steel (QTHSLAWS)
21	Normalized Ni/Cr/Mo wrought steel (NNCMWS)
22	Quenched & tempered Ni/Cr/Mo wrought steel (QTNCMWS)
23	Austenitic stainless steel (ASS)
24	Ferritic stainless steel (FSS)
25	Martensitic stainless steel (MSS)

CODE	Description
26	Annealed plain carbon wrought steel, 0.2-0.4% carbon (APCWS24)
27	Annealed plain carbon wrought steel, 0.4-0.7% carbon (APCWS47)
28	Normalized carbon/manganese steel (MCMS)
29	Quenched and tempered carbon/manganese steel (QTCMS)
30	Hardened chromium steel (HCS)
31	Quenched and tempered chromium steel (QTCS)
99	Steel of unknown heat treatment (STEEL)
100	Wrought aluminium (WA)
101	Wrought aluminium-copper alloy (WACA)
102	Wrought aluminium-manganese alloy (WAMNA)
103	Wrought aluminium-magnesium alloy (WAMGA)
104	Wrought aluminium-magnesium-silicon alloy (WAMGSA)
105	Wrought aluminium-zinc alloy (WAZA)
106	Cast aluminium alloy (CAA)
107	Wrought complex special purpose aluminum alloys (WCSPAA)
200	Wrought copper (WCU)
201	Wrought brass (WBR)
202	Wrought aluminium bronze (WABR)
203	Cupronickel (CUPNI)
204	Nickel silver (NIAG)
205	Wrought phosphor bronze (WPHBR)
206	Wrought copper beryllium (WCUBE)
207	Cast copper alloys (CCUA)
300	Titanium alloy (TA)
400	Wrought magnesium alloys (WMGA)
401	Cast magnesium alloys (CMGA)
500	Fusible alloys, solders (FUSSOL)
600	Cast zinc alloys (CZINCA)
700	Wrought nickel alloys (WNIA)
701	Cast nickel alloys (CNIA)
800	Precious metals (PRECMET)

CODE	Description
900	Clad materials (CLADMAT)
1000	Thermoplastics (THERPLAS)
1001	Thermosetting plastics (TSETPLAS)

Table 9-23 Spot and Seam Weld Fatigue Analysis SN Curve Defaults

Property	Generic Nugget Spot Weld		Generic Top/Bottom Sheet Spot Weld		Generic Flexible Seam Weld (r=1)		Generic Stiff Seam Weld (r=0)	
CODE	steel <100	alum 100-199	steel <100	alum 100-199	steel <100	alum 100-199	steel <100	alum 100-199
UTS (MPa)*	4000	4000	4000	4000	1.8E4	4000	1.8E4	4000
YS (MPa)	355	150	355	150	Blank	Blank	Blank	Blank
E (MPa)	2.1E5	7E4	2.1E5	7E4	2.1E5	7E4	2.1E5	7E4
SRI1 (MPa)	2100	2462	2900	2462	3.6E4	3140	1.8E4	1275
b1	-0.1667	-0.2	-0.1667	-0.2	-0.3333	-0.1734	-0.3333	-0.1625
Nc1	1E6	1E7	1E6	1E7	1E7	1E7	1E7	1E7
b2	-0.09091	-0.1111	-0.09091	-0.1111	-0.3333	-0.1734	-0.3333	-0.1625
SE	0.334	0.33	0.33	0.33	0.4	0.4	0.4	0.4
RR	0.0	0.0	0.0	0.0	-1.0	-1.0	-1.0	-1.0
Nfc	1E30	1E30	1E30	1E30	1E30	1E30	1E30	1E30
MSS	0.1	0.2	0.1	0.2	Blank	Blank	Blank	Blank
M1	Blank	Blank	Blank	Blank	Blank	Blank	Blank	Blank
M2	-0.1	-0.2	-0.1	-0.2	-0.25	-0.25	-0.25	-0.25
M3	-0.1	-0.2	-0.1	-0.2	-0.1	Blank	-0.1	Blank
M4	Blank	Blank	Blank	Blank	-0.1	Blank	-0.1	Blank
BTHRESH	Blank	Blank	Blank	Blank	Blank	Blank	Blank	Blank
RTHICK	Blank	Blank	Blank	Blank	1 (mm)	1 (mm)	1 (mm)	1 (mm)
nTHICK	Blank	Blank	Blank	Blank	0.16667	0.16667	0.16667	0.16667

*Both Spot Weld and Seam Weld fatigue analyses use structural stresses, which do not relate to UTS. They both should have static failure set to false or to ensure static failure is not active, the UTS should be set to a high value, thus the high default values for these data sets. Corrections based on UTS do not apply to welds.

Table 9-24 Steel Scale Factor, Diameter & Thickness Exponent Defaults for Fatigue Analysis of Spot Welds

Steel (CODE < 100)	Scale Factor	Diameter Exponent	Thickness Exponent
FX,FY	SF-FXY = 1.0	DE-FXY = 0.0	TE-FXY = 0.0
MX,MY	SF-MXY = 0.6	DE-MXY = 0.0	TE-MXY = 0.5
FZ	SF-FZ = 0.6	DE-FZ = 0.0	TE-FZ = 0.5
MZ	SF-MZ = 0.0	DE-MZ = 0.0	TE-MZ = 0.0

Aluminum Scale Factor, Diameter & Thickness Exponent Defaults
for Fatigue Analysis of Spot Welds

Aluminum ($100 \leq \text{CODE} \leq 200$)	Scale Factor	Diameter Exponent	Thickness Exponent
FX,FY	SF-FXY = 0.4	DE-FXY = 0.5	TE-FXY = -0.25
MX,MY	SF-MXY = 0.4	DE-MXY = 0.5	TE-MXY = -0.25
FZ	SF-FZ = 1.0	DE-FZ = 0.0	TE-FZ = 1.0
MZ	SF-MZ = 0.0	DE-MZ = 0.0	TE-MZ = 0.0

MATG**Gasket Material Properties**

Specifies gasket material properties to be used in SOL 600 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATG	MID	IDMEM	BEHAV	TABLD	TABLUI	TABLUS	TABLUS	TABLUS	TABLUS
	TABLUS	TABLUS	TABLUS	TABLUS	TABLUS	TABLUS	YPRS	EPL	
	GPL	GAP	TABYPRS	TABEPL	TABGPL	TABGAP	N/A	N/A	

Example:

MATG	100	10	0	1001	1002	1003			
							100.	2500.	
	950.	0.0							

Descriptor	Meaning
MID	Material ID number. (Integer)
IDMEM	ID of MAT1 providing material behavior for membrane behavior. See Remarks. (Integer)
BEHAV	Behavior type (presently only type 0 is supported). (Integer)
TABLD	ID of a TABLES1 table providing loading path of the gasket (pressure versus displacement). See Remarks 1. and 3. (Integer)
TABLUI	ID of TABLES1 table providing unloading path(s) of the gasket (pressure versus displacement) can range from 1 to 10. If there is no unloading, no unloading tables need be entered. Leave fields blank for tables that are not required. See Remarks. (Integer)
YPRESS	Yield pressure. See Remark 4. (Real)
EPL	Tensile modulus (pressure per unit length). (Real)
GPL	Transverse shear modulus (force per unit area). (Real)
GAP	Initial gap (if present). (Real)
TABYPRS	ID of TABLES1 table associated with yield pressure (not presently used). (Integer)
TABEPL	ID of TABLES1 table associated with tensile modulus (not presently used). (Integer)
TABGPL	ID of TABLES1 table associated with transverse shear modulus (not presently used). (Integer)
TABGAP	ID of TABLES1 table associated with initial gap (not presently used). (Integer)

Remarks:

1. MATG defines nonlinear properties in the thickness direction for compression only, designed for gasket-like materials. MATG has anisotropy only in the thickness direction, which is called normal anisotropy.
2. The MATG option can only be used with continuum composite elements, use PLCOMP.
3. The MATG entry defines the compressive behavior in thickness. The thickness direction is the principal direction (3) in 3-dimensional solids and (2) for 2-dimensional solids (plane strain and axisymmetric elements). Since MATG material allows only normal anisotropy, linear properties in MAT1 are required for in-plane behavior.
4. The initial yield pressure should match a point in table TABLD.
5. The loading path for the gasket is always in compression. However, it starts from the origin to initial yield pressure (nonlinear elastic range) and continues with strain hardening slope into the plastic region. All the data points are specified in the first quadrant.
6. As many as 10 unloading paths may be defined in the thickness direction using TABLS1 in pressure vs. gasket closure distance as in the loading path. All the unloading paths must start from zero pressure and end at the loading path in the plastic region. Unloading behavior at undefined paths will be interpolated between two adjacent unloading paths. The last point of the last specified unloading path signifies full compression, which does not allow any further closure beyond the point.

