

QVOL**Volume Heat Addition**

Defines a rate of volumetric heat addition in a conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
QVOL	SID	QVOL	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

Example:

QVOL	5	10.0	101	10	12	11	9		
------	---	------	-----	----	----	----	---	--	--

Descriptor	Meaning
SID	Load set identification. (Integer > 0)
QVOL	Power input per unit volume produced by a heat conduction element. (Real)
CNTRLND	Control point used for controlling heat generation. (Integer ≥ 0 ; Default = 0)
EIDi	A list of heat conduction elements. (Integer > 0 or “THRU” or “BY”)

Remarks:

1. EIDi has material properties (MAT4) that include HGEN, the element material property for heat generation, which may be temperature dependent. This association is made through the element EID. If HGEN is temperature dependent, it is based on the average element temperature.
2. QVOL provides either the constant volumetric heat generation rate or the load multiplier. QVOL is positive for heat generation. For steady-state analysis, the total power into an element is
 - If CNTRLND = 0, then $P_{in} = \text{volume} \cdot \text{HGEN} \cdot \text{QVOL}$.
 - If CNTRLND > 0, then $P_{in} = \text{volume} \cdot \text{HGEN} \cdot \text{QVOL} \cdot u_{\text{CNTRLND}}$.
where u_{CNTRLND} is the temperature multiplier.
3. For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD = SID).
4. In transient analysis SID is referenced by a TLOADi Bulk Data entry. A function of time $F[t - \tau]$ defined on the TLOADi entry multiplies the general load where τ specifies time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
5. For “THRU” or “BY”, all intermediate referenced heat conduction elements must exist.
6. The CNTRLND multiplier cannot be used with any higher-order elements.
7. For RC network solver in thermal analysis, CNTRLND can only be used as the ID in CONTROLT as thermostats controller.

8. QVOL is not supported for CQUADX and CTRIAX axisymmetric elements. It is supported for the CTRIA6 element.

RADBC

Space Radiation Specification

Specifies an CHBDYi element face for application of radiation boundary conditions.

Format:

1	2	3	4	5	6	7	8	9	10
RADBC	NODAMB	FAMB	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	EID7	-etc.-						

Example:

RADBC	5	1.0	101	10					
-------	---	-----	-----	----	--	--	--	--	--

Descriptor	Meaning
NODAMB	Ambient point for radiation exchange. (Integer > 0)
FAMB	Radiation view factor between the face and the ambient point. (Real ≥ 0.0)
CNTRLND	Control point for radiation boundary condition. (Integer ≥ 0 ; Default = 0)
EIDi	CHBDYi element identification number. (Integer > 0) Key words "THRU" and "BY" can be used to assist the listing with ascending order of EIDi

Remarks:

1. The basic exchange relationship is:
 - if CNTRLND = 0, then $q = \sigma \cdot FAMB \cdot \epsilon_e \cdot (T_e^4 - T_{amb}^4)$
 - if CNTRLND > 0, then

$$q = \sigma \cdot FAMB \cdot u_{CNTRLND} \cdot \epsilon_e \cdot (T_e^4 - T_{amb}^4)$$
2. NODAMB is treated as a black body with its own ambient temperature for radiation exchange between the surface element and space. No surface element that is a member of a radiation enclosure cavity may also have a radiation boundary condition applied to it.
3. Two PARAM entries are required when stipulating radiation heat transfer:
 - ABS defines the absolute temperature scale; this value is added internally to any specified temperature given in the problem. Upon solution completion, this value is subtracted internally from the solution vector.
 - SIGMA (σ) is the Stefan-Boltzmann constant.
4. RADBC allows for surface radiation to space in the absence of any cavity behavior. The emissivity is supplied from a RADM entry.
5. When using "THRU" or "BY", all intermediate referenced CHBDYi surface elements must exist.

RADBND**Radiation Wavelength Band Definition**

Specifies Planck's second radiation constant and the wavelength breakpoints used for radiation exchange problems.

Format:

1	2	3	4	5	6	7	8	9	10
RADBND	NUMBER	PLANCK2	LAMBDA1	LAMBDA2	LAMBDA3	LAMBDA4	LAMBDA5	LAMBDA6	
	LAMBDA7	etc.							

Example:

RADBND	6	14388.0	1.0	2.0	4.0	8.0	12.0		
--------	---	---------	-----	-----	-----	-----	------	--	--

Descriptor	Meaning
NUMBER	Number of radiation wave bands. See Remarks. (Integer > 1)
PLANCK2	Planck's second radiation constant. See Remarks. (Real > 0.0)
LAMBDAi	Highest wavelength of the i-th wave band. See Remarks. (Real \geq 0.0)

Remarks:

- Only one RADBND entry may be specified in the Bulk Data Section and must always be used in conjunction with the RADM entry.
- PLANCK2 has the units of wavelength times temperature. The same units of length must be used for LAMBDAi as for PLANCK2. The units of temperature must be the same as those used for the radiating surfaces. For example: $25898.\mu\text{m } ^\circ\text{R}$ or $14388.\mu\text{m } ^\circ\text{K}$.
- The first wavelength band extends from 0 to LAMBDA1 and the last band extends from LAMBDA n to infinity, where $n = \text{NUMBER} - 1$.
- Discontinuous segments in the emissivity versus wavelength piecewise linear curve must be treated as a wavelength band of zero width.
- LAMBDAi must be specified in ascending order, and all LAMBDAi fields where i is greater than or equal to NUMBER must be blank.
- If Modules are present then this entry may only be specified in the main Bulk Data section.

RADC

Thermal Radiative Coating Properties - SOL400 - RC Network solver

Defines the radiative properties of advanced materials such as coatings and multilayer insulation, commonly used in the aerospace market.

Format: (COAT)

1	2	3	4	5	6	7	8	9	10
RADC	RADMID	Emis	Absorptivity	IR Spec	UV Spec				
	"COAT"	IR TraspA	IR Transluc	UV TranspA	UV Transluc	IR Refrac Ind	UV Refrac Ind		

Example:

RADC	101	1.	1.	0.					
	COAT	0.	0.	0.	0.	1.	1.		

Format: (MLI)

1	2	3	4	5	6	7	8	9	10
RADC	RADMID	Emis	Absorptivity	IR Spec	UV Spec				
	"MLI"	Estar							

Example:

RADC	102	1.	1.	0.					
	MLI	0.02							

Descriptor	Meaning
RADMID	Radiation material identification number (Integer > 0; Required)
COAT	Identification that a coating type material has been started (Character)
Emis	Emissivity (Real \geq 0.0; Default 1.0)
Absorptivity	Absorptivity (Real \geq 0.0; Default 1.0)
IR Spec	IR Specularity (0.0 \leq Real \leq 1.0; Default 0.0)
UV Spec	UV Specularity (0.0 \leq Real \leq 1.0; Default 0.0)
IR TranspA	IR Transparency (0.0 \leq Real \leq 1.0; Default 0.0)
IR Transluc	IR Translucency (0.0 \leq Real \leq 1.0; Default 0.0)
UV TranspA	UV Transparency (0.0 \leq Real \leq 1.0; Default 0.0)
UV Transluc	UV Translucency (0.0 \leq Real \leq 1.0; Default 0.0)
IR Refrac Ind	IR Refraction Index (Real \geq 1.0; Default 1.0)
UV Refrac Ind	UV Refraction Index (Real \geq 1.0; Default 1.0)
MLI	Identification that an MLI type material has been started (Character)
Estar	MLI E-star parameter (Real; Default 0.0)

Remarks:

1. This entry is for RC Network solver only.
2. Estar is defined as the effective emissivity from the wall to MLI outer surface. The general value is around 0.01 to 0.03, the typical value is 0.02.
3. Emis is usually for the IR waveband, and Absorptivity is for the UV waveband.

RADCAV**Radiation Cavity Identification**

Identifies the characteristics of each radiant enclosure.

Format:

1	2	3	4	5	6	7	8	9	10
RADCAV	ICAVITY	ELEAMB	SHADOW	SCALE	PRTPCH	NFECI	RMAX	NCOMP	
	SET11	SET12	SET21	SET22	SET31	SET32	etc.		

Example:

RADCAV	1	1					.99		
	3	5	4	5	7	5			

Descriptor	Meaning
ICAVITY	Unique cavity identification number associated with enclosure radiation. (Integer > 0)
ELEAMB	CHBDYi surface element identification number for radiation if the view factors add up to less than 1.0. (Unique Integer > 0 among all CHBDYi elements or blank.)
SHADOW	Flag to control third body shading calculation during view factor calculation for each identified cavity. (Character = "YES" or "NO"; Default = "YES")
SCALE	View factor that the enclosure sum will be set to if a view factor is greater than 1.0. (0.0 ≤ Real ≤ 1.0; Default = 0.0)
PRTPCH	Facilitates the blocking of view factor printing and punching onto RADLST and RADMTX entries. (Integer = 0, 1, 2, 3, 4, or 5; Default = blank):

Hemi-Cube & VIEW3D		
Value	Printout in .f06 file	Printout in .pch file
Blank	No	Yes
0	Full Print	Yes
1	No	Yes
2	Full Print	No
3	No	No
4	Summary Print	Yes
5	Summary Print	No

NFECI	Controls whether finite difference or contour integration methods are to be used in the calculation of view factors in the absence of a VIEW3D Bulk Data entry. (Character = "FD" or "CONT"; See Remark 4. for default.)
RMAX	Subelement area factor. See Remark 5. (Real ≥ 0.0 ; Default = 0.1)
NCOMP	Total number of computational element for one-half ring. See Remark 8. (Default = 32)
SETij	Set identification pairs for the calculation of global view factors. Up to 30 pairs may be specified (i = 1 to 2 and j = 1 to 30). (Integer > 0)

Remarks:

1. For the surfaces of an incomplete enclosure (view factors add up to less than 1.0), a complete enclosure may be achieved ($\text{SUM} = 1.0$) by specifying an ambient element, ELEAMB. When multiple cavities are defined, each cavity must have a unique ambient element if ambient elements are desired. No elements can be shared between cavities.
2. Third-body shadowing is ignored in the cavity if SHADOW = "NO". In particular, if it is known a priori that there is no third-body shadowing, SHADOW = NO overrides KSHD and KBSD fields on the VIEW Bulk Data entry as well as reduces the calculation time immensely.
3. The view factors for a complete enclosure may add up to slightly more than 1.0 due to calculation inaccuracies. SCALE can be used to adjust all the view factors proportionately to acquire a summation equal to the value specified for SCALE. If SCALE is left blank or set to 0.0, no scaling is performed.
4. If the VIEW3D Bulk Data entry is not specified, the view factors are calculated using finite difference and contour integration methods. If NFECI = "FD", then all view factors are calculated using the finite difference technique. NFECI = "CONT" invokes contour integration for all view factor calculations. If NFECI is blank, the program selects a method to use between any two particular elements based on RMAX.
5. The comparison value for RMAX is equal to A_s/d_{rs}^2 where A_s is the area of a subelement and d_{rs} is the distance between two subelements r and s for which view factors are being computed. When NFECI is blank, the program selects the contour integral method only if $A_s/d_{rs}^2 > \text{RMAX}$.
6. When a number of elements are grouped together and considered as a conglomerate surface, view factors can be calculated between these groups. These are referred to as global view factors. The SET1 Bulk Data entry is used to define the conglomerate. When using this feature, negative EIDs are not allowed.
7. If a RADLST and RADMTX entry exists for this cavity ID, new view factors are not computed and the existing RADLST and RADMTX are used in the thermal analysis.
8. The VIEW3D Bulk Data entry must be specified for the calculation of axisymmetric view factors. The process relies on the internal construction of a semi-circle of computational elements. NCOMP specifies the number of such elements desired.
9. For SOL 600, fields 4-8 of the primary entry and all continuation lines are ignored.

10. For SOL 600 field 9 of the primary entry (NCOMP) is used to indicate if the cavity is open or closed according to the following designation:
 - 0 Cavity is closed
 - 1 Cavity is closed and view factors are scaled such that they sum to exactly 1.0
 - 2 Cavity is open
11. For RC network solver in thermal analysis, RADCAV is only used to specify radiation space node, the ELEAMB must be a POINT type CHBDYP element. The SHADOW, SCALE, PRTPCH, NFECI, RMAX, NCOMP, SET11, SET12, SET21, SET22, SET31 and SET32 are ignored.

RADCOL

Radiation Collective Entity - SOL400 - RC Network solver

Specifies a collection of boundary elements to be used as a single face in the radiation calculation. This will decrease computation time at the small cost of accuracy. Computational savings and accuracy are dependent on the coarseness of the collection versus the constituents. View factors of the collection are redistributed across the elements for calculation of the radiative energy transfer.

Format:

1	2	3	4	5	6	7	8	9	10
RADCOL	RADCOLID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			

Example:

RADCOL	101	5	6	2	3	7			
--------	-----	---	---	---	---	---	--	--	--

Descriptor	Meaning
RADCOLID	Radiation Collection identification number. (Integer > 0; Required)
IVIEWF	A VIEW entry identification number for the front face. (Integer ≥ 0 ; Default 0)
IVIEWB	A VIEW entry identification number for the back face. (Integer ≥ 0 ; Default 0)
RADMIDF	RADM identification number for the front face. (Integer ≥ 0 ; Default 0)
RADMIDB	RADM identification number for the back face. (Integer ≥ 0 ; Default 0)
SET3 idn	ID of the element collection to be considered a super element. (Integer > 0)

Remarks:

1. This entry is for RC Network solver only.
2. IVIEWF/IVIEWB will default to 0 if left blank. This would indicate that the corresponding front and/or back do not participate in the radiation.
3. If an IVIEWF is specified, there must also be a RADMIDF for surface material properties. If a IVIEWB is specified, there must also be a RADMIDB for surface material properties.

RADCT Thermal Radiative Functionally-Dependent Properties - SOL400 - RC Network solver

Specifies table references for RADCT properties that are functionally dependent.

Format:

1	2	3	4	5	6	7	8	9	10
RADCT	RADMID	Emis f(T)	Abs f(T)						

Example:

RADCT	11	101	102						
-------	----	-----	-----	--	--	--	--	--	--

Descriptor	Meaning
RADMID	Radiation material identification number (Integer > 0; Required)
Emis f(T)	Emissivity TABLEMj table ID (Integer ≥ 0 ; Default.0)
Absorptivity f(T)	Absorptivity TABLEDj table ID (Integer ≥ 0 ; Default 0)

Remarks:

1. This entry is for RC Network solver only.
2. RADCT always comes with the RADCT entry. They have the same RADMID so that they can be referenced together.
3. Currently, RC Network solver does not support this entry for enclosure radiation.

RADLST**Listing of Enclosure Radiation Faces**

Identifies the individual CHBDYi surface elements that comprise the entire radiation enclosure.

Format:

1	2	3	4	5	6	7	8	9	10
RADLST	ICAVITY	MTXTYP	EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	-etc.-							

Example:

RADLST	2	1	2	3	4	5	6	7	
--------	---	---	---	---	---	---	---	---	--

Descriptor	Meaning
ICAVITY	Unique cavity identification number that links a radiation exchange matrix with its listing of enclosure radiation faces. (Integer > 0)
MTXTYP	Type of radiation exchange matrix used for this cavity. (Integer ≤ 4 and $\neq 0$; Default = 1 for an enclosure without an ambient element. Default = 4 for an enclosure with an ambient element as specified on the RADCAV entry.)
	1 Symmetric view factor matrix [F] and nonconservative radiation matrix [R]. 2 Symmetric exchange factor matrix [\mathfrak{J}] and conservative radiation matrix [R]. 3 Unsymmetric exchange factor matrix [\mathfrak{J}] and conservative radiation matrix [R]. 4 Symmetric view factor matrix [F] and conservative radiation matrix [R]. -n The first n CHBDYi elements may lose energy to space but the remainder may not. Symmetric exchange factor matrix [F] and nonconservative radiation matrix [R].
EIDI	Identification numbers for the CHBDYi elements in this radiation cavity. (Integer $\neq 0$ or "THRU")

Remarks:

1. A radiation EIDI list isolates those CHBDYi surface element faces that are to communicate in a radiation enclosure. View-factor calculation and RADMTX formation for an enclosure is performed only for (or among) those faces identified within the same RADCAV.
2. A radiation exchange matrix (RADMTX) can only reference one radiative face list (RADLST). The companion RADCAV, RADLST, and RADMTX must share a unique ICAVITY.
3. For each EIDI, the appropriate CHBDYi element is located, and the proper RADM entry ID field found.
4. If the radiation exchange matrix or any radiation boundary conditions are available from an external source, the RADMTX must be user generated.

5. Multiple RADLST entries may be specified.
6. If any RADLST entry is changed or added on restart then a complete re-analysis may be performed. Therefore, RADLST entry changes or additions are not recommended on restart.
7. RC network solver does not support RADLST for thermal analysis.

RADM**Radiation Boundary Material Property**

Defines the radiation properties of a boundary element for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RADM	RADMID	ABSORP	EMIS1	EMIS2	EMIS3	EMIS4	EMIS5	EMIS6	
	EMIS7	-etc.-							

Example:

RADM	11		.45	.33	.29	.20	.17	.13	
------	----	--	-----	-----	-----	-----	-----	-----	--

Descriptor	Meaning
RADMID	Material identification number. (Integer > 0)
ABSORP	Surface absorptivity or the temperature function curve multiplier if ABSORP is variable. See Remark 2. ($0.0 \leq \text{Real} \leq 1.0$)
EMISi	Surface emissivity at wavelength LAMBDAi or the temperature function curve multiplier if EMISi is variable (See the RADBND entry.) ($0.0 \leq \text{Real} \leq 1.0$)

Remarks:

1. The RADM entry is directly referenced only by one of the CHBDYE, CHBDYG, or CHBDYP type surface element entries.
2. For radiation enclosure problems, ABSORP is set equal to emissivity. For QVECT loads, absorptivity is specified by ABSORP.
3. If there is more than one EMISi, then:
 - There must be a RADBND entry.
 - The number of EMISi may not exceed the number of LAMBDAi on the RADBND entry.
 - The emissivity values are given for a wavelength specified by the corresponding LAMBDAi on the RADBND entry. Within each discrete wavelength band, the emissivity is assumed to be constant.
 - At any specific wavelength and surface temperature, the absorptivity is exactly equal to the emissivity.
4. To perform any radiation heat transfer exchange, the user must furnish PARAM entries for:
 - TABS to define the absolute temperature scale.
 - SIGMA (σ) to define the Stefan-Boltzmann constant in appropriate units.
5. RC network solver only supports RADMID, ABSORP and EMIS1 for thermal analysis, other fields are ignored.

RADMT**Radiation Boundary Material Property Temperature Dependence**

Specifies table references for temperature dependent RADM entry radiation boundary properties.

Format:

1	2	3	4	5	6	7	8	9	10
RADMT	RADMID	T(A)	T(ε_1)	T(ε_2)	T(ε_3)	T(ε_4)	T(ε_5)	T(ε_6)	
		T(ε_7)	-etc.-						

Example:

RADMT	11		1	2	3	4	5	6	
-------	----	--	---	---	---	---	---	---	--

Descriptor	Meaning
RADMID	Material identification number. (Integer > 0)
T(A)	TABLEMj identifier for surface absorptivity. (Integer ≥ 0 or blank)
T(ε_i)	TABLEMj identifiers for surface emissivity. (Integer ≥ 0 or blank)

Remarks:

1. The basic quantities on the RADM entry of the same RADMID are always multiplied by the corresponding tabular function.
2. Tables T(A) and T(ε_i) have an upper bound that is less than or equal to one and a lower bound that is greater than or equal to zero.
3. The TABLEMj enforces the element temperature as the independent variable. Blank or zero fields means there is no temperature dependence of the referenced property on the RADM entry.
4. RC network solver only supports RADMID, T(A) and T(ε_1) for thermal analysis, other fields are ignored.

RADMTX**Radiation Exchange Matrix**

Provides the $F_{ji} = A_j f_{ji}$ exchange factors for all the faces of a radiation enclosure specified in the corresponding RADLST entry.

Format:

1	2	3	4	5	6	7	8	9	10
RADMTX	ICAVITY	INDEX	Fi,j	Fi+1,j	Fi+2,j	Fi+3,j	Fi+4,j	Fi+5,j	
	Fi+6,j	-etc.-							

Example:

RADMTX	2	1	0.0	0.1	0.2	0.2	0.3	0.2	
--------	---	---	-----	-----	-----	-----	-----	-----	--

Descriptor	Meaning
ICAVITY	Unique cavity identification number that links a radiation exchange matrix with its listing of enclosure radiation surface elements. (Integer > 0)
INDEX	Column number in the matrix. (Integer > 0)
F _{k,j}	If symmetric, the matrix values start on the diagonal (i = j) and continue down the column (k = i + 1, i + 2, etc.). If unsymmetric, the values start in row (i = 1). i refers to EIDi on the RADLST entry. (Real ≥ 0)

Remarks:

1. If the matrix is symmetric, only the lower triangle is input, and i = j = INDEX. If the matrix is unsymmetric, i = 1, and j = INDEX.
2. Only one ICAVITY may be referenced for those faces that are to be included in a unique radiation matrix.
3. Coefficients are listed by column with the number of columns equal to the number of entries in the RADLST.
4. All faces involved in any radiation enclosure must be defined with an CHBDYi element.
5. If any RADMTX entry is changed or added on restart then a complete re-analysis may be performed. Therefore, RADMTX entry changes or additions are not recommended on restart.
6. Set NASTRAN SYSTEM (87) = 3 is a new option in MSC Nastran 2005 that prevents radiation energy from being lost to space.
7. RC network solver does not support RADMTX for thermal analysis.

RADSET

Identifies a Set of Radiation Cavities

Specifies which radiation cavities are to be included for radiation enclosure analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RADSET	ICAVITY1	ICAVITY2	ICAVITY3	ICAVITY4	ICAVITY5	ICAVITY6	ICAVITY7	ICAVITY8	
	ICAVITY9	-etc.-							

Example:

RADSET	1	2	3	4					
--------	---	---	---	---	--	--	--	--	--

Descriptor	Meaning
ICAVITYi	Unique identification number for a cavity to be considered for enclosure radiation analysis. (Integer > 0)

Remark:

- For multiple radiation cavities, RADSET specifies which cavities are to be included in the analysis.

RANDPS**Power Spectral Density Specification**

Defines load set power spectral density factors for use in random analysis having the frequency dependent form

$$S_{jk}(F) = (X + iY)G(F)$$

Format:

1	2	3	4	5	6	7	8	9	10
RANDPS	SID	J	K	X	Y	TID			

Example:

RANDPS	5	3	7	2.0	2.5	4			
--------	---	---	---	-----	-----	---	--	--	--

Descriptor	Meaning
SID	Random analysis set identification number. See Remarks 1 and 9. (Integer > 0)
J	Subcase identification number of the excited load set. See Remarks 6 through 9. (Integer > 0)
K	Subcase identification number of the applied load set. See Remarks 6 through 9. (Integer ≥ 0 ; K $\geq J$)
X, Y	Components of the complex number. See Remarks 2 and 7. (Real)
TID	Identification number of a TABRNDi entry that defines G(F). See Remarks 3 and 7. (Integer > 0 or Blank)

Remarks:

- Set identification numbers must be selected with the Case Control command (RANDOM = SID) or through use of FTGLOAD when doing random vibration fatigue analysis.
- For auto spectral density, J = K, X must be greater than zero and Y must be equal to zero.
- For TID = Blank, G(F) = 1.0.
- RANDPS may only reference subcases included within a single loop (a change in direct matrix input is not allowed).
- See the [MSC Nastran Dynamic Analysis User's Guide](#) for a discussion of random analysis.
- In the case of cyclic symmetry Solution Sequence 118, J and K must refer to the coded subcase IDs.
- In superelement analysis, J and K must refer to the superelement subcases. For example, if superelement 10 has SUBCASEs 1 and 2 and superelement 20 has SUBCASEs 3 and 4, then a separate RANDPS entry is required for each superelement, even though X, Y, and TID may be identical.
- For uncoupled PSDF (no J < K entries) only one J = K entry is allowed for unique value of J. For coupled PSDF (some J < K entries) any number of entries are allowed.

9. J and K must reference valid subcase IDs. In superelement analysis, J and K must reference valid subcase IDs that pertain to the applicable superelement. If this requirement is not met, the program issues an appropriate user warning message and ignores the associated RANDOM=SID command.
10. If Modules are present then this entry may only be specified in the main Bulk Data section.

RANDT1**Autocorrelation Function Time Lag**

Defines time lag constants for use in random analysis autocorrelation function calculation.

Format:

1	2	3	4	5	6	7	8	9	10
RANDT1	SID	N	T0	TMAX					

Example:

RANDT1	5	10	3.2	9.6					
--------	---	----	-----	-----	--	--	--	--	--

Descriptor	Meaning
SID	Random analysis set identification number. (Integer > 0)
N	Number of time lag intervals. (Integer > 0)
T0	Starting time lag. (Real ≥ 0.0)
TMAX	Maximum time lag. (Real $> T0$)

Remarks:

1. Time lag sets must be selected with the Case Control command RANDOM = SID.
 2. At least one RANDPS entry must be present with the same set identification number.
 3. The time lags defined on this entry are given by
- $$T_i = T0 + \frac{TMAX - T0}{N}(i - 1), i = 1, N + 2$$
4. See the [MSC Nastran Dynamic Analysis User's Guide](#) for a discussion of random analysis.
 5. If Modules are present then this entry may only be specified in the main Bulk Data section.

RBAR**Rigid Bar**

Defines a rigid bar with six degrees-of-freedom at each end.

Format:

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CNA	CNB	CMA	CMB	ALPHA	

Example:

RBAR	5	1	2	123456				6.5-6	
------	---	---	---	--------	--	--	--	-------	--

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)
GA, GB	Grid point identification number of connection points. ($\text{Integer} > 0$)
CNA, CNB	Component numbers of independent degrees-of-freedom in the global coordinate system for the element at grid points GA and GB. See Remark 3. (Integers 1 through 6 with no embedded blanks, or zero or blank.)
CMA, CMB	Component numbers of dependent degrees-of-freedom in the global coordinate system assigned by the element at grid points GA and GB. See Remarks 4. and 5. (Integers 1 through 6 with no embedded blanks, or zero or blank.)
ALPHA	Thermal expansion coefficient. See Remark 11. (Real or blank)

Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create the Lagrange multiplier degrees-of-freedom internally in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.
- For the linear method, the total number of components in CNA and CNB must equal six; for example, CNA = 1236, CNB = 34. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. For the Lagrange method, the total number of components must also be six. However, only CNA = 123456 or CNB = 123456 is allowed. If both CNA and CNB are blank, then CNA = 123456. For this method, RBAR1 gives the simpler input format.
- If both CMA and CMB are zero or blank, all of the degrees-of-freedom not in CNA and CNB will be made dependent. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be member of the m-set, depending on the method selected in the RIGID Case Control command. However, the rules regarding the m-set described below apply to both methods.

5. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
6. Element identification numbers should be unique with respect to all other element identification numbers.
7. RBAR, among other eligible rigid element types, can be selected via MPC and SET3.
8. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
9. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
10. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
11. For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.
There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBAR.
12. When there is large rotation, CMA and CMB must have all “456” if any rotational degrees of freedom is used.

RBAR1

Rigid Bar

Alternative format for RBAR.

Format:

1	2	3	4	5	6	7	8	9	10
RBAR1	EID	GA	GB	CB	ALPHA				

Example:

RBAR1	5	1	2	123	6.5-6				
-------	---	---	---	-----	-------	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB, which are constrained to move as the rigid bar. See Remark 4. (Integers 1 through 6 with no embedded blanks or blank.)
ALPHA	Thermal expansion coefficient. See Remark 8. (Real or blank)

Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CB.
- RBAR1 is a preferred input format to define the Lagrange method for a rigid bar.
- When CB = "123456" or blank, the grid point GB is constrained to move with GA as a rigid bar. For default CB = "123456". Any number of degrees-of-freedom at grid point GB can be released not to move with the rigid body.
- For the Lagrange method, the theory is formulated such that a consistent rigid body motion for grid points GA and GB will be computed even if these two points have different global coordinate systems.
- For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.

There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBAR1.

7. Element identification numbers should be unique with respect to all other element identification numbers.
8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
9. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

RBAX3D

Axisymmetric Harmonic Element Rigid Connection

Defines a rigid connection between a point on an axisymmetric harmonic element (CQUADX or CTRIAX) and a point on the axis of symmetry of that element.

Format:

1	2	3	4	5	6	7	8	9	10
RBAX3D	EID	G3D	GAX						

Example:

RBAX3D	100	10	20						
--------	-----	----	----	--	--	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000). See Remark 1..
G3D	Identification number of a grid point on the axis of symmetry (Integer > 100,000,000). See Remarks 2. and 4. through 7.
GAX	Identification number of a grid point referenced on an axisymmetric harmonic element (CQUADX or CTRIAX). (0 < Integer < 100,000,000). See Remarks 3. through 7..

Remarks:

1. The EID should be unique with respect to all other elements.
2. The G3D point may not be referenced on an axisymmetric harmonic element. If it is, the program terminates the execution with an appropriate user fatal message.
3. The GAX point must be referenced on an axisymmetric harmonic element. If it is not, the program terminates the execution with an appropriate user fatal message. Further, the axisymmetric harmonic element (or elements) that reference GAX must have harmonic index 1 specified on the associated PAXSYMH entry. If this condition is not satisfied, the program terminates the execution with an appropriate user fatal message.
4. The G3D point must be on the axis of symmetry. The GAX point need not be on the axis of symmetry.
5. The G3D and GAX points must both have the same axial coordinate along the axis of symmetry.
6. The G3D point is regarded as the independent point and the GAX point is regarded as the dependent point. The dependent degrees of freedom of GAX will be selected from the axisymmetric harmonic degrees of freedom.

7. If the GAX point is on the axis of symmetry, the connection is made only to the G3D displacement components perpendicular to the axis. However, if the GAX point is not on the axis of symmetry, then connection is made not only to the G3D displacement components perpendicular to the axis, but also to the slopes. The tolerance to determine if the GAX point is on the axis of symmetry is 1.0E-6.

RBE1**Rigid Body Element, Form 1**

Defines a rigid body connected to an arbitrary number of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RBE1	EID	GN1	CN1	GN2	CN2	GN3	CN3		
		GN4	CN4	GN5	CN5	GN6	CN6		
	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	-etc.-	ALPHA				

Example:

RBE1	59	59	123456						
	UM	61	246	6.5-6					

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GNi	Grid points at which independent degrees-of-freedom for the element are assigned. (Integer > 0)
CNi	Independent degrees-of-freedom in the global coordinate system for the rigid element at grid point(s) GNi. See Remark 1. (Integers 1 through 6 with no embedded blanks.)
“UM”	Indicates the start of the dependent degrees-of-freedom. (Character)
GMj	Grid points at which dependent degrees-of-freedom are assigned. (Integer > 0)
CMj	Dependent degrees-of-freedom in the global coordinate system at grid point(s) GMj. (Integers 1 through 6 with no embedded blanks.)
ALPHA	Thermal expansion coefficient. See Remark 13. (Real or blank)

Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CMj.
- For the linear method, the total number of components in CN1 to CN6 must equal six; for example, CN1 = 123, CN2 = 3, CN3 = 2, CN4 = 3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. The first continuation entry is not required if there are fewer than four GN points. For the Lagrange method, the total number of components must also be six. In addition, CN1 must be 123456, and CN2 through CN6 must be blank.

4. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be member of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding to m-set described below apply to both types of methods.
5. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
6. A degree-of-freedom cannot be both independent and dependent for the same element. However, both independent and dependent components can exist at the same grid point.
7. Element identification numbers should be unique with respect to all other element identification numbers.
8. RBE1, among other eligible rigid element types, can be selected via MPC and SET3.
9. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
10. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer subsstep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
11. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
12. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
13. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the grid point GN1 and any dependent grid point are taken as the average temperature of the two connected grid points.
There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBE1.

RBE2**Rigid Body Element, Form 2**

Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
	GM6	GM7	GM8	-etc.-	ALPHA				

Example:

RBE2	9	8	12	10	12	14	15	16	
	20	6.5-6							

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GN	Identification number of grid point to which all six independent degrees-of-freedom for the element are assigned. (Integer > 0)
CM	Component numbers of the dependent degrees-of-freedom in the global coordinate system at grid points GMi. (Integers 1 through 6 with no embedded blanks.) See Remark 12.
GMi	Grid point identification numbers at which dependent degrees-of-freedom are assigned. (Integer > 0)
ALPHA	Thermal expansion coefficient. See Remark 11. (Real or blank)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom which is obtained by CM multiplied with the number of dependent grid points.
3. For the linear method, the dependent degrees-of-freedom indicated by CM will be made members of the m-set at all grid points. For the Lagrange method, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.
4. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.

5. Element identification numbers should be unique with respect to all other element identification numbers.
6. RBE2, among other eligible rigid element types, can be selected via MPC and SET3.
7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
9. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
10. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
11. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the grid point GN and any dependent grid point are taken as the average temperature of the two connected grid points.
There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBE2.
12. It can be any combination in 1, 2, 3, but must have all "456" if any rotation dof is used when there is large rotation.

RBE2GS**Internally Generate an RBE2 Element**

Defines an RBE2 connecting the two closest grids to GS.

Format:

1	2	3	4	5	6	7	8	9	10
RBE2GS	EID	GS	TYPE			R	CM	ALPHA	
	XS	YS	ZS	GNi	GMj				

Example:

RBE2GS	3	17				1.3		6.5-6	
				end/	56	99			

Alternate Formats and Examples:

RBE2GS	EID	GS	TYPE			R	CM	ALPHA	
	XS	YS	ZS	GNi	THRU	GNj	end/		
	GMk	THRU	GM/						

RBE2GS	15	35				-.66			
				88	THRU	107	end/	15	
	76	88							

RBE2GS	25					.66			
	5.173	0.0	19.3185	88	99	108	end/		
	15	THRU	88						

RBE2GS	35	28				-.66			
				56	THRU	102	end/	19	
	21	THRU	200						

RBE2GS	45	16				-.66			
				56	THRU	102	1129		
	1146	THRU	1200	end/					

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GS	Search POINT or GRID point. (Integer > 0 or blank)
TYPE	Connectivity: (Character)
blank	Search the complete model. (Default)

Descriptor	Meaning
NMFLIP	The independent and dependent DOF's are interchanged.
IIRBE2	The grids chosen will be the independent GN's of the two closest existing RBE2 elements.
NMIIRBE2	Same as 'IIRBE2' except the independent and dependent grids are interchanged.
R	Radius. (Real > 0, or < 0)
CM	Component number of dependent degrees-of-freedom for grid GM. (Integers 1 through 6 with no embedded blanks. Blank defaults to 123456)
ALPHA	Thermal expansion coefficient. (Real or blank)
XS, YS, ZS	Location of search point if GS is blank. Used only if GS=0 or blank. (Real or blank)
GNi	List of grids to be excluded from candidate grids for GN. If a GNi list is given it must end with an <i>endl</i> . (Integer > 0 or blank or "THRU")
GMj	List of grids to be excluded from candidate grids for GM. If no GNi list is given and a GMj list is present, then GNi must have an <i>endl</i> entry. (Integer > 0 or blank or "THRU")

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. This entry will internally define a RBE2 element with the same ID of the RBE2GS entry. The grid assigned to GN will always be the independent grid. GM will be the dependent grid. If GS is a 'POINT' entry (or GS is blank and XYZ is specified), the first two grids that fall within the search radius $|R|$ about GS (or XYZ) will be chosen as GN and GM. The closest to the search location will be the independent grid GN the next closest will be the dependent grid GM. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with or without use of the exclusion lists for GNi and GMj (and if TYPE = 'NMFLIP', then GN and GM will be reversed). If GS is 'GRID' entry, and is part of the physical model, i.e., it has physical structural elements attached to it, it will become GN the independent grid for the RBE2 to be generated. If it is in the GNi exclusion list, the next closest grid will be chosen. the closest grid within the search radius about $|R|$ GN will be chosen as GM, the dependent grid. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with our without use of the exclusion lists for GNi and GMj) and if TYPE = 'NMFLIP', then GN and GM will be reversed. If GS is a 'GRID' entry, and is determined not to be part of the physical model, the first two grids that fall within the search radius $|R|$ about GS (or XYZ) will be chosen as GN and GM. The closest to the search location will be the independent grid, GN, the next closest will be the dependent grid, GM. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with or without use of the exclusion lists for GNi and GMj) and if TYPE='NMFLIP', then GN and GM will be reversed. The GS grid will remain on the GEOM1 table for post-processing viewing purposes, but will not be part of the Nastran solution g-set. If TYPE='IIRBE2', the two grids chosen for the

RBE2 will be the independent GN's of existing RBE2 elements whose independent grids lie within the search radius. If TYPE='NMIIRB2', then GN and GM will be reversed. If no existing RBE2 elements have independent grids within the search radius or if only one existing RBE2 has an independent grid within the search radius, then a fatal message will be issued. POINT and GRID entries must be unique with respect to all other POINT and GRID entries. If TYPE='IIRB2', it is recommended that PARAM,AUTOMSET,YES be used in the analysis run.

3. If $R < 0$, the two located grids GN and GM will be made coincident to the GS (or XYZ) location.
4. If CM is 123456 and GM is touching only solid elements, CM will be internally changed to 123. For solid elements, the grids GN and GM should always be coincident.
5. When Module GP4 is run, checks are made to insure that the selected grids, GN and GM, do not violate existing constraint sets. If a violation occurs a fatal message will be issued for the offending grids. These grids can be excluded from further search inclusion by specifying them on the GNi or GMj list. PARAM,AUTOMSET,YES will often prevent any violation.
6. If GS=0 or blank and XS, YX, ZS is not specified or if both GS and XS, YS, ZS are specified a fatal error will occur.
7. The end of a grid exclusion lists is indicated by the existence of "end/" in the field following the last entry in the list. In the "thru" option, not all grids in the range need exist.
8. For superelement or part superelement connection the independent grid assigned to GN will be exterior to the superelement. The dependent grid GM will be an interior grid to the superelement. If the resulting RBE2 element, connects two different superelements, the element will be assigned to the superelement with the lower ID and the grid attached to the superelement with the higher ID will be moved to the superelement with the lower ID.
9. If the RBE2GS is listed on a SEELT entry, it will be placed as the SEELT directs. If say grid G1 lies in another superelement and G2 lies in the SEELT superelement, G1 will be moved to the SEELT defined superelement.
10. "THRU" should not be the first nonblank field of a continuation line. Blank fields are allowed in the exclusion lists for readability.
11. Use PARAM,SEP1XOVR,16 to print the grids found by each RBE2GS entry.
12. RBE2GS is not supported in the presence of part (BEGIN SUPER) superelements.

For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the grid point GN1 and any dependent grid point are taken as the average temperature of the two connected grid points.

There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBE2GS.

RBE3

Interpolation Constraint Element

Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
	G1,3	WT2	C2	G2,1	G2,2	-etc.-	WT3	C3	
	G3,1	G3,2	-etc.-	WT4	C4	G4,1	G4,2	-etc.-	
	"UM"	GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	GM5	CM5	-etc.-			
	"ALPHA"	ALPHA							

Example:

RBE3	14		100	1234	1.0	123	1	3	
	5	4.7	1	2	4	6	5.2	2	
	7	8	9	5.1	1	15	16		
	UM	100	14	5	3	7	2		
	ALPHA	6.5-6							

Descriptor	Meaning
EID	Element identification number. Unique with respect to all elements. (0 < Integer < 100,000,000)
REFGRID	Reference grid point identification number. (Integer > 0)
REFC	Component numbers at the reference grid point. (Any of the integers 1 through 6 with no embedded blanks.)
WTi	Weighting factor for components of motion on the following entry at grid points Gi,j. (Real)
Ci	Component numbers with weighting factor WTi at grid points Gi,j. (Any of the integers 1 through 6 with no embedded blanks.)
Gi,j	Grid points with components Ci that have weighting factor WTi in the averaging equations. (Integer > 0)
"UM"	Indicates the start of the degrees-of-freedom belonging to the dependent degrees-of-freedom. The default action is to assign only the components in REFC to the dependent degrees-of-freedom. (Character)
GMi	Identification numbers of grid points with degrees-of-freedom in the m-set. (Integer > 0)

Descriptor	Meaning
CMi	Component numbers of GMi to be assigned to the m-set. (Any of the Integers 1 through 6 with no embedded blanks.)
"ALPHA"	Indicates that the next number is the coefficient of thermal expansion. (Character)
ALPHA	Thermal expansion coefficient. See Remark 14. (Real or blank)

Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, the REFC must be "123", "456", or "123456". No other combination is allowed.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of degrees-of-freedom given by REFC.
- For the linear method, the dependent degrees-of-freedom indicated by REFC will be made members of the m-set. For Lagrange rigid element, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.
- We recommend that for most applications only the translation components 123 be used for Ci. An exception is the case where the Gi,j are colinear. A rotation component may then be added to one grid point to stabilize its associated rigid body mode for the element.
- Blank spaces may be left at the end of a Gi,j sequence.
- For the Lagrange method, the default for "UM" must be used. For the linear method, the default for "UM" should be used except in cases where the user wishes to include some or all REFC components in displacement sets exclusive from the m-set. If the default is not used for "UM":
 - The total number of components in the m-set (i.e., the total number of dependent degrees-of-freedom defined by the element) must be equal to the number of components in REFC (four components in the example).
 - The components specified after "UM" must be a subset of the components specified under REFC and (Gi,j, Ci).
 - The m-rows by m-columns partition [Rm,m] of the global stiffness containing the dependent degrees-of-freedom of multipoint constraints and rigid elements must be nonsingular. PARAM,CHECKOUT in SOLs 101 through 200 may be used to check for this condition. For some complex constraints, forming a well-conditioned [Rm,m] is difficult for a user to perform manually and PARAM,AUTOMSET,YES will automatically determine a well-conditioned partition by employing a rectangular decomposition of the [Rm,g] matrix.
- Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
- RBE3, among other eligible rigid element types, can be selected via MPC and SET3.

10. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
11. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
12. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
13. The formulation for the RBE3 element was changed in Version 70.7. This change allowed the element to give consistent answers that are not dependent upon the units of the model. Only models that connected rotation degrees-of-freedom for Ci were affected. Note that these models are ignoring the recommendation in Remark 5. The formulation prior to Version 70.7 may be obtained by setting SYSTEM(310)=1.
14. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the reference grid point REFGRID and any other grid point Gij are taken as the average temperature of the two connected grid points.

There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBE3.

RBE3U

Defines Method to Distribute Applied Loads to a Surface in SOL 600

Format:

1	2	3	4	5	6	7	8	9	10
RBE3U	EID	IOPT	GREF	G1	G2	G3	G4	G5	
	G6	G7	G8	G9	etc.				

Alternate Format:

1	2	3	4	5	6	7	8	9	10
RBE3U	EID	IOPT	GREF	G1	THRU	G3	BY	G4	
	G5	THRU	G6	BY	G7				
	G8	THRU	G9	BY	G10	etc.			

Examples:

RBE3U	250	1	1000	100	THRU	120			
-------	-----	---	------	-----	------	-----	--	--	--

Descriptor	Meaning
EID	Identification number, unique among all elements. (Integer; no Default; Required)
IOPT	Option of how to distribute the load. (Integer; Default = 1)
1	Equal loads values will be applied to all grid points on the surface.
2	Loads will be distributed according to how close each grid point on the surface is to GREF.
Gi	List of grids to which forces at GREF will be distributed. (Integer; Default = 123)

Remarks:

- The sum of the loads on all grid points is the same as the applied to GREF in each of the CREF directions. If IOPT=1 the force on each grid point will be the total loads divided by the number of applicable grid points. If IOPT=2 the load of each grid point will be weighted according to its distance from GREF, however the sum of all loads will be the same as that applied to GREF. The load is removed from GREF after distribution.
- RBE3U is only used to distributed applied loads. No actual rigid elements are included in the analysis. If rigid elements are desired, use RBE2 or RBE3.
- Only FORCE and MOMENT will be distributed using this entry. All other types of forces such as FORCE1, MOMENT1, TEMP, etc. applied to GREF will not be distributed.

RBJOINT

Defines a Joint Between Two Rigid Bodies

Defines a joint between two rigid bodies. Used in SOL 700 only.

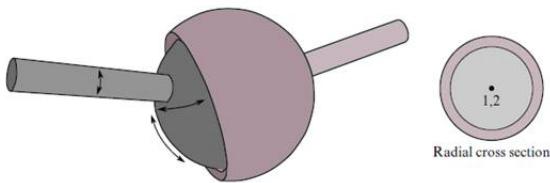
Format:

1	2	3	4	5	6	7	8	9	10
RBJOINT	ID	TYPE	N1	N2	N3	N4	N5	N6	
		RPS							

Examples:**Example 1 - Cylindrical Joint**

RBJOINT	1	CYLIND	101	201	103	204			
---------	---	--------	-----	-----	-----	-----	--	--	--

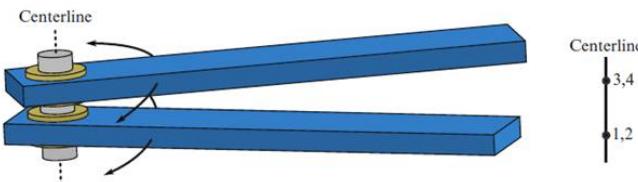
Descriptor	Meaning
ID	RBJOIN identification number. (Integer > 0; Required)
TYPE	Type of RBJOIN. (Character; Required) Types available are (node numbers in the images correspond to N1, N2, etc.): TYPE=SPHER(spherical), The relative motion of the rigid bodies are constrained so that nodes which are initially coincident remain coincident. In the below figure the socket's node is not interior to the socket-SOL700 does not require that a rigid body's nodes be interior to the body.



Spherical joint

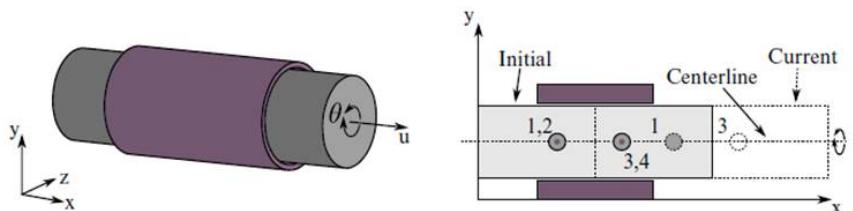
Descriptor	Meaning
------------	---------

TYPE=REVOLUTE, Both nodal pairs (1,2) and (3,4) are constrained to remain coincident. Consequently, the relative motion of these rigid bodies is restricted to rotations about the line segment formed by the two pairs of coincident nodes. This segment is labeled the "centerline".



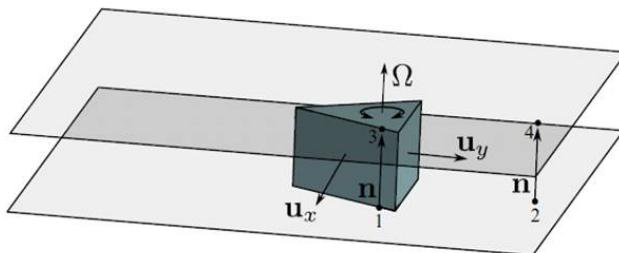
Revolute joint

TYPE=CYLINDRICAL, This joint is derived from the rotational joint by relaxing the constraints along the centerline. This joint admits relative rotation and translation along the centerline.



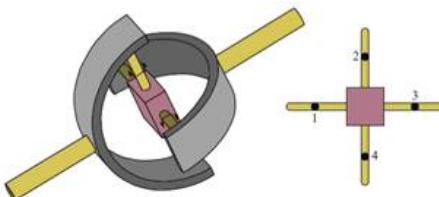
Cylindrical joint

TYPE=PLANAR, This joint is derived from the rotational joint by relaxing the constraints normal to the centerline. Relatively displacements along the direction of the centerline are excluded.



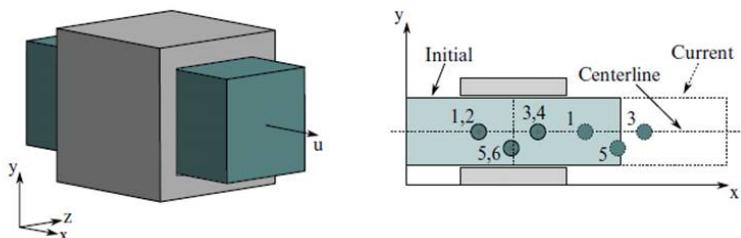
Planar joint

Descriptor	Meaning
	TYPE=UNIVERS (universal joint), In contrast with the preceding joints, nodal pairs (1,2) and (3,4) are not initially coincident. Rather, the segments formed by (1,3) and (2,4) must be orthogonal; and they serve as axes about which the two bodies may undergo relative rotation. The universal joint excludes all other relative motion and the axes remain orthogonal at all time.



Universal joint

TYPE=TRANSL (translational), This is a cylindrical joint with a third pair of off-centerline nodes which restrict rotation. Aside from translation along the centerline the two rigid bodies are stuck together.



Translational joint

- N1 GRID id of Node 1, in rigid body A. Define for all joint types
- N2 GRID id of Node 2, in rigid body B. Define for all joint types
- N3 GRID id of Node 3, in rigid body A. Define for all joint types except SPHER.
- N4 GRID id of Node 4, in rigid body B. Define for all joint types except SPHER.
- N5 GRID id of Node 5, in rigid body A. Define only for joint types TRANSL.
- N6 GRID id of Node 6, in rigid body B. Define only for joint types TRANSL.
- RPS Relative penalty stiffness (Default = 1.0)

RCONN**Rigid Connection**

Defines a rigid connection between the different parts of Lagrangian meshes (tied surfaces). Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
RCONN	CID	STYPE	MTYPE	SID	MID	OPTION			
	CLSGAP	GAPDIS	GAPDISV						

Example:

RCONN	7	GRID	SURF	3	7	NORMAL			
-------	---	------	------	---	---	--------	--	--	--

Field	Content	
CID	Unique rigid-connection number (Integer; Required)	
STYPE	Type of entity used to define the slave surface (Character; Default = SURF)	
	SURF	The faces of the elements are used for the slave surface. SID refers to BSURF ID. See Remark 1.
	GRID	Grid points will be tied to the master surface. SID then refers to a SET or BCGRID ID containing the list of grid points to be used. See Remarks 2. and 3.
MTYPE	Type of entity used to define the master surface (Character; Default = SURF)	
	SURF	The faces of the elements are used for the master surface. MID refers to BSURF ID.
SID	Slave BSURF ID or SET1 ID containing the list of grid points (Integer; Required)	
MID	Master BSURF ID (Integer; Required)	
OPTION	Only used if discrete grid points are tied to a surface (STYPE is equal to GRID). (Character; Default = NORMAL)	
	NORMAL	The grid points are tied to the master surface. See Remark 2.
	SHELL	The grid points are attached to the edge of shell or beam elements, which are tied to the shell surface. See Remark 3.
CLSGAP	Switch to automatically close any gaps that are present between the master-slave surface (Character; Default = NO)	
	YES	Gaps are automatically closed
	NO	Gaps are not closed. See Remark 2.

Field	Content
GAPDIS	Defines the tolerance used in the search for a master face. If the distance between a slave point and a master face falls within this tolerance, the master face is accepted. If not, the search for a correct master face continues (Character; Default = DISTANCE) DISTANCE The tolerance has the length as specified at GAPDISV
GAPDISV	The value of the gap tolerance or a factor to calculate this tolerance depending on the value of GAPDIS (Real; Default = 1.0E20)

Remarks

1. The RCONN entry can be used to define three types of connection:
 - a. Two Surfaces Tied Together.
 - b. Define slave and master segments representing the two surfaces to be tied together. There should not be a gap between the two sets of segments. The two surfaces are tied together during the analysis.
 - c. Grid Points Tied to a Surface.
 - d. If STYPE is set to GRID and OPTION is set to NORMAL, the slave entities comprise discrete grid points that are tied to the master surface during the analysis. The grid points must lie on the surface.
 - e. Shell Edge Tied to a Shell Surface.
 - f. If STYPE is set to GRID and OPTION is set to SHELL, the edges of shell or beams elements can be tied to the faces of other shells. The grid points attached to the edge of the shells/beams must be selected as the slave grid points. The shell surface to which they are tied must be selected as the master surface. The two sets will then be tied together throughout the analysis. All degrees of freedom will be coupled.
2. The CLSGAP entry enables you to define two different meshes that are not coincident over the master/slave interface. If the option is set to YES, the slave surface becomes coincident (according to projections) with the master surface.
3. The search method of the contact algorithm is used to find the closest master face. The tolerance defined with the GAPDIS/GAPDISV fields is similar to the monitoring distance defined on the CONTACT entry with the MONDIS/ MONDISV fields.
4. The use of the gap closing CLSGAP can cause an element to collapse. This may happen if the GAPDISV tolerance is set to a value greater than the length of the side of an element.
5. When a solid and a shell mesh are tied together, the rotational degrees of freedom of the shell grid points are not coupled.
6. When OPTION=SHELL and CLSGAP=NO, the time step scale factor will be set to 0.4. This can be overwritten by:
PARAM*, STEPFCTRCONN, xxx

RCPARM

Parameters for RC Heat Transfer Analysis Control

Defines a set of parameters for nonlinear resistance-capacitance based analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RCPARM	ID	SOLVER	DRLXCA	ARLXCA	BALENG	NLOOP	DAMPD	GRVCON	
	TIMEND	OUTPUT	CSGFAC	DTIMEI					

Example:

RCPARM	1	SNDUFR	1.0-2	1.0-2	1.0-4	2000	0.7	9.81	
	1800.	60.	1.2	0.0					

Descriptor	Meaning
ID	Identification number. See Remark 2. (Integer > 0; Required)
Solver	Equivalent Sinda Solution name. See Remarks 3. and 7. (String; Required) Possible values: RCNS “SNSOR”, “STDSTL” RCNT “SNFRDL”, “FWDBKL”, “SNADE”, “SNDUFR”, “ATSDUF”, “ATSFBK”, “SNTSM”, “SNTSM3”, “SNTSM1”, “SNTSM4”, Maintained for “SNSOR1”, “SNSORA”, “SNSOR1A”, “SNHOSD”, “SNSOSS”, legacy, but not “SNHOSS”, “SCROUT”, “SNDSNR”, “TRSPMA” shown in GUI
DRLXCA	Diffusion node convergence criterion (Real ≥ 0.0 ; Default 1.0e-3 degrees) (SimX: Default 1.0e-4 for transient)
ARLXCA	Arithmetic node convergence criterion (Real ≥ 0.0 ; Default 1.0e-3 degrees) (SimX: Default 1.0e-4 for transient)
BALENG	Allowable system energy imbalance (Real ≥ 0.0 ; Default 0.0 energy/time)
NLOOP	Number of iterations allowable (Integer ≥ 0 ; Default 5000 loop) (SimX: Default 50 for transient)
DAMPD	Damping constant (Real ≥ 0.0 ; Default 0.0 non dimensional)
GRVCON	Gravitation constant (Real ≥ 0.0 ; Default 9.81 length/time^2)
TIMEND	Problem end time (Real; Default 3600.0 time)
OUTPUT	Output interval (Real ≥ 0.0 ; Default 60.0 time)
CSGFAC	Time step control factor (Real ≥ 0.0 ; Default 1.0 non dimensional)
DTIMEI	Time step (Real ≥ 0.0 ; Default 0.0 time)

Remarks:

1. This entry is for RC Network solver only.
2. ID can be referenced by the case control command. Different cases may reference different RCPARM entries. It is possible to have multiple RCPARM entries inside one BDF file.
3. RCNS stands for RC Network Steady-state (static), RCNT stands for RC Network Transient
4. The default values are for Nastran only. The modeler (Patran or SimXpert) will have their default values. Some are related with the model units. The default values will not be all 0.0.
5. Default solvers and result checking SNSOR and SNDUFR are the default steady and transient solvers. They are good for most of the cases. Users are encouraged to check the results by the following methods.

Obtain nearly identical results with a different solver

Obtain nearly identical results with tighter convergence

Obtain nearly identical results with a smaller time step (transient)

Obtain nearly identical results with more rays or different random seeds (external radiation solvers)

6. About non-default solvers:

For non-spacecraft, non-radiation problems:

Solid type model – SNSOR, SNTSM1

For spacecraft/radiation models:

SNSOR (with user-specified DAMPD if necessary)

FWDBKL if thermo-stats are present

ATSDUF, SNTSM1 for most other cases

7. About the time step

- a. The default computed time step (DTIMEU) = CSGMIN* CSGFAC. CSGMIN can be checked in the .sot file. If CSGFAC is not specified, it is internally set to 1.0. The DTIMEU in the .sot file has been truncated by the OUTPUT point.
- b. In a normal sized model, CSGMIN usually is small enough for the time step which will assure a convergent transient run.
- c. CSGFAC is used to adjust the time step. It is recommended to determine the best CSGFAC to the model while maintaining acceptable temperature errors.
- d. If OUTPUT < CSGFAC*CSGMIN or OUTPUT < DTIMEI, then OUTPUT becomes the time step. All the OUTPUT points are automatically required to be calculated.
- e. DTIMEI is the forced time step which will ignore any other factors. Sometimes it may lead to inaccurate answer if it is too large. DTIMEI does not affect the automatic time step solvers.
- f. If the model size is very small, CSGMIN may be too big for the time step. A small CSGFAC or DTIMEI should be used to adjust the time step.
- g. CSGFAC*CSGMIN or DTIMEI should be small enough to “catch” any details in time fields, temperature fields or orbital flux arrays.

8. For more details about these parameters, please reference *MSC SINDA User's Guide and Library Reference*.

RCROSS**Cross-Power Spectral Density and Cross-Correlation Functions Output**

Defines a pair of response quantities for computing the cross-power spectral density and cross-correlation functions in random analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RCROSS	SID	RTYPE1	ID1	COMP1	RTYPE2	ID2	COMP2	CURID	

Example:

RCROSS	10	DISP	100	3	STRESS	200	10	2	
--------	----	------	-----	---	--------	-----	----	---	--

Descriptor	Meaning
SID	Case Control RCROSS identification number for cross-power spectral density function and cross-correlation function. (Integer > 0)
RTYPEi	Type of response quantity. At least one field must be selected. See Remark 2. (Character or blank)
IDi	Element, grid or scalar point identification number. (Integer > 0)
COMPi	Component code (item) identification number. See Remark 3. (Integer > 0)
CURID	Curve identification number. See Remark 4. (Integer ≥ 0 or blank)

Remarks:

1. This entry is required for computing the cross-power spectral density function and cross-correlation function. SID must be selected with the Case Control command (RCROSS = SID). Fields RTYPE1, ID1, and COMP1 represent the first response quantity, and fields RTYPE2, ID2, and COMP2 the second in the correlation.
2. The keywords for field RTYPEi are listed as follows:

Keyword	Meaning
DISP	Displacement Vector
VELO	Velocity Vector
ACCEL	Acceleration Vector
OLOAD	Applied Load Vector
SPCF	Single-point Constraint Force Vector

Keyword	Meaning
MPCF	Multi-point Constraint Force Vector
STRESS	Element Stress
STRAIN	Element Strain
FORCE	Element Force

If anyone of RTYPE1 or RTYPE2 is blank, then the default is the one same as the other field.

3. For elements, the item code COMPi represents a component of the element stress, strain, and force and is described in Tables [Element Stress-Strain Item Codes, 1048](#) and [Element Force Item Codes Part 1, 1085](#). For an item having both a real and imaginary part, the code of the real part must be selected. This is required for computing both the cross-power spectral density function and cross-correlation function.
For grid point, the item code is one of 1, 2, 3, 4, 5, and 6, which represent the mnemonics T1, T2, T3, R1, R2, and R3, respectively. For scalar point, always use 1.
4. Field CURID is optional. It is for the user's convenience to identify the output by using a single index.

RELEASE**Superelement Boundary Grid Point Release**

Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.

Format:

1	2	3	4	5	6	7	8	9	10
RELEASE	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	-etc.-						

Example:

RELEASE	15	456	3	7	11	2	156	9	
	152	162							

Alternate Formats and Examples:

RELEASE	SEID	C	G1	“THRU”	G2				
RELEASE	6	2	15	THRU	127				

(ALL must be in FIELD 4 and no continuation is allowed)

RELEASE	SEID	C	“ALL”						
RELEASE	127	156	ALL						

Descriptor	Meaning
SEID	Superelement identification number. (Integer > 0)
C	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)
Gi	Grid point identification numbers. (Integer > 0; “THRU”, or “ALL”; For THRU option, G1 < G2.)

Remarks:

1. A grid point referenced on this entry must be an exterior grid point of the superelement referenced on the entry.
2. In the first alternate format, all grid points in the sequence G1 through G2 are not required to be exterior grid points. Such grid points will collectively produce a warning message but will otherwise be ignored.
3. If the “ALL” option is used, all exterior grid points are released for the referenced degrees-of-freedom.
4. The RELEASE entry is applicable to only the superelement solution sequences (SOLs 101 through 200). It may not reference the residual structure (SEID = 0).
5. This entry is not supported for partitioned superelements.

RELEX

External Definition of a Rigid Ellipsoid- SOL 700

Defines a rigid ellipsoid whose properties and motion are defined by either ATB. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
RELEX	NAME	PROG					etc		

Example:

RELEX	HEAD	ATB							
-------	------	-----	--	--	--	--	--	--	--

Field	Content
NAME	This name is used within the input file to define the interactions between the external ellipsoid and grid points and rigid bodies. This name is also used in the output requests. When coupled to ATB, the name must correspond to the name of the ATB segment. (Character; required)
PROG	Name of the external program. (Character; required) ATB SOL700 runs coupled with ATB

RESTART

Restart Data for Marc Executed from SOL 600

Specifies writing or reading of restart data for Nonlinear Analysis when Marc is executed from SOL 600. If this Bulk Data entry is found in the job stream, the type of "restart" specified by KIND and KTYPE will be performed. Only one RESTART entry is allowed.

Format:

1	2	3	4	5	6	7	8	9	10
RESTART	ID	KIND	NINC	NBEGIN					
	NAME		TSTEP	ENDTIME	NSTEPS	NDCYC	STEPMAX	PERCENT	

Example(s):

RESTART	101	1	1						
---------	-----	---	---	--	--	--	--	--	--

The above example writes a restart file. The original run named is abcde.dat.

RESTART	201	3	1	15					
	abcde								

The above example restarts the original abcde.dat run. The name of the restart run must not be abcde.dat in this case.

RESTART	151	2		11					
	my_first	_run							

The above example reads the restart file and prints out results not printed in the original run.

Descriptor	Meaning
ID	Identification number of the restart entry -- Not presently used (Integer)
KIND (2,1)	Type of restart (Integer > 0; required field; no Default) 1 = Write a restart file 2 = Restart a previous analysis (Read an existing restart file) 3 = Restart a previous analysis and write new data on restart file. 11 = Only write restart file for the last converged increment of the run. 12 = Read a restart file written with KIND=11 13 = Read a restart file written with KIND=11 and write the last increment or time step of the present run on that file as well.
NBEGIN (2,3)	The "time" increment at which the restart run begins (used only if KIND=2 or 3). (Integer > 0. See Remarks 3, 4.)
NINC (2,2)	Number of increments between writing of restart data. (Integer > 0; Default = 1)

Descriptor	Meaning
IPRINT (2,8)	Set this field to 1 if the restart data is to be printed (All data from increment INCBEG to LAST will be printed if IPRINT=1). Use this option if printing on a previous run was suppressed but now is desired. (Integer ≥ 0 ; Default = 0)
LAST (2,9)	The last time increment is printed if IPRINT=1, otherwise this field is ignored. (Integer > 0 , Default = 0. See Remarks 3., 4.)
NAME	Name of input file for the original MSC Nastran run without extension. NAME is limited to 16 characters and may not contain imbedded blanks. If the small field format is used, NAME may span fields 2 and 3 of the continuation entry. If the large field is used, NAME should be coded in the 2nd field. NAME is required for a restart run, see Remarks 3., 7. If NAME exceeds 8 characters, the continuation line must be coded in small-field fixed format or in large field fixed or free-format.
TSTEP [2,1]	Time step size after restart -- For dynamic problems only. (Real ≥ 0.0 or blank; if the value is ≤ 0.0 the original step size is used)
ENDTIME [2,2]	Ending time for this restart run (Real ≥ 0.0 or blank, if the value is ≤ 0.0 the original end time is used)
NSTEPS [2,3]	Total number of time steps for the restart run plus the original run. (Integer ≥ 1 ; Default = 1)
NDCYC [2,6]	Desired number of recycles if AUTO INCREMENT options were specified on the original run. (Integer ≥ 0 ; if the value is ≤ 0 the original is used)
STEPMAX [2,7]	Maximum step size if AUTO INCREMENT options were specified on the original run. (Real ≥ 0.0 ; if the value is ≤ 0 the original is used)
PERCENT [2,8]	Percentage of total load to be applied.(Real ≥ 0.0 ; if the value is ≤ 0 the original is used)

Remarks:

1. RESTART is available only when Marc is executed from within MSC Nastran Implicit Nonlinear (SOL 600).
2. There should only be one RESTART entry in the bulk data. If more than one exists, the first one will be used.
3. (i,j) Indicate the field in Marc's RESTART model definition options. [i,j] Indicate the field in Marc's REAUTO model definition options.
4. The jid.marc.t16 and jid.marc.t08 files must be saved from the first run when a restart run is executed. Both original and restart runs must be located in the same directory. File extensions are .t08 and t16.
5. A restart run may not have the same input file name (jid) as that of the original run. In other words, if the input file for the original run is named abcd.dat, the input file for a restart run may not be named abcd.dat.

6. For static analyses, normally each load case has a total time of 1.0. The first case goes from 0.0 to 1.0, the second from 1.0 to 2.0, etc. If the first run has two static load cases and terminates at 1.6, it is in the middle of the second load case. The original run should be examined to determine which increment (before 1.6) to begin the restart run.
7. In the second example above, the original run was named my_first_run.dat (or first_run.bdf, etc). This is a small field example. There are 8 characters in the 2nd field of the continuation line and 4 characters in the third field. The name can start anywhere within the two fields. There must not be any blank spaces in the name. If the name exceeds 8 characters, the continuation line must be coded in small-field fixed format or in large-field fixed or free format. NAME is limited to a maximum of 16 characters.
8. Both original and restart run names (jid's) should use only lower case letters except on computer systems that are not case sensitive. "NAME" (continue line, fields 2-3) will be converted automatically to lower case.
9. The restart run must use the same values of Marc's DIST LOAD parameter as the original run or it may fail. Be sure to use parameters MARCDIS2, MARCDIS3 and MARCDIS4 to set these values to be exactly the same as the original run (examine jid.marc.dat from the original run to determine these values before submitting the new run).
10. The RSF options creates a complete database which is necessary for restarts. When RSF=0 is specified, the same file is overwritten after each interval. When RSF=1 is specified, a new restart file is created after each interval, thus a "family" of files is created numbered sequentially jid.dytr.d3dump10, jid.dytr.d3dump02, etc. These files can take significant disk space but are important if a model might need to be modified prior to the end time.
11. It is not necessary to enter continuation lines if not needed for the particular job to be run.
12. Restarts are not available for models with bolt loading using entries MBOLT, MBOLTUS or BOLT.

RFORCE

Rotational Force

Defines a static loading condition due to an angular velocity and/or acceleration.

Format:

1	2	3	4	5	6	7	8	9	10
RFORCE	SID	G	CID	A	R1	R2	R3	METHOD	
	RACC	MB	IDRF						

Example:

RFORCE	2	5		-6.4	0.0	0.0	1.0	2	
	1.0								

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number through which the rotation vector acts. (Integer ≥ 0)
CID	Coordinate system defining the components of the rotation vector. See Remark 16. (Integer ≥ 0 ; Default = 0)
A	Scale factor of the angular velocity in revolutions per unit time. (Real)
R1, R2, R3	Rectangular components of rotation vector \vec{R} . The vector defined will pass through point G. (Real; $R1^2 + R2^2 + R3^2 > 0.0$ unless A and RACC are both zero)
METHOD	Method used to compute centrifugal forces due to angular velocity, see Remark 20.. For angular acceleration, see Remark 13. (Integer = 1 or 2; Default = 1)
RACC	Scale factor of the angular acceleration in revolutions per unit time squared. (Real; Default = 0.0)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 15. (Integer; Default = 0)
IDRF (SOL 600 only)	ID indicating to which portion of the structure this particular RFORCE entry applies. It is possible to have multiple RFORCE entries in the same subcase for SOL 600 to represent different portions of the structure with different rotational accelerations. IDRF corresponds to a SET3 entry specifying the elements with this acceleration. A BRKSQ entry may also be specified with a matching IDRF entry. (Integer; Default = 0)

Remarks:

- The forces that are created with the RFORCE entry act on the structure as follows: the forces that are defined with the RFORCE entry for a constant angular velocity (A), act in the positive radial direction. These forces represent the inertia forces on the structure due to a constant angular velocity. The forces that are defined with the RFORCE entry for a constant angular acceleration (RACC), act in the same direction as the angular acceleration. These forces would be opposite to the inertia forces on the structure due to a constant angular acceleration. In [Figure 9-151](#), the force vector at grid point Gi is given by

$$\{\vec{F}\}_i = [m]_i [\vec{\omega} \times (\vec{\omega} \times (\vec{r}_i - \vec{r}_a)) + \vec{\alpha} \times (\vec{r}_i - \vec{r}_a)] \quad (9-33)$$

where:

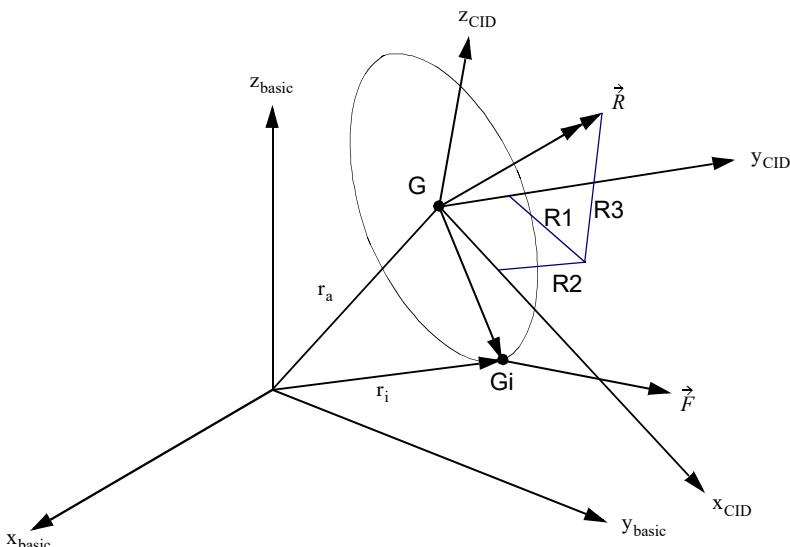
$$\text{angular velocity} = \vec{\omega} = 2\pi A \cdot \vec{R} \text{ (radians/unit time)}$$

$$\text{angular acceleration} = \vec{\alpha} = 2\pi RACC \cdot \vec{R} \text{ (radians/unit time}^2\text{)}$$

$$[m]_i = 3 \times 3 \text{ translational mass matrix at grid point Gi}$$

Note:

The equation for \vec{F}_i will have additional terms if the mass is offset or I_{23}^i , I_{13}^i terms exist relative to the rotation axes and METHOD = 1 is selected.



[Figure 9-151](#) RFORCE Vector at Grid Point Gi

2. In the static solution sequences, SID must be selected by the LOAD Case Control command.
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. G = 0 signifies that the rotation vector acts through the origin of the basic coordinate system.
4. CID = 0 (Default) signifies that the rotation vector is defined in the basic coordinate system.
5. The load vector generated by this entry can be printed with an OLOAD command in the Case Control Section.
6. METHOD = 1 yields correct results only when there is no coupling in the mass matrix. This occurs when the lumped mass option is used with or without the ZOFFS option (see the CQUAD4 entry for a description of ZOFFS). METHOD = 2 yields correct results for lumped or consistent mass matrix only if the ZOFFS option is not used. The acceleration terms due to the mass offset (X1, X2, X3) on the CONM2 entry are not computed with METHOD = 2. All the possible combinations of mass matrices and offset and the correct method to be used are shown below.

	No Offset	Offset
Lumped	METHOD = 1 or METHOD = 2	METHOD = 1
Coupled	METHOD = 2	Neither

7. In cyclic symmetry analyses, the T3 axis of the basic coordinate system must be coincident with the axis of symmetry. In the DIH type of cyclic symmetry, the T1 axis also must be parallel to side 1 of segment 1R of the model.
8. For superelement analysis, G should reference a residual structure point that is exterior to all superelements when loading Superelement 0 and a separate G must be interior to each superelement when loading an upstream superelement else centrifugal loads will not be generated for that superelement. However, in cyclic analysis, User Fatal Message 4347 will be issued.
9. In a geometric nonlinear static analysis (SOL 106 and 400 when PARAM LDGISP is set to +1), this type of loading is a follower force type of loading. However, the orientation of coordinate system CID is not updated.
10. In nonlinear static solutions when there is more than one increment (INC) specified on the NLPARM entry for a given subcase, the load vector resulting from the RFORCE input (and not the angular velocity vector) is scaled linearly. This means that loading by increments in the angular velocity can only be achieved by having subcases where the RFORCE loading is applied in a single increment.
11. The continuation entry is optional.
12. Forces due to angular acceleration (RACC) are computed with METHOD = 2 even if METHOD = 1 is specified.
13. Loads derived from this entry do not include effects due to mass specified for scalar points.

14. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter **FOLLOWK** (Ch. 6)). In addition, follower force effects are included in the force balance in the nonlinear solution sequences, SOLs 106, 129, 153, 159 and 400, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) and general nonlinear solution sequence (SOL 400) for both ANALYSIS=NSTAT and NLTRAN but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
15. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.
16. If CID is not a rectangular coordinate system, RFORCE will treat it as if it were and unexpected answers may result.
17. Follower force stiffness (param,followk,yes) is supported for method 2 only.
18. Multiple RFORCE entries with different SID's may be used in SOL 600 in the same subcase or SOL 400 in the same loadcase (STEP and/or SUBCASE). They should be combined using the LOAD entry similar to the way FORCE or PLOAD4 with different ID's are combined.
19. Fields CID, METHOD, RACC, MB and IDR will be ignored for SOL 700.
20. For Axisymmetric Harmonic elements, (elements selected with PAXSYMH entry), METHOD=2 is required. A selection of METHOD=1 will automatically be overridden and be replaced by METHOD=2 for all elements of the model.
21. When used with rotor dynamics, the angular velocity creates axial terms that are proportional to ω^2 and moments that are proportional to ω . In order to keep consistent spin rate, if the scaling is requested on the LOAD entry, the on the axial terms will be scaled by this scale factor, and the ω on the moment terms will be scaled by the square root of this scale factor. See Remark 6. of the LOAD entry.
22. If Modules are present then this entry may only be specified in the main Bulk Data section.

RGYRO

Rotordynamic Analysis Parameters

Specifies synchronous or asynchronous analysis, reference rotor, and rotation speed of the reference rotor.

Format:

1	2	3	4	5	6	7	8	9	10
RGYRO	RID	SYNCFLG	REFROTR	SPDUNIT	SPDLLOW	SPDHIGH	SPEED	ROTRSEID	
	WR3WRL	WR4WRL	WRHWRL						

Example:

1	2	3	4	5	6	7	8	9	10
RGYRO	100	SYNC	1	RPM	1000.0	5000.0		10	

Descriptor	Meaning
RID	Identification number of RGYRO entry. Selected by Case Control command, RGYRO. (Required; no Default). See Remarks 1. and 2.
SYNCFLG	Specifies whether the analysis is synchronous or asynchronous analysis. Required input for frequency response and complex modes analyses. Not required for static analyses. (Character: 'SYNC', 'ASYNC', or blank). See Remarks 2. through 4.
REFROTR	Specifies the reference rotor ID for the analysis. (Integer > 0; Required; no Default)
SPDUNIT	Specifies whether the entries SPDLLOW, SPDHIGH, and SPEED are given in terms of RPM (revolutions/minute) or frequency (revolutions (cycles)/unit time). (Character: 'RPM' or 'FREQ'; no Default). See Remark 2.
SPDLLOW	Specifies the low speed for synchronous analysis. See Remark 2. (Real; Default = 0.0)
SPDHIGH	Specifies the high speed for synchronous analysis. See Remark 2. (Real; Default = 99999.0)
SPEED	Specifies reference rotor speed for asynchronous analysis. Also required for static analyses. See Remark 2. (Default = 0)
ROTRSEID	Identification number of the superelement in which the rotor specified in the REFROTR field is defined. (Integer >= 0, Default = 0).
WR3WRL	Specifies whirl frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through GR field for asynchronous complex eigenvalue analysis. (Real, Default = 0.)
WR4WRL	Specifies whirl frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through material entries for asynchronous complex eigenvalue analysis. (Real, Default = 0.)
WRHWRL	Specifies whirl frequency for calculation of rotor damping and circulation terms for rotor structural hybrid damping specified through ROTHYBD card for asynchronous complex eigenvalue analysis. (Real, Default = 0.)

Remarks:

1. Multiple RGYRO entries with the same RID value are not allowed.
2. The required information on the RGYRO entries varies for different analyses. Values for the RID and SPDUNIT fields are always required. Values for SPDLOW, SPDHIGH and SPEED are analysis dependent as shown in the table below:

Solution Sequence	Type of Analysis	PARAM, GYROAVG	Required Entry	COMMENT
Frequency Response	SYNC	0	None	--
	SYNC	-1	SPDLOW, SPDHIGH	a, b
	ASYNC	0	SPEED	--
	ASYNC	-1	SPEED	b
Complex Modes	SYNC	--	SPDLOW, SPDHIGH	a, b
	ASYNC	--	SPEED	b, Remark 4.
Static Analysis	--	--	SPEED	Remark 3.

- a. The relative rotor speeds will be treated as linearly dependent on the reference rotor speed ($\Omega = A0 + A1\Omega_{\text{reference}}$). The scale factors A0 and A1 will be determined by a least-mean-square fit of the relative rotor speeds input on the RSPINR entries between SPDLOW and SPDHIGH of the reference rotor. If SPDLOW or SPDHIGH are outside the range specified on the RSPINR entry, the values will be extrapolated from the RSPINR entry values.
- b. PARAM, WR3 and PARAM, WR4 are required for rotor damping.
3. For static analysis, the SYNCFLG field must be left blank or must have the ASYNC option specified in it. Otherwise, the program terminates the execution with an appropriate fatal message.
4. When there is a Case Control request for Campbell diagram, the selected RGYRO Bulk Data entry must have the ASYNC option specified in its SYNCFLG field. Otherwise, the program terminates the execution with an appropriate fatal message.
5. In the presence of GR field, the rotor damping matrix (BR) in complex eigenvalue analysis is given by:

$$BR = KR1 \cdot GR1 / |\Omega_1(\Omega_{\text{ref}}) - WR3WRL|$$

where,

KR is the rotor stiffness matrix.

GR is the structural damping parameters specified in the RSPINR entry

$\Omega_1(\Omega_{\text{ref}})$ is the specified rotor speed.
6. If non-zero WR3WRL is specified in RGYRO, then PARAM,WR3 and WR3R (in RSPINR) cannot be specified. Appropriate FATAL message will be issued, in case these entries are specified.

RINGAX

Conical Shell Ring

Defines a ring for conical shell problems.

Format:

1	2	3	4	5	6	7	8	9	10
RINGAX	ID		R	Z			PS		

Example:

RINGAX	3		2.0	-10.0			162		
--------	---	--	-----	-------	--	--	-----	--	--

Descriptor	Meaning
ID	Ring identification number. See Remark 6. (Integer > 0)
R	Ring radius. (Real > 0.0)
Z	Ring axial location. (Real)
PS	Permanent single-point constraints. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)

Remarks:

1. RINGAX is allowed only if an AXIC entry is also present.
2. The number of degrees-of-freedom defined is $(6 - \text{NPS}) \cdot H$ where H is the harmonic count and NPS is the number of digits in field 8. (See [AXIC, 1245](#)).
3. RINGAX identification numbers must be unique with respect to all other POINTAX, RINGAX, and SECTAX identification numbers.
4. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.
5. Constraints may be necessary to avoid matrix singularities. The CONEAX element has no stiffness for rotation about the normal. In addition, there is no stiffness for rotation about V (see [Figure 9-152](#)) when transverse shear flexibility is not included.

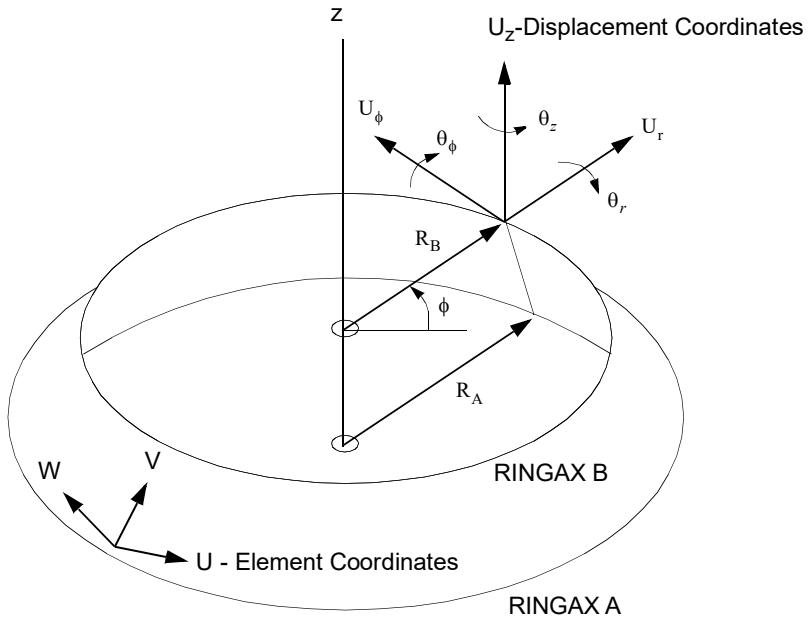


Figure 9-152 RINGAX Coordinate System

6. In order to reference this entry on a SET Case Control command, the ID must be modified by $ID(n) = ID + 1000000 \cdot n$ where n is the harmonic number plus one and ID(n) is the value specified on the SET entry.

RINGFL**Axisymmetric Fluid Point**

Defines a circle (fluid point) in an axisymmetric fluid model.

Format:

1	2	3	4	5	6	7	8	9	10
RINGFL	IDFA	XA1	XA2	XA3	IDFB	XB1	XB2	XB3	

Example:

RINGFL	3	1.0		30.0					
--------	---	-----	--	------	--	--	--	--	--

Descriptor	Meaning
IDFA, IDFB	Unique identification number of the fluid points. (0 < Integer < 500000)
XA _i , XB _i	Coordinates of the point defined in the coordinate system specified on the AXIF entry. (Real; XA1 and XB1 > 0.0)

Remarks:

1. RINGFL is allowed only if an AXIF entry is also present.
2. All fluid point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. X1, X2, X3 are (r, ϕ , z) for a cylindrical coordinate system and (ρ , θ , ϕ) for a spherical coordinate system. θ is in degrees. The value of ϕ must be blank or zero.
4. One or two fluid points may be defined per entry.

RJOINT**Rigid Joint**

Defines a rigid joint element connecting two coinciding grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RJOINT	EID	GA	GB	CB					

Example:

RJOINT	5	1	2	12345					
--------	---	---	---	-------	--	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (Integer > 0)
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB. These degrees-of-freedom are constrained to move with the same degrees-of-freedom at GA. See Remarks 4. and 5. (Integers 1 through 6 with no embedded or blank.)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CB.
3. The length between grid points GA and GB must be zero.
4. When CB = "123456" or blank, the grid point GB is constrained to move with GA and the two grid points moves as a single point. For default CB = "123456".
5. If any degree-of-freedom is released on CB, RJOINT becomes a mechanical joint element. For example, CB = "12345", then RJOINT becomes a hinge. CB = "1234", then RJOINT becomes a universal joint. And CB = "123", RJOINT becomes a spherical joint.
6. For the Lagrange method, the theory for the RJOINT is formulated such that a consistent mechanical joint is created even if the user requests different global coordinate systems at grid points GA and GB.
7. Thermal expansion effect is not applicable for the RJOINT element, since the distance between grid points GA and GB is zero.
8. Element identification numbers should be unique with respect to all other element identification numbers.

RLOAD1

Frequency Response Dynamic Excitation, Form 1

Defines a frequency-dependent dynamic load of the form

$$\{P(f)\} = \{A\}[C(f) + iD(f)]e^{i\{\theta - 2\pi f\tau\}}$$

for use in frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID	EXCITEID	DELAYI/ DELAYR	DPHASEI/ DPHASER	TC/RC	TD/RD	TYPE		

Example:

RLOAD1	5	3	2.0	10	1				
--------	---	---	-----	----	---	--	--	--	--

Descriptor	Meaning
SID	Set identification number. See Remarks 1. and 3. (Integer > 0)
EXCITEID	Identification number of a static or thermal load set or a DAREA or FBALOAD (in FRF Based Assembly or FBA process) or SPCD entry set that defines $\{A\}$. See Remarks 4. and 5. (Integer > 0)
DELAYI	Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay τ . See Remark 2. (Integer > 0 or blank)
DELAYR	Value of time delay τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
DPHASEI	Identification number DPHASE or FBAPHAS (in FRF Based Assembly or FBA process) Bulk Data entry that defines phase angle θ . (See Remark 2. (Integer > 0 or blank)
DPHASER	Value of phase angle θ (in degrees) that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
TC	Set identification number of the TABLEDi entry that gives $C(f)$. See Remark 2. (Integer > 0 or blank)
RC	Value of C to be used for all frequencies. See Remark 2.. (Real or blank)
TD	Set identification number of the TABLEDi entry that gives $D(f)$. See Remark 2. (Integer > 0 or blank)
RD	Value of D to be used for all frequencies. See Remark 2.. (Real or blank)
TYPE	Defines the type of the dynamic excitation. See Remarks 4. and 5. (Integer, character or blank; Default = 0)

Remarks:

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.
2. If any of DELAYI/DELAYR, DPHASEI/DPHASER, TC/RC, or TD/RD fields are blank or zero, the corresponding τ , θ , $C(f)$ or $D(f)$ will be zero. Either TC/RC or TD/RD may be blank or zero, but not both.
3. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
4. The type of the dynamic excitation is specified by TYPE (field 8) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS or DISP	Enforced displacement using SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration SPC/SPCD data

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 108, 111, 146, 200, and 400. For other solution sequences such as SOL118 (Cyclic Frequency Response) the large mass method must still be used.

5. TYPE (field 8) also determines the manner in which EXCITEID (field 3) is used by the program as described below:

Excitation specified by TYPE is applied load

- There is no LOADSET request in Case Control

EXCITEID may reference DAREA, FBALOAD (in FRF Based Assembly or FBA process), static and thermal load set entries.

- There is a LOADSET request in Case Control

The program may reference DAREA entries as well as static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.

Excitation specified by TYPE is enforced motion

- There is no LOADSET request in Case Control

EXCITEID will reference SPCD entries.

- There is a LOADSET request in Case Control

The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID.

6. If Modules are present then this entry may only be specified in the main Bulk Data section.

RLOAD2

Frequency Response Dynamic Excitation, Form 2

Defines a frequency-dependent dynamic excitation of the form.

$$\{P(f)\} = \{A\} \cdot B(f) e^{i\{\phi(f) + \theta - 2\pi f \tau\}}$$

for use in frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID	EXCITEID	DELAYI/ DELAYR	DPHASEI/ DPHASER	TB/RB	TP/RP	TYPE		

Example:

RLOAD2	5	3	15	5.0	7				
--------	---	---	----	-----	---	--	--	--	--

Descriptor	Meaning
SID	Set identification number. See Remarks 1. and 3. (Integer > 0)
EXCITEID	Identification number of a static or thermal load set or a DAREA or FBALOAD (in FRF Based Assembly or FBA process) or SPCD entry set that defines $\{A\}$. See Remarks 4. and 5. (Integer > 0)
DELAYI	Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay τ . See Remark 2. (Integer > 0 or blank)
DELAYR	Value of time delay τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
DPHASEI	Identification number DPHASE or FBAPHAS (in FRF Based Assembly or FBA process) Bulk Data entry that defines phase angle θ . (See Remark 2. (Integer > 0 or blank))
DPHASER	Value of phase angle θ (in degrees) that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
TB	Set identification number of the TABLEDi entry that gives $B(f)$. (Integer > 0)
RB	Value of B to be used for all frequencies. (Real, non-zero)
TP	Set identification number of the TABLEDi entry that gives $\phi(f)$ in degrees. (Integer ≥ 0)
RP	Value of ϕ to be used for all frequencies. See Remark 2. (Real or blank)
TYPE	Defines the type of the dynamic excitation. See Remarks 4. and 5. (Integer, character or blank; Defaults = 0)

Remarks:

- Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.

2. If any of DELAYI/DELAYR, DPHASEI/DPHASER, or TP/RP fields are blank or zero, the corresponding τ , θ , or $\phi(f)$ will be zero.
3. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
4. The type of the dynamic excitation is specified by TYPE (field 8) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS or DISP	Enforced displacement using SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration SPC/SPCD data

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 108, 111, 146, 200, and 400. For other solution sequences such as SOL118 (Cyclic Frequency Response) the large mass method must still be used.

5. TYPE (field 8) also determines the manner in which EXCITEID (field 3) is used by the program as described below:
 - Excitation specified by TYPE is applied load
 - There is no LOADSET request in Case Control

EXCITEID may reference DAREA, FBALOAD (in FRF Based Assembly or FBA process), static and thermal load set entries.
 - There is a LOADSET request in Case Control

The program may reference DAREA entries as well as static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.
 - Excitation specified by TYPE is enforced motion
 - There is no LOADSET request in Case Control

EXCITEID will reference SPCD entries.
 - There is a LOADSET request in Case Control

The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

ROTHYBD

Hybrid damping for rotors

Specifies hybrid damping data for rotors.

Format:

1	2	3	4	5	6	7	8	9	10
ROTHYBD	ROTORID1	HYBDAMP1	ROTORID2	HYBDAMP2	ROTORID3	HYBDAMP3	ROTORID4	HYBDAMP4	
	ROTORID5	HYBDAMP5	-etc.-						

Examples:

ROTHYBD	1	15							
ROTHYBD	10	100	20	200	30	300			

Descriptor	Meaning
ROTORIDI	Identification number of rotor. (Integer > 0). See Remarks 1 and 2.
HYBDAMPi	Identification number of a HYBDAMP entry defining hybrid modal damping data. (Integer > 0). See Remarks 1 and 2.

Remarks:

1. ROTORIDI - HYBDAMPi pair values referencing non-existent rotors are ignored.
2. If there is no HYBDAMP entry defined in the data for a HYBDAMPi specified for a valid ROTORIDI, the program terminates the execution with an appropriate fatal error.
3. Hybrid damping can result in very densely populated damping matrix causing significant performance penalty.

ROTOR**Rotor Model Definition**

Specifies list of grids, elements or properties that comprise the rotor 3D model.

Format:

1	2	3	4	5	6	7	8	9	10
ROTOR	ROTORID	FRAME							
	LTYPE	ID1	ID2	ID3	etc.				
	AXIS	GID1	GID2	etc.					

Example:

ROTOR	10	ROT							
	ELEM	10	THRU	12					
	PROP	1	THRU	5					
	AXIS	101	102						

Descriptor	Meaning
ROTORID	Identification number of rotor. (Integer > 0).
FRAME	Analysis frame (Char, ROT or FIX, Required, Default: FIX)
LTYPE	ELEM or PROP or both, indicating whether the specified list references element IDs or property IDs. (Character; Required; No default)
Idi	Note that the order is important. In case both ELEM and PROP are specified, ELEM should be specified first.
AXIS	IDs of elements or properties comprising the rotor. (Integer > 0; Required; No default)
GIDI	Defines grid points which define the axis of rotation.
	IDs of grids comprising the axis of the rotor (Integer > 0; Required; No default)

Remarks:

- Supported element types for analysis in rotating reference frame:
 - 0D elements : CONM1, CONM2
 - 1D elements : CBEAM, CBAR
 - 2D elements : CQUAD4, CQUAD8, CTRIA3, CTRIA6
 - 3D elements : CHEXA, CPENTA, CTETRA, CPYRAM
- Supported element types for analysis in fixed reference frame:
 - 0D elements : CONM1, CONM2
 - 1D elements : CBEAM, CBAR

2D elements : CQUAD4, CQUAD8

3D elements : CHEXA, CPENTA, CTETRA, CPYRAM

3. Current limitation for analysis in fixed reference frame:
 - a. If shell elements are defined perpendicular to the rotor axis for a 3D rotor defined using ROTOR entry, then gyroscopic matrix is not available for those elements. For example, a flat disk like CD spinning about the center. If the shell elements are replaced by SOLID elements, then correct gyroscopic effects can be obtained.
 - b. Bar elements (default version) do not have polar moment of inertia. Thus, if CBAR elements are defined along the axis of rotation (for both ROTOR and ROTORG entries), then they do not provide any contribution to gyroscopic matrix. The CABR elements can be replaced by CBEAM elements in case gyroscopic terms are needed for those elements.
4. THRU option is supported in ROTOR entry. Note that the order is important for LTYPE. In case both ELEM and PROP are specified, ELEM should be specified first.
5. Analysis can be performed using coupled mass or diagonal mass for all the elements.
6. ROTORAX, ROTORG and ROTORSE should not be used along with ROTOR in ROT frame. They can only be used with ROTOR in FIX frame. (since ROTORAX, ROTORG and ROTORSE assume fixed reference frame.)
7. At least 2 grid points need to be defined on AXIS to complete ROTOR definition, these points may not be part of rotor.
8. For unbalance loads the grid point, at which UNBALNC is defined, should be part of ROTOR AXIS list.
9. In order to include CONM1/2 elements as part of a rotor, its element ID should be listed using ELEM in ROTOR definition.
10. Stator portion of the model should only be defined in residual for external superelement runs in rotating system.
11. Only permanent glue option is supported for contact analysis for rotating portion of the model.