Element Thickness Direction for MATG	Runs Parallel from Face	To Face
Element T direction	G1-G2-G3-G4	G5-G6-G7-G8

7. If creep analysis is required, Bulk Data entry, MPCREEP, must also be entered to activate Marc's CREEP parameter.

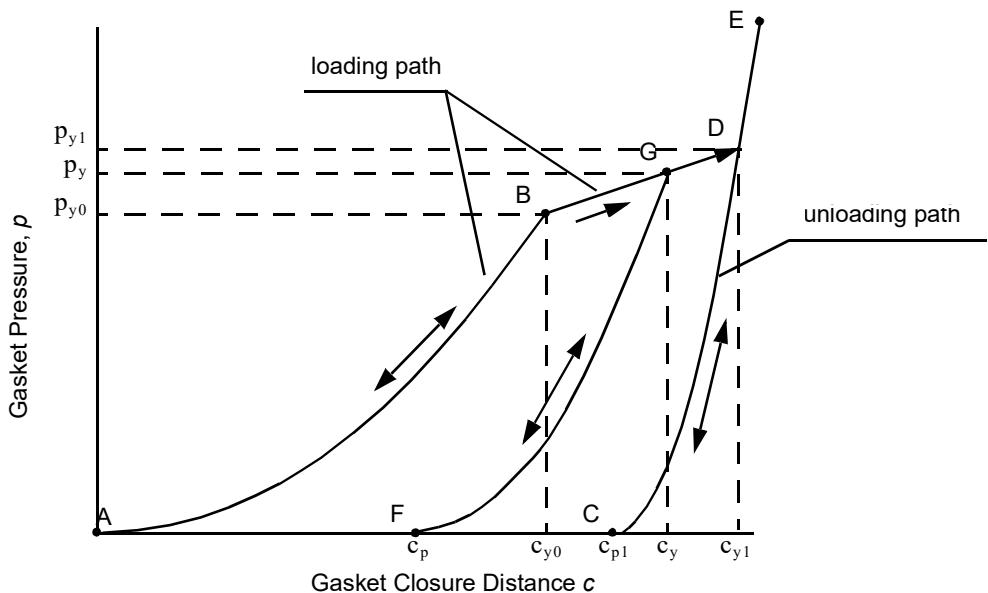


Figure 9-110 Pressure-closure Relation of a Gasket

8. See associated MATTG entry for temperature variation of these properties.
9. All continuation cards must be entered.
10. MID, IDMEM, BEHAV, TABLD, TABLU1, YPRS, EPL and GPL must be non-zero.
11. Each unloading curve must begin with gasket pressure of 0.0. Subsequent unloading curves must start with larger closure distances (when gasket pressure is 0.0) than previous unloading curves.
12. Points on loading and unloading curves must be defined in order of increasing gasket pressure.
13. MATG may be referenced by “solid” composite elements only, via PCOMPLS linked to PSOLID and PSHLN2 linked to PLPLANE.
14. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATTIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.
15. Result data recovery is not supported in SOL 400 when ANALSYS=MODES, MFREQ or DFREQ.

MATHE**Hyperelastic Material Properties**

Specifies hyperelastic (rubber-like) material properties for nonlinear (large strain and large rotation) analysis in SOL 600 and SOL 400 only.

Format 1 (Default): Generalized Mooney-Rivlin Model (Model = Mooney). The first two lines are required, the others may be omitted depending on how many terms are desired.)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO	Texp	Tref	GE	
	C10	C01	D1	TAB1	TAB2	TAB3	TAB4	TABD	
	C20	C11	C02	D2	NA	ND			
	C30	C21	C12	C03	D3				
	C40	C31	C22	C13	C04	D4			
	C50	C41	C32	C23	C14	C05	D5		

Format 2: (Model = Ogden or Foam)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model	NOT	K	RHO	Texp	Tref	GE	
	Mu1	Alpha1	Beta1						
	Mu2	Alpha2	Beta2	Mu3	Alpha3	Beta3			
	Mu4	Alpha4	Beta4	Mu5	Alpha5	Beta5			
	D1	D2	D3	D4	D5				

Note that Foam does not support D1~D5.

Format 3: Arruda-Boyce model or Gent Model (Model = Aboyce or Gent)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO	Texp	Tref	GE	
	NKT	N/E	Im						
	D1	D2	D3	D4	D5				

Format 4: User Strain Energy Function (Model = GHDMI) (SOL 400 only)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO	Texp	Tref	GE	

Example - Format 1:

MATHE	2	Foam		3.30E+07	0.00035	4.00E-06	75	0.02	
	2.00E+04	1.5	1.15						
	1.50E+04								

Example - Format 2:

MATHE	1	Ogden	2	200000					
	16	2	1.15						
	-4	-2							

Example - Format 3:

MATHE	2	Gent		3.30E+07	0.00035	4.00E-06	75	0.02	
	2.00E+04	1250							

Example - Format 4:

MATHE	2	GHEM1		3.30E+07	0.00035	2.50E-06	70.5	0.015	
-------	---	-------	--	----------	---------	----------	------	-------	--

Descriptor	Meaning
MID	Identification number of a MATHE entry. (Integer > 0; no Default)
Model	Select hyperelastic material model from (Character; Default = Mooney):
Mooney	for generalized Mooney-Rivlin hyperelastic model. See Remark 1.
Ogden	for Ogden hyperelastic model. See Remark 2.
Foam	for foam model. See Remark 3.
Aboyce	for Arruda-Boyce strain energy model. See Remarks 4. and 17.
Gent	for Gent strain energy model. See Remarks 5. and 17.
GHEM1	Invariant-base user subroutine (foam models only) $W = W(I_1, I_2, I_3)$
GHEM2	Principal-stretch based user subroutine (foam models only) $W = W(\lambda_1, \lambda_2, \lambda_3)$
GHEM3	Invariant-based with volumetric and deviatoric split user subroutine (foam models only) $W = W_{dev}(I_1, I_2) + U(J)$
GHEM4	Principal-stretch based with volumetric and deviatoric split user subroutine (foam models only) $W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + U(J)$
GHEM5	Invariant-based with deviatoric part only (rubber models only) $W = W_{dev}(I_1, I_2)$

Descriptor	Meaning												
GHEM6	<p>Principal-stretch based with deviatoric part only (rubber models only)</p> $W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3)$ <p>where I_1, I_2, I_3 (λ_1, λ_2, and λ_3) are strain invariants (principal stretches), and \bar{I}_1 and \bar{I}_2 ($\bar{\lambda}_1, \bar{\lambda}_2$, and $\bar{\lambda}_3$) are their deviatoric parts, defined by and $\bar{I}_2 = J^{-4/3}I_2$ ($\bar{\lambda}_i = J^{-1/3}\lambda_i$, $i = 1, 2, 3$); J is the determinant of the deformation gradient.</p> <p>For GHEMi models, see Remarks 13., 14., and 15.</p>												
NOT	Number of terms to be used in Ogden and Foam model. ($0 < \text{Integer} \leq 5$; Default = 1)												
K	Specifies a bulk modulus. See Remarks 7. and 16. (Real > 0 or -1.0; Default = automatically set for nearly incompressible condition. Enter -1.0 if nonlinear volumetric strain energy function defined by Di will be used.)												
RHO	Mass density in original configuration. (Real; Default = 0.0)												
Texp	Coefficient of thermal expansion. See Remark 8. (Real; Default = 0.0)												
Tref	Reference temperature at which the thermal expansion coefficient is measured. Tref is used only if the thermal expansion coefficient is temperature-dependent. (Real; Default = 0.0)												
GE	<p>If GE is real it represents the structural damping coefficient. See Remark 12. (Real; Default = 0.0).</p> <p>For SOL 600 only, if GE is integer, it represents the form of the user subroutine uelastomer.f as follows:</p> <table> <tbody> <tr> <td>1</td><td>Foam Material - Invariant-based model $W = W(I_1, I_2, I_3)$</td></tr> <tr> <td>2</td><td>Foam Material - Principal stretch based $W = W(\lambda_1, \lambda_2, \lambda_3)$</td></tr> <tr> <td>3</td><td>Foam Material -Invariant-based with volumetric and deviatoric split $W = W_{dev}(I_1, I_2) + U(J)$</td></tr> <tr> <td>4</td><td>Foam Material - Principal stretch with volumetric and deviatoric split $W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + U(J)$</td></tr> <tr> <td>5</td><td>Rubber Material (Mooney, Arruda-Boyce or Gent) - Invariant based $W = W_{dev}(\bar{I}_1, \bar{I}_2)$</td></tr> <tr> <td>6</td><td>Rubber Material (Ogden) - Principal stretch based) $W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3)$</td></tr> </tbody> </table> <p>where I_1, I_2, and I_3 (λ_1, λ_2, and λ_3) are strain invariants (principal stretches), and \bar{I}_1 and \bar{I}_2 ($\bar{\lambda}_1, \bar{\lambda}_2$, and $\bar{\lambda}_3$) are their deviatoric parts, defined by $\bar{I}_1 = J^{-2/3}I_1$ and $\bar{I}_2 = J^{-4/3}I_2$ ($\bar{\lambda}_i = J^{-1/3}\lambda_i$, $i = 1, 2, 3$); J is the determinant of the deformation gradient.</p> <p>If GE is integer, Bulk Data entry USRSUB6 is also required (see Marc Vol D for more details.)</p>	1	Foam Material - Invariant-based model $W = W(I_1, I_2, I_3)$	2	Foam Material - Principal stretch based $W = W(\lambda_1, \lambda_2, \lambda_3)$	3	Foam Material -Invariant-based with volumetric and deviatoric split $W = W_{dev}(I_1, I_2) + U(J)$	4	Foam Material - Principal stretch with volumetric and deviatoric split $W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + U(J)$	5	Rubber Material (Mooney, Arruda-Boyce or Gent) - Invariant based $W = W_{dev}(\bar{I}_1, \bar{I}_2)$	6	Rubber Material (Ogden) - Principal stretch based) $W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3)$
1	Foam Material - Invariant-based model $W = W(I_1, I_2, I_3)$												
2	Foam Material - Principal stretch based $W = W(\lambda_1, \lambda_2, \lambda_3)$												
3	Foam Material -Invariant-based with volumetric and deviatoric split $W = W_{dev}(I_1, I_2) + U(J)$												
4	Foam Material - Principal stretch with volumetric and deviatoric split $W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + U(J)$												
5	Rubber Material (Mooney, Arruda-Boyce or Gent) - Invariant based $W = W_{dev}(\bar{I}_1, \bar{I}_2)$												
6	Rubber Material (Ogden) - Principal stretch based) $W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3)$												