ROTORAX

Axisymmetric Model Rotor Definition

Specifies list of elements, properties or grid points that comprise an axisymmetric model rotor.

Format:

1	2	3	4	5	6	7	8	9	10
ROTORAX	ROTORID	LTYPE	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	ID8	-etc.-						

or

ROTORAX	ROTORID	LTYPE	ID1	THRU	ID2	BY	INC		
---------	---------	-------	-----	------	-----	----	-----	--	--

Example:

ROTORAX	100	ELEM	5	THRU	25				
ROTORAX	10	PROP	10						
ROTORAX	15	GRID	50	69	70				

Descriptor	Meaning
ROTORID	Identification number of rotor. (Integer > 0). See Remarks 1. and 2..
LTYPE	ELEM, PROP or GRID, indicating whether the specified list references element IDs, property IDs or grid point IDs, respectively. (Character; Required; No default). See Remarks 3. through 6..
IDi	IDs of elements, properties or grid points comprising the rotor. (Integer > 0; Required; No default)
THRU	Implies a range of identification numbers. (Optional).
BY	Implies that an increment is being specified for the THRU option (Optional)
INC	ID increment. (Integer > 0; Optional)

Remarks:

1. ROTORIDs of ROTORAX entries may not be the same as the ROTORIDs of ROTORG entries or ROTORSE entries.
2. Multiple ROTORAX entries with the same ROTORID are supported.
3. When the GRID option is used for the LTYPE field, a minimum of two grid points must be specified in the list. If this condition is not satisfied, the program terminates the execution with an appropriate user fatal message.
4. For every rotor defined by a ROTORAX entry, there should be at least one entry using the GRID option for the LTYPE field to define the axis of symmetry.

5. Grid points specified on ROTORAX entries using the GRID option for the LTYPE field must be collinear. If this condition is not satisfied, the program terminates the execution with an appropriate user fatal message.
6. Only grid points specified on ROTORAX entries using the GRID option for the LTYPE field may be referenced by RSPINR, RSPINT and UNBALNC entries.

ROTORG

Rotor Line Model Grids

Specifies grids that compose the rotor line model.

Format:

1	2	3	4	5	6	7	8	9	10
ROTORG	ROTORID	GRID1	GRID2	GRID3	GRID4	GRID5	GRID6	GRID7	
		GRID8	GRID9	-etc.-					

or

ROTORG	ROTORID	GRID1	THRU	GRID2	BY	INC			
--------	---------	-------	------	-------	----	-----	--	--	--

Example:

ROTORG	100	101	1002	103	4001				
--------	-----	-----	------	-----	------	--	--	--	--

ROTORG	200	1001	THRU	1100	BY	2			
--------	-----	------	------	------	----	---	--	--	--

Descriptor	Meaning
ROTORID	Identification number of rotor. (Integer > 0; Required). See Remarks 1. and 2..
GRIDi	Grids comprising the rotor. (Integer > 0; Required; no Default). See Remarks 3. through 7..
THRU	Specifies a range of identification numbers. (Optional)
BY	Specifies an increment for a THRU specification (Optional)
INC	Grid point number increment. (Integer)

Remarks:

1. ROTORIDs of ROTORG entries may not be the same as the ROTORIDs of ROTORAX entries or ROTORSE entries.
2. Multiple ROTORG entries with the same ROTORID are supported.
3. Grid IDs must be unique. Duplicate grid IDs will produce a fatal error.
4. All grids specified on ROTORG entries for a specific ROTORID must be collinear. Collinearity will be checked.
5. If not using superelements, no element stiffness can be connected between any GRID listed on a ROTORG entry and any GRID not listed on the ROTORG. In this case, any connections to GRIDs listed on a ROTORG must be done using MPC equations or R-elements.
6. If superelements are used, no element stiffness in the residual structure may connect between a GRID listed on a ROTORG and any GRID not listed on the ROTORG. In this case, any connections in the residual structure to GRIDs listed on a ROTORG must be done using MPC equations or R-elements.

7. All mass for any rotor defined using a ROTORG entry should be defined on the GRIDs listed on the ROTORG entry. Any mass which should belong to the rotor, but is placed on GRIDs not listed on the ROTORG entry, will be ignored when calculating the gyroscopic terms for the rotor. Only the mass shown in the ROTOR DYNAMICS MASS SUMMARY in the .f06 file is included in the calculation of the gyroscopic terms for the rotor.
8. Bar elements (default version) do not have polar moment of inertia. Thus, if CBAR elements are defined along the axis of rotation (for both ROTOR and ROTORG entries), then they do not provide any contribution to gyroscopic matrix. The CABR elements can be replaced by CBEAM elements in case gyroscopic terms are needed for those elements.

ROTORSE

Rotor Superelement Identification

Specifies grids that compose the rotor line model. An alternate to the ROTORG entry when superelements are used.

Format:

1	2	3	4	5	6	7	8	9	10
ROTORSE	ROTORID	SEID	SEOPT						

Example:

ROTORSE	10	1							
---------	----	---	--	--	--	--	--	--	--

Descriptor	Meaning
ROTORID	Identification number of rotor line model. (Integer > 0; Required). See Remark 1.
SEID	Superelement identification number of rotor superelement. (Integer > 0; Required)
SEOPT	Form of superelement for calculation of gyroscopic terms. (Integer = 1 or 2; Default = 1) See Remark 3.

Remarks:

1. ROTORIDs of ROTORSE entries may not be the same as the ROTORIDs of ROTORAX entries or ROTORG entries.
2. A ROTORSE entry can be used when the rotor is placed in a superelement.
3. The rotor line model may be the boundary of a 3D rotor superelement or the rotor line model may be a superelement itself. SEOPT is chosen to distinguish between these cases. The options are:
 - 1- If the user has a 3D model of the rotor and places it in a superelement with ID SEID, the boundary (a-set) of this superelement must consist of no more and no less than the collinear rotor line model. This will be checked. Specify SEOPT as 1 to identify this configuration. When this option is used, the A-set matrices of the superelement are used to calculate the gyroscopic terms. This option should be used for external superelements.
 - 2- If the user has a line model of the rotor and places it in a superelement with ID SEID, this superelement (g-set) may be partially or completely reduced in the SE reduction process. This superelement must consist of no more and no less than the rotor line model. Specify SEOPT as 2 to identify this configuration. When this option is used, the G-set matrices of the superelement are used to calculate the gyroscopic terms.
4. Rotors specified using the ROTORSE entry can be connected directly to the support structure. In contrast, rotors specified using the ROTORG entry must employ rigid elements to keep the rotor disconnected from the support in the G-set of the residual structure.
5. Static and component mode reduction of the rotor line model are supported when using ROTORSE entries.

RROD**Rigid Pin-Ended Element Connection**

Defines a pin-ended element that is rigid in translation.

Format:

1	2	3	4	5	6	7	8	9	10
RROD	EID	GA	GB	CMA	CMB	ALPHA			

Example:

RROD	14	1	2	2		6.5-6			
------	----	---	---	---	--	-------	--	--	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB	Grid point identification numbers of connection points. (Integer > 0)
CMA, CMB	Component number of one and only one dependent translational degree-of-freedom in the global coordinate system assigned by the user to either GA or GB. (Integer 1, 2, or 3. Either CMA or CMB must contain the integer, and the other must be blank for the linear RROD. For Lagrange RROD, both CMA and CMB can be blank.) See Remark 3.
ALPHA	Thermal expansion coefficient. See Remark 11. (Real or blank)

Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally one Lagrange multiplier degree-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points.
- For the Lagrange method, if both CMA and CMB are blanks, Nastran will compute the best degree-of-freedom for the dependent degree-of-freedom.
- The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
- Element identification numbers should be unique with respect to all other element identification numbers.
- RROD, among other eligible rigid element types, can be selected via MPC and SET3.
- Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
- Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.

9. The degree-of-freedom selected to be dependent must have a nonzero component along the axis of the element. This implies that the element must have finite length.
10. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
11. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.

There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RROD.

RSPINR**Relative Spin Rates Between Rotors**

Specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis and rotor damping parameters.

Format:

1	2	3	4	5	6	7	8	9	10
RSPINR	ROTORID	GRIDA	GRIDB	SPDUNT	SPTID	ROTRSEID			
	GR	ALPHAR1	ALPHAR2	WR3R	WR4R	WRHR			

Example:

1	2	3	4	5	6	7	8	9	10
RSPINR	100	1001	1002	RPM	100	5			
	0.02	1	3	1000.	1500.	1200.			

Descriptor	Meaning
ROTORID	Identification number of rotor. (Integer > 0; Required). See Remark 1.
GRIDA/GRIDB	Positive rotor spin direction is defined from GRIDA to GRIDB. See Remark 4. (Integer > 0; Required)
SPDUNIT	Specifies whether the listing of relative spin rates is given in terms of RPM (revolutions/minute) or frequency (revolutions (cycles)/sec). (Character; 'RPM' or 'FREQ'; Required)
SPTID	Table for relative rotor spin rates. See Remark 5. (Real or Integer, if integer, must be > 0; Required)
GR	Rotor structural damping factor. See Remarks 6. and 8. (Real; Default = 0.0)
ALPHAR1	Scale factor applied to the rotor mass matrix for Rayleigh damping. See Remarks 7. and 8. (Real; Default = 0.0)
ALPHAR2	Scale factor applied to the rotor stiffness matrix for Rayleigh damping. See Remarks 7. and 8. (Real; Default = 0.0)
ROTRSEID	Identification number of the superelement in which the rotor specified in the ROTORID field is defined. (Integer >= 0, Default = 0). See Remark 1.
WR3R	Specifies "average" excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through GR field. (Real, Default = 0.)

Descriptor	Meaning
WR4R	Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through material entries. (Real, Default = 0.)
WRHR	Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural hybrid damping specified through ROTHYBD card. (Real, Default = 0.)

Remarks:

1. The ROTORID-ROTRSEID pair must be unique across all RSPINR entries, but the ROTORID and ROTRSEID fields individually need not be unique.
2. Depending upon the type of analysis being performed, a RSPINR/RSPINT entry should be present for each rotor defined by a ROTORG or ROTORAX entry whose gyroscopic effects are to be included in the analysis. If missing for a rotor, then that rotor is assumed to be stationary and the analysis proceeds accordingly. The absence of a RSPINR/RSPINT entry for a rotor is noted in the .f06 file via an appropriate user information message.
3. RSPINR/RSPINT entries are honored and processed only in residual or assembly jobs and are ignored in external superelement creation jobs. For rotor defined in external SE, the GRIDA and GRIDB specified in RSPINR must be part of the SEs ASET during creation run. This requirement is relaxed for rotors defined in PART SE.
4. The rotor spin axis is determined from the ROTORG or ROTORAX entries. The positive rotation vector is from GRIDA to GRIDB. GRIDA and GRIDB must be specified on the ROTORG or ROTORAX entry.
If ROTRSEID refers to a secondary external superelement, then GRIDA and GRIDB must be points associated with the ROTORID of the primary external superelement since the secondary external superelement derives all of its properties from the primary external superelement.
5. If SPTID is a real number, the value is considered constant. If SPTID is an integer number, the value references a DDVAL entry that specifies the relative rotor spin rates. The number of spin rates for each rotor must be the same. Relative spin rates are determined by correlation of table entries. The ith entry for each rotor specifies the relative spin rates between rotors at RPMi/FREQi. Spin rates for the reference rotor must be in ascending or descending order.
6. Rotor structural damping specified by the GR entry will be added as equivalent viscous damping or structural damping depending on the solution. That is,

$$[B_{rotor}]_{structural} = \left(\frac{GR}{WR3} \right) [K_{rotor}]$$

where WR3 is a user parameter, or

$$[K_{rotor}] = (1 + iGR)[K_{rotor}]$$

depending on the solution sequence, SYNC/ASYNC and value of PARAM, GYROAVG. See Remark for all the damping and circulation terms added to the equation in the different cases.

In case WR3R is specified in RSPINR, then WR3R is used for determining equivalent viscous damping for this particular rotor instead of WR3.

7. Rayleigh damping for the rotor will be calculated as

$$[B_{rotor}]_{Rayleigh} = \alpha_{R1}(M_{rotor}) + \alpha_{R2}[K_{rotor}]$$

8. The various rotor damping and circulation matrices applicable to frequency response and complex eigenvalue solutions are shown in the following table.

Solution	Damping	Circulation
Frequency Response - ASYNC option	$i\omega([B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R]) + i(GR[K_R] + [K4_R] + [KH_R])$	$\Omega_R(\Omega_{ref}) \begin{pmatrix} [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{\omega}\right)[K_R^C] + \left(\frac{1}{\omega}\right)[K4_R^C] + \left(\frac{1}{\omega}\right)[KH_R^C] \end{pmatrix}$
Frequency Response - ASYNC option w/ PARAM,GYROAVG,-1	$i\omega \begin{pmatrix} [B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \\ + \left(\frac{GR}{WR3}\right)[K_R] + \left(\frac{1}{WR4}\right)[K4_R] \\ + \left(\frac{1}{WRH}\right)[KH_R] \end{pmatrix}$	$\Omega_R(\Omega_{ref}) \begin{pmatrix} [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] \\ + \left(\frac{1}{WRH}\right)[KH_R^C] \end{pmatrix}$
Frequency Response - SYNC option	$i\omega([B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R]) + i(GR[K_R] + [K4_R] + [KH_R])$	$\Omega_R(\omega) \begin{pmatrix} [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{\omega}\right)[K_R^C] + \left(\frac{1}{\omega}\right)[K4_R^C] \\ + \left(\frac{1}{\omega}\right)[KH_R^C] \end{pmatrix}$

Solution	Damping	Circulation
Frequency Response - SYNC option w/ PARAM,GYROAVG,-1	$i\omega \left([B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \right) + \left(\frac{GR}{WR3} \right) [K_R] + \left(\frac{1}{WR4} \right) [K4_R] + \left(\frac{1}{WRH} \right) [KH_R]$	$\omega \beta_R \left([B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \right) + \left(\frac{GR}{WR3} \right) [K_R^C] + \left(\frac{1}{WR4} \right) [K4_R^C] + \left(\frac{1}{WRH} \right) [KH_R^C]$ $+ \alpha_R \left([B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \right) + \left(\frac{GR}{WR3} \right) [K_R^C] + \left(\frac{1}{WR4} \right) [K4_R^C] + \left(\frac{1}{WRH} \right) [KH_R^C]$
Complex Modes - ASYNC option	$i\omega \left([B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \right) + \left(\frac{GR}{WR3} \right) [K_R] + \left(\frac{1}{WR4} \right) [K4_R] + \left(\frac{1}{WRH} \right) [KH_R]$	$\Omega_R(\Omega_{ref}) \left([B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \right) + \left(\frac{GR}{WR3} \right) [K_R^C] + \left(\frac{1}{WR4} \right) [K4_R^C] + \left(\frac{1}{WRH} \right) [KH_R^C]$
Complex Modes - SYNC option	$i\omega \left([B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \right) + \left(\frac{GR}{WR3} \right) [K_R] + \left(\frac{1}{WR4} \right) [K4_R] + \left(\frac{1}{WRH} \right) [KH_R]$	$\omega \beta_R \left([B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \right) + \left(\frac{GR}{WR3} \right) [K_R^C] + \left(\frac{1}{WR4} \right) [K4_R^C] + \left(\frac{1}{WRH} \right) [KH_R^C]$ $+ \alpha_R \left([B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \right) + \left(\frac{GR}{WR3} \right) [K_R^C] + \left(\frac{1}{WR4} \right) [K4_R^C] + \left(\frac{1}{WRH} \right) [KH_R^C]$

where:

- $[B_R]$ = the rotor viscous damping
- $[M_R]$ = the rotor mass
- $[K_R]$ = the rotor stiffness
- $[K4_R]$ = the rotor material damping
- $[BH_R]$ = the rotor viscous hybrid damping
- $[KH_R]$ = the rotor structural hybrid damping
- $[B_R^C]$ = the circulation due to rotor viscous damping
- M_R^C = the circulation due to rotor 'mass'
- K_R^C = the circulation due to rotor structural 'stiffness'
- $[K4_R^C]$ = the circulation due to rotor material damping
- $[BH_R^C]$ = the circulation due to rotor viscous hybrid damping
- $[KH_R^C]$ = the circulation due to rotor structural hybrid damping
- $\alpha 1_R, \alpha 2_R$ = used to specify Rayleigh viscous damping
 $([B_R]_{Rayleigh} = \alpha 1_R[M_R] + \alpha 2_R[K_R])$

- α_R, β_R = scale factors of linear fit of rotor speed to reference rotor speed. The linear fit is calculated between the SPDLOW and SPDHIGH speeds (values specified on RGYRO entry) of the reference rotor.
- $WR3, WR4, WRH$ = User parameters specified by PARAM statement. If the parameter values are zero (Default), the corresponding damping and circulation terms are not added.
- a. Parameter and hybrid damping applied to the rotor does not apply to the support and vice versa.
- b. For hybrid damping of the rotors, only the rotor mass and stiffness are used for the modes calculation.
- c. All rotor damping terms are cumulative. Multiple damping options should be selected with caution.

RSPINT**Rotor Spin Rates**

Specifies rotor spin rates for transient analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RSPINT	ROTORID	GRIDA	GRIDB	SPDUNT	SPTID	SPDOUT	ROTSEID		
	GR	ALPHAR1	ALPHAR2	WR3R	WR4R	WRHR			

Example:

1	2	3	4	5	6	7	8	9	10
RSPINT	100	1001	1002	RPM	1001				
	0.01	0.01	0.002	1000.	1500.	1200.			

Descriptor	Meaning
ROTORID	Identification number of rotor. (Integer > 0; Required). See Remark 1.
GRIDA/GRIDB	Positive rotor spin direction is defined from GRIDA to GRIDB. See Remark 4. (Integer > 0; Required)
SPDUNIT	Specifies whether the spin rates are given in terms of RPM (revolutions/minute) or frequency (revolutions(cycles)/sec). (Character; 'RPM' or 'FREQ'; Required)
SPTID	Rotor spin rate. See Remark 5. (Integer > 0; Required)
SPDOUT	EPOINT to output the rotor speed vs. time. Output will be in SPDUNITs (Integer > 0 or blank)
GR	Rotor structural damping factor. See Remark 6. and 8. (Real; Default = 0.0)
ALPHAR1	Scale factor applied to the rotor mass matrix for Rayleigh damping. See Remark 7. and 8. (Real; Default = 0.0)
ALPHAR2	Scale factor applied to the rotor stiffness matrix for Rayleigh damping. See Remark 7. and 8. (Real; Default = 0.0)
ROTRSEID	Identification number of the superelement in which the rotor specified in the ROTORID field is defined. (Integer >= 0, Default = 0). See Remark 1.
WR3R	Specifies "average" excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through GR field. (Real, Default = 0.)

Descriptor	Meaning
WR4R	Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through material entries. (Real, Default = 0.)
WRHR	Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural hybrid damping specified through ROTHYBD card. (Real, Default = 0.)

Remarks:

1. The ROTORID-ROTRSEID pair must be unique across all RSPINT entries, but the ROTORID and ROTRSEID fields individually need not be unique.
2. Depending upon the type of analysis being performed, a RSPINR/RSPINT entry should be present for each rotor defined by a ROTORG or ROTORAX entry whose gyroscopic effects are to be included in the analysis. If missing for a rotor, then that rotor is assumed to be stationary and the analysis proceeds accordingly. The absence of a RSPINR/RSPINT entry for a rotor is noted in the .f06 file via an appropriate user information message.
3. RSPINR/RSPINT entries are honored and processed only in residual or assembly jobs and are ignored in external superelement creation jobs. For rotor defined in external SE, the GRIDA and GRIDB specified in RSPINT must be part of the SEs ASET during creation. This requirement is relaxed for rotors defined in PART SE.
4. The rotor spin axis is determined from the ROTORG or ROTORAX entries. The positive rotation vector is from GRIDA to GRIDB. GRIDA and GRIDB must be specified on the ROTORG or ROTORAX entry.
If ROTRSEID refers to a secondary external superelement, then GRIDA and GRIDB must be points associated with the ROTORID of the primary external superelement since the secondary external superelement derives all of its properties from the primary external superelement.
5. SPTID references a TABLED1 entry that specifies the rotor spin rate history.
6. Rotor structural damping specified by the GR entry will be added as equivalent viscous damping. The equivalent damping will be calculated using:

$$[B_{rotor}]_{structural} = \left(\frac{GR}{WR3} \right) [K_{rotor}]$$

where WR3 is a user parameter.

In case WR3R is specified in RSPINT, then WR3R is used for determining equivalent viscous damping for this particular rotor instead of WR3.

7. Rayleigh damping for the rotor will be calculated as
$$[B_{rotor}]_{Rayleigh} = \alpha_{R1}(M_{rotor}) + \alpha_{R2}[K_{rotor}]$$
8. The various rotor damping and circulation matrices used in transient analysis are shown in the following table.

Damping	Circulation (added to stiffness)
$[B_R] + [BH_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R]$ $+ \left(\frac{GR}{WR3}\right)[K_R] + \left(\frac{1}{WR4}\right)[K4_R]$ $+ \left(\frac{1}{WRH}\right)[KH_R]$	$\Omega_R(t) \begin{pmatrix} [B_R^C] + [BH_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] \\ + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] \\ + \left(\frac{1}{WRH}\right)[KH_R^C] \end{pmatrix}$

where:

$[B_R]$	= the rotor viscous damping
$[M_R]$	= the rotor mass
$[K_R]$	= the rotor stiffness
$[K4_R]$	= the rotor material damping
$[BH_R]$	= the rotor viscous hybrid damping
$[KH_R]$	= the rotor structural hybrid damping
$[B_R^C]$	= the circulation due to rotor viscous damping
M_R^C	= the circulation due to rotor 'mass'
K_R^C	= the circulation due to rotor structural 'stiffness'
$[K4_R^C]$	= the circulation due to rotor material damping
$[BH_R^C]$	= the circulation due to rotor viscous hybrid damping
$[KH_R^C]$	= the circulation due to rotor structural hybrid damping
$\alpha 1_R, \alpha 2_R$	= used to specify Rayleigh viscous damping $([B_R]_{Rayleigh} = \alpha 1_R[M_R] + \alpha 2_R[K_R])$
α_R, β_R	= scale factors of linear fit of rotor speed to reference rotor speed. The linear fit is calculated between the SPDLOW and SPDHIGH speeds (values specified on RGYRO entry) of the reference rotor.
$WR3, WR4, WRH$	= User parameters specified by PARAM statement. If the parameter values are zero (Default), the corresponding damping and circulation terms are not added.

- a. Parameter and hybrid damping applied to the rotor does not apply to the support and vice versa.
- b. For hybrid damping of the rotors, only the rotor mass and stiffness are used for the modes calculation.

- c. All rotor damping terms are cumulative. Multiple damping options should be selected with caution.

RSPLINE**Interpolation Constraint Element**

Defines multipoint constraints for the interpolation of displacements at grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RSPLINE	EID	D/L	G1	G2	C2	G3	C3	G4	
	C4	G5	C5	G6	-etc.-				

Example:

RSPLINE	73	.05	27	28	123456	29		30	
	123	75	123	71					

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
D/L	Ratio of the diameter of the elastic tube to the sum of the lengths of all segments. (Real > 0.0; Default = 0.1)
Gi	Grid point identification number. (Integer > 0)
Ci	Components to be constrained. See Remark 2. (Blank or any combination of the Integers 1 through 6.)

Remarks:

1. Displacements are interpolated from the equations of an elastic beam passing through the grid points. This is a linear method only element, and not controlled with the Case Control command RIGID.
2. A blank field for Ci indicates that all six degrees-of-freedom at Gi are independent. Since G1 must be independent, no field is provided for C1. Since the last grid point must also be independent, the last field must be a Gi, not a Ci. For the example shown G1, G3, and G6 are independent. G2 has six constrained degrees-of-freedom while G4 and G5 each have three.
3. Dependent (i.e., constrained) degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
4. Degrees-of-freedom declared to be independent by one rigid body element can be made dependent by another rigid body element or by a multipoint constraint.
5. EIDs must be unique.
6. RSPLINE, among other eligible rigid element types, can be selected via MPC and SET3.
7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer subsstep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
9. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
10. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
11. The constraint coefficient matrix is affected by the order of the Gi Ci pairs on the RSPLINE entry. The order of the pairs should be specified in the same order that they appear along the line that joins the two regions. If this order is not followed then the RSPLINE will have folds in it that may yield some unexpected interpolation results.
12. The independent degrees-of-freedom that are the rotation components most nearly parallel to the line joining the regions should not normally be constrained.
13. The RSPLINE has a limit of 100 grid points.

RSSCON

Shell-to-Solid Element Connector

Defines multipoint constraints to model clamped connections of shell-to-solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
RSSCON	RBID	TYPE	ES1	EA1	EB1	ES2	EA2	EB2	

Examples:

RSSCON	110	GRID	11	12	13	14	15	16	
RSSCON	111	GRID	31	74	75				
RSSCON	115	ELEM	311	741					

Descriptor	Meaning
RBID	Element identification number. (0 < Integer < 100,000,000)
TYPE	Type of connectivity: ELEM Connection is described with element identification numbers. GRID Connection is described with grid point identification numbers. (Character: “GRID” or “ELEM”; Default = “ELEM”)
ES1	Shell element identification number if TYPE = “ELEM”. Shell grid point identification number if TYPE = “GRID”. See Figure 9-153 . (Integer > 0)
EA1	Solid element identification number if TYPE = “ELEM”. Solid grid point identification number if TYPE = “GRID”. (Integer > 0)
EB1	Solid grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)
ES2	Shell grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)
EA2	Solid grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)
EB2	Solid grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)

Remarks:

1. RSSCON generates a multipoint constraint that models a clamped connection between a shell and a solid element. The shell degrees-of-freedom are put in the dependent set (m-set). The translational degrees-of-freedom of the shell edge are connected to the translational degrees-of-freedom of the upper and lower solid edge. The two rotational degrees-of-freedom of the shell are connected to the translational degrees-of-freedom of the lower and upper edges of the solid element face. Poisson’s ratio effects are considered in the translational degrees-of-freedom.

2. The shell grid point must lie on the line connecting the two solid grid points. It can have an offset from this line, which can not be more than 5% of the distance between the two solid grid points. The shell grid points that are out of the tolerance will not be constrained, and a fatal message will be issued. This tolerance is adjustable. Please see PARAM,TOLRSC and PARAM,SEPIXOVR.
3. When using the TYPE = "ELEM" option
 - The elements may be p-elements or h-elements. The solid elements are CHEXA, CPENTA, and CTETRA with and without midside nodes. The shell elements are CQUAD4, CTRIA3, CQUADR, CTRIAR, CQUAD8, or CTRIA6. If the solid element has mid-side nodes, then the shell element needs mid-side nodes on the common edge.
 - In case of p-elements, the p-value of the shell element edge is adjusted to the higher of the p-value of the upper or lower solid p-element edge. If one of the elements is an h-element, then the p-value of the adjacent edge is lowered to 1.
 - Both the shell and solid elements have to belong to the same superelement. This restriction can be bypassed using SEELT entry to reassign the downstream boundary element to an upstream superelement.
 - When a straight shell p-element edge and a solid p-element are connected, the geometry of the shell edge is not changed to fit the solid face. When a curved shell p-element edge and a solid p-element are connected, the two solid edges and solid face are not changed to match the shell edge.
 - It is not recommended to connect more than one shell element to the same solid using the ELEM option. If attempted, conflicts in the multipoint constraint relations may lead to UFM 6692.
4. When using TYPE = "GRID" option
 - The GRID option does not verify that the grids used are valid shell and/or solid grids.
 - The hierarchical degrees-of-freedom of p-element edges are not constrained. The GRID option is therefore not recommended for p-elements.
 - The grids in the GRID option can be in different superelements. The shell grid must be in the upstream superelement.
5. It is recommended that the height of the solid element's face is approximately equal to the shell element's thickness of the shell. The shell edge should then be placed in the middle of the solid face.
6. The shell edge may coincide with the upper or lower edge of the solid face.
7. RSSCON, among other eligible rigid element types, can be selected via MPC and SET3.
8. Forces of multipoint constraints may be recovered with the MPCFORCE Case Control command.
9. The RSSCON is ignored in heat-transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
10. The m-set coordinates (shell degrees-of-freedom) may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

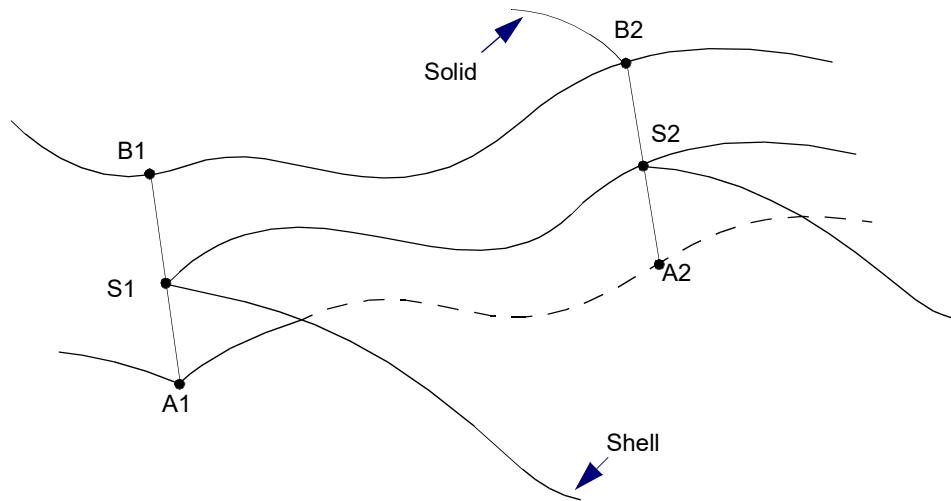


Figure 9-153 Shell Elements Connected to the Faces of Solid Elements

RTRPLT**Rigid Triangular Plate**

Defines a rigid triangular plate.

Format:

1	2	3	4	5	6	7	8	9	10
RTRPLT	EID	GA	GB	GC	CNA	CNB	CNC		
	CMA	CMB	CMC	ALPHA					

Example:

RTRPLT	7	1	2	3	1236	3	3		
--------	---	---	---	---	------	---	---	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB, GC	Grid point identification number of connection points.
CNA, CNB, CNC	Independent degrees-of-freedom in the global coordinate system for the element at grid points GA, GB, and GC, indicated by any of the Integers 1 through 6 with no embedded blanks. See Remark 3. (Integer ≥ 0 or blank)
CMA, CMB, CMC	Component numbers of dependent degrees-of-freedom in the global coordinate system. (Any of the Integers 1 through 6 with no embedded blanks, or 0 or blank.)
ALPHA	Thermal expansion coefficient. See Remark 12. (Real or blank)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 18 displacement degrees-of-freedom given by grid points GA, GB, and GC. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.
3. For the linear method, the total number of components in CNA, CNB, and CNC must equal six; for example, CNA = 1236, CNB = 3, CNC = 3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. For the Lagrange method, the total number of components must also be six. However, only CNA= 123456 or CNB = 123456 or CNC = 123456 is allowed. For this type of element, RTRPLT1 gives a simpler input format.
4. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.

5. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
6. Element identification numbers should be unique with respect to all other element identification numbers.
7. RTRPLT, among other eligible rigid element types, can be selected via MPC and SET3.
8. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
9. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
10. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
11. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
12. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is defined as the following, the bar GA-GB will have the average temperature of grid points GA and GB. The bar GA-GC will have the average temperature of the grid points GA and GC.
There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RTRPLT.

RTRPLT1**Rigid Triangular Plate (Alternative Format)**

Alternative format to define a rigid triangular plate element connecting three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RTRPLT1	EID	GA	GB	GC	CMB	CMC	ALPHA		

Example:

RTRPLT1	7	1	2	3	1236	3	6.0-6		
---------	---	---	---	---	------	---	-------	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB, GC	Grid point identification number of connection points. (Integer > 0)
CMB, CMC	Component numbers at GB and GC in the global coordinate systems, which are constrained to move with the rigid body. See Remark 4. (Integers 1 through 6 with no embedded blanks or blank.)
ALPHA	Thermal expansion coefficient. See Remark 9. (Real or blank)

Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 18 displacement degrees-of-freedom given by grid points GA, GB, and GC. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.
- RTRPLT1 is a preferred input format for the Lagrange method.
- When CMB = “123456” or blank, CMC = “123456” or blank, the grid points GB and BC are constrained to move with GA as a rigid triangular plate. For default, CMB = “123456” and CMC = “123456”. Any number of degrees-of-freedom at grid points GB and GC can be released not to move with the rigid body.
- The length of any two connected grid points must be greater than zero.
- For the Lagrange method, the theory is formulated such that a consistent rigid body motion for grid points GA, GB, and GC will be computed even if these three points have different global coordinate systems.

7. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The bar GA-GB will have the average temperature of grid points GA and GB. The bar GA-GC will have the average temperature of the grid points GA and GC.
There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RTRPLT1.
8. Element identification numbers should be unique with respect to all other element identification numbers.
9. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.

RVDOF

Degrees-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are to be applied to obtain static solutions for use in residual vector computations.

Format:

1	2	3	4	5	6	7	8	9	10
RVDOF	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

RVDOF	800	1	850	2					
-------	-----	---	-----	---	--	--	--	--	--

Descriptor	Meaning
IDi	Grid or scalar identification number. (Integer > 0)
Ci	Component numbers. (Any one of the integers 1 through 6 for grid points and integer zero or blank for scalar points)

Remarks:

1. In multiple superelement analysis, the IDi points may be interior to any superelement. The program automatically partitions the data for allocation to the appropriate superelements. Separate entries for separate superelements are not required as in the case of USETi,U6 and SEUSETi,U6 entries.
2. The unit loads applied to the interior points of a superelement due to the RVDOF/RVDOF1 entries are passed downstream all the way down to the residual for the purpose of residual vector processing by all superelements in its downstream path, resulting in more accurate results. This is in contrast to the way residual vector processing is performed when USETi,U6 or SEUSETi,U6 entries are employed. In the latter case, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

RVDOF1

Degrees-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are to be applied to obtain static solutions for use in residual vector computations.

Format:

1	2	3	4	5	6	7	8	9	10
RVDOF1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Example:

RVDOF1	3	100	210	450					
--------	---	-----	-----	-----	--	--	--	--	--

Alternate Format and Example:

RVDOF1	C	ID1	"THRU"	ID2					
RVDOF1	12	6	THRU	21					

Descriptor	Meaning
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0 or blank for scalar points.)
IDi	Grid or scalar identification numbers. (Integer > 0 or "THRU"; for "THRU" option, ID1 < ID2)

Remarks:

1. In multiple superelement analysis, the IDi points may be interior to any superelement. The program automatically partitions the data for allocation to the appropriate superelements. Separate entries for separate superelements are not required as in the case of USETi,U6 and SEUSETi,U6 entries.
2. The unit loads applied to the interior points of a superelement due to the RVDOF/RVDOF1 entries are passed downstream all the way down to the residual for the purpose of residual vector processing by all superelements in its downstream path, resulting in more accurate results. This is in contrast to the way residual vector processing is performed when USETi,U6 or SEUSETi,U6 entries are employed. In the latter case, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.
3. If the alternate format is used, points in the sequence ID1 through ID2 are not required to exist. Points that do not exist will collectively produce a warning message but will otherwise be ignored.
4. When alternative format is used, single point IDs cannot be specified.

SANGLE

Analytical contact threshold angle in SOL 600

Defines automatic analytical contact threshold angle for multiple subcases in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
SANGLE	IDC	IDB	Angle	IDC	IDB	Angle			

Example:

SANGLE	1	4	50.0	1	6				
	2	4	-1.0	2	6	55.0			

Descriptor	Meaning
IDC	Identification number of a SUBCASE Case Control command. (Integer; no Default) To enter a value corresponding to Marc's increment zero, set IDC=0.
IDB	Identification of a contact body (must be the same as a BCBODY ID) (Integer; no Default)
Angle	Threshold automatic analytical contact angle (SANGLE). (Real; Default = 60.0) A value of -1.0 turns off analytical.

Remarks:

1. This entry should only be made if IDSPL=1 and if SANGLE is a non-zero integer value on one or more BCBODY entry.
2. This entry is available in SOL 600 only.
3. For the example, BCBODY with id=4 has a threshold angle of 50.0 degrees in subcase 1 and analytical contact is turned off in subcase 2. For bcbbody=6, the analytical contact is on for subcaes 1 and 2 and the threshold angle is 60.0 degrees (the default) and 55.0 degrees for subcases 1 and 2 respectively.
4. Only those contact bodies whose SANGLE changes from subcase to subcase or is turned on/off need be described here. Those with constant SANGLE may be described on the BCBODY entry.

SEBNDRY

Superelement Boundary-Point Definition

Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.

Format:

1	2	3	4	5	6	7	8	9	10
SEBNDRY	SEIDA	SEIDB	GIDA1	GIDA2	GIDA3	GIDA4	GIDA5	GIDA6	
	GIDA7	GIDA8	-etc.-						

Example 1:

SEBNDRY	400	4	10	20	30	40			
---------	-----	---	----	----	----	----	--	--	--

Example 2:

SEBNDRY	400	ALL	10	20	30	THRU	40		
---------	-----	-----	----	----	----	------	----	--	--

Descriptor	Meaning
SEIDA	Partitioned superelement identification number. (Integer > 0)
SEIDB	Superelement identification. See Remark 2. (Integer > 0 or Character “ALL”; Default = “ALL”)
GIDAi	Identification number of a boundary grid point in superelement SEIDA. (Integer > 0 or “THRU”; For “THRU” option, G1 < G2.)

Remarks:

1. SEBNDRY may only be specified in the main Bulk Data Section and is not recognized after the BEGIN SUPER=n.
2. SEIDB may reference partitioned superelements or superelements in the main Bulk Data Section.

SEBSET

Fixed Boundary Degree-of-Freedom

Defines boundary degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
SEBSET	SEID	ID1	C1	ID2	C2	ID3	C3		

Example:

SEBSET	5	2	135	14	6				
--------	---	---	-----	----	---	--	--	--	--

Descriptor	Meaning
SEID	Superelement identification number. (Integer > 0)
Ci	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer zero or blank for scalar points)
IDi	Grid or scalar point identification numbers. (Integer > 0)

Remarks:

1. If there are no SECSETi or SEBSETi entries present, all boundary points are, by default, fixed during component mode analysis. If only SEBSETi are entries present, any boundary degrees-of-freedom not listed are placed in the free boundary set (c-set). If both SEBSETi and SECSETi entries are present, the c-set degrees-of-freedom are defined by the SECSETi entries and any remaining boundary points are placed in the b-set.
2. Degrees-of-freedom listed on SEBSETi entries must be exterior degrees-of-freedom of the superelement and may not be specified on SECSETi entries.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SEBSET1

Fixed Boundary Degree-of-Freedom, Alternate Form of SEBSET

Defines boundary degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode calculations.

Format:

1	2	3	4	5	6	7	8	9	10
SEBSET1	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example:

SEBSET1	5	2	135	14	6	23	24	25	
	122	127							

Alternate Format and Example:

SEBSET1	SEID	C	G1	"THRU"	G2				
SEBSET1	5	3	6	THRU	32				

Descriptor	Meaning
SEID	Superelement identification number. (Integer > 0)
C	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points, 0 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. (Integer > 0 or "THRU"; for THRU option G1 < G2.)

Remarks:

1. If there are no SECSETi or SEBSETi entries present, all boundary points are, by default, fixed during component mode analysis. If there are only SEBSETi entries present, any boundary degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both SEBSETi and SECSETi entries present, the c-set degrees-of-freedom are defined by the SECSETi entries, and any remaining boundary points are placed in the b-set.
2. Degrees-of-freedom listed on SEBSETi entries must be exterior degrees-of-freedom of the superelement and may not be specified on SECSETi entries.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SEBULK

Partition Superelement Connection

Defines superelement boundary search options and a repeated, mirrored, or collector superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEBULK	SEID	TYPE	RSEID	METHOD	TOL	LOC	UNITNO		

Example:

SEBULK	14	REPEAT	4	AUTO	1.0E-3				
--------	----	--------	---	------	--------	--	--	--	--

Descriptor	Meaning
SEID	Partitioned superelement identification number. (Integer > 0). See Remark 11.
TYPE	Superelement type. (Character; no Default)
	PRIMARY Primary. See Remarks 2. and 5.
	COLLCTR Collector. See Remark 3.
	EXTERNAL External. See Remarks 4. and 5.
	EXTOP2 External using an OUTPUT2 file created in an earlier run. See Remarks 4. and 5.
	EXTOP4 External using an OUTPUT4 file created in an earlier run. See Remarks 4. and 5.
	REPEAT Identical. See Remark 6.
	MIRROR Mirror. See Remark 6.
RSEID	Identification number of the reference superelement, used only if TYPE = "REPEAT" or "MIRROR". (Integer \geq 0; Default = 0). See Remark 6.
METHOD	Method to be used when searching for boundary grid points. (Character: "AUTO" or "MANUAL"; Default = "AUTO"). See Remarks 7. and 8.
TOL	Location tolerance to be used when searching for boundary grid points. (Real; Default = 10E-5). See Remarks 10. and 11.
LOC	Coincident location check option for manual connection option. (Character: "YES" or "NO"; Default = "YES"). See Remark 10.
UNITNO	FORTRAN unit number for the OUTPUT2 or OUTPUT4 file (applicable and meaningful only when TYPE = "EXTOP2" or "EXTOP4"). See Remarks 4. and 5.

Remarks:

1. This specification must be specified in the MAIN Bulk Data and is meaningful only if part superelements (using BEGIN SUPER) or external superelements created by employing the EXTSEOUT Case Control command exist in the data.

2. TYPE = PRIMARY indicates that the superelement is a part superelement (using BEGIN SUPER).
 3. TYPE = "COLLCTR" indicates a collector superelement which does not contain any grids or scalar points.
 4. TYPE = "EXTERNAL", "EXTOP2" or "EXTOP4" indicates an external superelement created by employing the EXTSEOUT Case Control command in an earlier job. See discussion under the description of the EXTSEOUT Case Control command. (For employing external superelements using the old three-step procedure, see discussion under the description of parameter EXTDROUT in Chapter 5.)
 5. A superelement whose TYPE is "PRIMARY", "EXTERNAL" or "EXTOP2" may itself be re-positioned by the use of an SELOC or an SEMPLN entry or both.

If it references an SELOC entry, then the primary superelement will be positioned at the location implied by the SELOC entry.

If it references an SEMPLN entry, then a mirror image of the primary superelement will be positioned using the plane defined by the SEMPLN entry.

If it references both an SELOC entry and an SEMPLN entry, then a mirror image of the primary superelement will first be created using the plane defined by the SEMPLN entry and then it will be positioned at the location implied by the SELOC entry.
 6. TYPE = "REPEAT" and TYPE = "MIRROR" have many common features, but they also have some important differences. These are described below.
- The following comments apply to both TYPE= "REPEAT" and TYPE = "MIRROR":
- a. An SEID whose TYPE is "REPEAT" or "MIRROR" is referred to as a secondary superelement. The RSEID specified in this case is regarded as the primary superelement.
 - b. The primary superelement must be either a part superelement or an external superelement created by employing the EXTSEOUT Case Control command in an earlier job. It must be defined via its own SEBULK entry wherein TYPE must be either "PRIMARY", "EXTERNAL" or "EXTOP2".
 - c. The primary superelement does not include superelements upstream of the primary superelement.
 - d. The primary superelement may be re-positioned as indicated in Remark 5.
 - e. If the primary superelement specified by RSEID is a part superelement, then the secondary superelement is a "G-set" copy of the primary superelement. In this case, the boundary, loads, constraints and reduction procedure of the secondary superelement can be different from those of its primary superelement.
 - f. If the primary superelement specified by RSEID is an external superelement resulting from the use of the EXTSEOUT Case Control command in an earlier job, then the secondary superelement is an "A-set" copy of the primary superelement. In this case, the boundary, loads, constraints and reduction procedure of the secondary superelement are set and are the same as those of its primary superelement.

The following comments apply only to TYPE= "REPEAT":

- g. The secondary superelement in this case may reference an SELOC entry, an SEMPLN entry or both or none.

If it references an SELOC entry, then an identical copy of its primary superelement will be positioned at the location implied by the SELOC entry.

If it references an SEMPLN entry, then a mirror image copy of the primary superelement will be positioned using the plane defined by the SEMPLN entry.

If it references both an SELOC entry and an SEMPLN entry, then a mirror image copy of the primary superelement will first be created using the plane defined by the SEMPLN entry and then it will be positioned at the location implied by the SELOC entry.

If it references neither an SELOC entry nor an SEMPLN entry, then the secondary superelement will merely be a duplicate of the primary superelement positioned at the same location as the primary superelement. This usage is extremely uncommon. Hence the program cautions the user about this usage by issuing a user warning message.

The following comments apply only to TYPE= "MIRROR":

- h. The secondary superelement in this case must reference an SEMPLN entry. (Otherwise, the program will terminate the execution with an appropriate user fatal message.) In addition, the secondary superelement may also reference an SELOC entry.

If the secondary superelement references only an SEMPLN entry, then a mirror image copy of the primary superelement will be positioned using the plane defined by the SEMPLN entry.

If it references both an SELOC entry and an SEMPLN entry, then a mirror image copy of the primary superelement will first be created using the plane defined by the SEMPLN entry and then it will be positioned at the location implied by the SELOC entry.