Descriptor	Meaning
NA	Order of the distortional strain energy polynomial function. See Remark 6. (0 < Integer < or =5; Default = 2; Default = 5 for SOL 600)
ND	Order of the Volumetric strain energy polynomial function. See Remark 6. (0 < Integer < or =5; Default = 1; Default = 5 for SOL 600)
Cij	Material constants related to distortional deformation for generalized Mooney-Rivlin model. (Real; Default = 0.0) SOL 600 uses only five constants (C10, C01, C20, C11, and C30) and ignores others.
Di	Material constants related to volumetric deformation. (Real \geq 0.0) Needed only if K = -1.0. Not used for foam model. See Remark 7. For SOL 600, only Mooney model supports nonlinear volumetric strain energy function.
TAB1	Table identification number of a TABLES1 entry that contains simple tension-compression data to be used in the estimation of the material constants Cij. The x-values in the TABLES1 entry must be stretch ratios l/l_0 and y-values must be values of the engineering stress F/A_0 . l_0 is the initial length and A_0 is the initial cross-sectional area. See Remark 9. (Integer > 0 or blank)
TAB2	Table identification number of a TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants Cij. The x-values in the TABLES1 entry must be stretch ratios l/l_0 and y-values must be values of the engineering stress $F/A_0 \cdot l_0$. l_0 is the initial length and A_0 is the initial cross-sectional area. See Remark 9. (Integer > 0 or blank)
TAB3	Table identification number of a TABLES1 entry that contains simple shear data to be used in the estimation of the material constants Cij. The x-values in the TABLES1 entry must be values of the shear strain and y-values must be values of the engineering shear stress. (Integer > 0 or blank)
TAB4	Table identification number of a TABLES1 entry that contains pure shear data to be used in the estimation of the material constants Cij. The x and y values in the TABLES1 entry must be stretch ratios $\lambda_1 = l/l_0$ and the values of the nominal stress $F/A_0 \cdot l_0$ and A_0 are the initial length and cross-sectional area, respectively, in the l-direction. See Remark 9. (Integer > 0 or blank)
TABD	Table identification number of a TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constant Di. The x-values in the TABLES1 entry must be values of the volume ration $J = \lambda^3$ where $\lambda = l/l_0$ is the stretch ratio in all three directions; y-values must be values of the pressure, assumed positive in compression. See Remark 9. (Integer > 0 or blank)
Muk	Coefficients μ_k of the strain energy function for Ogden or foam material. See Remarks 2. and 3. (Real; Default = 0)
Alphak	Coefficients α_k of the strain energy function for Ogden or foam material. See Remarks 2. and 3. (Real; Default = 0)
Betak	Coefficients β_k of the strain energy function for foam material. These fields should be left blank for Ogden mode. See Remarks 2. and 3. (Real; Default = 0)

Descriptor	Meaning
NTK	Material constant for Arruda-Boyce strain energy model. (Real > 0; Default = 1)
N/E	Material constant representing the number (N) of statistical links of the chain for Arruda-Boyce model; or tensile modulus (E) for Gent strain energy model. (Real > 0; Default = 1)
Im	Maximum first invariant for Gent strain energy model. (Real > 0; Default = 0)

Remarks:

1. The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$W(J, \bar{I}_1, \bar{I}_2) = \sum_{i+j=1}^3 C_{ij} (\bar{I}_1 - 3)^i (\bar{I}_2 - 3)^j + 4.5K(J^{1/3} - 1)^2$$

with

$$\bar{I}_1 = \bar{\lambda}_1^2 + \bar{\lambda}_2^2 + \bar{\lambda}_3^2$$

$$\bar{I}_2 = \bar{\lambda}_1^2 \bar{\lambda}_2^2 + \bar{\lambda}_2^2 \bar{\lambda}_3^2 + \bar{\lambda}_3^2 \bar{\lambda}_1^2$$

where K and J are bulk modulus and volume ratio, respectively.

For small strains, the shear modulus G is related to the Mooney-Rivlin constants by

$$G = 2(C_{10} + C_{01}) \text{ with } C_{01} \approx 0.25 C_{10}$$

The model reduces to a Mooney-Rivlin material with only two constants (C10 and C01), and to a Neo-Hookean material with one constant (C10). The third order Mooney-Rivlin model in SOL 600 uses only five distortional constants (C10, C01, C11, C20, C30) and the bulk modulus K for volumetric deformation. Instead of MATHE, the hyperelastic material can be specified using MATHP Bulk Data entry in SOLs 106, 129, and 600.

2. For the Ogden material model, the strain energy function is

$$W = \sum_{k=1}^5 \frac{\mu_k}{\alpha_k} (\bar{\lambda}_1^{\alpha_k} + \bar{\lambda}_2^{\alpha_k} + \bar{\lambda}_3^{\alpha_k} - 3) + 4.5K(J^{1/3} - 1)^2$$

where α_k and μ_k are the moduli and exponent constants, while $\bar{\lambda}_i$ is the deviatoric stretch ratio defined as:

$$\bar{\lambda}_i = J^{-\frac{1}{3}} \lambda_i$$

and J and K are the determinant of the deformation gradient and the bulk modulus, respectively. A two-term Ogden model is equivalent to a simple Mooney-Rivlin model

$$\mu_1 = 2C_{10} \quad \text{and} \quad \mu_2 = 2C_{01}$$

with $\alpha_1 = 2$. and $\alpha_2 = 2$.

3. For foam material model, the distortional strain energy function is the same as Ogden. The volumetric energy function of foam model is defined by a polynomial function with coefficients β_i
4. For the Arruda-Boyce model, the strain energy function is

$$W = N_{KT} \left[\frac{1}{2}(\bar{I}_1 - 3) + \frac{1}{20N}(\bar{I}_1^2 - 9) + \frac{1}{1050N^2}(\bar{I}_1^3 - 27) \right. \\ \left. + \frac{19}{7000N}(\bar{I}_1^4 - 81) + \frac{519}{67375N^4}(\bar{I}_1^5 - 243) \right] + 4.5K(J^{1/3} - 1)^2$$

with $\bar{I}_1 = \tilde{\lambda}_1^2 + \tilde{\lambda}_2^2 + \tilde{\lambda}_3^2$

where N_{KT} is a material constant and N is a material parameter representing the number of statistical links of the material chain.

If the material test data are available from multiple experiments such as uniaxial and equi-biaxial tests, the Ogden model is more accurate in fitting experimental results. If only uniaxial tension data is available, the Arruda-Boyce model provides more accurate data fitting for multiple modes of deformation.

5. For the Gent model, the strain energy function is

$$W = -\frac{1}{6}EI_m \log \left[\frac{I_m}{I_m - \bar{I}_1 + 3} \right]$$

where E and I_m are tensile modulus and maximum first invariant, respectively.

6. The curve fitting is activated if any of the TAB1, TAB2, TAB3, TAB4 and/or TABD are specified. The NA and ND fields are used in case of the curve fitting for Mooney from the experimental data.
7. Although the conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible, SOL 600 provides a compressible rubber model. Nearly incompressible material may be simulated with a large value of K . The default value for the Mooney-Rivlin model represents a nearly incompressible condition, which is $K = 10^4(C_{10} + C_{01})$. In case of Ogden model, the default is $2500 \cdot \sum \mu_k \alpha_k$. If any of the D_i are non-zero, then the entered value of the Bulk Modulus must be -1.0 and the nonlinear volumetric strain energy function will be used. See Remark 16.
8. The thermal expansion coefficient is a secant value measured with respect to a temperature, Tref.

The thermal strain is computed by $\varepsilon_{th} = \bar{\alpha}(T - T_0)$

where T_0 is an initial temperature. The secant coefficient of thermal expansion is related to the instantaneous coefficient of thermal expansion by

$$\alpha = \frac{d\varepsilon_{th}}{dT} = \bar{\alpha} + \frac{d\bar{\alpha}}{dT}(T - T_0)$$

The choice of secant or instantaneous coefficient (CTE) of thermal expansion is determined by PARAM,MARALPHA in SOL 600. All CTE's in SOL 600 models must be either secant or instantaneous.

9. All the material constants may be obtained from least squares fitting of experimental data. One or more of four experiments (TAB1 to TAB4) may be used to obtain constants for distortional properties. The bulk modulus K may be obtained from pure volumetric compression data (TABD). If all TAB1 through TAB4 fields are blank, the material constants must be specified by the user. Parameter estimation, specified through any of the TABLES1 entries, supersedes the manual input of the parameters. It is recommended that the GUI be used to determine the materials constants, so one can visualize the correlation between the numerical representation and the experimental data. It is recommended that the experimental data span the range of anticipated strains to reduce the chances of adverse numerical calculations.
10. All the alphanumeric fields are recognizable by the first four letters.
11. Enter NKT and N/E for Aboyce (leave Im blank). Enter N/E and Im for Gent (leave NKT blank).
12. For SOL 600, Bulk Data entry USRSUB6 is also required.
13. The GHEMi models using the user subroutine are only available if the updated Lagrange formulation is used for the hyperelastic material. The user subroutine MD UELASTOMER defined with the GHEMi must be used along with the Bulk Data entry MATUDS.
14. The GHEMi are not available for plane stress, shell, membrane, and beam elements.
15. For the GHEM3 and GHEM4 models, the user subroutine is called twice, once with either a 3 or 4 to evaluate the deviatoric strain energy and once with a 7 to evaluate the volumetric strain energy.
16. The series representation of the volumetric strain energy is only available for the updated Lagrange formulation.
17. The Gent and Arruda-Boyce models are not available for plane stress, shell, membrane, and beam elements.
18. The Ogden, Foam, Arruda-Boyce and Gent models are not available in SOLs 106 and 129.
19. In SOL 400, MATHE is only supported for nonlinear elements with property extensions. This implies that for such elements, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Also, for incompressible rubber models, linear 2D triangular elements specified on PSHLN2 and 3D tetrahedral elements specified on PSLDN1 should be associated with an incompressible formulation (IPS for 2D plane strain, IAX for 2D axisymmetric and ISOL for 3D tetrahedral. Note that only PSOLID with PSLDN1 should be used for 3D solid nonlinear elements and PLSOLID cannot be used for MATHE. Note also that,

prior to MD Nastran 2010, if the property extensions were missing, then the analysis would stop with an error. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

20. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.

Notes (SOL 600 only):

1. The structural damping constant GE is specified in Marc under the option DAMPING as a numerical damping γ , i.e.,

$$C = \alpha M + \left(\beta + \frac{2\gamma}{\omega} \right) K$$

in which 2γ is equivalent to GE; α and β are equivalent to user parameters ALPHA1 and ALPHA2.

2. Material curve-fitting from experimental data is not supported in SOL 400. Direct material property input is required.
3. This Bulk Data entry accommodates Marc's input data under the model definition options MOONEY, OGDEN, and FOAM as well as the parameter ELASTICITY. It also accommodates MATHP input data in Nastran.