7. METHOD = "MANUAL" requires SECONCT entries. SEBNDRY and SEEXCLD, which reference SEID, will produce a fatal message.
8. SECONCT, SEBNDRY, and SEEXCLD entries can be used to augment the search procedure and/or override the global tolerance.
9. For combined automatic and manual boundary search, the METHOD = "AUTO" should be specified and connections should be specified on a SECONCT entry.
10. TOL and LOC are the default values that can be modified between two superelements by providing the required tolerance on the SECONCT entry.
11. A SEID=0 is valid for the residual part only if it is desired to set a TOL value for the residual.

SECONCT

Partitioned Superelement Boundary-Point Connection

Explicitly defines grid and scalar point connection procedures for a partitioned superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SECONCT	SEIDA	SEIDB	TOL	LOC					
	GIDA1	GIDB1	GIDA2	GIDB2	GIDA3	GIDB3	-etc.-		

Example:

SECONCT	10	20	1.0E-4	YES					
	1001	4001			2222	4444			

Alternate Format and Example:

SECONCT	SEIDA	SEIDB	TOL	LOC					
	GIDA1	'THRU'	GIDA2	GIDB1	'THRU'	GIDB2			
SECONCT	10	20							
	101	'THRU'	110	201	'THRU'	210			

Descriptor	Meaning
SEIDA	Identification number of superelement for connection to SEIDB. (Integer ≥ 0)
SEIDB	Identification number of superelement for connection to SEIDA. (Integer ≥ 0)
TOL	Location tolerance to be used when searching for or checking boundary grid points. (Real; Default = 10E-5)
LOC	Coincident location check option for manual connection. (Character; "YES" or "NO"; Default = "YES")
GIDAi	Identification number of a grid or scalar point in superelement SEIDA, which will be connected to GIDBi.
GIDBi	Identification number of a grid or scalar point in superelement SEIDB, which will be connected to GIDAi.

Remarks:

1. SECONCT can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER = n command.
2. TOL and LOC can be used to override the default values specified on the SEBULK entries.
3. The continuation entry is optional.
4. The (GIDAi, GIDBi) pair must both be grids or scalar points.

5. All six degrees-of-freedom of grid points will be defined as boundary degrees-of-freedom.
6. This entry will only work if PART superelements (BEGIN SUPER) exist.
7. Blank fields are allowed after the first GIDA1-GIDB1 pair. Blank fields must also occur in pairs. This remark does not apply to the alternate format.
8. For Alternate Format 1, the thru ranges must be closed sets. That is, all IDs listed between 101 and 110 in the example must exist in the model.

SECSET

Free Boundary Degree-of-Freedom

Defines boundary degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
SECSET	SEID	ID1	C1	ID2	C2	ID3	C3		

Example:

SECSET	3	124	1	5	23	6	15		
--------	---	-----	---	---	----	---	----	--	--

Descriptor	Meaning
SEID	Superelement identification number. (Integer > 0)
Ci	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
IDi	Grid or scalar point identification number. (Integer > 0)

Remarks:

1. Exterior grid and scalar points are, by default, fixed during component mode analysis and placed in the b-set unless listed on SECSETi or SESUP entries. Coordinates listed on this entry are considered free (c-set) during component mode calculations. Exterior grid and scalar points are determined by the program and listed in the SEMAP table output.
2. Degrees-of-freedom specified on this entry are assigned to the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
3. There must be a sufficient number of degrees-of-freedom specified on SESUP entries to discard any free body modes of the superelement.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SECSET1

Free Boundary Degree-of-Freedom, Alternate Form of SECSET

Defines boundary degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
SECSET1	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example:

SECSET1	5	2	135	14	6	23	24	25	
	122	127							

Alternate Formats and Example:

SECSET1	SEID	C	G1	"THRU"	G2				
SECSET1	5	3	6	THRU	32				
SECSET1	SEID		"ALL"						
SECSET1	SEID		ALL						

Descriptor	Meaning
SEID	Superelement identification number. (Integer > 0)
C	Component numbers of degree-of-freedoms. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification number. (Integer > 0)

Remarks:

1. Exterior grid and scalar points are, by default, fixed during component mode analysis and placed in the b-set unless listed on SECSET*i* or SESUP entries. Degrees-of-freedom listed on this entry are considered free (c-set) during component mode calculations. Exterior grid and scalar points are determined automatically and listed in the SEMAP table output.
2. If the alternate formats are used, the grid points Gi are not required to exist or to be exterior degrees-of-freedom and may be listed on SECSET1 entries. Points of this type will cause one warning message but will otherwise be ignored.
3. Degrees-of-freedom specified on this entry are assigned to the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. There must be a sufficient number of degrees-of-freedom specified on SESUP entries to discard any free body modes of the superelement.
5. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SECTAX

Conical Shell Sector

Defines a sector of a conical shell.

Format:

1	2	3	4	5	6	7	8	9	10
SECTAX	ID	RID	R	PHI1	PHI2				

Example:

SECTAX	1	2	3.0	30.0	40.0				
--------	---	---	-----	------	------	--	--	--	--

Descriptor	Meaning
ID	Sector identification number. (Unique Integer > 0)
RID	Ring identification number. See RINGAX entry. (Integer > 0)
R	Effective radius. (Real)
PHI1, PHI2	Azimuthal limits of sector in degrees. (Real)

Remarks:

1. SECTAX is allowed only if an AXIC entry is also present.
2. SECTAX identification numbers must be unique with respect to all other POINTAX, RINGAX and SECTAX identification numbers.
3. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.

SEDLINK**Multiple Design Variable Linking Across PART SE Boundary**

Relates one design variable of a PART SE to one or more other design variables from other PART SEs.

Format:

1	2	3	4	5	6	7	8	9	10
SEDLINK	ID	DSEID	DDVID	C0	CMULT	ISEID1	IDV1	C1	
	ISEID2	IDV2	C2	ISEID3	IDV3	C3			
	ISEID4	IDV4	C4	-etc.-					

Example:

SEDLINK	10	10	2	0.1	0.33	8	8	-1.0	
	11	6	-1.0	20	8	7.0			
	11	2	2.0						

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
DSEID	PART SE identification number for DDVID (Integer ≥ 0)
DDVID	Dependent design variable identification number. (Integer > 0)
C0	Constant term. (Real; Default = 0.0)
CMULT	Constant multiplier. (Real; Default = 1.0)
ISEID _i	PART SE identification number for IDVi (Integer ≥ 0)
IDVi	Independent design variable identification number. (Integer > 0)
C _i	Coefficient i corresponding to IDVi. (Real)

Remarks:

1. SEDLINK is provided for inter-PART SE design variables linking and must be placed in main Bulk Data Section for consideration. SEDLINK will be ignored if it shows up under 'BEGIN SUPER=seid' where seid>0.
2. SEDLINK defines the relationship

$$DDVID = C0 + CMULT \sum_i C_i ISEID_i \cdot IDVi_{ISEID_i}$$

3. This capability provides a means of linking physical design variables such as element thicknesses to nonphysical design variables such as the coefficients of interpolating functions.
4. CMULT provides a simple means of scaling the C_i . For example if $C_i = 1/7, 2/7, 4/7$, etc. is desired, then CMULT = 1/7 and $C_i = 1, 2, 4$, etc., may be input.

5. An independent $IDV_{i,ISEID_i}$ must not occur on the same SEDLINK entry more than once.
6. ID is for user reference only.
7. If a design variable of a PART SE is specified as dependent on a SEDLINK entry, then it cannot be specified as independent on another SEDLINK or DLINK entry.

SEDRSP2

Design Sensitivity Equation Response Quantities for PART SE

Defines equation responses that are used in the design, either as constraints or as an objective with quantities from multiple PART SEs.

Format:

1	2	3	4	5	6	7	8	9	10
SEDRSP2	ID	LABEL	EQID or FUNC	REGION	METHOD	C1	C2	C3	
	"DESVAR"	DVSEID1	DVID1	DVSEID2	DVID2	DVSEID3	DVID3		
		DVSEID4	DVID4	-etc.-					
	"DTABLE"	LBSEID1	LABL1	LBSEID2	LABL2	LBSEID3	LABL3		
		LBSEID4	LABL4	-etc.-					
	"DRESP1"	R1SEID1	NR1	R1SEID2	NR2	R1SEID3	NR3		
		R1SEID4	NR4	-etc.-					
	"DNODE"	NDSEID1	G1	CMP1	NDSEID2	G2	CMP2		
		NDSEID3	G3	CMP3	-etc.-				
	"DVPREL1"	P1SEID1	DPIP1	P1SEID2	DPIP2	P1SEID3	DPIP3		
		P1SEID4	DPIP4	-etc.-					
	"DVCREL1"	C1SEID1	DCIC1	C1SEID2	DCIC2	C1SEID3	DCIC3		
		C1SEID4	DCIC4	-etc.-					
	"DVMREL1"	M1SEID1	DMIM1	M1SEID2	DMIM2	M1SEID3	DMIM3		
		M1SEID4	DMIM4	-etc.-					
	"DVPREL2"	P2SEID1	PDI2P1	P2SEID2	DPI2P2	P2SEID3	DPI2P3		
		P2SEID4	DPI2P4	-etc.-					
	"DVCREL2"	C2SEID1	DC12C1	C2SEID2	DC12C2	C2SEID3	DC12C3		
		C2SEID4	DC12C4	-etc.-					
	"DVMREL2"	M2SEID1	DMI2M1	M2SEID2	DMI2M2	M2SEID3	DMI2M3		
		M2SEID4	DMI2M4	-etc.-					

Example:

SEDRSP2	1	LBUCK	5	3					
	DESVAR	0	101	1	3	1	4		
		10	201						
	DTABLE	1	YM	10	L				
	DRESP1	0	14	1	1	1	4		
	DNODE	10	14	0	1				
	DVPREL1	0	101	1	102				

	DVCREL1	1	201	10	202				
	DVMREL1	0	301						
	DVPREL2	10	401	1	402				
	DVCREL2	0	501						
	DVMREL2	0	601	1	602	10	603		

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
LABEL	User-defined label. (Character)
EQID	DEQATN entry identification number. (Integer > 0)
FUNC	Function to be applied to the arguments. See Remark 8. of the DRESP2 entry. (Character)
REGION	Region identifier for constraint screening. See Remark 5. of the DRESP2 entry. (Integer > 0)
METHOD	When used with FUNC = BETA, METHOD = MIN indicates a minimization task while MAX indicates a maximization task. (Default = MIN) When used with FUNCT = MATCH, METHOD = LS indicated a least squares while METHOD = BETA indicated minimization of the maximum difference. (Default = LS)
Ci	Constants used when FUNC = BETA or FUNC = MATCH in combination with METHOD = BETA. See Remark 8. of the DRESP2 entry. (Real; Defaults: C1 = 100., C2 = .005)
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVSEIDI	PART SE identification number for DVIDI (Integer ≥ 0)
DVIDi	DESVAR entry identification number. (Integer > 0)
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LBSEIDj	PART SE identification number for LABLj. (Integer ≥ 0)
LABLi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)
“DRESP1”	Flag indicating DRESP1 entry identification numbers. (Character)
R1SEIDk	PART SE identification number for NRk. (Integer ≥ 0)
NRk	DRESP1 entry identification number. (Integer > 0)
“DNODE”	Flag indicating grid point and component identification numbers. (Character)
NDSEIDm	PART SE identification number for (Gm,Cm). (Integer ≥ 0)
Gm	Identification number for any grid point in the model. (Integer > 0)
Cm	Component number of grid point Gm. (1 < Integer < 3)
“DVPREL1”	Flag indicating DVPREL1 entry identification number. (Character)

Descriptor	Meaning
P1SEIDi	PART SE identification number for DPIP _i . (Integer ≥ 0)
DPIP _i	DVPREL1 entry identification number. (Integer > 0)
“DVCREL1”	Flag indicating DVCREL1 entry identification number. (Character)
C1SEIDi	PART SE identification number for DCIC _i . (Integer ≥ 0)
DCIC _i	DVCREL1 entry identification number. (Integer > 0)
“DVMREL1”	Flag indicating DVMREL2 entry identification number. (Character)
M1SEIDi	PART SE identification number for DMIM _i . (Integer ≥ 0)
DMIM _i	DVMREL1 entry identification number. (Integer > 0)
“DVPREL2”	Flag indicating DVPREL2 entry identification number. (Character)
P2SEIDi	PART SE identification number for DPI2P _i . (Integer ≥ 0)
DPI2P _i	DVPREL2 entry identification number. (Integer > 0)
“DVCREL2”	Flag indicating DVCREL2 entry identification number. (Character)
C2SEIDi	PART SE identification number for DCI2C _i . (Integer ≥ 0)
DCI2C _i	DVCREL2 entry identification number. (Integer > 0)
“DVMREL2”	Flag indicating DVMREL2 entry identification number. (Character)
M2SEIDi	PART SE identification number for DMI2M _i . (Integer ≥ 0)
DMI2M _i	DVMREL2 entry identification number. (Integer > 0)

Remarks:

1. SEDRSP2 is provided specifically for creating synthetic response with quantities from multiple PART SEs and must be placed in main bulk data section for consideration. SEDRSP2 will be ignored if it shows up under ‘BEGIN SUPER=seid’ where seid>0.
2. Items under DTABLE and DNODE flag requires companion items in the same SEID under flags of DESVAR, DVxRELY or DRESP1.
3. Other than the leading PART SE ID for each quantity, SEDRSP2 follows rules and shares limitations of DRESP2. Rules and limitations of DRESP2 will not be repeated here.
4. SEDRSP2 does not support DRESP2 flag.

SEDRSP3

Defines External Response with User-Supplied Routines

Defines constituents from multiple PART SE for an external response using user-supplied routine(s).

Format:

1	2	3	4	5	6	7	8	9	10
SEDRSP3	ID	LABEL	GROUP	TYPE	REGION				
	"DESVAR"	DVSEID1	DVID1	DVSEID2	DVID2	DVSEID3	DVID3		
		DVSEID4	DVID4	-etc.-					
	"DTABLE"	LBSEID1	LABL1	LBSEID2	LABL2	LBSEID3	LABL3		
		LBSEID4	LABL4	-etc.-					
	"DRESP1"	R1SEID1	NR1	R1SEID2	NR2	R1SEID3	NR3		
		R1SEID4	NR4	-etc.-					
	"DNODE"	NDSEID1	G1	CMP1	NDSEID2	G2	CMP2		
		NDSEID3	G3	CMP3	-etc.-				
	"DVPREL1"	P1SEID1	DPIP1	P1SEID2	DPIP2	P1SEID3	DPIP3		
		P1SEID4	DPIP4	-etc.-					
	"DVCREL1"	C1SEID1	DCIC1	C1SEID2	DCIC2	C1SEID3	DCIC3		
		C1SEID4	DCIC4	-etc.-					
	"DVMREL1:"	M1SEID1	DMIM1	M1SEID2	DMIM2	M1SEID3	DMIM3		
		M1SEID4	DMIM4	-etc.-					
	"DVPREL1"	P2SEID	DPI2P1	P2SEID2	DPI2P2	P2SEID3	DPI2P3		
		P2SEID4	DPI2P4	-etc.-					
	"DVCREL2"	C2SEID1	DC12C1	C2SEID2	DC12C2	C2SEID3	DC12C3		
		C2SEID4	DC12C4	-etc.-					
	"DVMREL2"	M2SEID	DMI2M1	M2SEID2	DMI2M2	M2SEID3	DMI2M3		
		M2SEID4	DMI2M4	-etc.-					
	"USRDATA"				String				
					-etc.-				

Example:

SEDRSP3	10	LBUCK	TAILWING	BUCK					
	DESVAR	0	101	1	3	1	4		
		10	201						
	DTABLE	1	YM	10	L				
	DRESP1	0	14	1	1	1	4		
	DNODE	10	14	0	1				
	DVPREL1	0	101	1	102				

	DVCREL1	1	201	10	202				
	DVMREL1	0	301						
	DVPREL2	10	401	1	402				
	DVCREL2	0	501						
	DVMREL2	0	601	1	602	10	603		
	USRDATA	Constants: 12345.67890.99.							

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
LABEL	User-defined label. (Character)
GROUP	Group name the external response type belongs to (Character). See Remark 2. of DRESP3.
TYPE	External response type (Character). See Remark 3. of the DRESP3.
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVSEID _i	PART SE identification number for DVID _i . (Integer ≥ 0)
DVID _i	DESVAR entry identification number. (Integer > 0)
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LBSEID _j	PART SE identification number for LABL _j . (Integer ≥ 0)
LABL _i	Label for a constant on the DTABLE or DTABLE2 entry. (Character)
“DRESP1”	Flag indicating DRESP1 entry identification numbers. (Character)
R1SEID _k	PART SE identification number for NR _k . (Integer ≥ 0)
NR _k	DRESP1 entry identification number. (Integer > 0)
“DNODE”	Flag indicating grid point and component identification numbers. (Character)
NDSEID _m	PART SE identification number for (G _m ,C _m). (Integer ≥ 0)
G _m	Identification number for any grid point in the model. (Integer > 0)
C _m	Component number of grid point G _m . (1 < Integer < 3)
“DVPREL1”	Flag indicating DVPREL1 entry identification number. (Character)
P1SEID _i	PART SE identification number for DPIP _i . (Integer ≥ 0)
DPIP _i	DVPREL1 entry identification number. (Integer > 0)
“DVCREL1”	Flag indicating DVCREL1 entry identification number. (Character)
C1SEID _i	PART SE identification number for DCIC _i . (Integer ≥ 0)
DCIC _i	DVCREL1 entry identification number. (Integer > 0)
“DVMREL1”	Flag indicating DVMREL2 entry identification number. (Character)
M1SEID _i	PART SE identification number for DMIM _i . (Integer ≥ 0)

Descriptor	Meaning
DMIMi	DVMREL1 entry identification number. (Integer > 0)
“DVPREL2”	Flag indicating DVPREL2 entry identification number. (Character)
P2SEIDi	PART SE identification number for DPI2Pi. (Integer ≥ 0)
DPI2Pi	DVPREL2 entry identification number. (Integer > 0)
“DVCREL2”	Flag indicating DVCREL2 entry identification number. (Character)
C2SEIDi	PART SE identification number for DCI2Ci. (Integer ≥ 0)
DCI2Ci	DVCREL2 entry identification number. (Integer > 0)
“DVMREL2”	Flag indicating DVMREL2 entry identification number. (Character)
M2SEIDi	PART SE identification number for DMI2Mi. (Integer ≥ 0)
DMI2Mi	DVMREL2 entry identification number. (Integer > 0)
“USRDATA”	Flag indicating user input data. (Character).

Remarks:

1. SEDRSP3 is provided specifically for creating external response with quantities from multiple PART SEs and must be placed in main Bulk Data Section for consideration. SEDRSP3 will be ignored if it shows up under ‘BEGIN SUPER=seid’ where seid>0.
2. Items under DTABLE and DNODE flag requires companion items in the same SEID under flags of DESVAR, DVxRELY or DRESP1.
3. Other than the leading PART SE ID for each quantity, SEDRSP3 follows rules and shares limitations of DRESP3. Rules and limitations of DRESP3 will not be repeated here.
4. SEDRSP3 does not support DRESP2 flag.

SEELT**Superelement Boundary Element Reassignment**

Reassigns superelement boundary elements to an upstream superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEELT	SEID	EID1	EID2	EID3	EID4	EID5	EID6	EID7	
	EID8	EID9	-etc.-						

Example:

SEELT	2	147	562	937					
-------	---	-----	-----	-----	--	--	--	--	--

Alternate Format and Example:

SEELT	SEID	EID1	"THRU"	EID2					
SEELT	5	12006	THRU	12050					

Descriptor	Meaning
SEID	Superelement identification number. See Remark 7. (Integer > 0)
EIDI	Element identification numbers. (Integer > 0 or "THRU"; for "THRU" option EID1 < EID2.)

Remarks:

- Elements connected entirely to the exterior points of an upstream superelement are called boundary elements and are assigned to the downstream superelement. The SEELT entry provides the means of reassigning the element to the upstream superelement. This entry may be applied to boundary elements only.
- Open sets are allowed with the "THRU" option.
- Elements processed with primary superelements will also be contained in any referencing secondary superelement.
- EIDI may refer to plot elements, general elements, and structural elements.
- This entry does not change the exterior grid point set of the superelement.
- SEELT can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
- If the model contains a BEGIN SUPER, the SEELT entry will assign the specified elements to the SEID (not just the boundary elements). This is an alternative to using SESET. In this case, SEID = 0 is a valid entry. For further information, refer to the [MSC Nastran Reference Manual](#).

SEECLD Partitioned Superelement Exclusion

Defines grids that will be excluded during the attachment of a partitioned superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEECLD	SEIDA	SEIDB	GIDA1	GIDA2	GIDA3	GIDA4	GIDA5	GIDA6	
	GIDA7	GIDA8	-etc.-						

Example1:

SEECLD	110	10	45	678	396				
--------	-----	----	----	-----	-----	--	--	--	--

Example 2:

SEECLD	400	ALL	10	20	30	THRU	40		
--------	-----	-----	----	----	----	------	----	--	--

Descriptor	Meaning
SEIDA	Partitioned superelement identification number. (Integer ≥ 0)
SEIDB	Superelement identification. (Integer ≥ 0 or Character = "ALL"; Default = "ALL")
GIDA _i	Identification number of a grid in superelement SEIDA to be excluded from connection to superelement SEIDB. (Integer > 0 or "THRU"; for "THRU" option GIDA1 < GIDA2.)

Remarks:

1. SEECLD can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER=n command.
2. SEIDA and SEIDB may reference only substructures or the residual structure, that is, parts defined after a BEGIN SUPER = entry.
3. This entry will only work if PART superelements (BEGIN SUPER) exist.

SELABEL

Superelement Output Label

Defines a label or name to be printed in the superelement output headings.

Format:

1	2	3	4	5	6	7	8	9	10
SELABEL	SEID			LABEL					

Example:

SELABEL	10	LEFT REAR FENDER, MODEL XYZ2000	
---------	----	---------------------------------	--

Descriptor	Meaning
SEID	Partitioned superelement identification number. (Integer > 0)
LABEL	Label associated with superelement SEID for output headings. (Character)

Remarks:

1. SELABEL can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. Only one SELABEL per superelement may be specified.
3. The label will appear in all superelement output headings. However, in some headings the label may be truncated.
4. This entry will only work if PART superelements (BEGIN SUPER) exist.

SELOC**Partitioned Superelement Location**

Defines a partitioned superelement relocation by listing three noncolinear points in the superelement and three corresponding points not belonging to the superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SELOC	SEID	PA1	PA2	PA3	PB1	PB2	PB3		

Example:

SELOC	110	10	100	111	1010	112	30		
-------	-----	----	-----	-----	------	-----	----	--	--

Descriptor	Meaning
SEID	Partitioned identification number of the partitioned superelement. (Integer > 0)
PAi	Identification numbers of three noncolinear grids (GRID entry) or points (POINT entry) which are in the partitioned superelement. (Integer > 0)
PBi	Identification numbers of three grids (GRID entry) or points (POINT entry) defined in the main Bulk Data Section to which PAi will be aligned. (Integer > 0)

Remarks:

1. SELOC can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER=n command.
2. The superelement will be rotated and translated for alignment of the GAi and GBi locations.
3. The PAi and PBi can be either GRIDs or POINTs.
4. PA1, PA2, and PA3 must be contained in superelement SEID.
5. PB1, PB2, and PB3 must be specified in the main Bulk Data Section. If they belong to a superelement that is also relocated, then the original (unmoved) positions of PB1, PB2, and PB3 are used.
6. PB1, PB2, and PB3 must have the same relative locations as PA1, PA2, and PA3.
7. Three grids or points are required even if the superelement connects to only one or two exterior grids.
8. Coordinate systems, global displacement directions, and element coordinate systems for the superelement will rotated and translated.
9. The global coordinate directions of the boundary grid points of the upstream superelement will be transformed internally to the global coordinate directions of the attachment grid points in the downstream superelement. For displacement data recovery, the output will be in the original global coordinate system.
10. The translation and rotation of the superelement to the new position is accomplished by defining local rectangular coordinate systems based on the specified grid locations:

- The local systems have their origin at PX1 and the x-axis points from PX1 to PX2.
 - The y-axis lies in the plane containing PX1, PX2, and PX3, is perpendicular to the x-axis, and points toward PX3.
 - The z-axis is defined by the cross product of the x-axis into the y-axis.
 - The rotation and translation transformation aligns the local system defined by the superelement grids with the local system defined by the main Bulk Data Section grids.
11. This entry will only work if PART superelements (BEGIN SUPER) exist.

SEMPLN**Superelement Mirror Plane**

Defines a mirror plane for mirroring a partitioned superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEMPLN	SEID	"PLANE"	P1	P2	P3				

Example:

SEMPLN	110	PLANE	12	45	1125				
--------	-----	-------	----	----	------	--	--	--	--

Descriptor	Meaning
SEID	Partitioned superelement identification number. (Integer > 0).
"PLANE"	Flag indicating that the plane is defined by three noncolinear points.
Pi	GRID or POINT entry identification numbers of three noncolinear points. (Integer > 0).

Remarks:

1. SEMPLN can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER=n command.
2. Grids or points referenced on this entry must be defined in the main Bulk Data Section.

SENQSET

Superelement Internal Generalized Degree-of-Freedom

Defines number of internally generated scalar points for superelement dynamic reduction.

Format:

1	2	3	4	5	6	7	8	9	10
SENQSET	SEID	N							

Example:

SENQSET	110	45							
---------	-----	----	--	--	--	--	--	--	--

Descriptor	Meaning
SEID	Partitioned superelement identification number. See Remark 3. (Integer > 0 or Character = "ALL")
N	Number of internally generated scalar points for dynamic reduction generalized coordinates. (Integer > 0; Default = 0)

Remarks:

1. SENQSET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER = n command.
2. SENQSET is only required if the user wants to internally generated scalar points used for dynamic reduction.
3. SEID = "ALL" will automatically generate N q-set degrees-of-freedom for all superelements, except the residual structure (SEID = 0). Specifying additional SENQSET entries for specific superelements will override the value of N specified on this entry.
4. If the user manually specifies q-set degrees-of-freedom using a SEQSETi or QSETi entries, then the internally generated scalar points will not be generated.
5. See PARAM,NQSET for an alternate method of specifying QSET degree-of-freedoms.
6. This entry will only work if PART superelements (BEGIN SUPER) exist.

SEQGP**Grid and Scalar Point Resequencing**

Used to manually order the grid points and scalar points of the problem. This entry is used to redefine the sequence of grid and scalar points to optimize bandwidth.

Format:

1	2	3	4	5	6	7	8	9	10
SEQGP	ID1	SEQID1	ID2	SEQID2	ID3	SEQID3	ID4	SEQID4	

Example:

SEQGP	5392	15.6	596	0.2	2	1.9	3	2	
-------	------	------	-----	-----	---	-----	---	---	--

Descriptor	Meaning
IDi	Grid or scalar point identification number. (Integer > 0)
SEQIDI	Sequenced identification number. (Real > 0.0 or Integer > 0)

Remarks:

1. The real format is used to insert a point ID between two consecutively numbered and existing point IDs. In the example above, point ID 5392 is inserted between IDs 15 and 16 by specifying 15.6 for SEQID. If the SEQID is real and < 1.0 the value must be entered as 0.2 not .2 (see field 5 of example).
2. The SEQIDI numbers must be unique and may not be the same as a point IDi which is not being changed. No grid point IDi may be referenced more than once.
3. From one to four grid or scalar points may be resequenced on a single entry.
4. If a point IDi is referenced more than once, the last reference will determine its sequence.
5. Automatic resequencing is also available. See [OLDSEQ, 963](#).

SEQSEP**Superelement Sequences**

Used with the CSUPER entry to define the correspondence of the exterior grid points between an identical or mirror-image superelement and its primary superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEQSEP	SSID	PSID	GP1	GP2	GP3	GP4	GP5	GP6	
	GP7	GP8	-etc.-						

Example:

SEQSEP	121	21	109	114	124	131			
--------	-----	----	-----	-----	-----	-----	--	--	--

Descriptor	Meaning
SSID	Identification number for secondary superelement. (Integer > 0).
PSID	Identification number for the primary superelement. (Integer ≥ 0).
GPi	Exterior grid point identification numbers for the primary superelement. (Integer > 0).

Remarks:

1. This entry is not needed if the grid points listed on the CSUPER entry with the same SSID are in the order of the corresponding exterior grid points of the primary superelement.
2. In [Figure 9-154](#), the exterior grid points of 10, 20, and 30 of SEID = 1 correspond to the points 13, 12, and 11, respectively, of image SEID = 2. The CSUPER entry may be defined alone or with a SEQSEP entry as shown in [Figure 9-154](#).

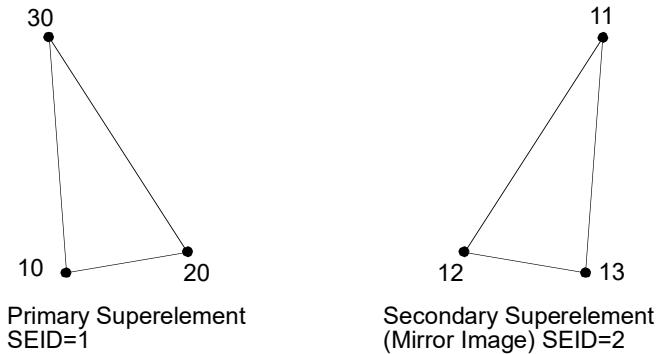


Figure 9-154 Grid Point Correspondence Between Primary and Secondary Superelements

CSUPER Entry Only:

1	2	3	4	5	6	7	8	9	10
CSUPER	2	1	13	12	11				

CSUPER and SEQSEP Entries:

CSUPER	2	1	11	12	13				
SEQSEP	2	1	30	20	10				

SEQSET

Superelement Generalized Degree-of-Freedom

Defines the generalized degrees-of-freedom of the superelement to be used in generalized dynamic reduction or component mode synthesis.

Format:

1	2	3	4	5	6	7	8	9	10
SEQSET	SEID	ID1	C1	ID2	C2	ID3	C3		

Example:

SEQSET	15	1	123456	7	5	22	3		
--------	----	---	--------	---	---	----	---	--	--

Descriptor	Meaning
SEID	Superelement identification number. Must be a primary superelement. (Integer > 0)
Ci	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer zero or blank for scalar points.)
IDi	Grid or scalar point identification numbers. Must be an exterior point. (Integer > 0)

Remarks:

- Degrees-of-freedom specified on this entry may not be specified for another superelement.
- Generalized degrees-of-freedom are interior to the residual structure.
- Connectivity to the superelement is provided by this entry. There is no need to use a CSUPEXT entry for this purpose.
- Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
- This entry describes the set used for generalized degrees-of-freedom only for the SEID listed. Degrees-of-freedom listed on this entry must also be members of a downstream superelement. The set used for these variables in downstream superelements must be prescribed by user action using other entries. If they are scalar points, they are automatically in the residual structure, which is the recommended procedure. If they are grid points, it is the user's responsibility to place them in a downstream superelement.

Generalized degrees-of-freedom of superelements that are also members of the residual structure are included as dynamic variables by placing them in the a-set. It is also necessary to place some or all residual structure physical degrees-of-freedom in the a-set to allow the boundary points to participate in the system mode shapes.

Grid points of downstream superelements used as generalized degrees-of-freedom may be used for advanced applications, such as omitting upstream generalized degrees-of-freedom from assembly into downstream superelements. Again, it is the user's responsibility to place these variables in the proper set in all downstream superelements of which they are members.

6. This entry may be applied only to primary superelements. The CSUPER entry automatically defines these degrees-of-freedom for secondary superelements.

SEQSET1

Superelement Generalized Degree-of-Freedom, Alternate Form

Defines the generalized degrees-of-freedom of the superelement to be used in generalized dynamic reduction or component mode synthesis.

Format:

1	2	3	4	5	6	7	8	9	10
SEQSET1	SEID	C	G1	G2	G3	G4	GS	G6	
	G7	G8	-etc.-						

Example:

SEQSET1	15	123456	1	7	9	22	105	6	
	52	53							

Alternate Format and Example:

SEQSET1	SEID	C	G1	"THRU"	G2				
SEQSET1	16	0	101	THRU	110				

Descriptor	Meaning
SEID	Superelement identification number. Must be a primary superelement. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. Must be exterior points. (Integer > 0 or "THRU"; for THRU option G1 < G2.)

Remarks:

- Degrees-of-freedom specified on this entry may not be specified for another superelement.
- Generalized degrees-of-freedom are interior to the residual structure.
- Connectivity to the superelement is provided by this entry. There is no need to use a CSUPEXT entry for this purpose.
- Degrees-of-freedom specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
- This entry describes the set used for generalized degrees-of-freedom only for the SEID listed. Degrees-of-freedom listed on this entry must also be members of a downstream superelement. The set used for these variables in downstream superelements must be prescribed by user action using other entries. If they are scalar points, they are automatically in the residual structure, which is the recommended procedure. If they are grid points, it is the user's responsibility to place them in a downstream superelement.

Generalized degrees-of-freedom of superelements that are also members of the residual structure are included as dynamic variables by placing them in the a-set. It is also necessary to place some or all residual structure physical degrees-of-freedom in the a-set, to allow the boundary points to participate in the system mode shapes.

Grid points of downstream superelements used as generalized degrees-of-freedom may be used for advanced applications, such as omitting upstream generalized degrees-of-freedom from assembly into downstream superelements. Again, it is the user's responsibility to place these variables in the proper set in all downstream superelements of which they are members.

6. This entry may be applied only to primary superelements. The CSUPER entry automatically defines these entries for secondary superelements.

SESET

Superelement Interior Point Definition

Defines interior grid points for a superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SESET	SEID	G1	G2	G3	G4	G5	G6	G7	

Example:

SESET	5	2	17	24	25	165			
-------	---	---	----	----	----	-----	--	--	--

Alternate Format and Example:

SESET	SEID	G1	“THRU”	G2					
SESET	2	17	THRU	165					

Descriptor	Meaning
SEID	Superelement identification number. Must be a primary superelement. (Integer ≥ 0)
Gi	Grid or scalar point identification number. ($0 < \text{Integer} < 100000000$; G1 < G2)

Remarks:

1. Interior grid points may also be defined via field 9 of the GRID and GRIDG Bulk Data entries. The SESET entry takes precedence over the SEID field on the GRID or GRIDG entries. SESET defines grid and scalar points to be included as interior to a superelement. SESET may be used as the primary means of defining superelements or it may be used in combination with SEELT entries which define elements interior to a superelement.
2. Gi may appear on an SESET entry only once.
3. Scalar points are ignored unless a BEGIN SUPER is in the file.
4. Open sets are allowed with the “THRU” option. Missing grid points (whether in “THRU” range or mentioned explicitly) are not identified.
5. All degrees-of-freedom for Gi are placed in the o-set of the superelement. See [Degree-of-Freedom Sets, 1111](#).
6. SESET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER = n command.

SESUP

Fictitious Support

Defines determinate reaction superelement degrees-of-freedom in a free-body analysis.

Format:

1	2	3	4	5	6	7	8	9	10
SESUP	SEID	ID1	C1	ID2	C2	ID3	C3		

Example:

SESUP	5	16	215						
-------	---	----	-----	--	--	--	--	--	--

Descriptor	Meaning
SEID	Superelement identification number. Must a primary superelement. (Integer > 0)
IDi	Grid or scalar point identification number. Must be exterior points. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points; Any unique combination of the Integers 1 through 6 for grid points.)

Remarks:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
2. The Ci degrees-of-freedom must be exterior degrees-of-freedom of the SEID superelement.
3. See [Rigid Body Supports](#) in the *MSC Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
4. There must be a sufficient number of degrees-of-freedom on SESUP entries to discard any free body modes of the superelement.
5. SESUP Bulk Data entries are not allowed for part (partitioned bulk data) superelements. Use the SUPPORT Bulk Data records to identify component rigid body modes.

SET1**Set Definition**

Defines a list of structural grid points or element identification numbers.

Format:

1	2	3	4	5	6	7	8	9	10
SET1	SID	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	-etc.-							

Example 1:

SET1	3	31	62	93	124	16	17	18	
	19								

Example 2:

SET1	6	29	32	THRU	50	61	THRU	70	
	17	57							

Example 3:

SET1	7	SKIN							
------	---	------	--	--	--	--	--	--	--

Descriptor	Meaning
SID	Unique identification number. (Integer > 0)
IDi	List of structural grid point or element identification numbers. (Integer > 0 or “THRU”; for the “THRU” option, ID1 < ID2 or “SKIN”; in field 3)

Remarks:

- When using the “THRU” option for SPLINEi or PANEL data entries, all intermediate grid points must exist.
- When using the “THRU” option for XYOUTPUT or AECOMP requests, missing grid points are ignored. The first and last points must exist.
- When using the “SKIN” option, a panel will be generated consisting of the structural portion of the fluid-structural boundary. This option works ONLY with all fields of ACMODL having default value.
- THRU may not appear in field 3 or 9 (2 or 9 for continuations).
- RC network solver does not support SET1 for thermal analysis.

SET2

Grid Point List

Defines a list of structural grid points in terms of aerodynamic macro elements.

Format:

1	2	3	4	5	6	7	8	9	10
SET2	SID	MACRO	SP1	SP2	CH1	CH2	ZMAX	ZMIN	

Example:

SET2	3	111	0.0	0.75	0.0	0.667	3.51		
------	---	-----	-----	------	-----	-------	------	--	--

Descriptor	Meaning
SID	Unique identification number. (Integer > 0)
MACRO	Element identification number of an aerodynamic macro element. (Integer > 0)
SP1, SP2	Lower and higher span division points defining the prism containing the set. (Real)
CH1, CH2	Lower and higher chord division points defining the prism containing the set. (Real)
ZMAX, ZMIN	Z-coordinates of top and bottom (using right-hand rule with the order of the corners as listed on a CAEROi entry) of the prism containing set. (Real)

Remarks:

1. The SET2 entry is referenced by the SPLINEi entry.
2. Every grid point within the defined prism and within the height range will be in the list. For example:

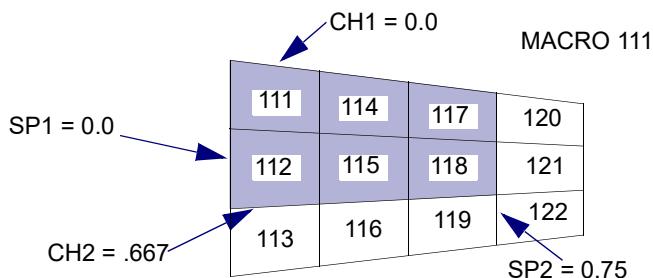


Figure 9-155 SET2 Entry Example.

The shaded area in Figure 9-155 defines the cross section of the prism for the sample data given above. Points exactly on the boundary may be missed; therefore, to get all the grid points within the area of the macro element, $SP1=-.01$, $SP2=1.01$, etc. should be used.

3. A zero value for ZMAX or ZMIN implies a value of infinity. Usually, $ZMAX \geq 0.0$ and $ZMIN \leq 0.0$.
4. To print the (internal) grid IDs found, use DIAG 18.

SET3**Labeled Set Definition**

Defines a list of grids, elements or points.

Format:

1	2	3	4	5	6	7	8	9	10
SET3	SID	DES	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	ID8	-etc-						

Example:

SET3	1	POINT	11	12	13	15	18	21	
------	---	-------	----	----	----	----	----	----	--

Alternate Format and Example:

SET3	SID	DES	ID1	"THRU"	ID2				
SET3	33	POINT	20	THRU	60				

Descriptor	Meaning
SID	Unique identification number. (Integer>0)
DES	Set description (Character). Valid options are "GRID", "ELEM", "POINT", "PROP", "RBEin", and "RBEx".
IDi	Identifiers of grids points, elements, points or properties. (Integer > 0)

Remarks:

1. If a SET3 entry is referenced on a PBMSECT or PBRSECT entry, the POINTs must lie in the (xy) plane of the basic coordinate system, and be in the order when traversing the boundary or the profile.
2. When the SET3 entry is referenced by a panel, describers can be "GRID", "ELEM" or "PROP".
3. THRU may not appear in field 4 or 9 (2 or 9 for continuations).
4. When SET3 is referenced by SOLs 400 entry only GRID or ELEM may be used.
5. When SET3 is referenced by RFORCE (IDRF field) for SOL 600, only ELEM may be used.
6. When SET3 is referenced by SOL 400 DEACTEL entries, only ELEM may be used.
7. When a SET3 is referenced by a ELSIDI or XELSIDi field on an FTGDEF entry, only ELEM may be used. When SET3 is referenced by a NDSIDI field on a FTGDEF entry, only GRID may be used.
8. When DES="RBEin", the SET selects rigid elements to be included for MPC=sid and is applicable to Rigid Element types of RBAR, RBAR1, RBE1, RBE2, RBE2GS, RBE3, RROD, RSPLINE, RSSCON, RTRPLT and RTRPLT1. Note that Rigid Elements with duplicate ID across Rigid Element types will all be utilized.

9. For DES="RBEex", the SET selects rigid elements to be excluded for MPC=sid and is applicable to Rigid Elements types of RBAR, RBAR1, RBE1, RBE2, RBE2GS, RBE3, RROD, RSPLINE, RSSCON, RTRPLT and RTRPLT1.
10. Note that "RBEin" and "RBEex" are mutually exclusive and should not appear together for a single SET.
11. By default, without SET3,mpcid,RBExx, all Rigid Elements in the input deck will be used.
12. SET selection for rigid elements does not cover additional IDs on MPCADD bulk data entry. This kind of selection is not supported in SOL 106 either.
13. Rigid element set section is supported in SOL 400 if 'RIGID=LINEAR' is present in case control deck.

SET4**Property Set Definition**

Defines a list of property IDs

Format:

1	2	3	4	5	6	7	8	9	10
SET4	ID	CLASS	TYPE	ID1	ID2	ID3	ID4	ID5	
	ID6	ID7	ID8	-etc-					

Example:

SET4	22	PROP	PSOLID	1	THRU	20			
------	----	------	--------	---	------	----	--	--	--

Descriptor	Meaning
ID	Unique identification number. (Integer>0).
CLASS	Set to "PROP". (Character = PROP; no default)
TYPE	Property type. Valid options are PSOLID, PHELL, PSHEAR, PBAR, PBEAM, and PWELD.
IDi	Property IDs of the specified TYPE flag.

Remarks:

1. THRU option may not appear in field 5 or 9 on first line or field 2 or 9 for continuation lines.
2. Currently referenced from the FTGDEF entry.

SETREE

Superelement Tree Definition (Alternate Form of DTI,SETREE)

Specifies superelement reduction order.

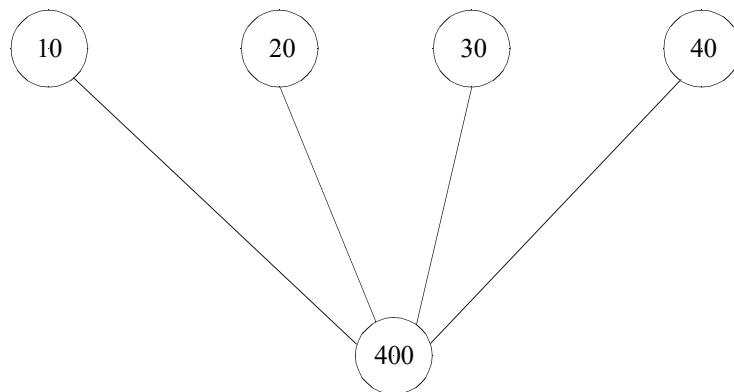
Format:

1	2	3	4	5	6	7	8	9	10
SETREE	SEID	SEUP1	SEUP2	SEUP3	SEUP4	SEUP5	SEUP6	SEUP7	
	SEUP8	SEUP9	-etc.-						

Example:

SETREE	400	10	20	30	40				
--------	-----	----	----	----	----	--	--	--	--

Descriptor	Meaning
SEID	Partitioned superelement identification number of a downstream superelement. (Integer ≥ 0)
SEUPI	Identification number of superelements that are upstream of SEID. (Integer > 0)

**Remarks:**

1. SETREE entries or DTI,SETREE entry are required for multilevel superelement configurations.
2. At least one SETREE entry is required for each nontip superelement, including the residual structure (SEID = 0). Multiple SETREE entries with the same SEID are allowed.
3. A superelement may appear only once in an SEUPI field on all SETREE entries.
4. If an DTI,SETREE entry is provided, then SETREE entries are not required.
5. If both SETREE entries and a DTI,SETREE entry exist, then the DTI,SETREE entry will be ignored.

6. If a superelement is not referenced on the DTI,SETREE or SETREE entry, then the manner in which it is handled depends on the type of that superelement. If it is a PART superelement, then the residual will be regarded as its downstream superelement and the undefined superelement will therefore be placed immediately above the residual in the tree. If it is a Main Bulk Data superelement, then it will also be handled like an undefined PART superelement as above *if all of its exterior points belong to the residual*. However, if one or more of its exterior points do not belong to the residual, then the program will terminate with a user fatal error complaining that one or more of the superelements are not in the same path.
7. The SETREE entry will only work if PART (BEGIN SUPER) superelements exist in the model. If there are no PARTs in the model, the SETREE entries will be ignored.

SEUSET

Superelement Degree-of-Freedom Set Definition

Defines a degree-of-freedom set for a superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEUSET	SEID	SNAME	ID1	C1	ID2	C2	ID3	C3	

Example:

SEUSET	15	U4	1	123456	7	5	22	3	
--------	----	----	---	--------	---	---	----	---	--

Descriptor	Meaning
SEID	Superelement identification number. (Integer > 0)
SNAME	Set name. (One to four characters or string “ZERO”, followed by the set name.)
IDi	Grid or scalar point identification numbers. (Integer > 0)
Ci	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blank for grid points; Integer 0 or blank for scalar points.)

Remarks:

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets, 1111](#) or their new names on the DEFUSET entry. However, in the Solution Sequences 0 through 200, it is recommended that SNAME refer only to the set names “U1” through “U6” or their new names on the DEFUSET entry.
2. If SNAME = “ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.

SEUSET1

Superelement Degree-of-Freedom Set Definition, Alternate Form

Defines a degree-of-freedom set for a superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEUSET1	SEID	SNAME	C	G1	G2	G3	G4	G5	
	G6	G7	-etc.-						

Example:

SEUSET1	15	U4	1	12	15	17	22	25	
	52	53							

Alternate Format and Example:

SEUSET1	SEID	SNAME	C	G1	"THRU"	G2			
SEUSET1	15	U4	1	12	THRU	27			

Descriptor	Meaning
SEID	Superelement identification number. (Integer > 0)
SNAME	Set name. (One to four characters or string "ZERO", followed by the set name.)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification number. (Integer > 0)

Remarks:

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets, 1111](#) or their new names on the DEFUSET entry. However, in the Solution Sequences 0 through 200, it is recommended that SNAME refer only to the set names "U1" through "U6" or their new names on the DEFUSET entry.
2. If SNAME= "ZERO*i*", where *i* is a set name, then the degrees-of-freedom are omitted from set *i*.
3. If the alternate format is used, all of the points G1 through G2 are assigned to the set.

SHREL**Elastic Shear Model**

Defines an elastic shear model with a constant shear modulus. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
SHREL	SID	G							

Example:

SHREL	250	80.E6							
-------	-----	-------	--	--	--	--	--	--	--

Descriptor	Meaning
SID	Unique shear model number referenced from a MATDEUL entry. (Integer > 0; Required)
G	Shear-modulus value. (Real; Default = 0.0)

Remark:

1. SID must unique among all SHRxx entries in one model.

SHRPOL**Polynomial Shear Model**

Defines an elastic shear model with a polynomial shear modulus. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
SHRPOL	SID	G_0	G_1	G_2	G_3				

Example:

SHRPOL	250	80.E6							
--------	-----	-------	--	--	--	--	--	--	--

Descriptor	Meaning
SID	Unique shear model number referenced from a MATDEUL entry. (Integer > 0; Required)
G_0	Coefficient G_0 . (Real; Default = 0.0)
G_1	Coefficient G_1 . (Real; Default = 0.0)
G_2	Coefficient G_2 . (Real; Default = 0.0)
G_3	Coefficient G_3 . (Real; Default = 0.0)

Remark:

1. SID must unique among all SHRx entries in one model.
2. The shear modulus is computed from

$$G = G_0 + G_1\gamma + G_2\gamma^2 + G_3\gamma^3$$

where γ = effective plastic shear strain

and G_0 , G_1 , G_2 and G_3 are constants

SHRUDS

User-defined Shear Model for Elements.

Specifies that a user subroutine is being used to define the shear modulus. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
SHRUDS	SID	GROUP	UNAME						

Example:

In FMS Section of the MSC Nastran input stream:

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

In Bulk Data:

1	2	3	4	5	6	7	8	9	10
SHRUDS	12	mymat	EXSHR						

Descriptor	Meaning
SID	Unique porosity model ID. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. (Character; default=EXSHR)

Remarks:

1. Only can be used for SOL 700.
2. The SID must be referenced by a MATDEUL or MAT1 entry.
3. UNAME=EXSHR can only be used.

SLBDY

Slot Boundary List

Format:

1	2	3	4	5	6	7	8	9	10
SLBDY	RHO	M	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	-etc.-							

Example:

SLBDY	0.002	6	16	17	18	25	20	21	
	22								

Descriptor	Meaning
RHO	Density of fluid at boundary. (Real > 0.0 or blank)
M	Number of slots. (Integer ≥ 0 or blank)
IDj	Identification numbers of GRIDS slot points at boundary with axisymmetric fluid cavity, $j = 1, 2, \dots, J$. (Integer > 0)

Remarks:

1. SLBDY is allowed only if an AXSLOT entry is also present.
2. If RHO or M is blank, the default value on the AXSLOT entry is used. The effective value must not be zero for RHO. If the effective value of M is zero, no matrices at the boundary will be generated.
3. The order of the list of points determines the topology of the boundary. The points are listed sequentially as one travels along the boundary in either direction. At least two points must be defined.

SLOAD

Static Scalar Load

Defines concentrated static loads on scalar or grid points.

Format:

1	2	3	4	5	6	7	8	9	10
SLOAD	SID	S1	F1	S2	F2	S3	F3		

Example:

SLOAD	16	2	5.9	17	-6.3	14	-2.93		
-------	----	---	-----	----	------	----	-------	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
Si	Scalar or grid point identification number. (Integer > 0)
Fi	Load magnitude. (Real)

Remarks:

1. In the static solution sequences, SID is selected by the LOAD Case Control command.
2. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.
3. Up to three loads may be defined on a single entry.
4. If Si refers to a grid point, the load is applied to component T1 of the displacement coordinate system (see the CD field on the GRID entry).

SLOADN1

Describes TOP/BOT/MID Scalar Load for Heat Shell Element in SOL 400

Defines concentrated static loads on grid points of heat shell elements with linear or quadratic temperature distribution through the thickness direction.

Format:

1	2	3	4	5	6	7	8	9	10
SLOADN1	SID	G1	C1	Q1	G2	C2	Q2		

Example:

SLOADN1	10	10	12	1300.	20	2	1300.		
---------	----	----	----	-------	----	---	-------	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
Gi	Grid point identification number. (Integer > 0)
Ci	Composite numbers. ($0 \leq$ Integer ≤ 3 ; up to 3 unique Integers may be placed in the field with no embedded blanks.) 1=TOP, 2=BOT, 3=MID. (Integer > -1; Default = 1)
Qi	Power. (Real)

Remarks:

1. This entry is for shell elements defined on a PSHLN1 heat transfer to specify power input.
2. In the steady-state heat transfer analysis, SID is selected by the LOAD Case Control command.
3. In the transient heat transfer analysis, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.

SNORM

Surface Normal Vector at Grid Point

Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
SNORM	GID	CID	N1	N2	N3				

Example:

SNORM	3	2	0.	-1.	0.				
-------	---	---	----	-----	----	--	--	--	--

Descriptor	Meaning
GID	Unique grid point identification number. (Integer > 0)
CID	Identification number of coordinate system in which the components of the normal vector are defined. See Remark 3. (Integer ≥ 0 ; Default = 0 for the basic coordinate system)
Ni	Components of normal vector. The three components of the normal need not define a unit vector. (Real; Default = 0.0)

Remarks:

1. The SNORM Bulk Data entry overrides any unique, internally-generated grid point normals that may have been requested with the user parameter SNORM, described in Chapter 6 of this guide.
2. The normal is used in CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements. For all other elements, the normal is ignored.
3. If CID is a cylindrical or spherical coordinate system, the components Ni are in the local tangent system at grid GID. For example, if CID=10 is a spherical coordinate system and normals must be defined pointing outwards in the radial direction of the sphere, see , then the SNORM entries for all grids GID on the sphere are simply
SNORM, GID, 10, 1., 0., 0.

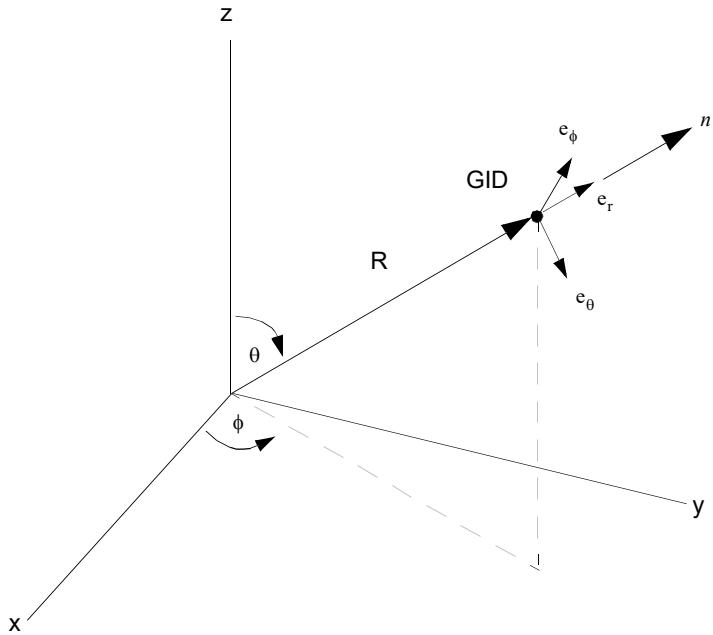


Figure 9-156

SPBLND1

Strip Based Spline Blending

Defines a strip based blending of two splines.

Format:

1	2	3	4	5	6	7	8	9	10
SPBLND1	SID	SID1	SID2	OPT	W1	GID	D1	D2	
	X1	X2	X3	CID					

Example:

SPBLND1	130	110	120	CUB		227	4.05	4.05	
	1.0	0.0	0.0	110					

Descriptor	Meaning
SID	Identification number of blended spline. (Integer > 0)
SID1	Identification number of first spline (may be a blended spline). (Integer > 0)
SID2	Identification number of second spline (may be a blended spline). (Integer > 0)
OPT	Blending Option: WAVG Weighted Average (Default) LIN Linear Blending Functions CUB Cubic Blending Functions
W1	Weight to be used with first spline. ($0.0 < \text{Real} \leq 1.0$; Default = 0.5) (Used only with option WAVG)
GID	Identification number of an aerodynamic grid to be used as reference grid. (Integer > 0)
D1	Blending Depth of first spline. (Real > 0.0)
D2	Blending Depth of second spline. (Real > 0.0)
X1, X2, X3	Components of a direction vector v, in the coordinate system defined by CID, which is used to measure the distance from the reference grid. (See Remark 3.)
CID	Identification number of a rectangular coordinate system used to define the direction vector. (Integer > 0; Default = 0; indicating the basic coordinate system)

Remarks:

1. The blending depth defines the maximum distance from the reference grid point of an aerodynamic grid point to be used in blending. It is also needed to evaluate the blending functions.
2. With option WAVG, the definition of GID, D1, D2 and the direction vector is optional. The weight W2 to be used with the second spline is computed from $W2 = 1 - W1$.

- The blended displacement is computed from

$$u_b = f_1(x)u_1 + f_2(x)u_2$$

where $f_1(x)$ and $f_2(x)$ are the blending functions (see [Figure 9-157](#)) and x is the distance from the reference grid point, measured in the direction of the direction vector v . Functions $f_1(x)$ and $f_2(x)$ sum up to 1.

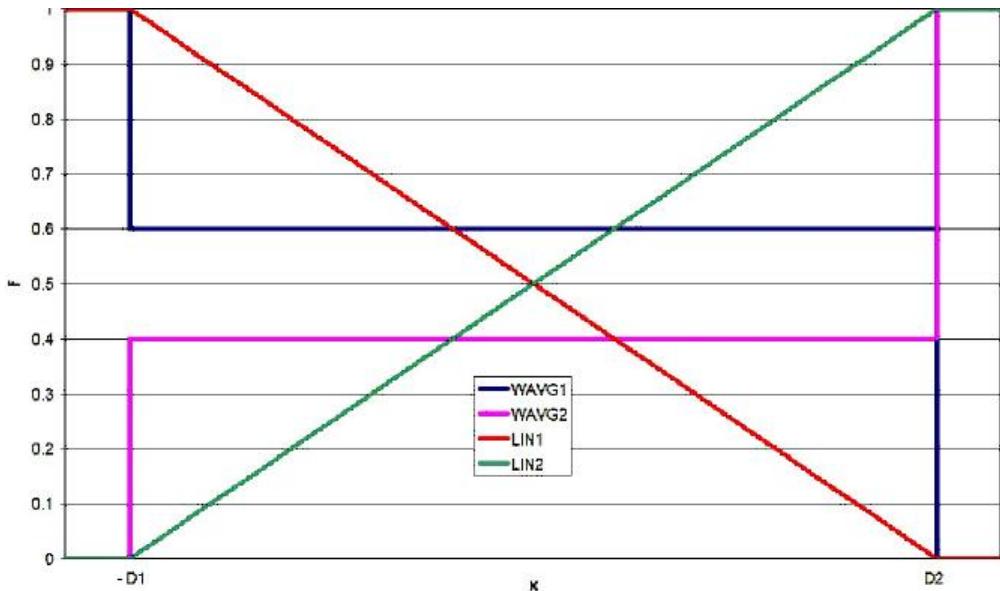


Figure 9-157 Blending Functions

- If the overlap region extends beyond $-D1 < x < D2$, then

$$f_1(x) = 1.0 \text{ and } f_2(x) = 0 \text{ for } x < -D1$$

$$f_1(x) = 0 \text{ and } f_2(x) = 1.0 \text{ for } x > D2$$
- The referenced splines must have the same USAGE flag. This USAGE flag defines the USAGE flag of the blended spline.
- If the splined aero components are of type CAERO, a MDLPRM,MLTSPLIN,1 entry must be used to enable blending. This is not required with AEGRID based aerodynamics.

SPBLND2

Curve Based Spline Blending

Defines a curve based blending of two splines.

Format:

1	2	3	4	5	6	7	8	9	10
SPBLND2	SID	SID1	SID2	OPT	AELIST	D1	D2		

Example:

SPBLND2	130	110	120	LIN	4	1.5	2.5		
---------	-----	-----	-----	-----	---	-----	-----	--	--

Descriptor	Meaning
SID	Identification number of blended spline. (Integer > 0)
SID1	Identification number of first spline (may be a blended spline). (Integer > 0)
SID2	Identification number of second spline (may be a blended spline). (Integer > 0)
OPT	Blending Option: LIN Linear Blending Functions (Default) CUB Cubic Blending Functions
AELIST	Identification number of an AELIST entry listing the aerodynamic grid points that define a reference curve. (Integer > 0)
D1	Blending Depth of first spline. (Real > 0.0)
D2	Blending Depth of second spline. (Real > 0.0)

Remarks:

1. The blending depth defines the maximum value of the distance of an aerodynamic grid point from the reference curve. It is also needed to evaluate the blending functions.
2. Blending functions are evaluated based on the distance of an aerodynamic grid point from the reference curve.
3. The reference curve is approximated by a polygon through the grid points listed on the AELIST entry referenced by AELIST. The list may contain coincident grid points. The order of the grid points is arbitrary.
4. The referenced splines must have the same USAGE flag. This USAGE flag defines the USAGE flag of the blended spline.
5. If the splined aero components are of type CAERO, a MDLPRM,MLTSPLIN,1 entry must be used to enable blending. This is not required with AEGRID based aerodynamics.

SPC**Single-Point Constraint**

Defines a set of single-point constraints and enforced motion (enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis).

Format:

1	2	3	4	5	6	7	8	9	10
SPC	SID	G1	C1	D1	G2	C2	D2		

Example:

SPC	2	32	3	-2.6	5				
-----	---	----	---	------	---	--	--	--	--

Descriptor	Meaning
SID	Identification number of the single-point constraint set. (Integer > 0)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. See Remark 8. ($0 \leq$ Integer ≤ 6 ; up to six Unique Integers, 1 through 6, may be placed in the field with no embedded blanks. 0 or 1 applies to scalar points and 1 through 6 applies to grid points.)
Di	Value of enforced motion for components Gi at grid Ci. (Real; Default = 0.0)

Remarks:

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
3. Single-point forces of constraint are recovered during stress data recovery.
4. From 1 to 12 degrees-of-freedom may be specified on a single entry.
5. Degrees-of-freedom on this entry may be redundantly specified as permanent constraints using the PS field on the GRID entry.
6. For reasons of efficiency, the SPCD entry is the preferred method for applying enforced motion rather than the Di field described here.
7. For SOL 400, the SPC entry requests enforced total displacement (Di) while the SPC1 entry requests null enforced relative displacements for a step. See the SPCD and SPCR entries for additional information.

Enforced Relative Displacement	
SPCR*	associated with SPC1 or GRID/PS
Enforced Total Displacement	
SPCD	associated with SPC and overwrite the value of enforced motion on SPC
SPCD	associated with SPC1 or GRID/PS
SPC	No association required

In SOL 400, when a GRID has an applied load associated with it in the previous STEP and the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is to use the SPCR. Alternatively, the user may apply instead an SPC1 to the GRID to lock in the position of the grid from the previous STEP.

For SOL 600, the SPC entry requests enforced total displacement just like SPCD, however SPC may only be used in SOL 600 to enforce displacements if there is just one subcase in the analysis. If two or more subcases exist, SPCD or SPCR must be used to prescribe the enforced motion.

8. For heat shell element with linear or quadratic nodal distribution (see option TEMPP for NLMOPTS entry 1=TOP, 2=BOT or 3=MID.)
9. In thermal analysis, this entry specifies a constant temperature boundary condition applied on the selected grid or scalar point. For SOL 400, transient thermal analysis with any time-varying boundary condition, a constant temperature condition should be specified using the SPCD and SPC1 Bulk Data entries. For SOLs 153 and 159 thermal analysis, no TEMPBC Bulk Data entries with TYPE="STAT" may be specified with this entry.

SPC1

Single-Point Constraint, Alternate Form

Defines a set of single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example:

SPC1	3	2	1	3	10	9	6	5	
	2	8							

Alternate Format and Example:

SPC1	SID	C	G1	"THRU"	G2				
SPC1	313	12456	6	THRU	32				

Descriptor	Meaning
SID	Identification number of single-point constraint set. (Integer > 0)
C	Component numbers. See Remark 7. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0, 1 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. (Integer > 0 or "THRU"; For "THRU" option, G1 < G2.)

Remarks:

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. Enforced displacements are available via this entry when used with the recommended SPCD entry.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. Degrees-of-freedom on this entry may be redundantly specified as permanent constraints using the PS field on the GRID entry.
5. If the alternate format is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will collectively produce a warning message but will otherwise be ignored.
6. For SOL 400, the SPC1 entry requests null enforced relative displacement for a step while the SPC entry requests enforced total displacements. SPC1 can always hold the displacement from the previous STEP. See the SPC, SPCD and SPCR entries for additional information.

Enforced Relative Displacement		
SPCR*	associated	SPC1 or GRID/PS
Enforced Total Displacement		
SPCD	associated	SPC
SPCD	associated	SPC1 or GRID/PS
SPC	No association required	

In SOL 400 if a “control” GRID has an applied load associated with it in the previous STEP and if the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is still to use the SPCR, however, the user may apply instead an SPC1 containing the “control” grid ID.

7. For heat shell element with linear or quadratic nodal distribution (see option TEMPP for NLMOPTS entry 1=TOP, 2=BOT or 3=MID.)
8. In thermal analysis, this entry is used with SPCD to specify a temperature boundary condition applied on the selected grid or scalar point. To define time-varying boundary conditions, the user should specify SPC1 and SPC Bulk Data entries for SOL 400 while using TEMPBC Bulk Data entries with TYPE=“TRAN” for SOL 159.

SPCADD**Single-Point Constraint Set Combination**

Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries.

Format:

1	2	3	4	5	6	7	8	9	10
SPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc.-						

Example:

SPCADD	101	3	2	9	1				
--------	-----	---	---	---	---	--	--	--	--

Descriptor	Meaning
SID	Single-point constraint set identification number. (Integer > 0)
Si	Identification numbers of single-point constraint sets defined via SPC or by SPC1 entries. (Integer > 0)

Remarks:

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. No Si may be the identification number of a single-point constraint set defined by another SPCADD entry.
3. The Si values must be unique.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.
5. SPCADD entries take precedence over SPC entries. If both have the same SID, only the SPCADD entry will be used.

SPCAX

Conical Shell Single-Point Constraint

Defines a set of single-point constraints or enforced displacements for conical shell coordinates.

Format:

1	2	3	4	5	6	7	8	9	10
SPCAX	SID	RID	HID	C	D				

Example:

SPCAX	2	3	4	13	6.0				
-------	---	---	---	----	-----	--	--	--	--

Descriptor	Meaning
SID	Identification number of a single-point constraint set. (Integer > 0)
RID	Ring identification number. See RINGAX entry. (Integer ≥ 0)
HID	Harmonic identification number. (Integer ≥ 0)
C	Component identification number. (Any unique combination of the Integers 1 through 6.)
D	Enforced displacement value. (Real)

Remarks:

1. SPCAX is allowed only if an AXIC entry is also present.
2. Single-point constraint sets must be selected with the Case Control command SPC = SID.
3. Coordinates appearing on SPCAX entries may not appear on MPCAX, SUPAX, or OMITAX entries.
4. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.

SPCD**Enforced Motion Value**

Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.

Format:

1	2	3	4	5	6	7	8	9	10
SPCD	SID	G1	C1	D1	G2	C2	D2		

Example:

SPCD	100	32	3	-2.6	5		2.9		
------	-----	----	---	------	---	--	-----	--	--

Descriptor	Meaning
SID	Set identification number of the SPCD entry. (Integer > 0)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. See Remark 10. ($0 \leq \text{Integer} \leq 6$; any unique combination of Integers 1 through 6 with no embedded blanks for grid points; Integer 0, 1 or blank for scalar points)
Di	Value of enforced motion for components Ci at grid Gi. (Real)

Remarks:

- In the static solution sequences, the set ID of the SPCD entry (SID) is selected by the LOAD Case Control command. For SOL 600, it may also be selected by Li on the Bulk Data entry LOAD.
- In dynamic analysis, the selection of SID is determined by the presence of the LOADSET request in Case Control as follows:
 - There is no LOADSET request in Case Control
SID is selected by the EXCITEID field of an RLOAD1, RLOAD2, TLOAD1 or TLOAD2 Bulk Data entry that has enforced motion specified in its TYPE field
 - There is a LOADSET request in Case Control
SID is selected by LID in the selected LSEQ entries that correspond to the EXCITEID entry of an RLOAD1, RLOAD2, TLOAD1 or TLOAD2 Bulk Data entry that has enforced motion specified in its TYPE field.
- The SPCD entry is not supported in SOL 200.
- A global coordinate (Gi and Ci) referenced on this entry must also be referenced on a SPC or SPC1 Bulk Data entry and selected by the SPC Case Control command. This requirement is optional for SOL 600.

5. Enforced motion may be specified on both SPCD and SPC Bulk Data entries. The SPCD entry is selected according to Remark 2 and the SPC entry is selected by the SPC Case Control command. The Di values from both of the selected SPCD and SPC entries will be combined to form a static (or spatial) loading condition. But if a degree-of-freedom is duplicated on both SPCD and SPC then the value of Di on the selected SPCD entry will override the value of Dion the selected SPC entry. In addition, any degrees-of-freedom from the selected SPC entries and not duplicated will still be included in the loading condition. This is illustrated in the examples below.

- Static analysis: In the example below in SUBCASE 11, D=1.0 at GRID 101 overrides D=1.2 on the SPC entry. But GRID 201 is not specified on the selected SPCD and therefore D=1.2 from the SPC entry is also enforced. Similary, in subcase 12, D=1.0 is enforced at GRID 201 and D=1.2 at GRID 101.

```
SPC=77
SUBCASE 11
LOAD=701
SUBCASE 12
LOAD=702
BEGIN BULK
SPCD,701,101,2,1.0
SPCD,702,201,2,1.0
SPC ,77 ,101,2,1.2
SPC ,77 ,201,2,1.2
```

- Dynamic analysis: In the example below, there are two unique EXCITEIDs specified on TLOAD2 and thusly two static loading conditions are created and identical to the conditions created in the static analysis example above.

```
SPC=77
DLOAD=201
BEGIN BULK
DLOAD,201,1.0,1.0,401,1.0,402
TLOAD2, 401, 701, , DISP, 0., 10.0, 60.,90.
TLOAD2, 402, 702, , DISP, 0., 10.0, 20.,270.
SPCD,701,101,2,1.0
SPCD,702,201,2,1.0
SPC ,77 ,101,2,1.2
SPC ,77 ,201,2,1.2
```

6. The LOAD Bulk Data entry will not combine an SPCD load entry except for SOL 600.
7. In static analysis, this method of applying enforced displacements is more efficient than the SPC entry when more than one enforced displacement condition is applied. It provides equivalent answers.

8. In dynamic analysis, this direct method of specifying enforced motion is more accurate, efficient and elegant than the large mass and Lagrange multiplier techniques.
9. For SOL 400, the SPCD entry requests enforced total displacement for a STEP while the SPCR entry requests enforced relative displacements. SPCD is the position at the end of the applied STEP for ANALYSIS=NLSSTAT.

Enforced Relative Displacement	
SPCR*	associated with SPC1 or GRID/PS
Enforced Total Displacement	
SPCD	associated with SPC and overwrite the value of enforced motion on SPC
SPCD	associated with SPC1 or GRID/PS
SPC	No association required

In SOL 400, when a GRID has an applied load associated with it in the previous STEP and the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is to use the SPCR. Alternatively, the user may apply instead an SPC1 to the GRID to lock in the position of the grid from the previous STEP.

For SOL 600, the SPCD entry requests enforced total displacement for the subcase and is the position at the end of the SUBCASE. SPCR requests displacement or rotation for the current subcase relative to that of the previous subcase. SPCD may be used for nonlinear statics or dynamics and does not need to be associated with SPC, SPC1 or GRID/PS. In other words, the association can be made but is not required for SOL 600.

10. For SOL 400 using heat shell element with linear or quadratic nodal distribution, see option TEMPP for NLMOPTS entry 1=TOP, 2=BOT or 3=MID).
11. In thermal analysis, this entry is used with SPC1 to specify a temperature boundary condition applied on the selected grid or scalar point. To define time-varying boundary conditions, the user should specify SPC1 and SPCD Bulk Data entries for SOL 400 while using TEMPBC Bulk Data entries with TYPE="TRAN" for SOL 159 and SOL 600.
12. If it is desired to enforce motion on the boundary point (a-set) of a superelement then define a high stiffness element attached to the boundary point and apply the SPCD to the opposite point of the high stiffness element. High stiffness elements may be easily defined with CBUSH or CBAR.

SPCOFF

Excludes Degrees-of-Freedom from the AUTOSPC Operation

Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide* for a description of the AUTOSPC operation.

Format:

1	2	3	4	5	6	7	8	9	10
SPCOFF	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

SPCOFF	32	436	5	1					
--------	----	-----	---	---	--	--	--	--	--

Descriptor	Meaning
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer 0, 1 or blank for scalar points; Integers 1 through 6 with no embedded blanks for grid points.)

Remarks:

- Degrees-of-freedom specified on this entry are to be excluded from the AUTOSPC operation. If any degree-of-freedom in this set is found to be singular, a warning message is issued and no constraint is applied.
- Degrees-of-freedom that are specified as both SPC and SPCOFF will be considered as SPC.

SPCOFF1

Excludes DOF's from AUTOSPC Processing, Alternate Form

Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide* for a description of the AUTOSPC operation.

Format:

1	2	3	4	5	6	7	8	9	10
SPCOFF1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	-etc.-						

Example:

SPCOFF1	2	1	3	10	9	6	5	4	
	8								

Alternate Format and Example:

SPCOFF1	C	G1	"THRU"	G2					
SPCOFF1	12456	6	THRU	32					

Descriptor	Meaning
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0, 1 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. (Integer > 0 or "THRU"; for "THRU" option, G1 < G2.)

Remarks:

1. Degrees-of-freedom specified on this entry are to be excluded from the AUTOSPC operation. If any degree-of-freedom in this set is found to be singular, a warning message is issued and no constraint is applied.
2. Degrees-of-freedom that are both specified as SPC and SPCOFF will be considered as SPC.
3. If the alternate format is used, points in the sequence G1 through G2 are not required to exist. Points which do not exist will collectively produce a warning message but will otherwise be ignored.

SPCR**Enforced Relative Motion Value**

Defines an enforced relative displacement value for a load step in SOL 400 and SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
SPCR	SID	G1	C1	D1	G2	C2	D2		

Example:

SPCR	100	32	436	-2.6	5		2.9		
------	-----	----	-----	------	---	--	-----	--	--

Descriptor	Meaning
SID	Set identification number of the SPCR entry. (Integer > 0)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. ($0 \leq \text{Integer} \leq 6$; up to six unique Integers may be placed in the field with no embedded blanks, a blank or 0 is treated the same as 1.)
Di	Value of enforced motion for Gi and Ci. (Real)

Remarks:

1. SPCR requests relative motion for a load step with respect to the previous step in SOL 400 or subcase in SOL 600. SPCR is the companion entry to SPCD. SPCD requests total motion while SPCR requests relative motion. SPCR is the position at activation of the boundary condition.

Enforced Relative Displacement	
SPCR*	associated with SPC1 or GRID/PS
Enforced Total Displacement	
SPCD	associated with SPC and overwrite the value of enforced motion on SPC

Enforced Relative Displacement	
SPCD	associated with SPC1 or GRID/PS
SPC	No association required

In SOL 400, when a GRID has an applied load associated with it in the previous STEP and the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is to use the SPCR. Alternatively, the user may apply instead an SPC1 to the GRID to lock in the position of the grid from the previous STEP.

For SOL 600, the SPCR entry requests displacement or rotation for the current subcase relative to that of the previous subcase. The SPCD requests enforced total displacement for the subcase and is the position at the end of the SUBCASE. SPCR may be used for nonlinear statics or dynamics and does not need to be associated with SPC, SPC1 or GRID/PS. In other words, the association can be made but is not required for SOL 600.

2. The enforced motion for a step, for SOL 400 or subcase for SOL 600, can be either total value or relative value. For SOL 400, SPC and SPCD request total enforced motion. SPC1 and SPCR request the relative value. For example, if a DOF is specified on a SPCR with 0.0 for step 2, the relative displacement of this DOF for step 2 with respect to step 1 is 0.0. The total displacement of step 2 is 0.2 if the solution of step 1 for this DOF is 0.2.
3. The SCPD and SPCR entries can have the same SID, but they cannot be specified on the same DOF. A user fatal error will be issued if SPCD and SPCR are specified on the same DOF.
4. In the static solution sequences, the SID of the SPCR entry (SID), same as SPCD, is selected by the LOAD Case Control command.
5. For SOL 400, a global coordinate (Gi and CI) referenced on this entry must also be referenced on a SPC1 Bulk Data entry and selected by the SPC Case Control command. Please note that, for this purpose, SPC cannot be used together with SPCR. If SPC is used, a user fatal error will be issued.
6. For SOL 400, the LOAD Bulk Data entry will not combine an SPCR load entry.
7. For SOL 600 if SPCR is entered, PARAM,MARCTOTT,1 must also be entered.

SPHERE

Defines the Shape of a Sphere

Spherical shape used in the initial condition definition on the TICEUL1 entry. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
SPHERE	VID		X	Y	Z	RADIUS			

Example:

SPHERE	100		1.	1.	1.	.5			
--------	-----	--	----	----	----	----	--	--	--

Descriptor	Meaning
VID	Number of the sphere. (Integer > 0; Required)
X, Y, Z	Coordinates of the center of the sphere. (Real; Default = 0.0)
RADIUS	Radius of the sphere. (Real > 0; Required)

SPLINE1

Surface Spline Methods

Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE1	EID	CAERO	BOX1	BOX2	SETG	DZ	METH	USAGE	
	NELEM	MELEM							

Example:

SPLINE1	3	111	115	122	14	0.			
---------	---	-----	-----	-----	----	----	--	--	--

Descriptor	Meaning
EID	Unique spline identification number. (Integer > 0)
CAERO	Aero-element (CAEROi entry ID) that defines the plane of the spline. (Integer > 0)
BOX1, BOX2	First and last box with motions that are interpolated using this spline; see Remark 3. when using Mach Box method. (Integer > 0; BOX2 ≥ BOX1)
SETG	Refers to the SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real ≥ 0.0 ; Default = 0.0)
METH	Method for the spline fit. IPS, TPS or FPS. See Remark 1. (Character; Default = IPS)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 5. (Character; Default = BOTH)
NELEM	The number of FE elements along the local spline x-axis if using the FPS option. (Integer > 0; Default = 10)
MELEM	The number of FE elements along the local spline y-axis if using the FPS option. (Integer > 0; Default = 10)

Remarks:

1. The default METHOD will result in the use of the Harder-Desmarais Infinite Plate Spline (IPS). The other options are the Thin Plate Spline (TPS) and the Finite Plate Spline (FPS). The continuation applies only to the FPS option and is required only if the defaults are not adequate.
2. The interpolated points (k -set) will be defined by aero boxes. Figure 9-158 shows the cells for which u_k is interpolated if BOX1 = 115 and BOX2 = 122.

111	114	117	120
112	115	118	121
113	116	119	122

Figure 9-158 SPLINE1 Entry Example

3. The attachment flexibility (units of area) is used for smoothing the interpolation. If DZ = 0.0, the spline will pass through all deflected grid points. If DZ is much greater than the spline area, a least squares plane fit will be applied. Intermediate values will provide smoothing.
4. When using the Mach Box method, BOX1 and BOX2 refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to structural grids. BOX1 and BOX2 do not refer to Mach Boxes.
5. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

$F_g = [GP_{kg}]^T \{P_k\}$ (FORCE/BOTH splines are in the transform)

$U_k = [GD_{kg}] \{U_g\}$ (DISP/BOTH splines are in the transform)

In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship $[GP_{gk}]^T = [GD_{kg}]$ satisfied. The default behavior (BOTH for all splines) is compatible with versions of MSC Nastran prior to Version 70.5.

In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from its effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCES from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). MSC Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform "rigid aerodynamic" analyses).

6. The SPLINE1 EID must be unique with respect to all SPLINEi entries.

SPLINE2

Linear Spline

Defines a beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE2	EID	CAERO	ID1	ID2	SETG	DZ	DTOR	CID	
	DTHX	DTHY		USAGE					

Example:

SPLINE2	5	8	12	24	60	0.	1.0	3	
	1.								

Descriptor	Meaning
EID	Unique spline identification number. (Integer > 0)
CAERO	Aero panel or body (CAERO <i>i</i> entry ID) that is to be interpolated. (Integer > 0)
ID1, ID2	First and last box or body element whose motions are interpolated using this spline. See Remark 6. when using the Mach Box method. (Integer > 0; ID2 ≥ ID1)
SETG	Refers to an SET <i>i</i> entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real ≥ 0.0 ; Default = 0.0)
DTOR	Torsional flexibility ratio (EI/GJ). (Real > 0.0 ; Default = 1.0; use 1.0 for bodies.)
CID	Rectangular coordinate system for which the y-axis defines the axis of the spline. Not used for bodies, CAERO2. (Integer ≥ 0)
DTHX, DTHY	Rotational attachment flexibility. DTHX is for rotation about the spline's x-axis (in-plane bending rotations); however, it is not used for bodies. DTHY is for rotation about the spline's y-axis (torsion); however, it is used for slope of bodies. (Real)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 9. (Character; Default = BOTH)

Remarks:

1. The interpolated points (k-set) will be defined by aero boxes.
2. The spline axis for panels is the projection of the y-axis of coordinate system CID, projected onto the plane of the panel. For bodies, the spline axis is parallel to the x-axis of the aerodynamic coordinate system.

3. The flexibilities DZ, DTHX, and DTHY are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing, whereas negative values of DTHX or DTHY will imply infinity, therefore, no attachment). See the [MSC Nastran Aeroelastic Analysis User's Guide](#) for a discussion of special cases.
4. The continuation entry is required.
5. The SPLINE2 EID must be unique with respect to all SPLINEi entries.
6. When using the Mach Box method, ID1 and ID2 refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to the structural grids. ID1 and ID2 do not refer to Mach Boxes.
7. DTOR is the ratio of rotational to linear deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily linear deflection will occur.
8. If a SPLINE2 element only references one grid point, the job will fail without a message in the GI Module.
9. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.

SPLINE3**Aeroelastic Constraint Equation**

Defines a constraint equation for aeroelastic problems. Useful for control surface constraints.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE3	EID	CAERO	BOXID	COMP	G1	C1	A1	USAGE	
	G2	C2	A2		-etc.				

Example:

SPLINE3	7000	107	109	6	5	3	1.0		
	43	5	-1.0						

Descriptor	Meaning
EID	Unique spline identification number. (Integer > 0)
CAERO	Identification number of the macro-element on which the element to be interpolated lies. (Integer > 0)
BOXID	Identification number of the aerodynamic element; i.e., the box number. (Integer > 0)
COMP	The component of motion to be interpolated. See Remark 4. (One of the Integers 1, 2, 3, 4, 5, or 6.)
Gi	Grid point identification number of the independent grid point. (Integer > 0)
Ci	Component numbers in the displacement coordinate system. (One of the Integers 1 through 6 for grid points, or 0 for scalar points.)
Ai	Coefficient of the constraint relationship. (Real)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 6. (Character; Default = BOTH).

Remarks:

1. The independent grid points and components must refer to degrees-of-freedom in the g-set.
2. The constraint is given by

$$u_d = \sum A_i u_i$$

where:

u_d = value of the dependent component of the aerodynamic element

u_i = displacement at grid Gi, component Ci.

3. The SPLINE3 EID must be unique with respect to all SPLINEi entries.

4. The allowable components by CAEROi entry type are indicated by an “X” in the table below:

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X	X	
CAERO2		X	X	X	X
CAERO3			X		
CAERO4			X	X	X
CAERO5			X	X	X
3D Geometry	X	X	X	X	X

COMP = 2: lateral displacement

COMP = 3 transverse displacement

COMP = 5: pitch angle

COMP = 6: relative control angle for CAERO4 and CAERO5 yaw angle for CAERO2.

For general 3D aerodynamic geometries the components numbers refer to axes of the Aerodynamic Coordinate System ($u_x, u_y, u_z, \theta_x, \theta_y, \theta_z$).

5. For Strip theory and Piston theory, the COMP = 6 control surface relative angle is positive when the trailing edge has a negative z-deflection in the element coordinate system (see the [MSC Nastran Aeroelastic Analysis User's Guide](#)).
6. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.

SPLINE4

Surface Spline Methods

Defines a curved surface spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE4	EID	CAERO	AELIST		SETG	DZ	METH	USAGE	
	NELEM	MELEM	FTYPE	RCORE					

Example:

SPLINE4	3	111	115		14	0.	IPS		
---------	---	-----	-----	--	----	----	-----	--	--

Descriptor	Meaning
SID	Unique spline identification number. (Integer > 0)
CAERO	Identification number of aerodynamic component that defines the interpolation surface. (Integer > 0)
AELIST	Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 2. (Integer > 0)
SETG	Identification number of a SET1 entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real ≥ 0.0 ; Default = 0.0)
METH	Spline method: <ul style="list-style-type: none"> IPS Infinite Plate Spline (Default) TPS Thin Plate Spline FPS Finite Plate Spline RIS Radial Interpolation Spline
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. Legal values are FORCE, DISP or BOTH. See Remark 3. (Character, Default = BOTH)
NELEM	The number of FE elements along the local x-axis if using the FPS option. (Integer > 0; Default = 10)
MELEM	The number of FE elements along the local y-axis if using the FPS option. (Integer > 0; Default = 10)
FTYPE	Selects the radial interpolation function to be used with the RIS option: <ul style="list-style-type: none"> WF0 C0 continuous Wendland function

Descriptor	Meaning
WF2	C2 continuous Wendland function (Default)
RCORE	Radius of support of radial interpolation function. (Real > 0.0; no Default)

Remarks:

1. The attachment flexibility (units of area) is used for smoothing the interpolation. If DZ = 0.0, the spline will pass through all deflected grid points. If DZ is much greater than the spline area, a least squares plane fit will be applied. Intermediate values will provide smoothing.
2. For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AFFACT entry). For all other aero methods, the AELIST items are box id's.
3. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
4. NELEM and MELEM are used only for the METH=FPS and are required only when the defaults are not adequate.
5. FTTYPE and RCORE used only with METH=RIS. FTTYPE=WF0 uses a Wendland function:

$$\phi\left(\frac{r}{r_c}\right) = \left(1 - \frac{r}{r_c}\right)_t^2$$

while FTTYPE = WF2 uses

$$\phi\left(\frac{r}{r_c}\right) = \left(1 - \frac{r}{r_c}\right)_t^4 \left(4\frac{r}{r_c} + 1\right)$$

where

$$(y)_t = \begin{cases} y & \text{if } y > 0 \\ o & \text{if } y < 0 \end{cases}$$

SPLINES5

Linear Spline

Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINES5	SID	CAERO	AELIST		SETG	DZ	DTOR	CID	
	DTHX	DTHY		USAGE	METH		FTYPE	RCORE	

Example:

SPLINES5	5	8	12		60			3	
	1.			BOTH					

Descriptor	Meaning
SID	Unique spline identification number (Integer > 0)
CAERO	Identification number of aerodynamic component that defines the interpolation surface. (Integer > 0)
AELIST	Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 6. (Integer > 0)
SETG	Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real ≥ 0.0 ; Default = 0.0)
DTOR	Torsional flexibility ratio (EI/GJ) for the bending in the zy-plane. This value is ignored for slender bodies since they have no torsion; see Remark 7. (Real > 0.0; Default = 1.0; ignored for CAERO2 bodies.)
CID	Rectangular coordinate system that defines the y-axis of the spline and the xy- and yz-planes for bending. Not used for bodies, CAERO2. (Integer ≥ 0)
DTHX, DTHY	Rotational attachment flexibility. DTHX is for rotation about the spline's x-axis (the bending rotations). DTHY is for rotation about the spline's y-axis (torsion); however, it is used for bending of bodies. (Real)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 8. (Character; Default = BOTH)
METH	Spline method: BEAM Beam Spline (Default) RIS Radial Interpolation Spline
FTYPE	Selects the radial interpolation function to be used with the RIS option:

Descriptor	Meaning
WF0	C0 continuous Wendland function
WF2	C2 continuous Wendland function. (Default)
RCORE	Radius of support of radial interpolation function. (Real > 0.0; no Default)

Remarks:

1. The interpolated points (k-set) will be defined by aero boxes.
2. The spline axis for panels is the projection of the y-axis of coordinate system CID, projected onto the plane of the panel. For bodies, the interpolating beam (y-axis) is parallel to the x-axis of the aerodynamic coordinate system; the z-axis is taken from the referenced CID and x is made orthogonal.
3. The flexibilities DZ, DTHX and DTHY are used for smoothing. (Zero attachment flexibility values imply rigid attachment; i.e., no smoothing, whereas negative values of DTHX or DTHY imply infinity, therefore, no attachment.) See the [MSC Nastran Aeroelastic Analysis User's Guide](#) for a discussion of special cases.
4. The continuation entry is required.
5. The SPLINE5 EID must be unique with respect to all SPLINEi entries.
6. For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFACT entry). For all other aero methods, the AELIST items are box id's.
7. DTOR is the ratio of axial rotational to bending deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily linear deflection will occur. The values will affect the results only if the structural grids over constrain the motion of the interpolating beam. Slender bodies have no torsional motion, so these values will not be used for CAERO2 entries.
8. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
9. FTYPE and RCORE are only used for METHOD=RIS. See Remark 5. on the SPLINE 4 entry for descriptions of the Wendland functions.

SPLINE6

3D Finite Surface Spline

Defines a 6DOF or 3DOF finite surface spline for interpolating motion and/or forces between two meshes.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE6	EID	CAERO	AELIST		SETG	DZ	METHOD	USAGE	
	VSTYPE	VSLIST	I2VNUM	D2VNUM	METHVS	DZR	METHCON	NGRID	
	ELTOL	NCYCLE	AUGWEI						

Example:

SPLINE6	5	8	12		60		FPS6	DISP	
	AERO	2	4	4	VS6				

Descriptor	Meaning
EID	Unique spline identification number. (Integer > 0)
CAERO	Aero panel (CAEROi entry ID) that is to be interpolated. See Remarks 2. and 4. (Integer > 0 or blank)
AELIST	Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 2. (Integer > 0)
SETG	Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real ≥ 0.0 ; Default = 0.0)
METHOD	Method for the spline fit. Either FPS3 or FPS6. See Remark 5. (Character; Default = FPS6)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 3. (Character; Default = BOTH)
VSTYPE	Virtual surface connectivity type. Either AERO or STRUC. (Character; Default = AERO)
VSLIST	Identification number of an AELIST entry listing quadrilateral and/or triangular shell elements of the VSTYPE mesh which define the connectivity of the virtual surface mesh. See Remark 2. (Integer > 0 or can be blank if CAERO is not blank and VSTYPE=AERO)
I2VNUM	The minimum number of structural mesh points to connect to each virtual mesh point. See Remarks 7., 8. and 9. (0 < Integer; Default = 3)
D2VNUM	The minimum number of aero mesh points to connect to each virtual mesh point. See Remarks 7., 8. and 9. (0 < Integer; Default = 3)

Descriptor	Meaning
METHVS	Similar to METHOD, this field chooses whether or not to include the rotational degrees-of-freedom of virtual surface. Either VS6 or VS3. See Remark 5. (Character, Default VS6)
DZR	Rotational attachment flexibility. (Real ≥ 0.0 ; Default = 0.0)
METHCON	Method used to determine RBE3 connecting points between the meshes. Either NODEPROX or CIRCBIAS. See Remarks 7., 8. and 9. (Character; Default = NODEPROX)
NGRID	Number of closest grids that are used to determine the element list that is used to define the RBE3 elements. Only valid for METHCON=CIRCBIAS. See Remarks 7. and 9. (Integer > 0 ; Default = 1)
ELTOL	Tolerance used to determine whether or not a node projects onto an element of the mesh. Specified as % of element size. Only valid for METHCON=CIRCBIAS. See Remarks 7. and 9. (Real; Default = 100.0)
NCYCLE	Maximum number of cycles used to find elements onto which the nodes project. Only valid for METHCON=CIRCBIAS. See Remark 7. and 9. (Integer > 0 ; Default = 3)
AUGWEI	RBE3 weighting factor augmentation parameter. Only valid for METHCON=CIRCBIAS. See Remarks 7. and 9. (Real ≥ 0.0 ; Default = 0.0)

Remarks:

1. The flexibilities DZ and DZR are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing). The DZ and DZR values are used to derive stiffness of the translational and rotational (respectively) bushing stiffnesses. Bushing elements are placed between the interpolating surface and the connections to the dependent and independent grids.
2. If an aerodynamic mesh is entered using AEGRID/AETRIA3/AEQUAD4 entries, the CAERO ID is not required and the AELIST refers to the element ID's of the aero mesh. In this scenario, if VSTYPE=AERO, VSLIST must point to an AELIST that defines the virtual surface mesh by identifying the aerodynamic elements that make up the surface. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFACT entry). For all other aero methods, the AELIST items are box id's.
3. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
4. CAERO2 entries are not supported. The CAERO entry may be blank unless points from a CAERO4 or a CAERO5 are specified. The list of splined points may span multiple aerodynamic components. If the spline defines points from multiple CAERO4 or CAERO5 entries, then any one of the referenced CAERO IDs is valid input.
5. The METHOD option provides a choice in using all 6 degrees of freedom (FPS6) on the independent points or only the translational degrees of freedom (FPS3) in connecting between the virtual surface and the independent points. Similarly, there is a choice in connecting the virtual surface to the dependent points (METHVS).

6. The connection between the independent points (structural) and the dependent points (aero) is made through a virtual surface whose mesh is defined by elements listed in the VSLIST (these are either AERO box id's or STRUCtural shell elements (CQUAD4, CQUADR, CTRIA3 ,CTRIAR)).
7. To bind the points to the virtual surface, a connection is made between the points and the surface using automatically generated virtual RBE3 elements. Two methods exist to choose which independent mesh points are connected to each dependent mesh point: nodal-proximity (NODEPROX) and circular bias (CIRCBIAS).
8. The nodal proximity method selects the closest independent mesh points to each dependent mesh point. The actual number of points will depend on the user inputs I2VNUM and D2VNUM as well as collinearity checks. Larger values will spread the connectivity (smearing). Smaller values allow for more concentration (with additional points added as necessary for collinearity).
9. The circular bias method uses elements of the virtual mesh in an attempt to select independent mesh points that encircle each dependent mesh point. This method will be restricted to the case where the virtual mesh is the target mesh.