MATHE**Hyperelastic Material Properties - SOL 700**

Specifies hyperelastic (rubber-like) material properties for nonlinear analysis. Use SOL700 only.

Format 1 (default): Mooney-Rivlin Model (Model=Mooney1)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model			RHO				
	A	B	NU						

Format 2: Ogden Model (Model=Ogden1)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	MODEL			RHO				
	MU1	ALPHA1	NU						
	MU2	ALPHA2		MU3	ALPHA3				
	MU4	ALPHA4		MU5	ALPHA5				
	MU6	ALPHA6		MU7	ALPHA7				
	MU8	ALPHA8							

Format 3: Foam Model (Model=FOAM1)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO				
	TYPE	VALUE	CUTOFF	TABLE	TABY				
	ALPHA	UNLOAD							

Example –Format 1:

MATHE	7	MOONEY1			100.				
	0.34	0.27	0.495						

Example –Format 2:

MATHE	7	OGDEN1			100.				
	6.3	1.3	0.495						
	0.12	5.0		-0.1	-2.0				

Example –Format 3:

MATHE	7	FOAM1		3.	100.				
	1.0	-100.0	1.0	111	112				
	0.4	2.0							

Descriptor	Contents
MID	Material identification number. (Integer > 0; required.)
Model	Select hyperelastic material model from (Character; Default = Mooney1)
Mooney1	Mooney-Rivlin hyperelastic model. It is only available for solid elements. See Remark 1.
Ogden1	Ogden hyperelastic model. It is only available for solid elements. See Remark 2.
FOAM1	Isotropic elastic foam material. It is only available for solid elements. See Remarks 3.-8.
RHO	Density. (Real > 0; required)
A	Strain-energy density function constant. (Real; required)
B	Strain-energy density function constant. (Real; required)
NU	Poisson's ratio. ($0.0 \leq R < 0.5$; required)
MU1-MU8	Strain energy density function coefficients. MU1 is required. (Real; default=0.0)
ALPHA1-ALPHA8	Strain energy density function coefficients. ALPHA1 is required. (Real; default=0.0)
TYPE	The type of data defined as the x-value in the table. (Real > 0.0; default=1.0) <ul style="list-style-type: none"> 1.0 Crush factor. (= 1-relative volume) 2.0 True strain
VALUE	The value for cut-off stress. (Real≤0.0; default=-0.1×Young's modulus)
CUTOFF	Cut-off stress. (Real > 0.0; default=3.0) <ul style="list-style-type: none"> 1.0 Stress for tensile failure 2.0 Minimum stress 3.0 Not used
TABLE	TABLED1 ID defining the variation of stress (y-value) with crush factor or true strain (x- value). (Integer > 0; required)
TABY	TABLED1 ID giving the variation of the scale factor for the stress (y-value) with the strain rate (x-value). (Integer > 0; default=not used)
ALPHA	Energy dissipation factor. ($0.0 \leq \text{Real} \leq 1.0$; default=0.0)
UNLOAD	Unloading option. (Real > 0.0; default=3.0) <ul style="list-style-type: none"> 1.0 Unloading via exponential curve 2.0 Unloading via piecewise linear curve 3.0 Unloading via quadratic curve

Remarks:

1. The Mooney-Rivlin constitutive behavior of this material is defined as a total stress/total strain relationship. The nonlinear elastic material response is formulated by a strain-energy density function for large strain components rather than by Hooke's law.

The strain-energy density function is formulated according to the Mooney-Rivlin model and is defined as

$$W(I_1, I_2, I_3) = A(I_1 - 3) + B(I_2 - 3) + C\left(\frac{1}{I_3^2} - 1\right) + D(I_3 - 1)^2$$

The constants C and D are defined as:

$$\begin{aligned} C &= \frac{1}{2}A + B \\ D &= \frac{A(5v - 2) + B(11v - 5)}{2(1 - 2v)} \end{aligned}$$

where A, B and v are input parameters.

I_1 , I_2 and I_3 are strain invariants in terms of stretches.

For a rubber-like material, the shear modulus G is much less than the bulk modulus K. As a result, Poisson's ratio is nearly equal to one half.

2. The constitutive behavior of this material is defined as a total stress/total strain relationship. The nonlinear elastic material response is formulated by a strain-energy density function for large strain components using Ogden function.

The strain-energy density function is formulated according to the Ogden model and is defined as

$$W = \sum_{i=1}^3 \sum_{j=1}^8 \frac{\mu_j}{\alpha_j} (\bar{\lambda}_i^{\alpha_j} - 1) + \frac{1}{2}K(J-1)^2$$

The over bar (-) indicates that the volumetric effects have been eliminated from the principal stretches.

3. Poisson's ratio for FOAM1 model is effectively zero. Therefore, the shear and elastic moduli are calculated from K, the bulk modulus.
4. For this model, the stress-strain curve is independent of the experimental test performed to obtain the material data (uniaxial, shear, or volumetric). The most common test is the uniaxial compression test where the stress-strain characteristic can either be defined in terms of the amount of crush, which is minus the engineering strain, or in terms of the true strain. Since Poisson's ratio is effectively zero, the amount of crush is defined as $\left(1 - \frac{V}{V_0}\right)$, with $\left(\frac{V}{V_0}\right)$ as the relative volume, and the true strain is defined as.

$$\int_{t_0}^t \frac{dV}{V}$$

or

$$\ln\left(\frac{V}{V_0}\right)$$

The crush factor must be between 0 and 1. The true strain must always be negative and the stress positive (absolute value).

5. The yield surface in three-dimensional space is a sphere in principal stresses, and is defined by

$$\tau_{11}^2 + \tau_{22}^2 + \tau_{33}^2 = R_s^2$$

where the radius of the sphere, R_s , depends on the strains and strain rates as follows:

$$R_s = f_1(R_e)f_2(R_r)$$

with

$$R_e = \varepsilon_{11}^2 + \varepsilon_{22}^2 + \varepsilon_{33}^2$$

and

$$R_r = \dot{\varepsilon}_{11}^2 + \dot{\varepsilon}_{22}^2 + \dot{\varepsilon}_{33}^2$$

and f_1 is the function supplied in the stress-strain table and f_2 (if defined) is the function supplied in the factor-strain rate table.

6. A minimum (CUTOFF=2.0) or failure (CUTOFF=1.0) tensile stress can be defined. In the first case this corresponds to a tensile cut-off where the stress cannot fall below the minimum value. In the second case, if the stress falls below the failure stress the element fails and cannot carry tensile loading for the remainder of the analysis. Thus the stress can never become negative again.
7. The unloading behavior is piecewise linear (UNLOAD=2.0), quadratic (UNLOAD=3.0) or exponential (UNLOAD=1.0). The unloading curve is constructed such that the ratio of the dissipated energy (area between compressive loading and unloading curve) to total energy (area under the loading curve) is equal to the energy dissipation factor alpha. In the case of piecewise linear unloading, Nastran SOL700 constructs an unloading curve whose segments are parallel to the supplied compression table, except for the first and last segments, which pass respectively through the origin and the point P on the compression curve where the unloading starts. In the case of quadratic unloading, Nastran SOL700 constructs a quadratic curve starting in the origin and ending in point P. If the quadratic unloading curve falls below the strain axis, then the unloading stress is set to zero. In the case of exponential unloading, the unloading curve is constructed in a similarly to quadratic unloading except for the shape of the curve, which is created from an exponential function instead of a quadratic polynomial.
8. Not default values for taby, alpha, unload options are allowed only when cutoff is set to 1.0 or 2.0.

MATHED

Damage Model Properties for Hyperelastic Materials - SOL 600

Specifies damage model properties for hyperelastic materials to be used for static, quasi static or transient dynamic analysis in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATHED	MID	method	Scale1	Relax1	Prop1	Scale2	Relax2	Prop2	
	Dinf	N/A	Scalem1	Relaxm1	N/A	Scalem2	Relaxm2		

Example:

MATHED	100		1.	0.3	0.5	0.8	0.4	1.0	
--------	-----	--	----	-----	-----	-----	-----	-----	--

Descriptor	Meaning
MID	Identification number of MATHE entry. See Remark 1. (Integer > 0)
Method	Select a method for damage calculation (Character) from: Multip for multiplicative decomposition. (Default) Additiv for additive decomposition. See Remarks 2. and 3.
Scale1,2	Scaling factor d_n for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real ≥ 0.0 ; < 1.0; Default = 0.0)
Relax1,2	Relaxation rate η_n for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real ≥ 0 or blank)
Prop1,2	Proportionality factor δ_n for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. See Remark 4. (Real > 0.0; Default = 1.0)
Dinf	d^∞ described in the equations of Remark 3. If Blank, the program will compute it; however even if set, the program usually calculates it. In most cases $d^\infty = 1.0 - scale1 - scale2$. (Real > 0.0 or blank)
Scalem1,2	Scaling factor d_m for m=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real ≥ 0.0 ; Default = 0.0)
Relaxm1,2	Relaxation rate λ_m for m=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real > 0.0 or blank)

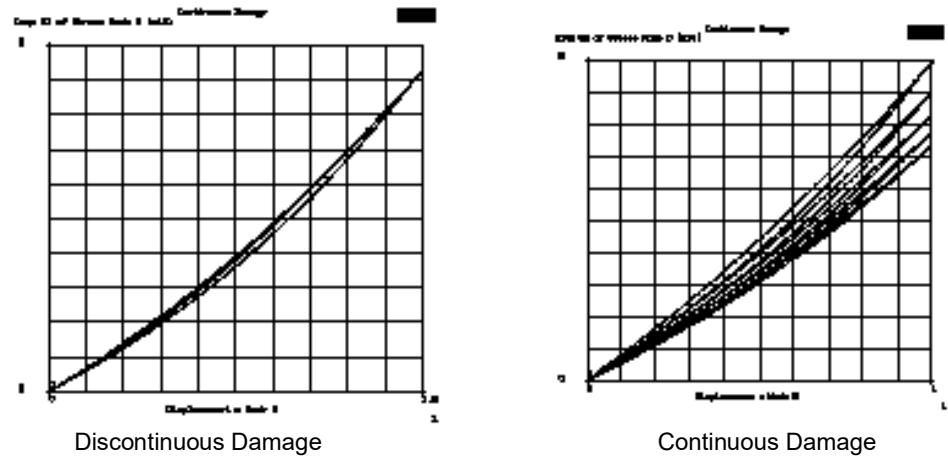
Remarks:

1. The MATHE Bulk Data entry with the same MID must exist for MATHED to be effective.
 The damage capability is available for all the elastomeric materials (Mooney-Rivlin, Ogden, Gent, Arruda-Boyce).