This method will do the following:

- For each splined dependent mesh node, find the closest NGRID splined independent mesh node(s).
- Assemble the list of virtual mesh elements that use the closest node(s).
- Check each of these elements to see if the dependent node projects onto the element in the element's mean plane normal direction. Note that this check may be computationally expensive, so it is performed only to the "possible" elements, not the entire virtual mesh. The projection check will contain a user-defined tolerance, ELTOL, to expand the area of the element that is acceptable for a match.
- If the dependent node does not project onto any element, use the candidate element's nodes to expand the list of elements to check. Repeat the projection check (the original elements will not be rechecked). Repeat this process up to NCYCLE times.
- All elements that are found to encompass the dependent node (and there may be more than one due to curvature) will be selected to move forward.
- Assemble the list of all splined nodes that connect the selected elements.
- Generate RBE3 elements based on this node list. An optional user-defined input parameter, AUGWEI, will be used to augment the RBE3 weighting factors with the following formula:

$$\text{weight} = (\text{NE} - 1) \cdot \text{AUGWEI} + 1$$

where NE is the number of elements that are connected with the RBE3 node.

SPLINE7

Finite Beam Spline

Defines a 6DOF finite beam spline for interpolating motion and/or forces between two meshes.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE7	EID	CAERO	AELIST		SETG	DZ	DTOR	CID	
				USAGE	METHOD	DZR	IA2	EPSBM	

Example:

SPLINE7	5	8	12		60			3	
				BOTH	FBS6				

Descriptor	Meaning
EID	Unique spline identification number. (Integer > 0)
CAERO	Aero panel (CAEROi entry ID) that is to be interpolated. See Remark 6. (Integer > 0 or blank)
AELIST	Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 2. (Integer > 0)
SETG	Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real ≥ 0.0 ; Default = 0.0)
DTOR	Ratio of the beam bending stiffness to the beam torsional stiffness. See Remark 3.. (Real > 0.0; Default = 1.0)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 4. (Character, Default = BOTH)
CID	Rectangular coordinate system that defines the y-axis of the spline and the xy- and yz- planes for bending. Not used for bodies, CAERO2. (Integer ≥ 0)
METHOD	Method for the spline fit. Either FBS3 or FBS6. See Remark 5. (Character; Default = FBS6)
DZR	Rotational attachment flexibility. (Real ≥ 0.0 ; Default = 0.0)
IA2	Ratio of the beam bending stiffness to the beam extensional stiffness. (Real > 0.0; Default = 1.0)
EPSBM	Ratio of the minimum beam length to the total beam length. See Remark 7. (Real > 0.0; Default = 0.01)

Remarks:

1. The flexibilities DZ and DZR are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing). The DZ and DZR values are used to derive stiffness of the translational and rotational (respectively) bushing stiffnesses. Bushing elements are placed between the interpolating beam and the connections to the dependent and independent grids.
2. For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFACT entry). For all other aero methods, the AELIST items are box id's.
3. DTOR is the ratio of axial rotational to bending deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily translational deflection will occur. The values will affect the results only if the structural grids over constrain the motion of the interpolating beam. Slender bodies have no torsional motion, so these values will not be used for CAERO2 entries.
4. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
5. The FBS3 method will map only the three translational degrees of freedom. The FBS6 method will map all six degrees of freedom.
6. CAERO2 entries are not supported. The CAERO entry may be blank unless points from a CAERO4 or a CAERO5 are specified. The list of splined points may span multiple aerodynamic components. If the spline defines points from multiple CAERO4 or CAERO5 entries, then any one of the referenced CAERO IDs is valid input.
7. EPSBM is the minimum length of a SPLINE7 beam as a fraction of the total SPLINE7 FEM length. Thus, an EPSBM value of 0.01 (Default), would produce a minimum beam length that is 1% of the total FEM length. Defining a value of EPSBM that is smaller than 0.01 will decrease the minimum length and potentially increase the number of beams used to define the SPLINE7 FEM.

SPLINEX**Externally-Evaluated Spline**

Defines the input for a spline that will be evaluated with a user-supplied procedure.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINEX	EID	GROUP	DGCOMP	IGCOMP	DECOMP	IECOMP		USAGE	
	AELIST	AEFACT	AELISTC						

Example:

SPLINEX	3	SPLNGRP4	GWNG1A	GWNG1S				BOTH	
	101	201	301						

Descriptor**Meaning**

EID	Element identification number. (Integer > 0)
GROUP	Group name to which the external spline type belongs. (Character; no Default)
DGCOMP	The name of an AECOMP or AECOMPL entry that defines the set of points for the dependent mesh. See Remarks 3. and 4. (Character; Default = Blank).
IGCOMP	The name of an AECOMP or AECOMPL entry that defines the set of points for the independent mesh. See Remarks 3. and 4. (Character; Default = Blank).
DECOMP	The name of an AECOMP or AECOMPL entry that defines the set of elements for the dependent mesh. See Remarks 3., 4. and 5. (Character; Default = Blank).
IECOMP	The name of an AECOMP or AECOMPL entry that defines the set of elements for the independent mesh. See Remarks 3., 4. and 5. (Character; Default = Blank).
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 2. (Character; Default = BOTH)
AELIST	ID of an AELIST that contains a list of user-defined integer data. See Remark 6. (Integer; no Default).
AEFACT	ID of an AEFACT that contains a list of user-defined real data. See Remark 6. (Integer; no Default).
AELISTC	ID of an AELISTC that contains a list of user-defined character data. See Remark 6. (Integer; no Default).

Remarks:

1. The SPLINEX EID must be unique with respect to all SPLINEi entries.

2. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

$F_g = [GPkg]^T \{Pk\}$ (FORCE/BOTH splines are in the transform)

$U_k = [GDkg] \{Ug\}$ (DISP/BOTH splines are in the transform)

In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship $[GPkg]^T = [GDkg]$ satisfied. The default behavior (BOTH for all splines) is compatible with version of MSC Nastran prior to Version 70.5.

In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from its effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCEs from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform "rigid aerodynamic" analyses).

3. Typically, for aero-to-structure splines "dependent" means "aerodynamic", and "independent" means "structural".
4. If the component defines a structural mesh, then the grid component may be left blank and the list of grids will be obtained from the element component member's connectivity. Both may not be left blank.
5. Structural elements referenced by DECOMP and IECOMP are limited to the following element types: CQUAD4, CQUADR, CTRIA3, CTRIAR. In one list, elements from the different types may not share the same ID.
6. The data that are defined on the AELIST, AEFACT, and AELISTC have no meaning to Nastran. These lists are generic containers for data that has meaning to the spline server. Note that the AELIST is limited to numbers greater than zero.

SPLINRB**Rigid Body Spline**

Defines a rigid body spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINRB	SID	CAERO	AELIST	USAGE	G1	C1	G2	C2	
	G3	C3	G4	C4	G5	C5	G6	C6	

Example:

SPLINRB	110	20	2	BOTH	1093	123456			
---------	-----	----	---	------	------	--------	--	--	--

Descriptor	Meaning
SID	Unique spline identification number. (Integer > 0)
CAERO	Identification number of aerodynamic component that defines the interpolation surface. (Integer > 0 or blank)
AELIST	Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 3. (Integer > 0)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. Legal values are FORCE, DISP or BOTH. (Character; Default = BOTH)
Gi	Identification number of a structural grid point. (Integer > 0)
Ci	Component numbers: Any unique combination of the Integers 1 through 6 with no embedded blanks.

Remarks:

- Up to six structural grid points can be used to select exactly 6 structural degrees-of-freedom that define the motion of the rigid body.
- The selected degrees-of-freedom must define statically determinate supports of the rigid body.
- For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFACT entry). For all other aero methods, the AELIST items are box id's.

SPOINT

Scalar Point Definition

Defines scalar points.

Format:

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

Example:

SPOINT	3	18	1	4	16	2			
--------	---	----	---	---	----	---	--	--	--

Alternate Format and Example:

SPOINT	ID1	"THRU"	ID2						
SPOINT	5	THRU	649						

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

Remarks:

1. A scalar point defined by its appearance on the connection entry for a scalar element (see the CELASI, CMASSi, and CDAMPi entries) need not appear on an SPOINT entry.
2. All scalar point identification numbers must be unique with respect to all other structural, scalar, fluid and extra (EPOINT) points. However, duplicate scalar point identification numbers are allowed in the input.
3. This entry is used primarily to define scalar points appearing in single-point or multipoint constraint equations to which no scalar elements are connected.
4. If the alternate format is used, all scalar points ID1 through ID2 are defined.
5. For a discussion of scalar points, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) (p. 174) in the *MSC Nastran Reference Guide*.

SPRELAX

Spline Relaxation

Defines relaxation of a spline based on an adjacent spline.

Format:

1	2	3	4	5	6	7	8	9	10
SPRELAX	SID1	SID2	LIST2	DREF	LIST1				

Example:

SPRELAX	140	130	50	5.0					
---------	-----	-----	----	-----	--	--	--	--	--

Descriptor	Meaning
SID1	Identification number of spline to be modified (may be a blended spline and need not be unique). (Integer > 0)
SID2	Identification number of adjacent spline (may be a blended spline). (Integer > 0)
LIST2	Identification of an AELIST entry listing aerodynamic grid points that define the curve used in the relaxation . (Integer > 0) (See Remark 1.)
DREF	Reference Distance (Real > 0.) (See Remark 2.)
LIST1	Identification number of an AELIST entry listing aerodynamic grid points of the spline to be modified (optional). (Integer > 0; Default = 0) (See Remark 3.)

Remarks:

1. The aerodynamic grid points referenced by LIST2 have to define a curve. The curve need not be contiguous, i.e., coincident grid points are allowed. The order of the grid points is arbitrary.
2. Displacements of spline 1 are modified according to

$$u_{1mod} = u_1 + f\left(\frac{r}{DREF}\right)(u_2 - u_1)$$

where r is the shortest distance of the aerodynamic grid point considered from the curve, u_2 is the displacement from spline 2, interpolated to the position on the curve which is closest to the aerodynamic grid point considered, and function f is defined by

$$f(x) = \begin{cases} 1-x & \text{if } x \leq 1 \\ 0 & \text{if } x > 1 \end{cases}$$

3. If LIST1 is defined, only aerodynamic grid points contained in the referenced list are processed. Otherwise, all aerodynamic grid points of spline 1 are processed.

STOCHAS

Randomization of Model Parameters

Specifies statistics used in randomization selected model parameters.

Format:

1	2	3	4	5	6	7	8	9	10
STOCHAS	SID	PENTRY	CDF	CoV	m				
		MENTRY	CDF	CoV	m				
		CENTRY	CDF	CoV	m				
		LOADS	CDF	CoV	m				
		SPCD	CDF	CoV	m				

Example 1: (Randomize all element and material properties with the default settings.)

STOCHAS	100	PENTRY							
		MENTRY							

Example 2: (Randomize material properties with CoV = 0.1 and loadings with CoV = 0.3 and default multipliers of standard deviations.)

STOCHAS	200	LOADS	GAUSS	0.3					
		MENTRY	GAUSS	0.1					

Descriptor	Meaning
SID	Unique identification number that is selected by the STOCHASTICS Case Control command. (Integer > 0)
“PENTRY”	Flag for randomizing real values on all the element property entries. (Character)
“MENTRY”	Flag for randomizing real values on all the independent material property entries. (Character)
“CENTRY”	Flag for randomizing real values on all the connectivity entries. (Character)
“LOADs”	Flag for randomizing real values on all the load entries. (Character)
“SPCD”	Flag for randomizing real values on all the SPCD entries. (Character)
CDF	Name of a cumulative distribution function. See Remark 2. (Character; Default = GAUSSIANS or blank).
CoV	Coefficient of variance. (Real > 0; Default = 0.05)
m	Number of standard deviations. See Remark 3. (Real > 0; Default = 3.0)

Remarks:

- At least one flag must exist but they can be placed in any order.
- Currently, only Gaussian distributions are supported.

3. The range of a random variable is defined as $(\mu - m \cdot \sigma, \mu + m \cdot \sigma)$ where μ is the mean of the random variable (or the value of one analysis model parameter on a Bulk Data entry), σ is the standard deviation that is related to μ , CoV by $\sigma = \text{CoV} \cdot \mu$ and m is the multiplier of the standard deviations.
4. $m \cdot \text{CoV}$ must be < 1.0 .

SUPAX

Conical Shell Fictitious Support

Defines determinate reaction degrees-of-freedom in free bodies for conical shell analysis.

Format:

1	2	3	4	5	6	7	8	9	10
SUPAX	RID1	HID1	C1	RID2	HID2	C2			

Example:

SUPAX	4	3	2						
-------	---	---	---	--	--	--	--	--	--

Descriptor	Meaning
RID _i	Ring identification number. (Integer > 0)
HID _i	Harmonic identification number. (Integer ≥ 0)
C _i	Conical shell degree-of-freedom numbers. (Any unique combination of the Integers 1 through 6.)

Remarks:

1. SUPAX is allowed only if an AXIC entry is also present.
2. Up to 12 degrees-of-freedom may appear on a single entry.
3. Degrees-of-freedom appearing on SUPAX entries may not appear on MPCAX, SPCAX, or OMITAX entries.
4. For a discussion of conical shell analysis, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.

SUPPORT

Fictitious Support

Defines determinate reaction degrees-of-freedom in a free body.

Format:

1	2	3	4	5	6	7	8	9	10
SUPPORT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

SUPPORT	16	215							
---------	----	-----	--	--	--	--	--	--	--

Descriptor	Meaning
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer 0 or blank for scalar points. Any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. The SUPPORT entry specifies reference degrees-of-freedom for rigid body motion. It is not intended to be used in place of a constraint (i.e., SPCi entry or PS on the GRID entry).
2. SUPPORT and/or SUPPORT1 entries are required to perform inertia relief in static analysis (SOL 101) if PARAM,INREL,-1 is specified. But if PARAM,INREL,-2 is specified, then SUPPORT and/or SUPPORT1 entries are not required.
3. Be careful not to spell SUPPORT with two Ps.
4. Degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
5. From 1 to 24 support degrees-of-freedom may be defined on a single entry.
6. See [Rigid Body Supports](#) in the *MSC Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
7. An alternative to SUPPORT is the SUPPORT1 entry, which is requested by the SUPPORT1 Case Control command.
8. The SUPPORT entry is not allowed in SOLs 106, 129, and 400.
9. The SUPPORT entry may not be used in contact in SOL 101 as this is tantamount to executing SOL 400. Additionally, INREL, -2 will not in general converge in SOL101 if run with contact. Inertial Relief is supported in SOL400 and invoked with the Case Control Command IRLOAD. Therefore, SOL101 runs with contact should be converted over to SOL400.

SUPPORT1

Fictitious Support, Alternate Form

Defines determinate reaction degrees-of-freedom (r-set) in a free body-analysis. SUPPORT1 must be requested by the SUPPORT1 Case Control command.

Format:

1	2	3	4	5	6	7	8	9	10
SUPPORT1	SID	ID1	C1	ID2	C2	ID3	C3		
	ID4	C4	-etc.-						

Example:

SUPPORT1	5	16	215						
----------	---	----	-----	--	--	--	--	--	--

Descriptor	Meaning
SID	Identification number of the support set. See Remark 1. (Integer > 0)
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer 0 or blank for scalar points. Any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. The SUPPORT entry specifies reference degrees-of-freedom for rigid body motion. It is not intended to be used in place of a constraint; (i.e., SPCi entry or PS on the GRID entry).
2. SUPPORT and/or SUPPORT1 entries are required to perform inertia relief in static analysis (SOL 101) if PARAM,INREL,-1 is specified. But if PARAM,INREL,-2 is specified, then SUPPORT and/or SUPPORT1 entries are not required.
- In SOL 101, PARAM,INREL,-1 must also be specified or the SUPPORTi entries will be treated as constraints.
3. SUPPORT1 must be requested by the SUPPORT1 Case Control command. The degrees-of-freedom specified on SUPPORT1 will be combined with those on the SUPPORT entry.
4. Be careful not to spell SUPPORT with two Ps.
5. Degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
6. From 1 to 18 support degrees-of-freedom may be defined on a single entry.
7. See [Rigid Body Supports](#) in the *MSC Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
8. In superelement analysis, SUPPORT1 may be specified for points belonging to the residual structure only.

9. The SUPPORT1 entry is not allowed in SOLs 106, 129, and 400.
10. The SUPPORT1 entry may not be used in contact in SOL 101 as this is tantamount to executing SOL 400. Additionally, INREL, -2 will not in general converge in SOL101 if run with contact. *Inertial Relief is supported in SOL400 and invoked with the Case Control Command IRLOAD.* Therefore, SOL101 runs with contact should be converted over to SOL400.

SUPPORT6

Inertia Relief for SOL 600

Inertia relief used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
SUPPORT6	SID	METH	IREMOV				IDS1		

Example:

SUPPORT6	0	3	1				101		
SUPPORT6	4	3	-2						

Descriptor	Meaning
SID	Set ID corresponding to a Case Control SUPPORT1 entry or zero. (Integer; Default = 0)
0	If this is the only SUPPORT6 entry, use this SUPPORT6 entry for all subcases. If there are multiple SUPPORT6 entries, use the one with SID=0 for Marc increment zero.
N	Use this SUPPORT6 entry for the subcase specified by Case Control SUPPORT1=N.
	Different SUPPORT6 entries can be used for each subcase if desired and different subcases can use different methods.
	If there is only one SUPPORT6 entry (with SID=0), no Case Control SUPPORT1 entries are necessary.
METH	Method to use (Integer; Default = 0)
0	Inertia relief is not active for this subcase.
3	Use the “Support Method”, usually specified using param,inrel,-1 for other solution sequences. (See Remark 1.) Input will come from all SUPPORT entries and those SUPPORT1 entries with ID=SID.
IREMOV	Method to retain or remove inertia relief from a previous subcase (Integer; Default = 1).
1	Retain inertia relief conditions from previous subcase.
-1	Remove inertia relief loads immediately.
-2	Remove inertia relief loads gradually

Descriptor	Meaning
	IREMOV should be blank or 1 unless METH is 0.
IDS1	ID of SUPORT1 entries to be used if METH=3 and SID=0 (Integer; no Default). For METH=3, only SUPPORT1 entries with ID=IDS1 will be used in Marc increment zero. All SUPPORT entries will be used. (Used for METH=3 when SID=0 ONLY.)

Remark:

1. The parameter INREL is ignored by SOL 600.

SURFINI**Eulerian Initialization Surface**

Defines a surface that is used for initialization of regions of an Eulerian mesh. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
SURFINI	VID	BSID	COVER	REVERSE	CHECK				

Example:

SURFINI	100	37							
---------	-----	----	--	--	--	--	--	--	--

Descriptor	Meaning	
VID	Unique number of an SURFINI region entry. Referenced from TICEUL1. (Integer > 0; Required)	
BSID	ID of a BSURF entry defining the initialization surface. (Integer > 0; Required)	
COVER	The processing strategy for Eulerian elements inside and outside of the initialization surface. (Character; Default = INSIDE)	
	INSIDE	The part of the Eulerian elements that lie inside the closed volume of the initialization surface will obtain the initial conditions belonging to that surface.
	OUTSIDE	The part of the Eulerian elements that lie outside the closed volume of the initialization surface will obtain the initial conditions that belong to that surface
REVERSE	Auto reverse switch for SURFINI surface segments. (Character; Default = ON)	
	ON	If necessary, the normals of the SURFINI surface segments are automatically reversed so that they all point in the same general direction and give a positive closed volume.
	OFF	The segments normals are not automatically reversed.
CHECK	Checking switch for SURFINI surface segments. (Character; Default = ON)	
	ON	The normals of the segments are checked to see whether they all point in the same general direction and give a positive closed volume.
	OFF	The segment normals are not checked.
	When "REVERSE" is set to "ON", "CHECK" is automatically set to "ON".	

Remarks:

1. All initialization surfaces must form a multifaceted closed volume.

2. An initialization surface can only be used to initialize regions in a Eulerian mesh with appropriate initial conditions. An initialization surface cannot be used as a coupling surface, contact surface or rigid surface.
3. The normal vectors of all segments that form the initialization surface must point in the same general direction, and result in a positive, closed volume. Setting the “REVERSE” option to “ON” ensures that this condition is satisfied, regardless of how segments are initially defined.
4. The “COVER” option determines how Eulerian elements that are (partially) inside or outside of the initialization surface are processed.

SWLDPRM

Parameters for CFAST, CWELD, and CSEAM Connector Elements

Overrides default values of parameters for CFAST, CWELD, and CSEAM connectivity search.

Format:

1	2	3	4	5	6	7	8	9	10
SWLDPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	PARAM4	VAL4	
	PARAM5	VAL5	-etc.-						

Example:

SWLDPRM	GSPROJ	15.0	GSMOVE	2	PRTSW	1			
---------	--------	------	--------	---	-------	---	--	--	--

Alternate Format and Examples:

1	2	3	4	5	6	7	8	9	10
SWLDPRM	PARAM1	VAL1	PARAM2	VAL2	etc.	CFAST	PARAM1	VAL1	
	PARAM2	VAL2	etc.	CWELD	PARAM1	VAL1	PARAM2	VAL2	
	etc.	CSEAM	PARAM1	VAL1	PARAM2	VAL2	etc.		

SWLDPRM	CHKRUN	2							
	CWELD	GSMOVE	2	PROJTOL	.04	PRTSW	1	CSEAM	
	PROJTOL	0.06	GMCHK	1					

SWLDPRM	CHKRUN	2							
	CWELD	GSMOVE	2	PROJTOL	.04	PRTSW	1		
	CSEAM	PROJTOL	0.06	GMCHK	1				

Descriptor	Meaning
PARAMi	Name of the connector parameter. Allowable names are listed in Table 9-36 . (Real or Integer)
VALi	Value of the parameter. See Table 9-36 (Real or Integer)
CFAST, CWELD, CSEAM	Keywords to control element type specific parameters. Any parameter following a keyword is applied only to that element type. See Remarks 2 and 3. (Character)

Table 9-36 PARAMi Names and Descriptions

Name	Type	Default	Description
ACTVTOL (SOL 600)	Integer ≥ 0 Integer ≤ 2211	1111	<p>Parameter controlling the behavior of PROJTOL for the different CWELD connection methods. This parameter is entered as an integer and is converted to a four-character string. If its value is less than 1000, the string will be prepended with zeros. The first character (from the left) controls the behavior when method PARTPAT is used. The second controls the behavior when method ELPAT is used. The third controls the behavior when method ELEMID is used and the fourth controls the behavior when method GRIDID is used.</p> <p>For ALIGN the PROJTOL tolerance has no significance. Each digit (d_i) in the string can have the value 0 or 1 or 2, where the value 2 only has significance for the methods ELPAT or PARTPAT. The values have the following meaning:</p> <ul style="list-style-type: none"> 0 = PROJTOL is completely deactivated 1 = PROJTOL is activated for ELEMID and GRIDID, PROJTOL is activated in initial projections for ELPAT, PROJTOL is only activated over free edges of the patch in auxiliary projections for ELPAT and in initial and auxiliary projections for PARTPAT. Free edges have no neighbors within the set that defines the complete surface. 2 = PROJTOL is always activated
CHKRUN	Integer 0, 1, 2	0	<p>Stop run or allow run to continue after the connectivity elements are generated.</p> <p>0=abort on first error; 1=stop after connectivity has been checked; 2=continue run if no errors are found.</p>
CNRAGLI	90.0 \leq Real \leq 180.0 or -1.0	160.0	CSEAM only. Minimum angle allowed between the free edges of shell elements EIDSA and EIDEA or shell elements EIDSB and EIDEB. The CSEAM will not be generated if the angle is less than the value of CNRAGLI. If set to -1.0, the check will be skipped.
CNRAGLO	0.0 \leq Real \leq 90.0 or -1.0	20.0	CSEAM only. Maximum angle allowed between the normal vectors of shell elements EIDSA and EIDEA or shell elements EIDSB and EIDEB. The CSEAM will not be generated if the angle is greater than the value of CNRAGLO. If set to -1.0, the check will be skipped.

Table 9-36 PARAMi Names and Descriptions

Name	Type	Default	Description
CSVOUT	Integer ≥ 0	0	<p>Print diagnostic output in a comma separated .csv file useful for reports. The users may view or manipulate data using Microsoft Excel, or other spread sheet programs that can process CSV format.</p> <p>0 no .csv output.</p> <p>unitnum > 0 unit number of the .csv file assigned via the ASSIGN statement in the File Management Section, for example,</p> <p>ASSIGN USERFILE=myfile.csv UNIT=<i>unitnum</i> FORM=FORMATTED DELETE STATUS=NEW</p>
CWSETS (SOL 600)	Integer > 0	0	<p>Parameter to control the automatic creation of four element sets with the elements involved in the CWELD connections.</p> <p>0 = the sets will not be created</p> <p>1 = four sets will be created automatically “fastener_all_beams_inc0000”, the set containing all connector beam elements. “fastener_all_faces_sidea_inc0000”, the set containing all elements with patches on side B of the connection. “all_fastener_warnings_inc0000”, the set containing all elements involved in CWELD warning messages.</p> <p>Defining sets with any of these names must be avoided and will be considered an error.</p>
CWSPOT (SOL 600)	$0 < \text{Integer} < 3$	1	<p>Parameter to choose the method for modifying the beam length.</p> <p>1 = scale the stiffness of the beam 2 = reposition the end nodes of the beam 3 = reposition the auxiliary patch nodes and dthe end nodes of the beam.</p>
DELMAX (SOL 600)	Real	0.1	Maximum allowable parametric coordinate change during the iteration process for finding the projection on a patch. At first DELMAX is not activated, i.e., the parametric coordinate change is not limited during the iteration process. The parameter is only activated when the full Newton Raphson iteration process for a projection did not converge. In that case the iteration process is restarted with DELMAX activated.
DLDMAX (SOL 600)	Real > 0.0	5.0	Default value for LDMAX, the largest ratio of length to characteristic diameter.

Table 9-36 PARAMi Names and Descriptions

Name	Type	Default	Description
DLLMIN (SOL 600)	Real > 0.0	0.2	Default value for LDMIN, the smallest ratio of length to characteristic diameter.
DRATIO	1.0 ≤ Real ≤ 10.0	1.0	CFAST or CWELD (PARTPAT and ELPAT) only. Increases the Connector patch diameter and is used to stiffen the connector patch when a loss of stiffness occurs for the FEM model as mesh size is made smaller. See Remark 10.
EPSITR (SOL 600)	Real > 0.0	1.0E-5	Tolerance to terminate the iteration process for finding the projection on a patch. If the parametric coordinate change in an iteration is less than EPSITR the projection is accepted as converged.
GMCHK	Integer 0, 1, 2, 3	0	<p>For CSEAM, CWELD with ELPAT or PARTPAT format, and CFAST only.</p> <p>0=no geometry error checks;</p> <p>1=check errors of the CSEAM across a cutout or over a corner with patch elements in plane or out of plane;</p> <p>2=check errors of CSEAM and output all candidate shell elements if an error is encountered.</p> <p>If GMCHK=1 or 2 and an error is detected, the program will loop back to search for next candidate element until a good pair of connection is found or all adjacent elements have been checked. In the latter case, a user fatal message 7595, 7638, or 7667 will be issued. A UFM 7595 is issued if the normal angles between the patches at end GS or the patches at end GE exceed the value of GSPROJ; a UFM 7638 is issued if either the length of the seam spans more than three elements or the seam spans a cutout; a UFM 7667 is issued if the normal angles between the top patches at GS and GE or the normal angles between the bottom patches at GS and GE exceeds CNRAGLO or if the angle between the free edges of the shell elements onto which GS and GE are projected is less than CNRAGLI.</p> <p>3=check backward projections for CWELD with ELPAT or PARTPAT format and CFAST. See Remark 8.</p>

Table 9-36 PARAMi Names and Descriptions

Name	Type	Default	Description
GSMOVE	Integer	0	Maximum number of times GS for the CFAST or CWELD (PARTPAT or ELPAT options only) or GS/GE for the CSEAM is moved in case a complete projection of all auxiliary points has not been found. See Remark 9.
MOVGAB	Integer 0, 1	0	<p>Option to correct the locations of user defined GA/GB for CFAST and CWELD elements.</p> <p>0 = keep the locations of the user specified GA/GB and connect them to shell elements directly.</p> <p>1 = generate new grids with corrected locations to connect shell elements.</p>
GSPROJ	$0.0 \leq \text{Real} \leq 90.0$ or -1.0	20.0 (-89.0 for SOL 600)	Maximum angle allowed between the normal vectors of shell A and shell B. The connector element will not be generated if the angle between these two normal vectors is greater than the value of GSPROJ. For CSEAM, see also GMCHK for additional error checks using GSPROJ. If GSPROJ is set to -1.0 (or -89.0 for SOL 600), the program will skip the checking of GSPROJ.
GSTOL	$\text{Real} \geq 0.0$	0.0	For CFAST or CWELD (PARTPAT and ELPAT only), if GSTOL > 0.0 and the distance between GS and the projected point GA or GB is greater than the GSTOL, a UFM 7549 is issued and the CFAST or CWELD is rejected. For CSEAM, if GSTOL > 0.0 and the distance between GS and the projected point GSA or GSB or the distance between GE and the projected point GEA or GEB is greater than the GSTOL, a UFM 7549 is issued and the CSEAM is rejected.
MAXEXP (SOL 600)	Integer > 0	2	Parameter to control the maximum number of expansions in the search for projections of the auxiliary nodes. First the master patch will be searched. If no projection is found on the master patch a first expansion will be made including all neighboring patches of the master patch. If no projection is found on any of the new patches a second expansion will be made including all neighbors of the patches tried so far. This process continues until the number of expansions exceeds MAXEXP. Two patches are neighbors if they share at least one node in their connectivities.

Table 9-36 PARAMi Names and Descriptions

Name	Type	Default	Description
MAXITR (SOL 600)	Integer > 0	20	The maximum number of iterations allowed in the iteration process for finding the projection on a patch.
NREDIA	Integer 0, 1, 2, 3, 4, 5, 6, 7, 8	0	CFAST or CWELD (PARTPAT and ELPAT) only. Maximum number of times the diameter D is reduced in half in case a complete projection of all points has not been found.
PROJTOL	$0.0 \leq \text{Real} \leq 1.0$	0.02 (0.199 for SOL 600)	<p>For CFAST or CWELD, tolerance to accept the projected point GA or GB if the computed coordinates of the projection point lie outside the shell element but is located within $\text{PROJTOL} \times (\text{dimension of the shell element forming the patch})$. For the CSEAM, a projection from GS/GE will always be attempted as if PROJTOL=0.0 and if one cannot be found then the non-zero value of PROJTOL will be used.</p> <p>Regardless of the value of SWLDPRM PROJTOL, the algorithm starts by assuming a zero projection tolerance for the projections of GA/GB for the CWELD option "PARTPAT" or the CFAST option "PROP" and for GAHi/GBHi for the CWELD options "PARTPAT" and "ELPAT" and any CFAST option. The tolerance is increased by 0.02 until a projection is found or the PROJTOL value is reached.</p> <p>This can be turned off while computing the auxiliary grid projection onto EIDA/EIDB or onto the adjacent elements of EIDA/EIDB by setting PROJTOL= - <i>value</i> where $0.0 \leq \text{value} \leq 1.0$. In this case, the projection calculation starts at tolerance = PROJTOL. For the rest of the projection search, the algorithm reverts back to the iteration starting at tolerance = 0.0.</p>
WMASS	Integer 0, 1	0	For CWELD to react to a non-zero density 1 value is required.

Table 9-36 PARAMi Names and Descriptions

Name	Type	Default	Description
PRTSW	Integer 0, 1, 2 , 11, 12, 100, 101, 111	0 (2 for SOL 600)	<p>Print diagnostic in output or punch out internally generated RBE3 elements and auxiliary grids in Bulk Data format for the connector elements.</p> <p>0=no diagnostic output (appropriate USER MESSAGES such as 'USER WARNING MESSAGE 7636 will continue to be issued);</p> <p>1=print diagnostic output in exponential format to f06 file;</p> <p>2=punch diagnostic output in exponential format to .pch file;</p> <p>11=print diagnostic output in real format to .f06 file;</p> <p>12=punch diagnostic output in real format to .pch file.</p> <p>100=punch out Bulk Data without diagnostic output.</p> <p>101= punch out Bulk Data and print diagnostic output in exponential format to f06 file.</p> <p>111= punch out Bulk Data and print diagnostic output in real format to f06 file.</p>
RBE3WT (SOL 600)	Real	0.0	<p>Default RBE3 distance weighting exponent.</p> <p>The weight factor for each retained node in a RBE3 involved in a CWELD connection is: $f_i = 1/d_i^n$, where:</p> <p>f_i is the weighting factor for retained node 1.</p> <p>d_i is the distance from the tied node to retained node i</p> <p>n is the weighting exponent RBE2WT</p> <p>Negative values for RBE3WT are not recommended, since they will result in heavier weighting for nodes further away. The default results in uniform weighting ($f_1 = 1$).</p>

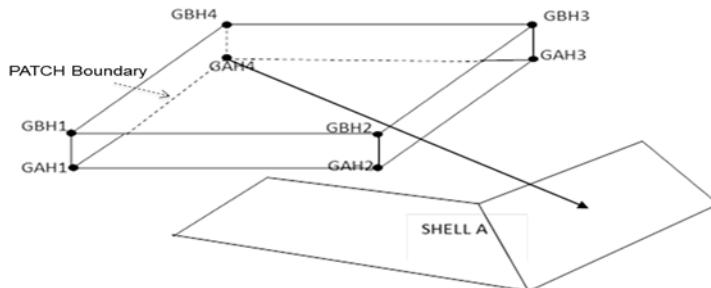
Table 9-36 PARAMi Names and Descriptions

Name	Type	Default	Description
GSCURV (SOL 600)	-90 < Real < 90	20.0	<p>Maximum angle allowed between the normal vectors of a patch to which an auxiliary node projects and its corresponding auxiliary and master patches. It provides a measure to monitor the curvature of a surface and to recognize patches that belong to, for example, stiffeners. A connection is not generated if the angle between the normal vectors is greater than 90-GSCURVE meaning that the patches are almost normal to each other. In that case, the patch is reflected and the search proceeds to the next patch in the list. If the angle is between zero and GSCURV, no message is displayed. If the angle is between GSCURV and 90-GSCURV, a large angle warning is displayed. The following three tests are performed in the order given below when GSCURV is positive:</p> <p>If $0 < \text{angle} < \text{GSCURVE} \geq \text{OK}$ If $\text{GSCURVE} < \text{angle} < 90-\text{GSCURV} >$ trigger a warning. If $\text{angle} > 90-\text{GSCURV} \geq \text{reject}$.</p> <p>Note that the warning condition is never triggered when $\text{GSCURV} > 45$ as it is overruled by the reject condition.</p> <p>If GSCURV is negative, the projection is always accepted and a warning is issued when the angle is larger than GSCURV.</p>
SCLSKIN	Real ≥ 0.0	0.0	CFAST or CWELD (PARTPAT and ELPAT) only. If set to a value > 0.0 the CFAST or CWELD connector will be stiffened by increasing the bending moment of inertia ratio by SCLSKIN of the shell elements involved in the connector patch. SCLSKIN is Factor used to stiffen the connector patch when a loss of stiffness occurs for the FEM model as mesh size is made smaller. An initial recommended value is 0.1; See Remark 10..

Remarks:

1. This entry changes the default settings of control variables for the CFAST, CWELD, and CSEAM connector elements. None of the parameters of this entry are required. Only one SWLDPRM entry is allowed in the Bulk Data Section.

2. If any of the key words CFAST, CWELD, and CSEAM does not appear on this entry, then a parameter set on this entry is considered "global" and applies to all the connector elements in the model. Any parameter set on this entry that comes before a key word CFAST, CWELD, or CSEAM is considered global.
3. Any parameter set on the entry that comes after a key word such as CFAST will only apply to that connector element type until another key word such as CSEAM is encountered. If a parameter is defined for a specific connector type that does not apply to that connector type then it will be ignored.
4. Blank fields are allowed for readability. However, a parameter name must be followed in the immediately following field by the corresponding parameter value. If the parameter name falls in the field just before a continuation field, then its parameter value must be placed in the first field after the continuation marker of the continuation entry.
5. Connectivity information is generated for the CFAST and CSEAM elements. For the CWELD elements, connectivity information is only generated for the PARTPAT, ELPAT, ELEMID, and GRIDID options.
6. The details of individual connector connectivity can be found on the appropriate CFAST, CWELD, and CSEAM Bulk Data entries.
7. The CHKRUN parameter must be global.
8. Backward connections sometimes occur if the patch is near the boundary of a structure and there is a "vertical" flange associated with the patch elements. In this case GMCHK=3 may be used to prevent backward projection. See the figure below. If GMCHK=3 and a backward projection is detected, the program will not connect this projected shell element. Instead, it will continue searching iterations until a satisfying connection is reached.

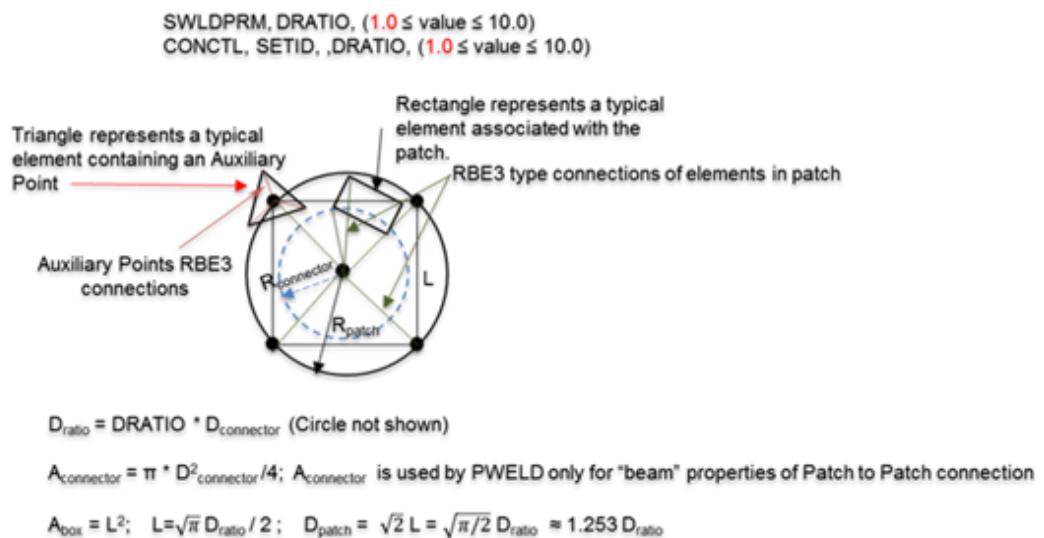


9. If the GMOVE specification limit is reached for the CFAST or the CWELD with options "PARTPAT" and "ELPAT" and SWLDPRM NREDIA ≥ 0 ; then the diameter of the connector will be reduced by half to compute new locations of auxiliary grids. If necessary this is repeated until the NREDIA specified value is reached.
 - a. When the NREDIA $\neq 0$ is initiated, the GS at its current location is used for GMOVE ≥ 0 .
 - b. When the NREDIA $\neq 0$ is initiated, the GS at its original location is used for the new option GMOVE < 0 .

10. Connector contribution to a structural model's overall stiffness is sensitive to the model's mesh size and the orientation of the connector relative to the mesh. Thus, the discretization process itself may cause, for example, a model using a fine mesh to be stiffer in torsion than a corresponding model using a coarse mesh. Also for production models that correlate well with test, refining the mesh may cause an inherent overall loss of stiffness due to mesh refinement and hence loss of correlation.

To allow the user some control over model stiffness, the new connectors (CWELD with ELPAT or PARTPAT or CFAST) are provided with two options to provide additional connector stiffness. The two options may be used individually or in combination.

The first stiffening technique is activated by "SWLDPRM, DRATIO, (1.0 ≤ value ≤ 10.0)" or "CONCTL, SETID, ,DRATIO, (1.0 ≤ value ≤ 10.0)". For this option the diameter, D_{ratio} , is defined as $D_{ratio} = DRATIO * D_{connector}$. This results in the diameter of the patch taking a value of $D_{patch} = \sqrt{\pi/2} D_{ratio}$. The default of DRATIO is a value=1.0 which implies the diameter of the patch is computed in the standard fashion. For the patch to patch connection for the "beam" properties of the CWELD, the area is still computed as $A_{connector} = \pi D_{connector}^2 / 4$ as defined in the PWELD entry.



A *disadvantage* of this method is that as DRATIO is increased using the global command SWLDPRM, DRATIO, *value*"; some connector elements may begin to fail because they may no longer be able to find a patch projection.

To overcome this, the "SWLDPRM, NREDIA, *Integer_value*" can be increased to a value as high as *Integer_value* = 8 to allow failing welds to halve their patch diameters up to eight times. Whenever a connector has its diameter reduced a message such as: "DA IS REDUCED BY HALF TO 8.8623E+00" when PRTSW diagnostic output is requested. In this message the DA is computed as : DA = L/2.

If the "SWLDPRM, NREDIA, *Integer_value*" is not an approach the user wishes to pursue, then for these failing elements, the bulk data entry CONCTL, SETID, ,DRATIO, value can be used to define a set for failing connectors and set a value of DRATIO for these connectors that allows them to find a projection.

The second stiffening algorithm attempts, based on the diameter of the connector, to determine a measure of the mesh discretization.

This feature is activated by setting "SWLDPRM, SCLSKIN > 0.0" or "CONCTL, SETID, SCLSKIN > 0.0". The default is a value = 0.0 which implies no stiffening. When activating the "SWLDPRM, SCLSKIN, real value" option for the first time with a new FEM, it is recommended that an initial value = 0.1 be used.

Depending on the complexity of the model and the overall mesh size and the number of connectors within the model and the diameter of the connectors relative to the mesh, the *default value* tends to stiffen a structural model from about 0.4% to about 4%. A value of SCLSKIN=10.0 stiffens coarser mesh models by about 10% to 11% and finer mesh models by about 2% to 6%.

The contribution of the stiffening algorithm to the overall stiffness of the FEM model eventually reaches a limit. For example, a very large value SCLSKIN=100 increases the stiffness of the models overall by only about 0.1% to 2% over the stiffness obtained for SCLSKIN=10.

For a correlated structural model evaluated at a specific mesh size, *with an aim to refine the mesh for some portion of this model containing connectors*, while leaving other portions containing connectors with an unmodified mesh, it is recommended that the "SCLSKIN, *real value*" be entered on the CONCTL bulk data entry referring to the connectors within the area of the refined mesh. Different refined mesh areas within the structural model can have different values of SCLSKIN associated to the specific connectors in each refined region.

For post processing affected shell elements, an updated EPT table is available after module MODGM2. It contains the PSKNSHL record that correlates the property data of the shells involved and a list of shell elements for each patch modified.

Neither of these stiffening methods have any effect on the mass computations of the model.

11. This entry is ignored in part super-elements. Inside the part super-elements, the default settings will be applied.
12. If Modules are present then this entry may only be specified in the main Bulk Data section.

Entries T-Y

TABD1MD

TABLED1 entries internal modification in SOL 600

Defines how TABLED1 entries are internally modified in SOL 600.

SOL 600 usually requires that the first point of all TABLED1 entries used to describe time histories start with time/amplitude of (0.0, 0.0). If the user omits this entry, all TABLED1 entries that do not start with (0.0, 0.0) will be modified internally to add two points at the start, the first at (0.0, 0.0) and the second at (0.001, V1) where V1 is the first amplitude of the original TABLED1 entry for that curve.

Format

1	2	3	4	5	6	7	8	9	10
TABD1MD	TID1	THRU	ID2	MOD	T2	A2	T3		

Example:

TABD1MD	1	THRU	5	1	.002	1.0	.003		
---------	---	------	---	---	------	-----	------	--	--

Descriptor	Meaning
ID1	First TABLED1 ID. (Integer > 0; no Default required)
ID2	Last TABLED1 ID to which MOD, T1 and A1 apply. See Remark 2. (Integer; Default = ID1)
MOD	Flag indicating whether or not to modify TABLED1 entries with ID in the range ID1 to ID2. (Integer; Default = 1) <ul style="list-style-type: none"> 0 Do not modify the table 1 Modify the table
T1	New time for the second point. See Remark 1. (Real; Default = 0.001)
A1	New amplitude of the second point. See Remark 2. (Real; Default = 1.0)
T2	New time for the third point. See Remark 1. (Real; Default = 0.002)

Remarks:

- If MOD=1, all TABLED1 entries with ID's in described by ID1 to ID2 will be modified to add two points to the beginning of the each table. The first point will be at (0.0, 0.0) the second point will be at (T1,A1). The original first point will be modified to be at time T3.
- All TABLED1 entries in the range ID1 to ID2 will be modified as indicated. Repeat this entry as many times as necessary to specify all TABLED1 ID's that should either be modified or not be modified.
- Make sure to set MOD=0 to all TABLED1 entries that are not used to describe time history loading.
- The ID1 to ID2 range may include values that do not have any TABLED1 ID's in the model.

5. If all TABLED1 entries are to be modified or if they do not all start with the same T2, A2, T3 values, PARAM,MTALBD1M PARAM,MTALBD1T may be entered.

TABDMP1

Modal Damping Table

Defines modal damping as a tabular function of natural frequency.

Format:

1	2	3	4	5	6	7	8	9	10
TABDMP1	TID	TYPE							
	f1	g1	f2	g2	f3	g3	-etc.-		

Example:

TABDMP1	2								
	2.5	.01057	2.6	.01362	ENDT				

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
TYPE	Type of damping units. (Character: "G", "CRIT", or "Q"; Default is "G")
fi	Natural frequency value in cycles per unit time. (Real ≥ 0.0)
gi	Damping value. (Real)

Remarks:

1. Modal damping tables must be selected with the Case Control command SDAMPING = TID.
2. The frequency values, fi, must be specified in either ascending or descending order, but not both.
3. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-159](#) discontinuities are allowed only between points f2 through f7. Also, if g is evaluated at a discontinuity, then the average value of g is used. In [Figure 9-159](#), the value of g at f = f3 is $g = (g_3 + g_4)/2$.
4. At least one continuation entry must be specified.
5. Any fi or gi entry may be ignored by placing "SKIP" in either of the two fields used for that entry.
6. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
7. The TABDMP1 uses the algorithm

$$g = g_T(f)$$

where f is input to the table and g is returned. The table look-up $g_T(f)$ is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points. See [Figure 9-159](#). No warning messages are issued if table data is input incorrectly. See Remark 11.

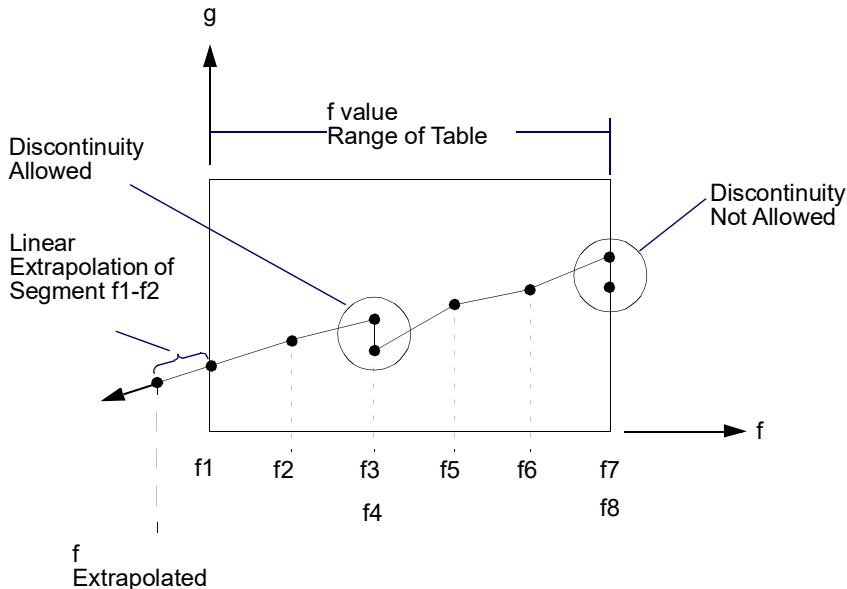


Figure 9-159 Example of Table Extrapolation and Discontinuity

8. This form of damping is used only in modal formulations of complex eigenvalue analysis, frequency response analysis, or transient response analysis. The type of damping used depends on the solution sequence (structural damping is displacement-dependent, and viscous damping is velocity-dependent). See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide* for the equations used.
9. PARAM,KDAMP may be used in solution sequences that perform modal frequency and modal complex analysis, to select the type of damping.

KDAMP	Result
1 (Default)	B Matrix
-1	$(1 + ig)K$

See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide* for a full explanation.

10. If TYPE is "G" or blank, the damping values g_i , etc., are in units of equivalent viscous dampers, as follows:

$$b_i = \frac{g_i}{\omega_i} K_i$$

(See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide*) If TYPE is "CRIT", the damping values g_i , etc., are in the units of fraction of critical damping C/C_0 . If TYPE is "Q", the damping values g_i are in the units of the amplification or quality factor, Q . These constants are related by the following equations:

$$C/C_0 = g/2$$

$$Q = \begin{cases} 1/(2C/C_0) \\ 1/g \end{cases}$$

11. A user warning message is used if either of the following conditions is satisfied:
 - a. The modal damping value is computed as a result of extrapolation.
 - b. The computed modal damping value is negative.

For any modal damping value that satisfies condition a or b, the program lists the cyclic frequency and the corresponding modal damping value and indicates whether this value was computed as a result of interpolation or extrapolation. For the latter case, it also indicates whether the extrapolation was beyond the left end of the table or beyond the right end of the table.

If a modal damping value satisfies *both* of the conditions, a and b above (that is, the modal damping value is computed as a result of extrapolation *and* it is negative), the program terminates the job with a user fatal message.

The user can prevent the program from terminating the job as above by specifying MDAMPEXT=1 [or SYSTEM(426)=1] on the NASTRAN statement. The user fatal message mentioned above does inform the user of this avoidance scheme.

12. The modal damping matrix generated by this approach is not affected by the presence of rotors in the model. Also, the circulation terms are not generated for damping corresponding to rotor degrees of freedom.
13. If Modules are present then this entry may only be specified in the main Bulk Data section.

TABL3D**Multi-Dimensional Table**

Specifies a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc for SOL 600 and SOL 400.

Format 0: Simple Table Type 0 - entry is a function of only one variable:

1	2	3	4	5	6	7	8	9	10
TABL3D0	ITID		KIND	EXTRP	ITIDS	ITIDB	SM		
	X1	Y1	X2	Y2	X3	Y3	X4	Y4	
	X5	Y5	-etc.-						

Format 1: Multi-Dimensional Table Type 1 - entry is a function of 2, 3, or 4 variables, data entered one row at a time

1	2	3	4	5	6	7	8	9	10
TABL3D1	ITID	NV	KIND1	KIND2	KIND3	KIND4	NW1	NW2	
	NW3	NW4	EXTRP1	EXTRP2	EXTRP3	EXTRP4			
	ITIDS1	ITIDB1	ITIDS2	ITIDB2	ITIDS3	ITIDB3	ITIDS4	ITIDB4	
	SM1	SM2	SM3	SM4					
	X11	X12	X13	X14	X15	X16	-etc.-		
	X21	X22	X23	X24	X25	X26	-etc.-		Enter if NW2>0
	X31	X32	X33	X34	X35	X36	-etc.-		Enter if NW3>0
	X41	X42	X43	X44	X45	X46	-etc.-		Enter if NW4>0
	Y1	Y2	Y3	Y4	Y5	Y6	-etc.-		See Remark 1

Format 2: Multi-Dimensional Table Type 2- entry is a function of 2, 3, or 4 variables, data entered one point at a time

1	2	3	4	5	6	7	8	9	10
TABL3D2	ITID	NV	KIND1	KIND2	KIND3	KIND4	NW1	NW2	
	NW3	NW4	EXTRP1	EXTRP2	EXTRP3	EXTRP4			
	ITIDS1	ITIDB1	ITIDS2	ITIDB2	ITIDS3	ITIDB3	ITIDS4	ITIDB4	
	SM1	SM2	SM3	SM4					
	X11	X12	X13	X14	X15	X16	-etc.-		
	X21	X22	X23	X24	X25	X26	-etc.-		Enter if NW2>0
	X31	X32	X33	X34	X35	X36	-etc.-		Enter if NW3>0

	X41	X42	X43	X44	X45	X46	-etc.-		Enter if NW4>0
	Y1	Y2	Y3	Y4	Y5	Y6	-etc.-		See Remark 2

Format 3: Multi-Dimensional Table Type 3- entry is specified by a formula
(SOL 600 only)

1	2	3	4	5	6	7	8	9	10
TABL3D3	ITID	NV	KIND1	KIND2	KIND3	KIND4	NW1	NW2	
	NW3	NW4	EXTRP1	EXTRP2	EXTRP3	EXTRP4			
	ITIDS1	ITIDB1	ITIDS2	ITIDB2	ITIDS3	ITIDB3	ITIDS4	ITIDB4	
	SM1	SM2	SM3	SM4					
	Formula								See Remark 3

Descriptor	Meaning
ITID	Table identification number. (Integer > 0; no Default)
SM or SMi	Flag to indicate smoothing of the table data. (Integer; Default = 0) <ul style="list-style-type: none"> 0 Do not smooth the data 1 Smooth the data
NV	Number of variables the entry is a function of (Integer, 1, 2, 3, or 4; no Default)
KIND or KINDi	“Independent” variable type (such as strain, temperature, Integer > 0; no Default; see Table 9-37 for application values)
EXTRP or EXTRPi	Extrapolation flag. (Integer; Default = 2) <ul style="list-style-type: none"> 1 Do not allow extrapolation 2 Allow extrapolation (both ends of curve)
ITIDS or ITIDSi, ITIDB or ITIDBi	Table IDs meant for future expansion to reference other tables from this table when trying to evaluate this table outside its defined range. Currently not used, leave blank.
Nwi	Number of X values of each variables. (i can range from 1 to 4) (Integer > 0; no Default)
Xi or Xij	Value of “independent” variable such as strain, temperature, ... (Real; no Default)
Yi	Value of the quantity desired such as stress, Poisson’s ratio, ... (Real; no Default)

The “Independent” variable(s) should be selected from [Table 9-37](#):

Table 9-37

Independent Variable Type					
1	time	26	z_0 coordinate	51	wavelength (used in spectral radiation)
2	normalized time	27	$s_0 = \sqrt{x_0^2 + y_0^2 + z_0^2}$	52	creep strain
3	increment number	28	contact force $ F $	53	pressure or primary quantity in diffusion
4	normalized increment time	29	not available	54	equivalent strain rate for non-Newtonian viscosity
5	x coordinate	30	σ_n (normal stress)	55	normalized arc distance
6	y coordinate	31	voltage**	56	distance to other contact surface (near contact only)
7	z coordinate	32	current**	57	terms of series
8	$s = \sqrt{x^2 + y^2 + z^2}$	33	$\left(\frac{\text{current radius}}{\text{radius of throat}}\right)^2$ (see throat)	58	hydrostatic stress
9	θ angle	34	Not available	59	hydrostatic strain
10	mode number	35	Not available	60	Not available
11	frequency	36	Not available	61	Not available
12	temperature	37	gasket closure distance	62	2nd state variable
13	function	38	displacement magnitude	63	3rd state variable
14	fourier	39	stress rate	64	4th state variable
15	$\bar{\varepsilon}^p$ (equivalent plastic strain)	40	experimental data	65	5th state variable
16	$\dot{\bar{\varepsilon}}$ (equivalent strain rate)	41	porosity	66	loadcase number*
17	Not available	42	void ratio	67	degree of cure*
18	arc length	43	$\dot{\bar{\varepsilon}}^c$ (equivalent creep strain rate)	68	magnetic field intensity**
19	relative density (not available for shells)	44	minor principal strain	69	equivalent mechanical strain
20	$\bar{\sigma}$ (equivalent stress)	45	distance from neutral axis (-1/2, +t/2)	70	1st strain invariant
21	magnetic induction**	46	normalized distance from neutral axis (-1, +1)	71	2nd strain invariant
22	velocity	47	local x-coordinate of layer point for open or closed section beam	72	3rd strain invariant

Table 9-37

Independent Variable Type					
23	parameter diameter**	48	local y-coordinate of layer point for open or closed section beam	73	local strain component
24	x_0 coordinate	49	not available	74	damage
25	y_0 coordinate	50	not available	75	accumulated crack growth
				76	relative sliding velocity

Remarks (General):

1. Independent Variable Types marked as 'NOT AVAILABLE' are not available in either SOL 600 or SOL 400. Independent variable types marked with * are only available in SOL 600. Independent variable types marked with ** are only available in SOL 600 through hand-editing of the Marc input file. The rest are available in SOL 600 and SOL 400.
2. Dependent quantities cannot be arbitrary functions of the Independent Variable Types shown in [Table 9-37](#). For e.g., Young's Modulus can be varied with Temperature and Space but cannot be varied with Time. Incorrect tabular settings that are not allowed will be errored out by the program.
3. For time-independent materials, dependent variables (i.e., yield stress) can typically be defined only as a function of temperature, stress/strain data or coordinates. For time-dependent materials, (e.g., creep coefficient), properties can also be defined as a function of time.
4. For contact dependent variables, (i.e., friction coefficient) can typically be defined as a function of temperature, contact body quantities like normal stress, body force, distance from body, relative velocity, etc.
5. If the independent variable is out of range of the table, the user can indicate if the last point in the table should be used or if the table should be extrapolated. Extrapolation means the table is continued with its first or last slope. Care should be taken with extrapolation, particularly for material properties.
6. Independent variable types for coordinates (5, 6, 7) depend on the type of analysis flagged. For a small strain or Total Lagrange or non-mechanical analysis, these coordinates refer to the original coordinates. For Updated Lagrange mechanical analysis, these coordinates refer to the updated
7. At present, options to specify additional tables using ITIDS, ITIDB, ITIDS1, ITIDB1, ... ITIDS4, ITIDB4 are not active and will be ignored if entered.

Remarks (for SOL 600 only):

1. The function is read by giving NW1 data points ($NW4^*NW3^*NW2$) times. The program reads the data using the following method.

```

do k4=1, nw4
  do k3=1, nw3
    do k2=1, nw2
      read nw1 values f(X1, K2, K3, K4)
    enddo
  enddo
enddo

```

2. The function is read one value at a time. There are NW1*NW2*NW3*NW4 values. The program uses the values as follows:

```

do k4=1, nw4
  do k3=1, nw3
    do k2=1, nw2
      do k1=1, nw1
        read one value f(K1, K2, K3, K4)
      enddo
    enddo
  enddo
enddo

```

3. The formula can extend from field 2 through field 9 and must be comprised of the items listed previously.
4. This parameter may also be used to control the BOLT entry. For BOLT, the default for MMBOLTUS=1 to achieve the same results as SOL 400. If the results appear to be backwards, set MMBOLTUS=-1. BOLT and MBOLTUS may not both be entered in the same model.

Remarks for SOL 400 only:

1. Independent variable types are typically specified for materials or contact. When material properties are varied through TABL3DX, the tables are honored only for elements with property extensions. For e.g., if TABL3D0 is used in conjunction with MATEP for a CHEXA mesh with PSOLID properties, then the elements should be given the PSLDN1 property extension for the table to be valid. Note that, with default property mapping (NLMOPTS,SPROPMAP,0), the property extensions will be added automatically based on the rules specified in Remark 9. of the NLMOPTS entry.
2. The strains and stresses identified as independent variables depend on the parameters flagged for large displacement.
 - a. For PARAM,LGDISP,-1, the strains and stresses are engineering quantities.
 - b. For PARAM,LGDISP,n ($n > 0$) or with NLMOPTS,LRGSTRN,n ($n > 0$), the strains are typically Logarithmic strains and the stresses are Cauchy Stresses.

TABLE3D

Tabular Function with Three Variables

Specify a function of three variables for the GMBC, GMLOAD, and TEMPF entries only.

Format:

1	2	3	4	5	6	7	8	9	10
TABLE3D	TID	X0	Y0	Z0	F0				
	X1	Y1	Z1	F1	X2	Y2	Z2	F2	
	X3	Y3	Z3	F3	X4	Y4	Z4	F4	
	-etc.-	ENDT							

Example:

TABLE3D	128	0.	0.	1.					
	7.	8.	9.	100.	12.	14.	11.	200.	
	17.	18.	19.	1100.	112.	114.	111.	1200.	
	ENDT								

Descriptor	Meaning	Type	Default
TID	Table identification number.	Integer > 0	Required
X0,Y0,Z0	Offset of the independent variables.	Real	0.0
F0	Offset of the dependent variables.	Real	0.0
Xi,Yi,Zi	Independent variables.	Real	0.0
Fi	Dependent variable.	Real	0.0

Remarks:

- At least two continuation entries must be specified.
- The value of the function at (x,y,z) is calculated as

$$f = \frac{\sum_{i=1}^4 \frac{F_i - F_0}{d_i}}{\sum_{i=1}^4 \frac{1}{d_i}}$$

where f are the function values at the four points with the lowest value of

$$d_i^2 = (x - X0 - Xi)^2 + (y - Y0 - Yi)^2 + (z - Z0 - Zi)^2$$

TABLED1

Dynamic Load Tabular Function, Form 1

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED1	TID	XAXIS	YAXIS						
	x1	y1	x2	y2	x3	y3	-etc.-	"ENDT"	

Example:

TABLED1	32								
	-3.0	6.9	2.0	5.6	3.0	5.6	ENDT		

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. See Remarks 6. and 10. (Character: "LINEAR" or "LOG"; Default = "LINEAR")
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. See Remarks 6. and 10. (Character: "LINEAR" or "LOG"; Default = "LINEAR")
xi, yi	Tabular values. (Real)
"ENDT"	Flag indicating the end of the table.

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-160](#) discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-160](#), the value of y at $x = x_3$ is $y = (y_3 + y_4)/2$. If the y-axis is a LOG axis then the jump at the discontinuity is evaluated as $y = \sqrt{y_3 y_4}$.
3. At least one continuation must be specified.
4. Any xi-yi pair may be ignored by placing the character string "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of the character string "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLED1 uses the algorithm

$$y = y_T(x)$$

where x is input to the table and y is returned. The table look-up is performed using interpolation within the table and extrapolation outside the table using the two starting or end points. See [Figure 9-160](#). The algorithms used for interpolation or extrapolation are:

XAXIS	YAXIS	$y_T(x)$
LINEAR	LINEAR	$\frac{x_j - x}{x_j - x_i} y_i + \frac{x - x_i}{x_j - x_i} y_j$
LOG	LINEAR	$\frac{\ln(x_j/x)}{\ln(x_j/x_i)} y_i + \frac{\ln(x/x_i)}{\ln(x_j/x_i)} y_j$
LINEAR	LOG	$\exp\left[\frac{x_j - x}{x_j - x_i} \ln y_i + \frac{x - x_i}{x_j - x_i} \ln y_j\right]$
LOG	LOG	$\exp\left[\frac{\ln(x_j/x)}{\ln(x_j/x_i)} \ln y_i + \frac{\ln(x/x_i)}{\ln(x_j/x_i)} \ln y_j\right]$

where x_j and y_j follow x_i and y_i .

No warning messages are issued if table data is input incorrectly.

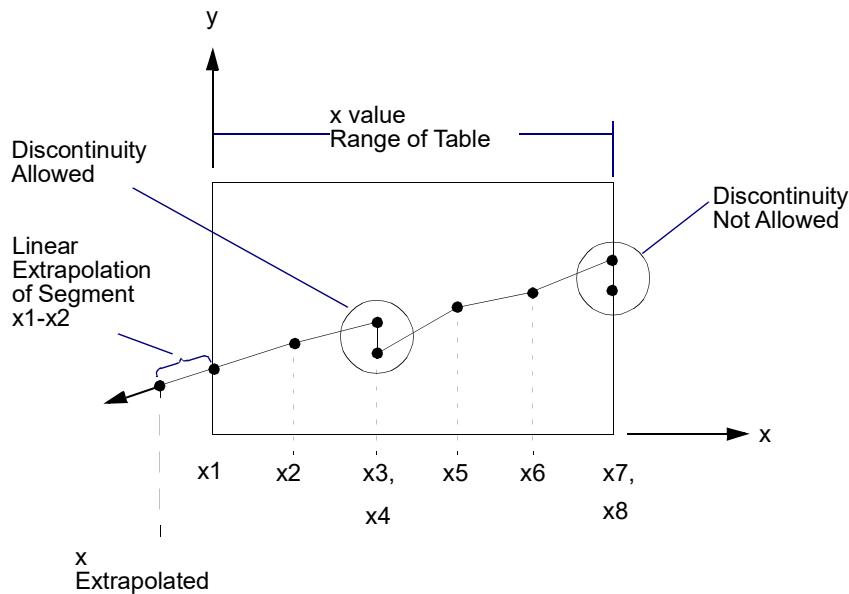


Figure 9-160 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads, x_i is measured in cycles per unit time.

9. Tabular values on an axis if XAXIS or YAXIS = LOG must be positive. A fatal message will be issued if an axis has a tabular value ≤ 0 .
10. LOG is not supported for SOL 600 or SOL 700. Fields 3 and 4 must be blank.
11. RC network solver only supports LINEAR type XAXIS and YAXIS for thermal analysis.
12. The X-Y plot information resulting from a dynamic response solution (like any response versus frequency in SOLs 108/111 or any response versus time in SOLs 109/112) can be generated in TABLED1 format by using the XYTRAN module via DMAP. The 6th parameter of this module, which is an integer value, can be used for this purpose. Details can be obtained by referring to the description of this module in the DMAP Programmer's Guide.

TABLED2

Dynamic Load Tabular Function, Form 2

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED2	TID	X1							
	x1	y1	x2	y2	x3	y3	-etc.-		

Example:

TABLED2	15	-10.5							
	1.0	-4.5	2.0	-4.2	2.0	2.8	7.0	6.5	
	SKIP	SKIP	9.0	6.5	ENDT				

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
X1	Table parameter. See Remark 6. (Real)
xi, yi	Tabular values. (Real)

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-161](#) discontinuities are allowed only between points x2 and x7. Also if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-161](#), the value of y at x = x3 is $y = (y3 + y4)/2$.
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLED2 uses the algorithm

$$y = y_T(x - X1)$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-161](#). No warning messages are issued if table data is input incorrectly.

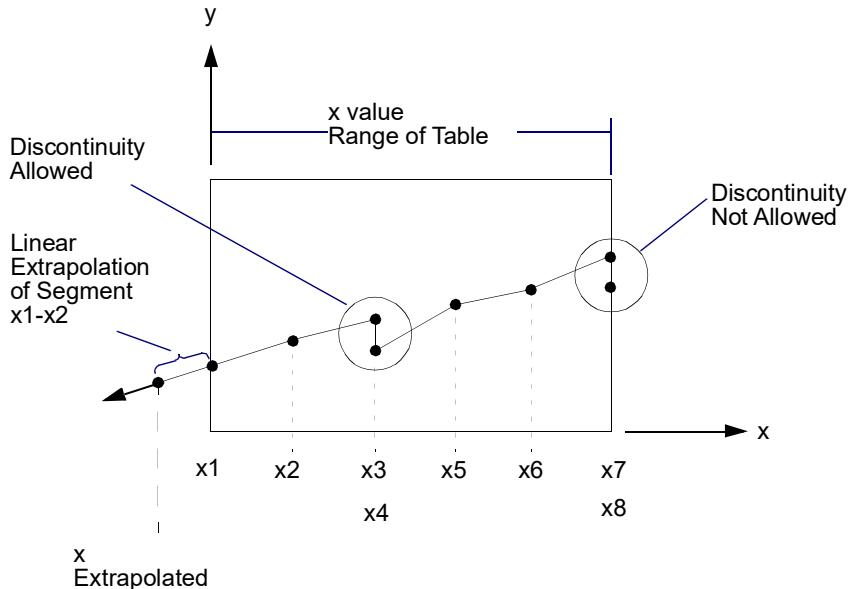


Figure 9-161 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads, x_1 and x_i are measured in cycles per unit time.

TABLED3

Dynamic Load Tabular Function, Form 3

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	-etc.-		

Example:

TABLED3	62	126.9	30.0						
	2.9	2.9	3.6	4.7	5.2	5.7	ENDT		

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
X1, X2	Table parameters. (Real; X2 ≠ 0.0)
xi, yi	Tabular values. (Real)

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-162](#) discontinuities are allowed only between points x2 and x7. Also if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-162](#), the value of y at x = x3 is y = (y3 + y4)/2 .
3. At least one continuation entry must be present.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLED3 uses the algorithm

$$y = y_T \left(\frac{x - X1}{X2} \right)$$

where x is input to the table and y is returned. The table look-up is performed using interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-162](#). No warning messages are issued if table data is input incorrectly.

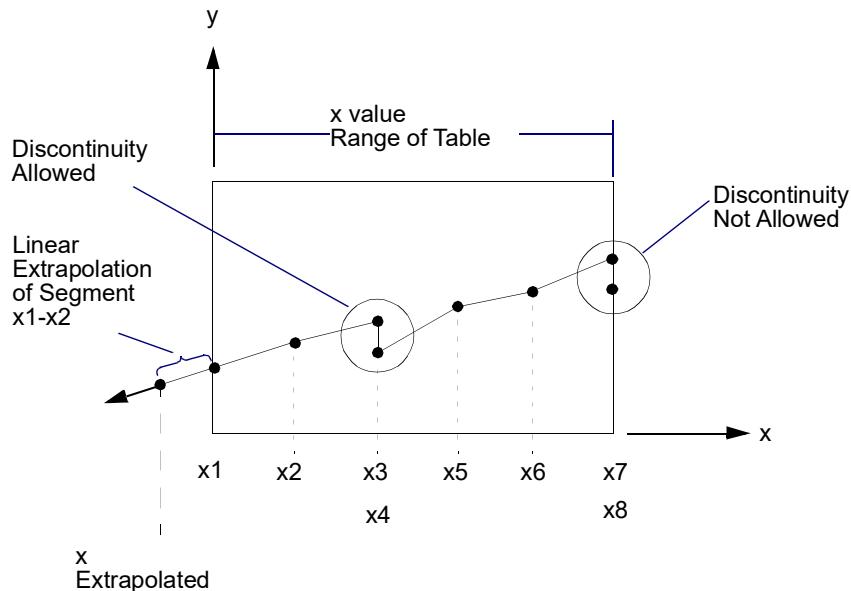


Figure 9-162 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads, X1, X2, and xi are measured in cycles per unit time.

TABLED4

Dynamic Load Tabular Function, Form 4

Defines the coefficients of a power series for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED4	TID	X1	X2	X3	X4				
	A0	A1	A2	A3	A4	A5	-etc.-		

Example:

TABLED4	28	0.0	1.0	0.0	100.				
	2.91	-0.0329	6.51-5	0.0	-3.4-7	ENDT			

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
Xi	Table parameters. (Real; X2 ≠ 0.0; X3 < X4)
Ai	Coefficients. (Real)

Remarks:

- At least one continuation entry must be specified.
- The end of the table is indicated by the existence of “ENDT” in the field following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
- TABLED4 uses the algorithm

$$y = \sum_{i=0}^N A_i \left(\frac{x - X_1}{X_2} \right)^i$$

where x is input to the table, y is returned, and N is the number of pairs. Whenever $x < X_3$, use X_3 for x ; whenever $x > X_4$, use X_4 for x . There are $N + 1$ entries in the table. There are no error returns from this table look-up procedure.

- For frequency-dependent loads, xi is measured in cycles per unit time.

TABLED5

Dynamic Load Tabular Function, Form 5

Defines a value as a function of two variables for use in generating frequency-dependent and time-dependent dynamic loads.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED5	TID								
	X(1)	TID(1)	X(2)	TID(2)	X(3)	TID(3)	X(4)	TID(4)	
	ENDT						

Example:

TABLED5	52								
	0.0	101	10.0	102	30.0	103	ENDT		

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
X(i)	X value for the function specified by TID(i) (Real; no Default).
TID(i)	ID of a TABLED1, TABLED2, TABLED3 or TABLED4 defining the function Y for the given value of X. (Integer > 0; no Default).

Remarks:

1. This table returns a value that is a function of 2 variables $f(x, y)$. The first variable value x is specified on this entry as X(i), the function versus y for the specified value for X(i) is specified on the referenced table TID(i).
2. When used with the NLRGAP entry to define a frequency dependent gap force as a function of penetration, the frequency values are input as X(i) and the variation of force vs. penetration is input on a series of TABLED1 entries referenced by the table values TID(i), one for each required change in properties with respect to frequency.
3. 2D linear interpolation is carried out for values of (x, y) not specified on the tabular entries.
4. X(i) values must be in ascending order.
5. TABLED5 Usage for frequency as function of temperature:

TABLED5 usage

	ID	E	G	NU				GE	
MAT1	33	7.2+10		.3				.02	

MAT1F	33	110	111	112				200	
-------	----	-----	-----	-----	--	--	--	-----	--

TABLED5	200								
	0.	3	40.	4	100.	5	...		

TABLED1	3	...							
---------	---	-----	--	--	--	--	--	--	--

TABLED1	4	...							
---------	---	-----	--	--	--	--	--	--	--

TABLED1	5	...							
---------	---	-----	--	--	--	--	--	--	--

Usage Example

E is dependent only on frequency, therefore points to a TABLED1 entry.

GE is both temperature and frequency dependent and therefore points to a TABLED5 entry

		E	G	NU				GE	
MAT1	1	7.2+10	2.8+10	.3			2.22-5	0.02	
MAT1F	1	110	111	112				200	
TABLED1	110								
	10.	7.2+10	200.	7.1+10	300.	6.9+10	ENDT		
TABLED1	111								
	10.	2.8+10	200.	2.7+10		2.6+10	ENDT		
TABLED1	112								
	10.	.3	200.	.3		.3	ENDT		

GE has frequency as a function of temperature

TABLED5	200								
	0.	3	40.	4	100.	5	ENDT		
TABLED1	3								
	10.	0.02	200.			0.025	ENDT		

TABLED1	4							
	10.	0.025	200.			0.03	ENDT	
TABLED1	5							
	10.	0.03	200.	0.04	300.	0.035	ENDT	

TABLED1 units are x=frequency, y=material value

TABLED5 input is numerical temperature - frequency table ID

For an element with average temperature of 15.0 degrees the GE value will be selected from TABLED1 ID=3;

For an element with average temperature of 30.0 degrees the GE value will be selected from TABLED1 ID=4;

For an element with average temperature of 20.0 degrees the GE value will be selected from TABLED1 ID=3;

6. TABLED5 in Solution Sequence SOL108, SOL111, SO200, or SOL400 with ANALYSIS=DFREQ or MFREQ:

Example (1): TABLED5 specification using TEMP(INIT) or TEMP(MATE) entry, no TEMP(LOAD) in Case Control section.

```
SOL 108
...
TEMP (INIT) =5
BEGIN BULK
...
TEMP, 5, ...
...
ENDDATA
```

The temperatures specified by TEMP(INIT) will be used for TABLED5 look up

Example (2): TABLED5 specification using TEMP(LOAD) entry.

```
SOL 108
...
TEMP (INIT) =5
SUBCASE 1
TEMP (LOAD) =12
...
```

```
BEGIN BULK
...
TEMP, 5, ...
TEMP, 12, ...
...
ENDDATA
```

The temperatures specified by TEMP(LOAD) will be used for TABLED5 look up.

User is reminded that for dynamic solutions sequences, the above will not apply thermal loading unless there is a DLOAD entry pointing to an RLOAD1 or RLOAD2 referencing TEMP(LOAD) with ID=12 specified.

Example (3): TABLED5 specification using TEMP(LOAD) entry and thermal loading desired.

```
SOL 108
...
TEMP (INIT)=5
SUBCASE 1
TEMP (LOAD)=12
DLOAD = 50
...
BEGIN BULK
...
RLOAD, 50, 12, , , 1.0, , LOAD
...
TEMP, 5, ...
TEMP, 12, ...
...
ENDDATA
```

The temperatures specified by TEMP(LOAD) will be used for TABLED5 look up and will also be used to compute thermal loads in elements.

TABLEHT**Heat Transfer Coefficient Table with Two Variables**

Specifies a function of two variables for convection heat transfer coefficient.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEHT	TID								
	x1	TID1	x2	TID2	x3	-etc.			

Example:

TABLEHT	85								
	10.0	101	25.0	102	40.0	110	ENDT		

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
xi	Independent variables. (Real)
TIDI	Table identification numbers of TABLEH1 entries. (Integer > 0)

Remarks:

1. xi must be listed in ascending order.
2. At least one continuation entry must be present.
3. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
4. This table is referenced only by PCONV entries that define free convection boundary condition properties.

TABLEH1

Heat Transfer Coefficient Table, Form 1

Defines a tabular function referenced by TABLEHT for convection heat transfer coefficient.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEH1	TID								
	y1	f1	y2	f2	y3	-etc.-			

Example:

TABLEH1	123								
	50.0	5.23	75.0	3.76	110.0	0.97	ENDT		

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
yi	Independent variables. (Real)
fi	Dependent variable. (Real)

Remarks:

1. yi must be listed in ascending order.
2. At least one continuation entry must be present.
3. Any yi-fi pair may be ignored by placing "SKIP" in either of the two fields used for that entry.
4. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
5. TABLEH1 is used to input a curve in the form of

$$f = f(y)$$
 where y is input to the table and f is returned. The table look-up is performed using linear interpolation within the table and is evaluated at the starting or end point outside the table. No warning messages are issued if table data is input incorrectly.
6. Discontinuities are not recommended and may lead to unstable results.

TABLE1

Specifies a Table of Amplitude vs Pseudo-Time for Static Loads Specified in LDTABL Entries

Specifies a table of amplitude vs pseudo-time for static loads specified in LDTABL entries SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEL1	TID								
	X1	Y1	X2	Y2	X3	Y3	etc.	"ENDT"	

Example:

TABLEL1	101								
	0.0	0.0	0.5	1.0	1.0	0.0	ENDT		

Descriptor	Meaning
TID	Table identification number. (Integer > 0; no Default)
Xi, Yi	Tabular values. (Real; no Default)
"ENDT"	Flag indicating end of the table. (Character)

Remarks:

1. TABLEL1 can only be used in SOL 600.
2. TABLED1 may be used instead of TABLEL1, however, XAXIS and YAXIS must be LINEAR or blank.
3. The TID must be unique among all TABLEL1's and TABLEDi's.
4. The string "SKIP" used in TABLED1 may not be used in TABLEL1.
5. Log X and Y is not available in TABLEL1.
6. No blank fields are allowed starting with the second field of the first continuation line until the ENDT string.
7. There must be at least two points in the table.

TABLEM1

Material Property Table, Form 1

Defines a tabular function for use in generating temperature-dependent material or fatigue related properties.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEM1	TID	XAXIS	YAXIS						
	x1	y1	x2	y2	x3	y3	-etc.-	"ENDT"	

Example:

TABLEM1	32								
	-3.0	6.9	2.0	5.6	3.0	5.6	ENDT		

Descriptor	Meaning
TID	Table identification number. See Remark 8. (Integer > 0 or Integer < 0)
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR") See Remark 9.
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR") See Remark 9.
xi, yi	Tabular values. (Real)
"ENDT"	Flag indicating the end of the table.

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-163](#) discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-163](#), the value of y at x = x3 is $y = (y_3 + y_4)/2$.
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLEM1 uses the algorithm

$$y = y_T(x) \quad (\text{for heat transfer, see Remark 7.})$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-163](#). No warning messages are issued if table data is input incorrectly.

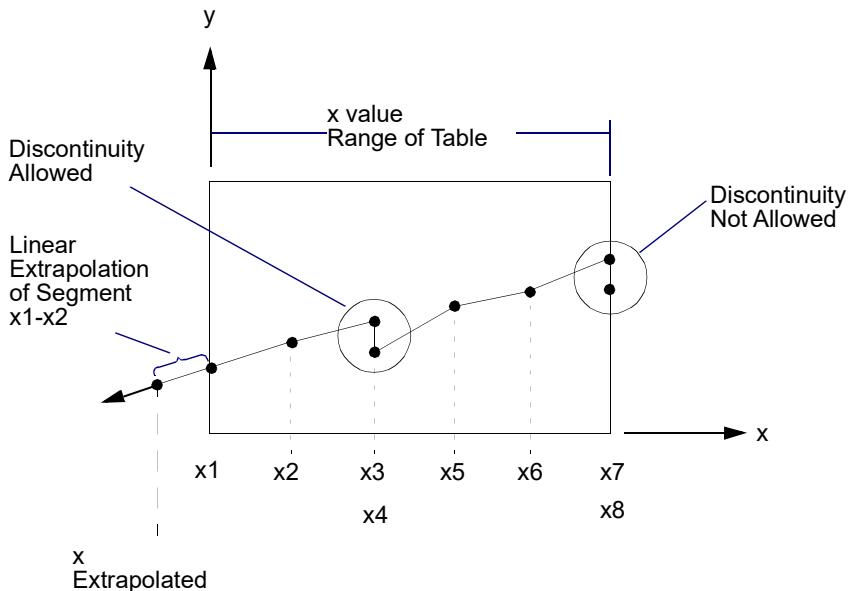


Figure 9-163 Example of Table Extrapolation and Discontinuity

7. For Nastran heat transfer, the TABLEM1 assumes

$$y = zy_T(x)$$

where x is input to the table, y is returned and z is supplied from MAT4 or MT5 entries.

8. A negative TID is used to associate thermal strain $\varepsilon(T)$ ordinate values instead of coefficient of thermal expansion ordinate values to the $T(A_i)$ fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative $ID_{T(A_i)}$ value will be changed to $|ID_{T(A_i)}| + 10000000$.
9. Logarithmic XAXIS and YAXIS is only recognized when specifying S-N curves referenced by MATFTG entries for TYPE=TABLE. All other uses use linear and ignore these fields.
10. TABLEM1 in RESTART job will cause recalculation of eigenvalues even in the cases where TABLEM1 does not change stiffness and/or mass matrix. An example is SOL 111 RESTART for fatigue analysis with TABLEM1 referenced on MATFTG.

TABLEM2

Material Property Table, Form 2

Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEM2	TID	X1							
	x1	y1	x2	y2	x3	y3	-etc.-		

Example:

TABLEM2	15	-10.5							
	1.0	-4.5	2.0	-4.5	2.0	2.8	7.0	6.5	
	SKIP	SKIP	9.0	6.5	ENDT				

Descriptor	Meaning
TID	Table identification number. See Remark 7. (Integer > 0 or Integer < 0)
X1	Table parameter. (Real)
xi, yi	Tabular values. (Real)

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-164](#), discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-164](#), the value of y at x = x3 is $y = (y3 + y4)/2$.
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLEM2 uses the algorithm

$$y = zy_T(x - X1)$$

where x is input to the table, y is returned and z is supplied from the MATi entry. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-164](#). No warning messages are issued if table data is input incorrectly.

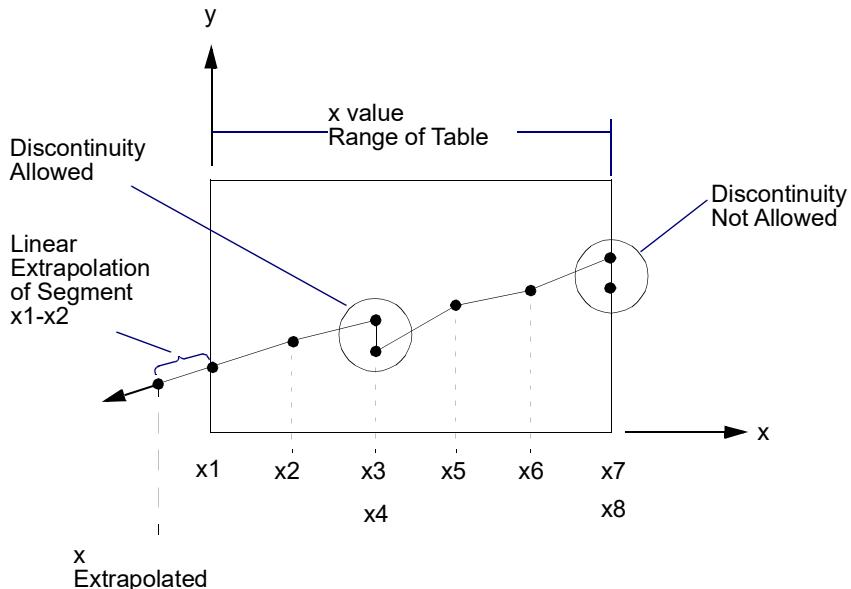


Figure 9-164 Example of Table Extrapolation and Discontinuity

7. A negative TID is used to associate thermal strain $\varepsilon(T)$ ordinate values instead of coefficient of thermal expansion ordinate values to the T(A_i) fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative ID_{T(A_i)} value will be changed to $|ID_{T(A_i)}| + 100000000$.

TABLEM3

Material Property Table, Form 3

Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEM3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	-etc.-		

Example:

TABLEM3	62	126.9	30.0						
	2.9	2.9	3.6	4.7	5.2	5.7	ENDT		

Descriptor	Meaning
TID	Table identification number. See Remark 7. (Integer > 0 or Integer < 0)
X1, X2	Table parameters. See Remark 6. (Real; X2 ≠ 0.0)
xi, yi	Tabular values. (Real)

Remarks:

1. Tabular values for xi must be specified in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-165](#) discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-165](#), the value of y at x = x3 is $y = (y3 + y4)/2$.
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLEM3 uses the algorithm

$$y = zy_T \left(\frac{x - X1}{X2} \right)$$

where x is input to the table, y is returned and z is supplied from the MATi entry. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-165](#). No warning messages are issued if table data is input incorrectly.

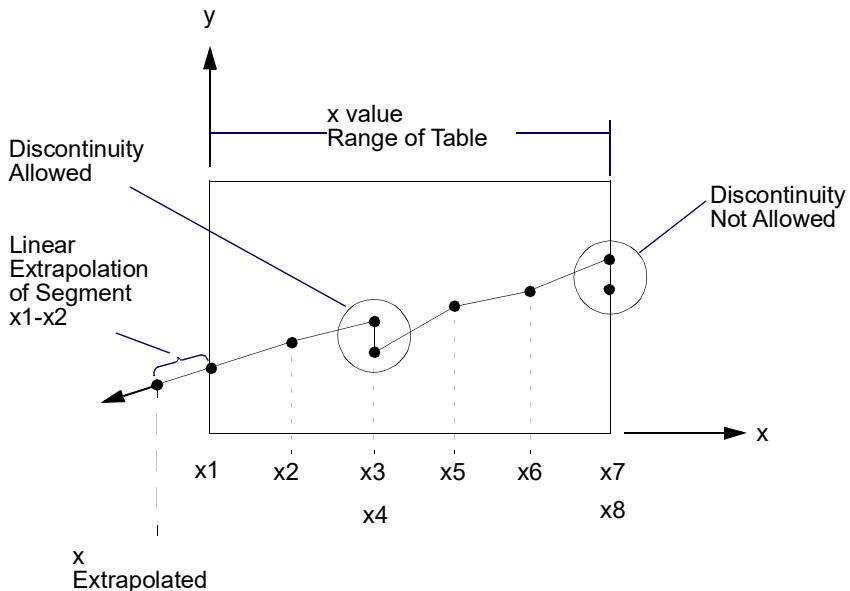


Figure 9-165 Example of Table Extrapolation and Discontinuity

7. A negative TID is used to associate thermal strain $\varepsilon(T)$ ordinate values instead of coefficient of thermal expansion ordinate values to the $T(A_i)$ fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative $ID_{T(A_i)}$ value will be changed to $|ID_{T(A_i)}| + 100000000$.

TABLEM4

Material Property Table, Form 4

Defines coefficients of a power series for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEM4	TID	X1	X2	X3	X4				
	A0	A1	A2	A3	A4	A5	-etc.-		

Example:

TABLEM4	28	0.0	1.0	0.0	100.				
	2.91	-0.0329	6.51-5	0.0	-3.4-7	ENDT			

Descriptor	Meaning
TID	Table identification number. See Remark 4. (Integer > 0 or Integer < 0)
Xi	Table parameters. (Real; X2 ≠ 0.0; X3 < X4)
Ai	Coefficients. (Real)

Remarks:

- At least one continuation entry must be specified.
- The end of the table is indicated by the existence of “ENDT” in the field following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
- TABLEM4 uses the algorithm

$$y = z \sum_{i=0}^N A_i \left(\frac{x - X_1}{X_2} \right)^i$$

where x is input to the table, y is returned and z is supplied from the MATi entry. Whenever $x < X_3$, use X_3 for x ; whenever $x > X_4$, use X_4 for x . There are $N + 1$ entries in the table. There are no error returns from this table look-up procedure.

- A negative TID is used to associate thermal strain $\varepsilon(T)$ ordinate values instead of coefficient of thermal expansion ordinate values to the $T(A_i)$ fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative $ID_{T(A_i)}$ value will be changed to $|ID_{T(A_i)}| + 100000000$.

TABLES1**Material Property Table, Form 1**

Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).

Format:

1	2	3	4	5	6	7	8	9	10
TABLES1	TID	TYPE							
	x1	y1	x2	y2	x3	y3	-etc.-	"ENDT"	

Example:

TABLES1	32								
	0.0	0.0	.01	10000.	.02	15000.	ENDT		

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
TYPE	Flag to define type of the stress-strain curve. See Remark 10. (Integer = 1 or 2; Default = 1)
xi, yi	Tabular values. (Real)
"ENDT"	Flag indicating the end of the table.

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-166](#) discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-166](#), the value of y at x = x3 is $y = (y_3 + y_4)/2$.
3. At least one continuation entry must be present.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields used for that entry.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
6. TABLES1 is used to input a curve in the form of

$$y = y_T(x)$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-166](#). No warning messages are issued if table data is input incorrectly.

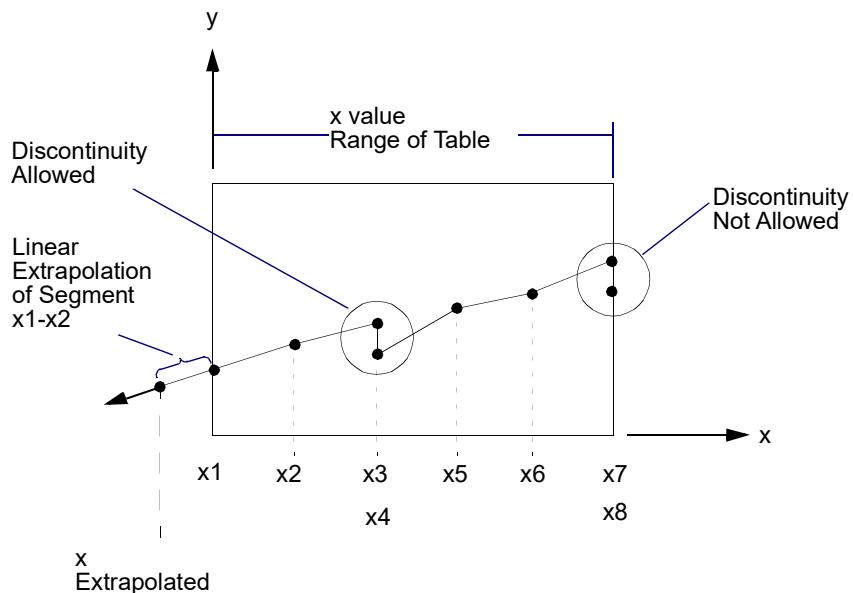


Figure 9-166 Example of Table Extrapolation and Discontinuity

7. Discontinuities are not recommended and may lead to unstable results. Discontinuities are not allowed in nonlinear solution sequences.
8. For SOL 600, general temperature-dependent stress vs. plastic strain curves may be entered using a combination of TABLEST and TABLES1 entries. Each TABLES1 entry is at a constant temperature. All entries must be in the form of stress vs. plastic strain using the stress and strain measures to be incorporated into the analysis. All sets of stress-strain values for a particular TABLES1 entry must be at the same temperature. One set is required for the lowest temperature in the model and another at or above the highest temperature in the model.
9. For SOL 600, the stress and strain values entered here depend on the stress and strain measures selected for the analysis. In addition, the strain is controlled using PARAM,MRTABLS1 which provides several methods of converting an engineering stress-strain curve to a stress vs. plastic strain curve (see MRTABLS1 in the Parameters Section).
10. For SOL 400, TYPE denotes the type of stress-strain curve; 1 - Cauchy (true) stress vs. total true strain; and 2 - Cauchy (true) stress vs. plastic true strain. For MATS1 Bulk Data entry, only TYPE = 1 can be used. A user fatal error will be issued if TYPE = 2 is used. For MATEP Bulk Data entry both TYPE = 1 and 2 can be used.