2. Under repeated application of loads, elastomers undergo damage by mechanisms involving chain breakage, multi-chain damage, micro-void formation, and micro-structural degradation due to detachment of filler particles from the network entanglement. The damage model for elastomeric materials is based on the undamaged strain energy function W_0 , multiplied by a Kachanov damage factor, K , i.e.,

$$W = K(\alpha, \beta)W_0$$

where α and β are parameters for discontinuous and continuous damage models, respectively. Two types of phenomenological models, discontinuous and continuous, exist to simulate the damage. The discontinuous damage model simulates the "Mullins' effect," which involves a loss of stiffness (represented by a parameter α) below the previously attained maximum strain. The higher the maximum attained strain the larger the loss of stiffness is. There is a progressive stiffness loss with increasing maximum strain amplitude. Most of the stiffness loss takes place in the first few cycles provided the maximum strain level is not increased. This phenomenon is observed in both filled as well as natural rubber although the higher level of carbon black particles increases the hysteresis and the loss of stiffness. The continuous damage model (Miehe's formulation) can simulate the damage accumulation for strain cycles for which the values of effective energy is below the maximum attained value of the past history. The evolution of continuous damage parameter is governed by the arc-length of the effective strain energy, represented by a parameter β .



3. Both the continuous damage as well as the discontinuous damage can be modeled by a cumulative Kachanov factor in multiplicative or additive decomposition form.

For multiplicative decomposition

$$K(\alpha, \beta) = d^\infty + \sum_{n=1}^2 d_n \exp\left(-\frac{\alpha + \delta_n \beta}{\eta_n}\right)$$

For additive decomposition

$$K(\alpha, \beta) = d^\infty + \sum_{n=1}^2 d_n^\alpha \exp\left(-\frac{\alpha}{\eta_n}\right) + \sum_{m=1}^2 d_m^\beta \exp\left(-\frac{\beta}{\lambda_m}\right)$$

where d_n , δ_n , η_n , d_m , and λ_m are constants specified by the user, and d^∞ is calculated by the program such that the Kachanov factor assumes a value of unity at zero damage if left blank.

4. The proportionality factor is not used by additive decomposition which requires the continuation fields to include the continuous damage model.
5. User subroutines must be called out using PARAM,MARCUSUB,CHAR where CHAR is a character variable such as UDAMAG.

Note:

1. This Bulk Data entry accommodates Marc's input data under the model definition option DAMAGE.

MATHP**Hyperelastic Material Properties**

Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).

Format:

1	2	3	4	5	6	7	8	9	10
MATHP	MID	A10	A01	D1	RHO	AV	TREF	GE	
		NA	ND						
	A20	A11	A02	D2					
	A30	A21	A12	A03	D3				
	A40	A31	A22	A13	A04	D4			
	A50	A41	A32	A23	A14	A05	D5		
	TAB1	TAB2	TAB3	TAB4				TABD	

Example:

MATHP	2	80.	20						
		1	1						

Descriptor	Meaning
MID	Identification number of a MATHP entry. (Integer > 0; No Default)
Aij	Material constants related to distortional deformation. (Real; Default = 0.0)
Di	Material constants related to volumetric deformation. (Real ≥ 0 ; Default for D1 is $10^3 \cdot (A10 + A01)$; Default for D2 through D5 is 0.0.)
RHO	Mass density in original configuration. (Real; Default = 0.0)
AV	Coefficient of volumetric thermal expansion. (Real; Default = 0.0)
TREF	Reference temperature. See MATBV, 2206 . (Real; Default = 0.0)
GE	Structural damping element coefficient. (Real; Default = 0.0)
NA	Order of the distortional strain energy polynomial function. ($0 < \text{Integer} \leq 5$; Default = 1)
ND	Order of the volumetric strain energy polynomial function. ($0 < \text{Integer} \leq 5$; Default = 1)
TAB1	Table identification number of a TABLES1 entry that contains simple tension/compression data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios l/l_0 and yi values must be values of the engineering stress F/A_0 . Stresses are negative for compression and positive for tension. (Integer > 0 or blank)

Descriptor	Meaning
TAB2	Table identification number of a TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios l/l_0 . yi values must be values of the engineering stress F/A_0 . l is the current length, F is the current force, l_0 is the initial length and A_0 is the cross-sectional area. In the case of pressure of a spherical membrane, the engineering stress is given by $P r_0 \lambda^2 / 2 t_0$ where P = current value of the pressure and r_0 , t_0 = initial radius and thickness. (Integer > 0 or blank)
TAB3	Table identification number of a TABLES1 entry that contains simple shear data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be values of the shear tangent γ and yi values must be values of the engineering shear stress F/A_0 . (Integer > 0 or blank)
TAB4	Table identification number of a TABLES1 entry that contains pure shear data to be used in the estimation of the material constants Aij. xi and yi values in the TABLES1 entry must be stretch ratios $\lambda_1 = l/l_0$ and values of the nominal stress F/A_0 . l is the current length, F is the current force, l_0 and A_0 are the initial length and cross-sectional area, respectively in the 1-direction. (Integer > 0 or blank)
TABD	Table identification number of a TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constants Di. xi values in the TABLES1 entry must be values of the volume ratio $J = \lambda^3$ where $\lambda = l/l_0$ is the stretch ratio in all three directions; yi values must be values of the pressure, assumed positive in compression. (Integer > 0 or blank)

Remarks:

1. The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$U(J, \bar{I}_1, \bar{I}_2) = \sum_{i+j=1}^{NA} A_{ij} (\bar{I}_1 - 3)^i (\bar{I}_2 - 3)^j + \sum_{i=1}^{ND} D_i (J - 1 - AV(T - T_0))^{2i}$$

$$A00 = 0$$

where \bar{I}_1 and \bar{I}_2 are the first and second distortional strain invariants, respectively; $J = \det F$ is the determinant of the deformation gradient; For small strains, the bulk modulus K and the shear modulus G are related to the Mooney-Rivlin Material.

$2D1 = K$ and $2(A10 + A01) = G$ at small strains, in which K is the bulk modulus and G is the shear modulus. The model reduces to a Mooney-Rivlin material if $NA=1$ and to a Neo-Hookean material if $NA = 1$ and $A01 = 0.0$. See Remark 2. For Neo-Hookean or Mooney-Rivlin materials no continuation entry is required. T is the current temperature and T_0 is the initial temperature.

2. Conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible. Full incompressibility is not presently available but may be simulated with a large enough value of D1. Note that a value of D1 lower than $10^3 \cdot (A10 + A01)$ is, however, not recommended.
3. The material constants A_{ij} and D_i are obtained from least squares fitting of experimental data. One or more of four experiments (TAB1 to TAB4) may be used to obtain constants for distortional properties A_{ij} . D_i may be obtained from pure volumetric compression data (TABD). If all TAB1 through TAB4 are blank, A_{ij} must be specified by the user. Parameter estimation, specified through any of the TABLES1 entries, supercedes the manual input of the parameters.
4. IF ND=1 and a nonzero value of D1 is provided or obtained from experimental data in TABD, then the parameter estimation of the material constants A_{ij} takes compressibility into account in the cases of simple tension/compression, equibiaxial tension, and general biaxial deformation. Otherwise, full incompressibility is assumed in estimating the material constants.
5. See Chapters [Hyperelastic Elements](#), 197, [Hyperelastic Material](#), 295 and [Hyperelastic Material](#) in the *MSC Nastran Reference Guide* for further details.
6. In SOL400, MATHP is supported for 2 sets of elements:
 - a. Incompressible conventional 2D (plane strain and axisymmetric) elements and 3D (continuum) elements without property extensions. They are prescribed through PLPLANE and PLSOLID, respectively. The original element formulations that are also available in SOL106 are used in these cases.
 - b. Incompressible elements that are supported through property extensions. For such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively.
Also, for incompressible rubber models, 2D 3-node triangular elements and 3D 4-node tetrahedral elements should be associated with an incompressible formulation by specifying IPS for 2D plane strain and IAX for 2D axisymmetric elements in PSHLN2, as well ISOL for 3D tetrahedral element in PSLDN1.
7. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.

Note:

1. PLSOLID is only used with the elements in (a) and only PSOLID with PSLDN1 can be used for 3D solid nonlinear elements in (b).
2. Prior to MD Nastran 2010, if the property extensions were missing, the analysis would stop with an error. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

MATNLE**Advanced NLELAST Options - SOL 600**

The MATNLEX entries specify advanced forms of nonlinear elastic materials. The simple NLELAST option uses MAT1, MATS1, TALBES1 in the same manner as data is entered for SOL 106 or SOL 129.

Format 1: Standard Nastran NLELAST Model

Use NLELAST on MATS1 entry.

Format 2: Invariant-based Material Model

1	2	3	4	5	6	7	8	9	10
MATNLE2	MID	E	PR	RHO	ALPH	G	TLIM	CLIM	
		TE	TPR	TRHO	TALPH	TG	TTLIM	TCLIM	
	KIND	TYPE	LIM	GE	NAME				

Format 3: Principal Strain-Based Material Model

1	2	3	4	5	6	7	8	9	10
MATNLE3	MID	REF	PR	RHO	ALPH	G	TLIM	CLIM	
		TE	TPR	TRHO	TALPH	TG	TTLIM	TCLIM	
	KIND	TYPE	LIM	GE	NAME				

Format 4: Linear Elasticity with Tension and/or Compression Limits

1	2	3	4	5	6	7	8	9	10
MATNLE4	MID	REF	PR	RHO	ALPH	G	TLIM	CLIM	
		TE	TPR	TRHO	TALPH	TG	TTLIM	TCLIM	
	KIND	TYPE	LIM	GE	NAME				

Format 5: Bi-Modulus Linear Elasticity with Tension/Compression Limits

1	2	3	4	5	6	7	8	9	10
MATNLE5	MID	E	PR	RHO	ALPH	G	TLIM	CLIM	
		TE	TPR	TRHO	TALPH	TG	TTLIM	TCLIM	
	KIND	TYPE	LIM	GE	NAME				
	EC	PRC	TEC	TPRC					

Format 6: Orthotropic Nonlinear Elasticity based upon strains in local directions

1	2	3	4	5	6	7	8	9	10
MATNLE6	MID	E11	E22	E33	PR12	PR23	PR31	RHO	
		TE11	TE22	TE33	TPR12	TPR23	TPR31	TRHO	
	G12	G23	G31	A11	A22	A33			

	TG12	TG23	TG31	TA11	TA22	TA33			
	KIND	TYPE	LIM	GE	NAME				

Example - Format 3:

MATNLE3	2	2.90E+03	0.49	1.33E-04	2.60E-06		999999	999999	
+									
+		1	1	0	MATNLE2A				

Example - Format 4:

MATNLE4	2	2.90E+03	0.49	1.33E-04	2.60E-06		300	25	
+									
+		1	1	0	MATNLE4A				

Example - Format 5:

MATNLE5	2	2.90E+03	0.49	1.33E-04	2.60E-06		300	25	
+									
+		1	1	0	MATNLE4A				

Example - Format 6:

MATNLE6	2	2.90E+03	2.80E+03	0.491	0.492	0.493	1.00E-04	999999	
+									
+	6.00E+01	5.00E+01	4.00E+01	2.00E-06	1.50E-06	1.00E-06			
+									
+		1	1	0	MATNLE6A				

Descriptor	Meaning
MID	Identification number of a MATNLEi entry. (Integer > 0; no Default)
E	Young's modulus. (Real; no Default)
PR	Poisson's ratio. (Real; no Default)
RHO	Mass density. (Real; no Default)
ALPH	Coefficient of thermal expansion. (Real; no Default)
G	Shear Modulus. (Real; no Default)
TLIM	Tensile stress limit. (Real; no Default)
CLIM	Compressive stress limit. (Real; no Default)
TE	ID of TABL3Di entry for E vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)

Descriptor	Meaning
TPR	ID of TABL3Di entry for PR vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TRHO	ID of TABL3Di entry for density vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TALPH	ID of TABL3Di entry for ALPH vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TG	ID of TABL3Di entry for G vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TTLIM	ID of TABL3Di entry for TLIM vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TCLIM	ID of TABL3Di entry for CLIM vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
KIND	Tension/compression usage flag. (Integer; Default = 1) 1 = Both tension and compression data will be used 2 = Only tension data will be used
TYPE	Poisson ratio/Bulk Modulus flag. (Integer; Default = 1) 1 = Constant Poisson's ratio will be used 2 = Constant bulk modulus will be used
LIM	Stress limit flag. (Integer; Default = 0) 0 = Material has no tension or compression limits 1 = Material has tension limits 2 = Material has compression limits 3 = Material has both tension and compression limits
GE	Structural damping coefficient. (Real; Default = 0.0)
NAME	Material name. (Character; no Default) Optional name of the material, may be blank or as long as 16 characters.
EC	Young's modulus in compression. (Real; no Default)
PRC	Poisson's ratio in compression. (Real; no Default)
TEC	ID of TABL3Di entry for EC vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TPRC	ID of TABL3Di entry for PRC vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
E11	Orthotropic modulus E11. (Real; no Default)
E22	Orthotropic modulus E22. (Real; no Default)
E33	Orthotropic modulus E33. (Real; no Default)

Descriptor	Meaning
PR12	Orthotropic Poisson's ratio in 12 direction. (Real; no Default)
PR23	Orthotropic Poisson's ratio in 23 direction. (Real; no Default)
PR31	Orthotropic Poisson's ratio in 31 direction. (Real; no Default)
TE11	ID of TABL3Di entry for E11 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TE22	ID of TABL3Di entry for E22 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TE33	ID of TABL3Di entry for E33 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TPR12	ID of TABL3Di entry for PR12 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TPR23	ID of TABL3Di entry for PR23 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TPR31	ID of TABL3Di entry for PR31 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
G12	Orthotropic shear modulus G12. (Real; no Default)
G23	Orthotropic shear modulus G23. (Real; no Default)
G31	Orthotropic shear modulus G31. (Real; no Default)
A11	Orthotropic coefficient of thermal expansion in 1 direction. (Real; no Default)
A22	Orthotropic coefficient of thermal expansion in 2 direction. (Real; no Default)
A33	Orthotropic coefficient of thermal expansion in 3 direction. (Real; no Default)
TG12	ID of TABL3Di entry for G12 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TG23	ID of TABL3Di entry for G23 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TG31	ID of TABL3Di entry for G31 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TA11	ID of TABL3Di entry for A11 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TA22	ID of TABL3Di entry for A22 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TA33	ID of TABL3Di entry for A33 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)

Remarks:

1. The structural damping constant GE is specified in MARC under the option DAMPING as a numerical damping γ ,

$$C = \alpha M + \left(\beta + \frac{2\gamma}{\omega} \right) K$$

in which 2γ is equivalent to GE; α and β are equivalent to user parameters ALPHA1 and ALPHA2.

2. This Bulk Data entry accommodates MARC's input data under the model definition option NLELAST for material model types 2-6.
3. 3D tables associated with this entry should normally select independent variable 69 (equivalent mechanical strain) or 73 (any strain component) to define the stress-strain curve. Normally the only other independent 3D table variables used with this entry are 12 (temperature) or 16 (equivalent strain rate).
4. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.

MATORT**Elastic 3D Orthotropic Material Properties**

Specifies elastic orthotropic material properties. Can be specified for all 3-D and 2-D continuum elements and shells in SOL400/600/700. Can also be specified with solid composite, solid shell composite and pyramid elements in all linear solution sequences between SOL101 and SOL112, and analysis only for SOL200.

Format:

1	2	3	4	5	6	7	8	9	10
MATORT	MID	E1	E2	E3	NU12	NU23	NU31	RHO	
	G12	G23	G31	A1	A2	A3	TREF	GE	
	IYLD	IHARD	SY		Y1	Y2	Y3	N/A	
	Yshr1	Yshr2	Yshr3	N/A	N/A	N/A	N/A	N/A	
	OPTION	FILE	X1	Y1	Z1	X2	Y2	Z2	

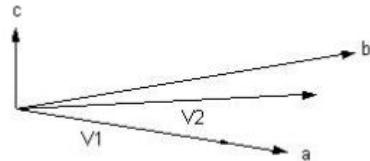
Example:

MATORT	100	3.e6	2.8e7	1.5e5	0.25				
--------	-----	------	-------	-------	------	--	--	--	--

Descriptor	Meaning
MID	Identification number of a MATORT entry. (Integer > 0; No Default)
E1	Modulus of elasticity in longitudinal or 1-direction. See Remark 1. (Real > 0)
E2	Modulus of elasticity in lateral direction or 2-direction. (Real > 0; no Default; must be entered)
E3	Modulus of elasticity in thickness direction or 3-direction. (Real > 0; no Default; must be entered)
NU12	Poisson's ratio ϵ_2/ϵ_1 for uniaxial loading in 1-direction. See Remark 2. (Real; no Default; must be entered)
NU23	Poisson's ratio ϵ_3/ϵ_2 for uniaxial loading in 2-direction. (Real; no Default; must be entered)
NU31	Poisson's ratio ϵ_1/ϵ_3 for uniaxial loading in 3-direction. (Real; Default = NU23)
RHO	Mass density. (Real; Default = 0.0)
G12	Shear modulus in plane 1-2. See Remark 3. (Real > 0; no Default; must be entered)
G23	Shear modulus in plane 2-3. (Real > 0.; no Default; must be entered)
G31	Shear modulus in plane 3-1. (Real > 0; no Default; must be entered)
Ai	Coefficient of thermal expansion in i-direction (Real; Default = 0.0).

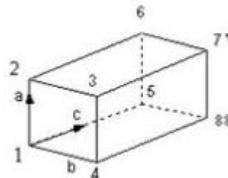
Descriptor	Meaning
TREF	Reference temperature at which the thermal expansion coefficient is measured. TREF is used only if the thermal expansion coefficient is temperature-dependent. (Real; Default = 0.0)
GE	Structural damping coefficient. (Real; Default = 0.0).
IYLD	Integer pertaining to one of the following yield criterion: (Integer) -1 = Elastic 1 = von Mises (Default) 2 = Hill (1948) yield 3 = Barlat (1991) yield 4 = Viscoplasticity through user subroutine UVSCPL
IHARD	Integer pertaining to one of the following work hardening rules: (Integer) 1 = Isotropic (Default) 2 = Kinematic 3 = Combined Isotropic/Kinematic
SY	Equivalent (von Mises) tensile yield stress. (Real > 0.0 or blank; Default = 0.0)
Y1	Hill's yield stress ratio in direction 1. (Real > 0.0 or blank; Default = 0.0)
Y2	Hill's yield stress ratio in direction 2. (Real > 0.0 or blank; Default = 0.0)
Y3	Hill's yield stress ratio in direction 3. (Real > 0.0 or blank; Default = 0.0)
Yshr1	Hill's yield shear stress ratio in direction 1. (Real > 0.0 or blank; Default = 0.0)
Yshr2	Hill's yield shear stress ratio in direction 2. (Real > 0.0 or blank; Default = 0.0)
Yshr3	Hill's yield shear stress ratio in direction 3. (Real > 0.0 or blank; Default = 0.0)

Descriptor	Meaning
OPTION	Material axes option used to determine how the local material axis system is defined. See Remark 7. (Character; default=ELEM)
VECT	Globally orthotropic with the material axes defined by two vectors V1 and V2, specified using the fields X1, Y1, Z1 and X2, Y2, Z2. The a-axis is defined by the first vector. The b- and c-axes are then defined as:



ELEM

Globally orthotropic material with the material axes defined by element topology. Grid point 1 defines the origin, grid point 5 lies on the c-axis, and grid point 2 lies in the ac-plane. The a, b, and c axes are defined as follows:



ELMAT

Orthotropic material properties and the material coordinate system is defined by the element. The material properties are read from a file (formatted). The filename is specified in the sixth field of the first line.

Format of material properties file:

```
Record#
EID, DENSITY, DUMMY, DUMMY, DUMMY,
E1, E2, E3, G12, G23, G31,
NU12, NU13, NU23, NU21, NU31, NU32
```

ELPROP

Globally orthotropic material with the material axes defined by element topology (see also ELEM). The elasticity matrix is available per element.

Descriptor	Meaning
FILE	Material file name (OPTION=ELMAT only). See Remark 7. (Character; default=blank)
X1, Y1, Z1	Components of the vector V1 in the basic coordinate system. See Remark 7. (Real; default=0.0)
X2, Y2, Z2	Components of the vector V2 in the basic coordinate system. See Remark 7. (Real; default=0.0)

Remarks:

1. All the material constants are specified in the orthotropic material coordinates in 1, 2, and 3 direction.
2. In general, v_{12} is not the same as v_{21} , but they are related by $v_{ij}/E_i = v_{ji}/E_j$. Furthermore, material stability requires that

$$E_i > v_{ij}^2 E_j$$

$$\text{and } 1 - v_{12}v_{21} - v_{23}v_{32} - v_{31}v_{13} - 2v_{21}v_{32}v_{13} > 0.$$

3. It may be difficult to find all nine orthotropic constants. In some practical problems, the material properties may be reduced to normal anisotropy in which the material is isotropic in a plane, e.g., in plane 1-2 and has different properties in the direction normal to the plane 1-2. In the plane of isotropy, the properties are reduced to

$$E_1 = E_2 = E_p$$

$$v_{31} = v_{32} = v_{np}$$

$$v_{13} = v_{23} = v_{pn}$$

$$G_{13} = G_{23} = G_n$$

$$\text{with } v_{np}/E_n = v_{pn}/E_p \text{ and } G_p = \frac{E_p}{2(1 + v_p)}.$$

There are five independent material constants for normal anisotropy (e.g., E_p , E_n , v_p , v_{np} , G_n).

In case the material has a planar anisotropy, in which the material is orthotropic only in a plane, the elastic constants are reduced to seven, e.g., E_1 , E_2 , E_3 , v_{12} , G_{12} , G_{23} , G_{31} .

4. If $Y2$ and/or $Y3$ are blank, then $Y1$ is assumed if entered. If $Ysh2$ and/or $Ysh3$ are blank, $Ysh1$ is assumed if entered.
5. Do not enter values for SY , $Y1$, $Y2$, $Y3$, $YSHR1$, $YSHR2$ or $YSHR3$ unless plasticity is to be taken into account.

6. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.
7. OPTION, FILE, X1, Y1, Z1, X2, Y2 and Z2 fields are only supported by new SOL700.
8. New SOL700 does not support A1, A2, A3, TREF, GE, IYLD, IHARD, SY, Y1, Y2, Y3, Yshr1, Yshr2 and Yshr3 fields.

Notes:

1. The structural damping constant GE is specified as a numerical damping γ , i.e.,

$$C = \alpha M + \left(\beta + \frac{2\gamma}{\omega} \right) K$$

in which 2γ is equivalent to GE.

2. This Bulk Data entry accommodates Marc's input data under the model definition option ORTHOTROPIC.

MATPE1**Isotropic Poroelastic Material Property definition**

Defines the material properties for an isotropic poroelastic material.

Format:

1	2	3	4	5	6	7	8	9	10
MATPE1	MID	MAT1	MAT10	BIOT					
	VISC	GAMMA	PRANDTL	POR	TOR	AFR	VLE	TLE	

Example:

MATPE1	101	1	10						
	1.84-8	1.4	7.13-1	9.5-1	1.4	2.5-5	9.32-2	9.32-2	

Descriptor	Meaning
MID	Material identification number, referenced by PSOLID entry. (Integer>0)
MAT1	Identification number of MAT1 (and MATF1 if it is frequency-dependent) entry for the skeleton (solid-phase) of porous material. (Integer>0, Required)
MAT10	Identification number of MAT10 entry for the fluid-phase of porous material. (Integer>0, Required)
BIOT	Biot factor. (Real>0.0, Default=1.0)
VISC	Fluid dynamic viscosity. See Remark 4. (Real>0.0, Required)
GAMMA	Fluid ratio of specific heats. See Remark 5. (Real>0.0, Default=1.402)
PRANDTL	Fluid Prandtl number, Pr. See Remark 6. (Real>0.0, Default=0.71)
POR	Porosity of porous material. (Real>0.0, Required)
TOR	Tortuosity of porous material. (Real>0.0, Default=1.0)
AFR	Air flow resistivity. (Real>0.0, Required)
VLE	Viscous characteristic length. (Real>0.0, Required)
TLE	Thermal characteristic length. (Real>0.0, Required)

Remarks:

1. MATPE1 entry is referenced by PSOLID cards when FCTN=PORO. It is used to define the isotropic poroelastic material properties for both solid and fluid phases of a poroelastic medium.
2. MATPE1 entry must be put under BEGIN BULK TRMC.
3. For a frequency-dependent skeleton material, MATF1 specifies the material properties at frequencies on matching MAT1 fields.
4. Fluid dynamic viscosity, VISC, is temperature dependent. At 15°C (Celsius), its value is 1.82×10^{-6} N.s/m² in the SI units.

5. GAMMA is defined as C_p/C_v where C_p is the specific heat at constant pressure and C_v the specific heat at constant volume. Its default value is 1.402 for the air at ambient temperature.
6. PRANDTL number, Pr, is defined as $\mu C_p/k$ where μ is the fluid dynamic viscosity (VISC) and k the thermal conductivity. For the dry air at ambient temperature, its value is around 0.7-0.8.
7. Porosity, Ω , is defined as V_f/V_t where V_f is the volume of fluid and V_t is the total volume of the poroelastic material. Hence, porosity is dimensionless.
8. Tortuosity measures the complexity of the path an air particle must follow to process from one point to another point. It is a dimensionless parameter.
9. Resistivity measures the resistance of poroelastic material against the flow of fluid. In SI unit, it is measured in N.s/m⁴.
10. The density of poroelastic material is defined as $\rho_{pem} = \Omega\rho_f + (1 - \Omega)\rho_s$ where ρ_f is the fluid density from MAT10 and ρ_s is the solid density from MAT1.
11. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.

MATTRIG**Rigid Material**

Defines the properties of a rigid material in SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
MATTRIG	MID	RHO	E	NU	MASS	XC	YC	ZC	
	IXX	IXY	IXZ	IYY	IYZ	IZZ	CID		
	VX	VY	VZ	WX	WY	WZ			
	XC-LOCAL	YC-LOCAL	ZC-LOCAL						

Example:

MATTRIG	7	7850.	210.E9	0.3	750	0.0	7.0	-3.0	
	17.0	13.2	14.3	20.9	15.7	10.0	12		
			13.3						

Descriptor	Meaning	Type	Default
MID	Unique material number.	I > 0	Required
RHO	Density	R > 0	1.0
E	Young's modulus	R > 0	1.0
NU	Poisson's ratio	0.0 R < 0.5	0.2
MASS	Mass of the rigid body.	R > 0.0	See Remark 2.
XC, YC, ZC	x, y, and z coordinates of the center of gravity.	R	See Remark 6.
IXX, IXY, IXZ, IYY, IYZ, IZZ	Inertia tensor of the rigid body about the center of gravity.	R	See Remark 6.
CID	Number of a coordinate system in which the inertia tensor and the center of gravity are defined.	I > 0	See Remarks 7. and 8.
VX, VY, VZ	Initial translational velocity of the center of gravity in the basic coordinate system.	R	0.0
WX, WY, WZ	Initial rotational velocities of the rigid body about the center of gravity in the basic coordinate system.	R	0.0
XC-LOCAL YC-LOCAL ZC-LOCAL	x, y, and z local coordinates of the center of gravity	R	See Remark 8.

Remarks:

1. All coordinates are defined in the basic coordinate system.

2. If MASS is blank or zero, the mass will be calculated from the density and the geometry of the mesh defining the rigid body.
3. The continuation lines are not required.
4. The MATRIG definition is used instead of a material definition and is referenced by properties PSOLIDn, PSHELLn, PBAR, and PBEAMn. Different properties can refer to the same MATRIG entry forming one rigid body.
5. If the fields VX, VY, VZ, WX, WY, and WZ are blank, then the initial conditions of the rigid body are calculated from the initial velocities on the TIC and TIC1 entries referring to grid points attached to the rigid body. The net initial conditions are the average of those for all the grid points attached to the rigid body.
If the initial conditions are set using the VX, VY, VZ, WX, WY, and WZ fields, the TIC and TIC1 entries referring to grid points attached to the rigid body are ignored.
6. If the inertia tensor or the coordinates of the center of gravity are undefined, then they will be computed from the density or mass and the geometry of the mesh defining the rigid body.
7. The inertia tensor can only be defined in a local rectangular coordinate system. If the entry for a local coordinate system is left blank, then the inertia tensor is defined in the global coordinate system.
8. The center of gravity can be defined in a local rectangular coordinate system (CID). However, XC YC ZC (x, y, and z coordinates of the center of gravity in the basic coordinate system) should be left blank when XC-LOCAL YC-LOCAL ZC-LOCAL (x, y, and z coordinates of the center of gravity in a local coordinate system) defined.

MATS1**Material Stress Dependence**

Specifies stress-dependent material properties for use in applications involving nonlinear materials. This entry is used if a MAT1 entry is specified with the same MID in a nonlinear solution sequence (SOLs 106, 129 and 400). For SOL 400 the MATEP option is the recommended approach for elastic-plastic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MATS1	MID	TID	TYPE	H	YF	HR	LIMIT1	LIMIT2	

Example:

MATS1	17	28	PLASTIC	0.0	1	1	2.+4		
-------	----	----	---------	-----	---	---	------	--	--

Descriptor	Meaning
MID	Identification number of a MAT1 entry. (Integer > 0)
TID	Identification number of a TABLES1 or TABLEST entry. If H is given, then this field must be blank. See Remark 3. (Integer ≥ 0 or blank)
TYPE	Type of material nonlinearity. See Remarks. (Character: "NLELAST" for nonlinear elastic or "PLASTIC" for elastoplastic.)
H	Work hardening slope (slope of stress versus plastic strain) in units of stress. For elastic-perfectly plastic cases, H = 0.0. For more than a single slope in the plastic range, the stress-strain data must be supplied on a TABLES1 entry referenced by TID, and this field must be blank. See Remark 2. (Real)
YF	Yield function criterion, selected by one of the following values (Integer): 1 von Mises (Default) 2 Tresca 3 Mohr-Coulomb 4 Drucker-Prager
HR	Hardening Rule, selected by one of the following values (Integer): 1 Isotropic (Default) 2 Kinematic 3 Combined isotropic and kinematic hardening
LIMIT1	Initial yield point. See Table 9-25 . (Real)
LIMIT2	Internal friction angle, measured in degrees, for the Mohr-Coulomb and Drucker-Prager yield criterion. See Table 9-25 . (0.0 \leq Real $<$ 45.0)

Table 9-25 Yield Functions Versus LIMIT1 and LIMIT2

Yield Function (YF)	LIMIT1	LIMIT2
von Mises (1) or Tresca (2)	Initial Yield Stress in Tension, Y_1	Not used
Mohr-Coulomb (3) or Drucker-Prager (4)	$2^* \text{Cohesion}, 2c$ (in units of stress)	Angle of Internal Friction ϕ (in degrees)

Remarks:

1. If TYPE = "NLELAST", then MID may refer to a MAT1 entry only. Also, the stress-strain data given in the TABLES1 entry will be used to determine the stress for a given value of strain. The values H, YF, HR, LIMIT1, and LIMIT2 will not be used in this case.

Thermoelastic analysis with temperature-dependent material properties is available for linear and nonlinear elastic isotropic materials (TYPE = "NLELAST") and linear elastic anisotropic materials. Four options of constitutive relations exist. The relations appear in [Table 23](#) along with the required Bulk Data entries.

Table 23 Constitutive Relations and Required Material Property Entries

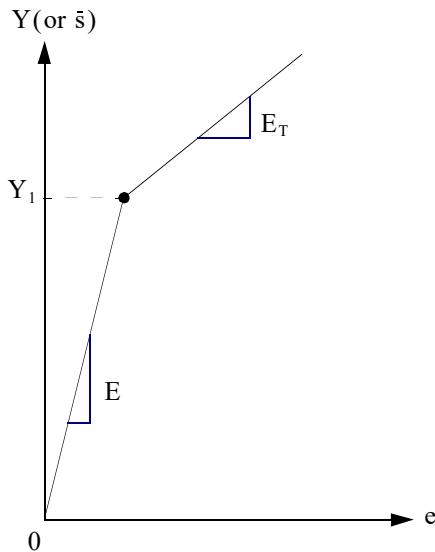
Constitutive Relation	Required Bulk Data Entries
$\{\sigma\} = [G_e(T)]\{\varepsilon\}$	MAT1 and MATT1
$\{\sigma\} = \frac{\bar{E}(\sigma, \varepsilon)}{E}[G_e(T)]\{\varepsilon\}$	MAT1, MATT1, MATS1, and TABLES1
$\{\sigma\} = \frac{\bar{E}(T, \sigma, \varepsilon)}{E}[G_e]\{\varepsilon\}$	MAT1, MATS1, TABLEST, and TABLES1
$\{\sigma\} = \frac{\bar{E}(T, \sigma, \varepsilon)}{E}[G_e(T)]\{\varepsilon\}$	MAT1, MATT1, MATS1, TABLEST, and TABLES1

In [Table 23](#) $\{\sigma\}$ and $\{\varepsilon\}$ are the stress and strain vectors, $[G_e]$ the elasticity matrix, \bar{E} the effective elasticity modulus, and E the reference elasticity modulus.

2. If TYPE = "PLASTIC", the elastic stress-strain matrix is computed from MAT1 entry, and then the isotropic plasticity theory is used to perform the plastic analysis. In this case, either the table identification TID or the work hardening slope H may be specified, but not both. If the TID is omitted, the work hardening slope H must be specified unless the material is perfectly plastic. The plasticity modulus (H) is related to the tangential modulus (E_T) by

$$H = \frac{E_T}{1 - \frac{E_T}{E}}$$

where E is the elastic modulus and $E_T = dY/d\varepsilon$ is the slope of the uniaxial stress-strain curve in the plastic region. See [Figure 9-111](#).



[Figure 9-111 Stress-Strain Curve Definition When H Is Specified in Field 5](#)

3. If TID is given, TABLES1 entries (X_i, Y_i) of stress-strain data (ε_k, Y_k) must conform to the following rules (see [Figure 9-112](#)):
 - If TYPE = "PLASTIC", the stress-total strain curve (i.e., TYPE in TABLES1 must be 1) must be defined in the first quadrant. The first point must be at the origin $(X_1 = 0, Y_1 = 0)$ and the second point (X_2, Y_2) must be at the initial yield point $(Y_1$ or $2c)$ specified on the MATS1 entry. The slope of the line joining the origin to the yield stress must be equal to the value of E . Also, TID may not reference a TABLEST entry.

The stress-total strain curve has 2 different definitions

- For conventional element, small strain formulation in terms of engineering strain and stress is used. Therefore, engineering stress-strain curve should be used here.
- For advanced nonlinear element, large strain formulation in terms of logarithmic strain and true stress is adopted. Then true stress-logarithmic strain curve should be used.

Note that SOL 106 and SOL 129 supports conventional element only. SOL 400 supports both of the conventional and advanced elements.

For TYPE = "PLASTIC", it should be noted that for conventional elements, strains should be below .05 to small strain formulation. Within this strain range, difference between logarithmic strain and engineering strain may be negligible. If the element strains exceed .05, then it is recommended to use the MATEP entry which will automatically switch to a large strain formulation in SOL 400

For SOL 600, if use of the MATS1 entry is desired, then it should be used in conjunction with PARAM,MRTABLS1,value.

- If TYPE = "NLELAST", the full stress-strain curve ($-\infty < x < \infty$) may be defined in the first and the third quadrant to accommodate different uniaxial compression data. If the curve is defined only in the first quadrant, then the curve must start at the origin (X1 = 0.0, Y = 0.0) and the compression properties will be assumed identical to tension properties.

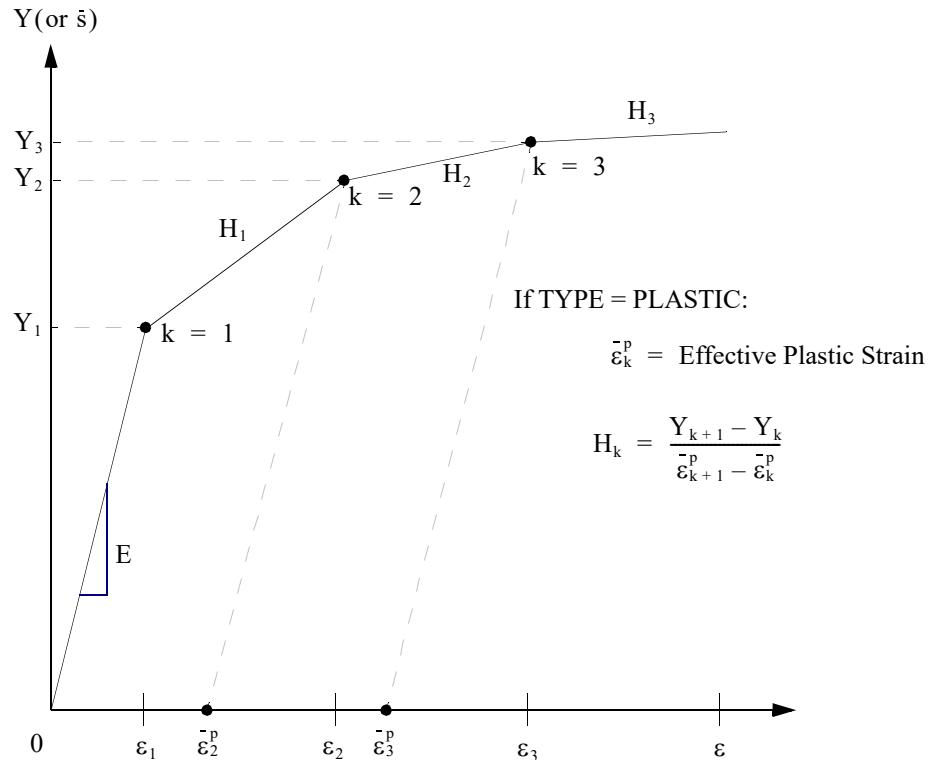


Figure 9-112 Stress-Strain Curve Definition When TID Is Specified in Field 3

4. In SOL 400, MATS1 is supported for 2 sets of elements:

- a. Elements with no property extensions that are prescribed through PROD, PBEAML, PSHELL and PSOLID. The original small-strain element formulations that are also available in SOL106 are used in these cases.
- b. Large-strain elements that are supported through property extensions. For such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Also, for TYPE = PLASTIC, linear 2D triangular elements specified on PSHLN2 and 3D tetrahedral elements specified on PSLDN1 should be associated with an incompressible formulation (IPS for 2D plane strain, IAX for 2D axisymmetric and ISOL for 3D tetrahedral. Note also that, prior to MD Nastran 2010, if the property extensions were missing, then for supported elements (i.e., PROD, PBEAML, PSHELL or PSOLID), the analysis would use the element formulations in (a), or ignore the MATS1 data for unsupported elements. From MD Nastran 2010 onwards, if the property extensions are missing, then they are automatically added with appropriate settings by the program unless the element belongs to group (a) or the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.