TABLEST**Material Property Temperature-Dependence Table**

Specifies the material property tables for nonlinear elastic temperature-dependent materials.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEST	TID								
	T1	TID1	T2	TID2	T3	-etc.-			

Example:

TABLEST	101								
	150.0	10	175.0	20	ENDT				

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
Ti	Temperature values. (Real)
TIDi	Table identification numbers of TABLES1 entries. (Integer > 0)

Remarks:

1. TIDi must be unique with respect to all TABLES1 and TABLEST table identification numbers.
2. Temperature values must be listed in ascending order.
3. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
4. This table is referenced only by MATS1 entries that define nonlinear elastic (TYPE = "NLELAST") materials.
5. For SOL 600, this entry provides IDs of TABLES1 curves as a function of temperature for use with Marc's AF_flowmat. The strains are plastic strain for all curves entered. The first curve must be entered at the lowest temperature encountered in the analysis run. Curves must be defined that equal or exceed the maximum temperature encountered in the run.

TABLFTG**Fatigue Loading Tabular Data**

Defines tabular data for specifying fatigue cyclic loading variation.

Format:

1	2	3	4	5	6	7	8	9	10
TABLFTG	TID								
	y1	y2	y3	y4	y5	y6	y7	"ENDT"	

Example:

TABLFTG	1								
	0.000	-1.0	1.0	0.0	ENDT				

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
yi	Y value of each point in the time history curve. (Real).
"ENDT"	Flag indicating the end of the table.

Remarks:

1. The TABLFTG is referenced by a FTGLOAD entry.
2. The x-values are assumed to be in ascending order. For rainflow cycle counting purposes the actual x values are inconsequential.
3. For modal analysis using SOL 103, this would define the modal participation factors for a particular mode.

TABLRPC**Dynamic Load Tabular Function Referencing Channel Data File**

Defines a tabular function for use in generating time-dependent loads from an externally defined channel data file.

Format:

1	2	3	4	5	6	7	8	9	10
TABLRPC	TID	XAXIS	YAXIS	UID	TYPE	CHAN	TOTIM	PUNCH	
	x _i 1	x _j 1	x _i 2	x _j 2	x _i 3	x _j 3	-etc.-	"ENDT"	

Examples:

TABLRPC	32			33	RPC	5			
UDNAME	33								
	C:\myrpcfile.rsp								

TABLRPC	32			33	DAC				
	0.0	20.0	35.0	60.0	ENDT				
UDNAME	33								
	C:\myrpcfile.dac								

TABLRPC	32			-33		5			
	0.0	20.0	35.0	60.0	ENDT				
UDNAME	33								
	C:\myrpcfile.rsp								

Descriptor	Meaning
TID	Table identification number. (Integer > 0, no default)
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR")
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR")
UID	Identification number of a UDNAME entry to specify the external file that defines the x-y pair values that define the actual tabular function. Required. (Integer != 0, no default)
TYPE	Type of external file to specify via UID field. (Character: "RPC", "DAC", or "S3T"; Default = "RPC")
CHAN	Channel number to read for Y (load) values from "RPC" and "S3T" files. Leave blank for "DAC" files. (Integer > 0, Default = 1)
TOTIM	Total time of the signal defined in channel CHAN. Optional. (Real > 0., Default=Blank). See remarks below.

Descriptor	Meaning
PUNCH	Specify whether equivalent TABLED1 entries containing the actual x-y pairs should be written to the PUNCH file. (Character: "YES" or "NO"; Default= "NO")
x_i^1/x_j^1	x-y pair Filter mechanism. The given (x_i^1, x_j^1) pairs reflect the start and end x-values for inclusion from the external file; or if UID is negative, the given pairs reflect the start and end x-values to exclude from reading from the external file. See remarks below. Optional. (Real).
"ENDT"	Flag indicating the end of the table.

Remarks:

1. The TABLRPC entry can be referenced by any entry that can reference a TABLED1. The TABLRPC is internally converted to a TABLED1 entry using the channel data specified. This internal TABLED1 entry can be written to the punch (.pch) file if the PUNCH field is set to YES.
2. The CHANnel specified contains the Y (load) data. The X (time) data for each point is automatically extracted from the channel file at each point to create the x-y pairs for a TABLED1 entry. The TOTIM (total time) of the signal can be optionally supplied, in which case the time increment of each point is calculated as TOTIM / (NPNTS-1) where NPNTS are the number of signal points. This will override any time specifications for the points from the original channel file.
3. x_i^1/x_j^1 field pairs are purely optional. If not needed, no continuation lines should be present, in which case the entire signal defined in the specified CHANnel is taken.
4. The x_i^1/x_j^1 field pairs represent sections of the channel to read and must be in the order in which they appear in the external file. For example, to read from only $x=0.0$ to $x=1.0$, specify $x_1^1=0.0$ and $x_1^2=1.0$ and $x_2^1=ENDT$. If the first x_i^1 is left blank, reading begins at the first data point. If the last x_i^1 value is left blank, then the rest of the data points are read to the end of the channel. If a specified x_i^1 or x_i^2 value does not exist, only the points in between are retained or excluded.
5. Any x_i^1/x_j^1 field pair may be ignored by placing the character string "SKIP" in either of the two fields.
6. The end of the table is indicated by the existence of the character string "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT"
7. As this data is converted to a TABLED1 entry, all comments pertaining to the TABLED1 entry are also valid.

TABLUDS

User Defined Failure For Nonlinear Explicit Analysis.

Specifies that a user routine is being used to define an arbitrary function. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
TABLUDS	TID	GROUP	UNAME						

Example:

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE myfunc 'SCA.MDSolver.Obj.Uds.Dytran.Loads'
```

In Bulk Data:

1	2	3	4	5	6	7	8	9	10
TABLUDS	12	myfunc	EXFUNC						

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

Remarks:

1. Since tables and user-defined functions belong to the same group, the table numbers must be unique.
2. UNAME can be:

Subroutine Name	Function
EXFUNC	Standard user defined function

TABRND1**Power Spectral Density Table**

Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

Format:

1	2	3	4	5	6	7	8	9	10
TABRND1	TID	XAXIS	YAXIS						
	f1	g1	f2	g2	f3	g3	-etc.-		

Example:

TABRND1	3								
	2.5	.01057	2.6	.01362	ENDT				

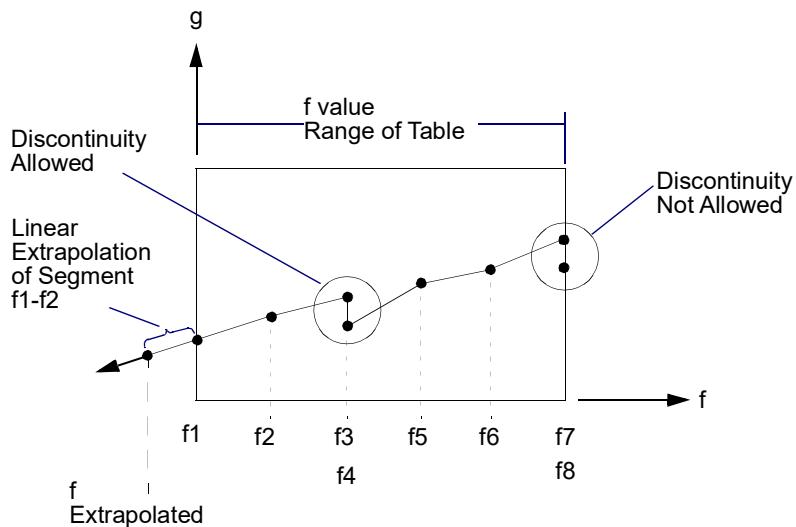
Descriptor	Meaning
TID	Table identification number. (Integer > 0)
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR")
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR")
f _i	Frequency value in cycles per unit time. (Real ≥ 0.0)
g _i	Power spectral density. (Real)

Remarks:

1. The f_i must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-167](#) discontinuities are allowed only between points f₂ through f₇. Also, if g is evaluated at a discontinuity, then the average value of g is used. In [Figure 9-167](#), the value of g at f = f₃ is $g = (g_3 + g_4)/2$. If the y-axis is a LOG axis then the jump at the discontinuity is evaluated as $y = \sqrt{y_3 y_4}$.
3. At least two entries must be present.
4. Any f_i-g_i pair may be ignored by placing "SKIP" in either of the two fields used for that entry.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABRND1 uses the algorithm

$$g = g_T(f)$$

where f is input to the table and g is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-167](#). No warning messages are issued if table data is input incorrectly.



[Figure 9-167](#) Example of Table Extrapolation and Discontinuity

7. For auto spectral density, the value of g returned must be greater than or equal to zero, as shown in [Remark 6](#).
8. Tabular values on an axis if XAXIS or YAXIS = LOG must be positive. A fatal message will be issued if an axis has a tabular value ≤ 0 .
9. The algorithms used are:

XAXIS	YAXIS	$f(x)$
LINEAR	LINEAR	$\frac{f_{i+1}-f}{f_{i+1}-f_i}g_i + \frac{f-f_i}{f_{i+1}-f_i}g_{i+1}$
LOG	LINEAR	$\frac{\ln(f_{i+1}/f)}{\ln(f_{i+1}/f_i)}g_i + \frac{\ln(f/f_i)}{\ln(f_{i+1}/f_i)}g_{i+1}$
LINEAR	LOG	$\exp\left[\frac{f_{i+1}-f}{f_{i+1}-f_i}\ln g_i + \frac{f-f_i}{f_{i+1}-f_i}\ln g_{i+1}\right]$
LOG	LOG	$\exp\left[\frac{\ln(f_{i+1}/f)}{\ln(f_{i+1}/f_i)}\ln g_i + \frac{\ln(f/f_i)}{\ln(f_{i+1}/f_i)}\ln g_{i+1}\right]$

where $f_i < f < f_{i+1}$.

10. If Modules are present then this entry may only be specified in the main Bulk Data section.

TABRNDG**Gust Power Spectral Density**

Defines the power spectral density (PSD) of a gust for aeroelastic response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TABRNDG	TID	TYPE	L/U	WG					

Example:

TABRNDG	1020	1	1.3	.1					
---------	------	---	-----	----	--	--	--	--	--

Descriptor	Meaning
TID	Table identification number. (Integer > 0)
TYPE	PSD type: von Karman (TYPE = 1) or Dryden model (TYPE = 2). (Integer = 1 or 2)
L/U	Scale of turbulence divided by velocity (units of time). See L/U in Remark 2. (Real)
WG	Root-mean-square gust velocity. (Real)

Remarks:

1. This entry must be referenced by a RANDPS entry.
2. The power spectral density is given by

$$S_q(\omega) = 2(WG)^2(L/U) \frac{1 + 2(p+1)k^2(L/U)^2\omega^2}{[1 + k^2(L/U)^2\omega^2]^{p+3/2}}$$

where:

Type	p	k
1=von Karman	1/3	1.339
2=Dryden	1/2	1.0

and $\omega = 2\pi f$. The units of $S_q(\omega)$ are velocity squared per frequency (f).

3. Other power spectral density functions may be defined using the TABRND1 entry.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.

TABSCTL**Load Stepping Control Table - SOL 400**

This option allows the user to provide user criteria for load stepping control. It is referred to by an NLSTEP entry. The criteria defined herein are used for controlling the load step size. The criteria come in two flavors as defined in the NLSTEP entry: limit and target. When used as limits, the time step will be reduced if the criterion would be violated. When used as target, the time step will also be increased if the calculated results are less than what is specified by the criteria.

The criteria are calculated for elements or grids as defined below. By default all elements or nodes are used for evaluating the respective criterion, but this can be limited to specific sets. (See the [NLSTEP, 2616](#) entry.)

Format:

1	2	3	4	5	6	7	8	9	10
TABSCTL	TID								
	ICRIT_1	SET3_ID1	YT1_1	MT1_1	YT2_1	MT2_1	YT3_1	MT3_1	
	YT4_1	MT4_1							
	ICRIT_2	SET3_ID2	YT1_2	MT_2	YT2_2	MT2_1	YT3_2	MT3_2	
	YT4_2	MT4_2							
					
	ICRIT_n	SET3_IDn	YT1_n	MT1_n	YT1_n	MT2_n	YT3_n	MT3_n	
	YT4_n	MT4_n							

Example:

TABSCTL	17								
	ESI	84	.03	21.	0.4	55.	0.6		
	ESRI	18	.03	21.					
	EPSI	ALL	1.1						

Descriptor	Meaning
TID	ID of TABSCTL entry. (Integer > 0)
ICRITi	Type of user criterion to use and the type of entity it refers to: (Character; no Default)
ESI	Element Strain Increment
EPSI	Element Plastic Strain Increment
ECSI	Element Creep Strain Increment
ENCSI	Element Normalized Creep Strain Increment
ESTRI	Element STRes Increment
ESRI	Element Strain Energy Increment
GTI	Grid Temperature Increment

Descriptor	Meaning
GDI	Grid Displacement Increment
GRI	Grid Rotation Increment
ENSTRI	Element Normalized STRes Increment
SET3_IDi	ID of a SET3 Bulk Data entry. (Integer > 0 or (ALL or BLANK); Default = ALL)
YT1_i	First limit or target value for the current criterion i. (Real; no Default)
MT1_i	First range or target value for the current criterion i. See Remark 2. (Real-default blank-always active)
YT2_i	Second limit or target value for criterion. (Real; no Default)
MT2_i	Second range of applicability for criterion i. See Remark 2. (Real-default blank-always active)

Remarks:

1. This entry is selected by the CRITTID field of the NLSTEP Bulk Data entry.
2. For each criterion ICRTT_n, up to four ranges of target/limit values can be given. This allows the use of different limit/target values for different ranges of the corresponding total quality. For example, for the first criterion type, YT1_1 is the strain increment while MT1_1 is the largest total strain for which YT1_1 will be used. Typically, the last MTn used should be zero in which case the corresponding YTn will be used for all larger total values.
3. The TID must be unique among all TABSCTL entries.

TEMP**Grid Point Temperature Field**

Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Format:

1	2	3	4	5	6	7	8	9	10
TEMP	SID	G1	T1	G2	T2	G3	T3		

Example:

TEMP	3	94	316.2	49	219.8				
------	---	----	-------	----	-------	--	--	--	--

Descriptor	Meaning
SID	Temperature set identification number. (Integer > 0)
Gi	Grid point identification number. (Integer > 0)
Ti	Temperature. (Real)

Remarks:

1. In the static solution sequences, SID must be selected by the TEMP Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. From one to three grid point temperatures may be defined on a single entry.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. If the element material is temperature dependent, its properties are evaluated at the average temperature.
6. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data are defined. Gauss point temperatures are averaged for solid elements instead of grid point temperature.
7. For steady state heat transfer analysis, this entry together with the TEMPD and TEMPN1 entries supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT) = SID requests selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.

8. For transient heat transfer analysis, this entry together with the TEMPD and TEMPN1 entries supplies the initial condition temperatures. The Case Control command IC = SID requests selections of this entry. The temperature values specified here must be coincident with any temperature boundary condition specified.
9. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS, 417](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.

TEMPAX**Conical Shell Temperature**

Defines temperature sets for conical shell problems.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPAX	SID1	RID1	PHI1	T1	SID2	RID2	PHI2	T2	

Example:

TEMPAX	4	7	30.0	105.3					
--------	---	---	------	-------	--	--	--	--	--

Descriptor	Meaning
SIDi	Temperature set identification number. (Integer > 0)
RIDI	Ring identification number (see RINGAX entry). (Integer > 0)
PHIi	Azimuthal angle in degrees. (Real)
Ti	Temperature. (Real)

Remarks:

1. TEMPAX is allowed only if an AXIC entry is also present.
2. SIDi must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. Temperature sets must be selected with the Case Control command TEMP=SID.
4. One or two temperatures may be defined on each entry.
5. For a discussion of the conical shell problem, see [Restart Procedures](#) in the *MSC Nastran Reference Guide*.
6. TEMP(INIT) is not used with this entry.

TEMPB3**CBEAM3 Element Temperature Field**

Defines a temperature field for the three-node beam element (CBEAM3 entry).

Format:

1	2	3	4	5	6	7	8	9	10
TEMPB3	SID	EID	T(A)	T(B)	T(C)	TPY(A)	TPZ(A)	TPY(B)	
	TPZ(B)	TPY(C)	TPZ(C)	TC(A)	TD(A)	TE(A)	TF(A)	TC(B)	
	TD(B)	TE(B)	TF(B)	TC(C)	TD(C)	TE(C)	TF(C)		
		Element	ID	List					

Example:

TEMPB3	101	23	45.9	10.0	0.0	1.3	23.9	3.8	
		2.5	68.0	91.0	45.0		48.0	80.0	
	20.0		33.9			45.6			
	9	10	THRU	30	41	51	67	78	
	THRU	110	BY	2					

Descriptor	Meaning
SID	Temperature set identification number. (Integer > 0; Required)
EID	Element identification number. (Integer > 0; Required)
T(<i>j</i>)	Temperature at <i>j</i> (<i>j</i> =A,B,C) on the neutral axis. (Real; Default = 0.0)
TP <i>i(j</i>)	Effective linear gradient in local direction <i>i</i> (<i>i</i> = y, z) at <i>j</i> (<i>j</i> = A, B, C). (Real; Default = 0.0)
T <i>i(j</i>)	Temperature at stress recovery point <i>i</i> (<i>i</i> =C, D, E, F) defined in PBEAM3 at location <i>j</i> (<i>j</i> =A, B, C). (Real; Default = 0.0; see Remark 3.)
Element ID	List of CBEAM3 element identification numbers. Character strings "THRU" and "BY"
List	may be used in the list. (Integer > 0; "THRU" or "BY". At least one element ID is required.)

Remarks:

1. In the static solution sequences, SID must be selected by the TEMP Case Control command.
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOAD*i* or TLOAD*i* entry.
2. SID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. If all T*i(j*) fields are blank, linear temperature gradients are assumed for stress recovery.

4. Temperature field defined by TEMPB3 entry always takes precedence over the grid point temperatures given by TEMP and TEMPD entries.
5. The effective thermal gradients are defined in the local coordinate system. For their definitions, see Remark 6 of Bulk Data entry TEMPRB for the details.

TEMPBC**Grid Point Temperatures**

Defines the temperature boundary conditions for heat transfer analysis. Applies to steady-state and transient conditions (SOLs 153 and 159 only).

Format:

1	2	3	4	5	6	7	8	9	10
TEMPBC	SID	TYPE	TEMP1	GID1	TEMP2	GID2	TEMP3	GID3	

Example:

TEMPBC	10	STAT	100.0	1	100.0	2	100.0	3	
--------	----	------	-------	---	-------	---	-------	---	--

Alternate Format and Example:

TEMPBC	SID	TYPE	TEMP1	GID1	"THRU"	GID2	"BY"	INC	
--------	-----	------	-------	------	--------	------	------	-----	--

TEMPBC	20	STAT	100.0	4	THRU	50	BY	2	
--------	----	------	-------	---	------	----	----	---	--

Descriptor	Meaning
SID	Temperature set identification number. (Integer > 0)
TYPE	Type of temperature boundary condition. See Remarks. (Character; Default = "STAT"): STAT - Constant temperature boundary condition TRAN - Time-varying temperature boundary condition
TEMPi	Temperature (Real)
GIDI	Grid point identification number. (Integer>0 or "THRU" or "BY")
INC	Grid point number increment. (Integer)

Remarks:

- For a constant Boundary Condition (TYPE = "STAT"), the temperature boundary load set (SID) is selected in the Case Control Section (SPC = SID). TYPE = "STAT" may be used in both steady-state (SOL 153) and transient analysis (SOL 159).
- For transient analysis (SOL 159), a constant boundary condition should be specified using the SPC Bulk Data entry.
- For a time-varying boundary condition (TYPE = "TRAN"), SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. TYPE="TRAN" is permitted only in transient analysis (SOL 159). A function of time $F(t - \tau)$ defined on the TLOADi entry multiplies the general load. τ provides any required time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis.

4. In the alternate format, TEMP1 is the nodal temperature for the grid points GID1,GID1+INC,...,GID2. If "BY" and INC are not specified, then the grid point number increment is unity.
5. If TYPE = "STAT", then no SPCi Bulk Data entries may be specified.
6. If TYPE = "TRAN", then no CELAS2 or DAREA Bulk Data entries may be specified. Also, "U" must be specified in the CONV field on the entry to obtain accurate results.
7. All TEMPBC entries in the Bulk Data Section must indicate either TYPE = "STAT" or TYPE = "TRAN" but not both.
8. In transient thermal analysis, the TEMPBC option is used to set a grid, known temperature as a function of time. Internally NASTRAN uses SLOAD and CELAS2 entries to enforce the temperature as a function of time. The $u=P/K$ or temperature is equal to SLOAD divided by CELAS2. The default stiffness for the CELAS2 entry is 1.0E10. This value is fine most of the time. However, if the user desired to run the model using thermal conductivity in the following unit (Btu/sec.inch.F), then it may run into a numerically convergence issue. This is because the thermal conductivity for this unit has conductivity value in the 1.0E-6 range. The avoidance is to set a NASTRAN system cell, TBCMAG to 1.0E2.
9. The TEMPBC with type=TRAN is no longer supported in SOL 400 transient thermal analysis. One should use the SPC1 and SPCD to enforced temperature. For example, convection coefficient as a function of time, mass flow rate as a function of time, or ambient temperature as a function of time all used the TEMPBC,TRAN option to apply a time varying nodal quantity in SOL 159. Following are the procedures for a user to convert SOL 159 into SOL 400 when there is TEMPBC,TRAN entries.

The SPC entry used to fixed a boundary temperature at a particular value at all time. However, if you have a TEMPBC,TRAN in your test file this means that all the permanent SPC must be converted into SPCD and SPC1 with a unit step function on the TABLED1 in SOL 400. The avoidance to this, using a constant temperature with SPC and a time-varying temperature in the same run, is using the large stiffness method to enforced the time varying quantity. The large stiffness used SLOAD and CELAS2 which $u = P / K$ which $P = \text{SLOAD}$, and $K = \text{CELAS2}$, and u is the desired temperature times the time-varying quantity in the TABLEDx.

To Convert SOL 159 Models to SOL 400 Models:

- a. Executive Control Section - change SOL 159 to SOL 400.
- b. Case Control Section - replace ANALYSIS=HEAT by ANALYSIS=HTRAN, also add SPC if all temperature boundary conditions are transient (the following Case 3b).
- c. Bulk Data Section - replace the "TRAN" type TEMPBC by SPC1 and SPCD. The details are explained below.

If all temperature boundary conditions are constant, no changes are required.

If all temperature boundary conditions are transient, replace TEMPBC by SPC1 and SPCD and modify TLOAD1.

For example, replace the following entries of SOL 159 model:

TLOAD1,40,400,,4000
TEMPBC,400,TRAN,300.0,99

by

SPC = 111 (Case CC)
:
TLOAD1,40,400,,1,4000
SPCD,400,99,,300.0
SPC1,111,,99

If a model has both constant and transient temperature boundary conditions, all boundary conditions must be converted into SPC1 and SPCD.

For example, replace the following entries of SOL 159 model:

DLOAD,222,1.0,1,0,30,1.0,40
TLOAD1,40,400,,4000
TEMPBC,400,TRAN,300.0,99
SPC,111,98,,20.0

by

DLOAD,222,1.0,1,0,30,1.0,40, **1.0,50**
TLOAD1,40,400,,1,4000
SPCD,400,99,,300.0
SPC1,111,,99
TLOAD1,50,500,,1,5000
SPCD,500,98,,20.0
SPC1,111,,98
TABLED1,5000,,,,
,0.0,1.0,1000.0,1.0,ENDT

10. If TYPE=TRAN and the initial temperature is non zero, then the initial temperature must be supplied. Use case control IC and bulk data TEMP entries.

TEMPD

Grid Point Temperature Field Default

Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP or TEMPN1 (for heat transfer analysis) entries.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPD	SID1	T1	SID2	T2	SID3	T3	SID4	T4	

Example:

TEMPD	1	216.3							
-------	---	-------	--	--	--	--	--	--	--

Descriptor	Meaning
SIDi	Temperature set identification number. (Integer > 0)
Ti	Default temperature value. (Real)

Remarks:

1. In the static solution sequences, SID must be selected by the TEMP Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. SIDi must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. From one to four default temperatures may be defined on a single entry.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. If the element material is temperature dependent, its properties are evaluated at the average temperature.
6. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data is defined.
7. For steady-state heat transfer analysis, this entry together with the TEMP and TEMPN1 entries supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT) = SID requests selection of this entry. The temperature values specified here must be coincident with any temperatures boundary conditions that are specified.

8. For transient heat transfer analysis, this entry together with the TEMP and TEMPN1 entries supplies the initial condition temperatures. The Case Control command IC=SID request selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.
9. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS, 417](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.
10. For partitioned Bulk Data superelements and auxiliary models, TEMPD must be specified in all partitioned Bulk Data Sections.
11. If Modules are present then this entry may only be specified in the main Bulk Data section.

TEMPF**p-Element Temperature Field with Function Definition**

Defines the thermal loading to be applied to a group of elements.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPF	SID	EID1	FTEMP	FTABID					
	EID2	EID3	-etc.-						

Example:

TEMPF	127	12	111						
-------	-----	----	-----	--	--	--	--	--	--

Alternate Format:

TEMPF	SID	EID1	FTEMP	FTABID					
	EID2	“THRU”	EIDn						

Descriptor	Meaning	Type	Default
SID	Temperature set identification number.	Integer > 0	Required
FTEMP	ID of a DEQATN entry describing the temperature field as a function of x,y,z. See Remark 1.	Integer > 0	
FTABID	ID of a TABLE3D entry describing the temperature field. See Remark 1.	Integer > 0	
EIDi	Identification numbers of the p-elements to which this thermal load is applied.	Integer > 0	Required

Remarks:

1. Either FTEMP or FTABID must be specified but not both.
2. The TEMPF entry overrides the temperature at the element vertices specified on the TEMP or TEMPD entries.

TEMPN1

TOP/BOT/MID Grid Point Temperature Field for Heat Shell Element in SOL 400

Defines initial temperature at grid points of heat shell elements with linear or quadratic temperature distribution across the thickness direction.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPN1	SID	G1	C1	T1	G2	C2	T2		

Example:

TEMPN1	10	100	123	1300.					
--------	----	-----	-----	-------	--	--	--	--	--

Descriptor	Meaning
SID	Temperature set identification number. (Integer > 0)
Gi	Grid point identification number. (Integer > 0)
Ci	Component numbers. ($0 \leq \text{Integer} \leq 3$; up to 3 unique Integers may be placed in the field with no embedded blanks.) 1=TOP, 2=BOT, 3=MID. (Integer > -1; Default = 1)
Ti	Temperature. (Real)

Remarks:

1. This entry is for shell elements defined on a PSHLN1 heat transfer.
2. In the steady-state solution sequences, SID is selected by the LOAD Case Control command.
3. In the transient solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.
4. The TEMPN1 Bulk Data entry is used to complement the TEMP Bulk Data entry. The TEMP entry will just initiate the TOP grids. This entry allows specifying of the TOP, BOT, and MID or any combination. See the NLMOPTS, TEMGO, vmaptg Bulk Data entry on how to list internally generated grids.
5. For steady-state heat transfer analysis, this entry together with the TEMPD and TEMP entries supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT)=SID requests selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.
6. For transient heat transfer analysis, this entry together with the TEMPD and TEMP entries supplies the initial condition temperatures. The Case Control command IC=SID requests selections of this entry. The temperature values specified here must be coincident with any temperature boundary condition specified.

TEMPP1**Plate Element Temperature Field, Form 1**

Defines a temperature field for plate, membrane, and combination elements (by an average temperature and a thermal gradient through the thickness) for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPP1	SID	EID1	TBAR	TPRIME	T1	T2			
	EID2	EID3	EID4	EID5	EID6	EID7	-etc.-		

Example:

TEMPP1	2	24	62.0	10.0	57.0	67.0			
	26	21	19	30					

Alternate Format and Example of Continuation Entry:

	EID2	“THRU”	EIDi	EIDj	“THRU”	EIDk			
	1	THRU	10	30	THRU	61			

Descriptor	Meaning
SID	Temperature set identification number. (Integer > 0)
EIDi, EIDj, EIDk	Unique element identification number(s). (Integer > 0 or the continuation entries may have “THRU” in fields 3 and/or 6, in which case EID2 < EIDi and EIDj < EIDk.)
TBAR	Temperature at the element’s reference plane as defined by ZOFFS on the connection entry. (Real, Default 0.0)
TPRIME	Effective linear thermal gradient. Not used for membranes. (Real, Default 0.0)
T1, T2	Temperatures for stress calculation at points defined on the element property entry. (Z_1 and Z_2 field on PSHELL entry.) T1 may be specified on the lower surface and T2 on the upper surface for the CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR elements. These data are not used for membrane elements. See Remark 9. If both T1 and T2 are blank, they are computed from the equation $T = \text{TBAR} + z \cdot \text{TPRIME}$, where z is the distance from the center fiber. The program replaces T1 with a flag, and z is computed in a later operation. (Real)

Remarks:

1. In the static solution sequences, SID must be selected by the TEMP Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry. TBAR and TPRIME are used for the analysis. If both are left blank the elements in essence see no thermal loading. T1 and T2 are used for post analysis for stress calculations.

2. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. If continuation entries are present, EID1 and elements specified on the continuation entry are used. Elements must not be specified more than once.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1 or TEMP RB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMP D entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. For temperature field other than a constant gradient, the “effective gradient” for a homogeneous plate is

$$\text{TPRIME} = \frac{1}{I} \int_z T(z) z dz$$

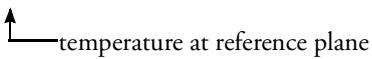
where I is the bending inertia and z is the distance from the neutral surface in the positive normal direction.

6. The “average” temperature for a homogeneous plate is

$$\text{TBAR} = \frac{1}{\text{Volume}} \int_{\text{Volume}} T d\text{Volume}$$

7. If the element material is temperature dependent, its properties are evaluated at the average temperature TBAR.
8. Large “THRU” ranges will lead to System Fatal Message 3008 (“Insufficient Core”) and should be avoided, particularly for open sets.
9. If the element material is nonlinear then T1 and T2 should be left blank (see the MATS1 entry).
10. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS, 417](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.
11. The bending and twisting moments can be reduced to outer fiber stresses and combined with membrane stresses in the composite plate elements. If, in addition, the temperature is specified by the user at a point where outer fiber stresses are calculated, the thermal expansion due to the difference between the specified temperature and the temperature that would be produced by a uniform gradient, T' , is assumed to be completely restrained. Stated differently, the second and higher order moments of the thermal expansion are assumed to be completely restrained by elastic stiffness. The resulting stress increment is

$$\{\Delta\sigma\} = -[G_e]\{\alpha_e\}(T - T_o - T'z)$$



temperature at reference plane

where $[G_e]$ and $\{\alpha_e\}$ are evaluated for the average temperature of the element \bar{T} .

[TEMPP3](#)

Plate Element Temperature Field, Form 3

TEMPP3 is no longer available. Use TEMPP1.

TEMPPRB**One-Dimensional Element Temperature Field**

Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPPRB	SID	EID1	TA	TB	TP1A	TP1B	TP2A	TP2B	
	TCA	TDA	TEA	TFA	TCB	TDB	TEB	TFB	
	EID2	EID3	EID4	EID5	EID6	EID7	-etc.-		

Example:

TEMPPRB	200	1	68.0	23.0	0.0	28.0		2.5	
	68.0	91.0	45.0		48.0	80.0	20.0		
	9	10							

Alternate Format and Example of Continuation Entry:

	EID2	"THRU"	EIDi	EIDj	"THRU"	EIDk			
	2	THRU	4	10	THRU	14			

Descriptor	Meaning
SID	Temperature set identification number. (Integer > 0)
EIDi, EIDj, EIDk	Unique element identification number(s). (Integer > 0 or the second continuation entry may have "THRU" in fields 3 and/or 6 in which case EID2 < EIDi and EIDj < EIDk.)
TA, TB	Temperature at end A and end B on the neutral axis. (Real, default 0.0)
TPij	Effective linear gradient in direction i on end j; used with CBAR, CBEAM, and CBEND only. (Real)
Tij	Temperature at point i (i=C, D, E, or F) as defined on the PBAR, PBEAM, and PBEND entries at end j (j=A or B). This data is used for stress recovery only with CBAR, CBEAM, and CBEND exclusively. See Remark 3. (Real)

Remarks:

1. In the static solution sequences, SID must be selected by the TEMP Case Control command.
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. SID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.

3. If at least one nonzero or nonblank T_{ij} is present, the point temperatures given are used for stress recovery. If no T_{ij} values are given, linear temperature gradients are assumed for stress recovery. The T_{ij} values are not used in the calculation of differential stiffness. The default for T_{ij} for $j=A$ is TA and the default for T_{ij} for $j=B$ is TB .
4. If the second (and succeeding) continuation is present, EID1 and elements specified on the second (and succeeding) continuations are used. Elements must not be specified more than once.
5. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1 or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
6. The effective thermal gradients in the element coordinate system for the CBAR element are defined by the following integrals over the cross section. For end "A" (end "B" is similar),

$$TA = \frac{1}{A} \int TA(y, z) dA$$

$$TP1A = \frac{I_2}{\Delta} \int_A (y - y_n) TA(y, z) dA - \frac{I_{12}}{\Delta} \int_A (z - z_n) TA(y, z) dA$$

$$TP2A = \frac{I_1}{\Delta} \int_A (z - z_n) TA(y, z) dA - \frac{I_{12}}{\Delta} \int_A (y - y_n) TA(y, z) dA$$

$$\Delta = I_1 I_2 - I_{12}^2$$

if $I_{12} = 0$

$$TP1A = \frac{1}{I_1} \int_A (y - y_n) TA(y, z) dA$$

$$TP2A = \frac{1}{I_2} \int_A (z - z_n) TA(y, z) dA$$

where $TA(y, z)$ is the temperature at point y, z (in the element coordinate system) at end "A" of the bar. See the CBAR entry description for the element coordinate system: I_1 , I_2 and I_{12} are the moments of inertia about the z and y axes, respectively. The temperatures are assumed to vary linearly along the length (x-axis). Note that if the temperature varies linearly over the cross section, then TP1A, TP1B, TP2A and TP2B are the actual gradients.

7. If the element material is temperature-dependent, the material properties are evaluated at the average temperature $(TA + TB)/2$.
8. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS, 417](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.
9. If any T_y is specified the stresses computed by the effective gradient are corrected by $\Delta\sigma$ such that:

$$\sigma = \sigma|_{T_A + y^{TP1A} + z^{TP2A}} + \Delta\sigma$$

where $\Delta\sigma$ is in the form

$\Delta\sigma = -\alpha E [T_{CA} - T_o - C_1 \cdot TP1A - C_2 \cdot TP2A]$ etc

for CBAR and CBEAM

$\Delta\sigma = -\alpha E [T_{CA} - T_o - (C_1 + \Delta N) \cdot TP1A - C_2 \cdot TP2A]$ etc

for CBEND.

TERMIN

Control to Terminate a SOL 600 Analysis Under Certain Conditions

Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
TERMIN	ID	NC							
	NTYPE	NBN	ICRIT	VAL					

Example:

TERMIN	2	2							
	7	1000	-1	2.0					
	7	1000	-2	0.8					

Descriptor	Meaning
ID	ID corresponding to a Case TERMIN entry. (Integer > 0; no Default)
NC	Number of termination conditions to be specified. (Integer > 0; Default = 1; Max number is 10)
NTYPE	<p>Termination Criteria Type (Integer; no Default)</p> <p>Enter 1 if termination occurs when a percentage of the boundary nodes are in contact.</p> <p>Enter 2 if termination occurs when the maximum force on a rigid body is exceeded.</p> <p>Enter 3 if termination occurs when the displacement of the rigid body exceeds the allowed displacement.</p> <p>Enter 5 if termination occurs when the distance between the reference points of two rigid bodies is less or greater than the specified value.</p> <p>Enter 6 if termination occurs, when any displacement in body, is greater than the specified value.</p> <p>Enter 7 if termination occurs, when the displacement in the node, is greater than the specified value.</p>
NBN	Body number, for criterion type 7, grid ID (Integer > 0; no Default)

Descriptor	Meaning
ICRIT	<p>Criteria specification. (Integer; no Default)</p> <p>For criterion type 1, enter the percentage of nodes to be in contact for termination; default = 100.</p> <p>For criterion type 2, enter direction 1/2/3 for the x, y, z global directions</p> <p>For criterion type 5, enter the second body.</p> <p>For criterion type 6 or 7, enter the degree of freedom.</p> <p>For criterion type 6 or 7, enter -1 if the total translational displacement.</p> <p>For criterion type 6 or 7, enter -2 if the total rotation.</p>
VAL	<p>Termination value. (Real; no Default)</p> <p>For criterion type 2, enter the critical force.</p> <p>For criterion type 3, enter the critical maximum displacement.</p> <p>For criterion type 5, enter the critical distance. If the value is positive, the termination occurs when the distance is less than the value. If the value is negative, the termination occurs when the distance is greater than the value in a positive sign.</p> <p>For criterion type 6 or 7, enter the critical distance (rotation).</p>

Remarks:

1. Different TERMIN entries may be used in different subcases.
2. Not all subcases require TERMIN entries if used in other subcases.

TF

Dynamic Transfer Function

Defines a dynamic transfer function of the form

$$(B_0 + B_1 \cdot p + B_2 \cdot p^2)u_d + \sum_i (A_0(i) + A_1(i)p + A_2(i)p^2)u_i = 0 \quad (9-34)$$

Where:

U_d = dependent coordinate

U_i = independent degree of freedom

p = differential operator ($p=d/dt$)

Can also be used as a means of direct matrix input. See Remark 4.

Format:

1	2	3	4	5	6	7	8	9	10
TF	SID	GD	CD	B0	B1	B2			
	G(1)	C(1)	A0(1)	A1(1)	A2(1)	-etc.-			

Example:

TF	1	2	3	4.0	5.0	6.0			
	3	4	5.0	6.0	7.0				

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
GD, G(i)	Grid, scalar, or extra point identification numbers. (Integer > 0)
CD, C(i)	Component numbers. (Integer zero or blank for scalar or extra points, any one of the Integers 1 through 6 for a grid point.)
B0, B1, B2	Transfer function coefficients. (Real)
A0(i), A1(i), A2(i)	

Remarks:

1. Transfer function sets must be selected with the Case Control command TFL = SID.
2. Continuation entries are optional.
3. The matrix elements defined by this entry are added to the dynamic matrices for the problem.
4. The constraint relation given in Eq. (9-34) will hold only if no structural elements or other matrix elements are connected to the dependent coordinate u_d . In fact, the terms on the left side of Eq. (9-34) are simply added to the terms from all other sources in the row for u_d .

5. See the [MSC Nastran Dynamic Analysis User's Guide](#) for a discussion of transfer functions.
6. For each SID, only one logical entry is allowed for each GD, CD combination.
7. For heat transfer analysis, the initial conditions must satisfy Eq. [\(9-34\)](#).
8. RC network does not support TF for thermal analysis.
9. For more information see [DMIGs, Extra Points, and Transfer Functions](#) (p. 658) in the *Dynamic Analysis User's Guide*.

THPAD**ROMAC's THPAD User Defined Service Element Property**

Allows the user to provide the parameters for a tilting pad bearing for use with ROMAC's THPAD service.

Format:

1	2	3	4	5	6	7	8	9	10
THPAD	RID	TLIMIT							
Title1									
Title2									
	RADIUS	CLEAR	OUTR	LENGT H	ALPHP	E	ALPHJ	ALPHS	
	NPADS	NEL	IECC						
	ARC1	OFFSET1	PRELOAD1	PVANG1	IP1	KP1	MP1	IT1	
	ARC2	OFFSET2	PRELOAD	PVANG2	IP2	KP2	MP2	IT2	
	
	KFILM	KPAD	TBACK	TJF	PSUMP	TIN	KCAV	CCAV	
	DENSIT Y	SPEC	TA	MUA	TB	MUB	ESUMP	TMANU	
	ERROR	XG	YG			NAX	FACTOR	XFACT	
	ITJ	IBC	ITUR	IDIM		ICOND	ITB	ITG	
			ICAV	ICROSS	IDEF	IFLEX	IUN	ISUMP	
	KTHETA	DEL0	HOTOVER	COLDOVE R					
	NCASE	MAXC	IAPH						
	RPM1	FX1	FY1	PGNU1	RELAX1	OFLOW1	DEREL1		
	RPM2	FX2	FY2	PGNU2	RELAX2	OFLOW2	DEREL2		
	

Example:

In the FMS Section of the Nastran input stream a connect statement is required:

```
CONNECT SERVICE GR1 'SCR.MDSolver.Obj.Uds.Elements.thpad'
```

In Bulk Data:

CBUSH2D	100	200	1	2		XY			
PBUSH2D	200								
ELEMUDS	200	PBUSH2D	GR1	THPAD	FREQ				
THPAD	200	1000.0							
	Tilting pad bearing used for the 8 stage centrifugal compressor								

	Load on Pad								
	2.5	0.0038	3.5	3.0	3.0E7	6.5E-6	6.5E-6	6.5E-6	
	5	30	0						
	60.0	0.5	0.3	54.0	0.001	0.0	0.0	0.0	
	60.0	0.5	0.3	126.0	0.001	0.0	0.0	0.0	
	60.0	0.5	0.3	198.0	0.001	0.0	0.0	0.0	
	60	0.5	0.3	270.0	0.001	0.0	0.0	0.0	
	60.0	0.5	0.3	342.0	0.001	0.0	0.0	0.0	
	2.0E-6	0.00067	120.0	175.0	0.0	175.0	5.0	0.0	
	8.0E-5	180.0	100.0	4.13E-6	210.0	6.8E-7	1.0	75.0	
	0.001	0.2	0.2			2.0	100.0	100.0	
	0	2	1	0		0	0	0	
			1	0	2	0	0	0	
	0.0	0.0	100.0	0.0					
	2	100	100						
	1000.0	0.0	931.5	1.0	1.0	0.0	1.0		
	2000.0	0.0	931.5	1.0	1.0	0.0	1.0		

Descriptor	Meaning
RID	ROMAC bearing identification number that matches the property identification number on an ELEMUDS entry as well as a PBUSH2D entry. (Integer > 0)
TLIMIT	Time limit for a call to THPAD. If the call takes longer than this, the service call will be killed. (Real)
TITLE1	User supplied title. (Character)
TITLE2	User supplied title. (Character)
RADIUS	Journal radius (Real, in)
CLEAR	Ground-in pad clearance. (Real, in)
OUTR	Outer radius of bearing pads. (Real, in)
LENGTH	Axial length of bearing pads. (Real, in)
E	Young's modulus for the pads. (Real, psi)
ALPHP	Pad thermal expansion coefficient. (Real, 1/F)
ALPHJ	Journal thermal expansion coefficient. (Real, 1/F)
ALPHS	Shell thermal expansion coefficient. (Real, 1/F)
NPADS	Number of pads. (Integer > 0)
NEL	Number of elements per pad. (Integer > 0)
IECC	Eccentricity variation flag. (Integer)
0	Allow eccentricity variation to match load.

Descriptor	Meaning
1	Eccentricity fixed at initial value.
ARCi	Arc length for the pad. (Real, deg)
OFFSETi	Offset factor for the pad. (Real)
PRELOADi	Preload factor for the pad. (Real)
PVANGi	Location of pad pivot angle. (Real, deg)
IPi	Pad polar moment. (Real, lb-sec ² -in)
KPi	Pad pivot stiffness. KPi=0 for rigid pivots. (Real, lb/in)
MPi	Pad mass. (Real, lb-sec ² /in)
ITi	Pad bending inertia. (Real, in ⁴)
KFILM	Fluid thermal conductivity. (Real, btu/in-sec-F)
KPAD	Pad thermal conductivity. (Real, btu/in-sec-F)
TBACK	Back of pad temperature. (Real, F)
TJF	Temperature of the journal surface. (Real, F)
PSUMP	Sump oil pressure. (Real, psi)
TIN	Pad inlet oil temperature. (Real, F)
KCAV	Thermal conductivities cavitation ratio. (Real)
CCAV	Cavitation latent heat ratio. (Real, btu/sec-in ² -F)
DENSITY	Lubricant density. (Real, lbf-sec ² /in ⁴)
SPEC	Lubricant specific heat. (Real, btu-in/lbf-sec ² -F)
TA	First temperature datum. (Real, F)
MUA	Viscosity at TA. (Real)
TB	Second temperature datum. (Real, F)
MUB	Viscosity at TB. (Real)
ESUMP	Sump heating factor. (Real)
TMANU	Clearance set temperature. (Real, F)
ERROR	Error criterion. (Real)
XG	Initial guess of x/cp. (Real)
YG	Initial guess of y/cp. (Real)
NAX	Axial pressure exponent. (Real, 2.0 is recommended)
FACTOR	Perturbation effect on position iteration. (Real)
XFACT	Percentage of cross-coupled terms in position update. (Real)
ITJ	Journal temperature flag. (Integer)
0	Journal temperature found as average of film temperature.

Descriptor	Meaning
	1 Journal temperature fixed by user. 2 Journal temperature set for zero heat flux to shaft.
IBC	Boundary condition flag. (Integer) 0 Reynolds boundary condition with flow correction. 1 Not used. 2 Reynolds boundary condition for pressure.
ITUR	Turbulence flag. (Integer) 0 Laminar solution. 1 Turbulence allowed.
IDIM	Dimension flag. (Integer) 0 Two-dimensional conduction in the pad. 1 Radial conduction in the pad.
ICOND	Conduction flag. (Integer) 0 Energy equation includes conduction. 1 Adiabatic (isothermal) energy equation.
ITB	Back temperature flag. (Integer) 0 Pad back temperature equals sump temperature. 1 Pad back temperature fixed by user.
ITG	Groove temperature flag. (Integer) 0 Groove temperature found iteratively by heat balance. 1 Groove temperature fixed at sump temperature.
ICAV	Cavitation flag. (Integer) 0 No cavitation effects in energy equation. 1 Include cavitation effects in energy equation.
ICROSS	Cross-film viscosity flag. (Integer) 0 Constant cross-film viscosity. 1 Variable cross-film viscosity.
IDEF	Deformation flag. (Integer) 0 No pad/pivot deformations. 1 Pivot deformations. 2 Pad, journal and shell thermal deformations. 4 Pad mechanical deformations.
IFLEX	Type of beam bending for pad deformations. (Integer)

Descriptor	Meaning
	0 Curved beam analysis. 1 Straight beam analysis. 2 Simple curvature calculation.
IUN	Unloaded pads flag. (Integer) 0 Effect of unloaded pads is considered. 1 Effect of unloaded pads is neglected.
ISUMP	Sump temperature flag. (Integer) 0 Sump temperature set to pad inlet oil temperature. 1 Sump temperature found iteratively by heat balance.
KTHETA	Pivot rotational stiffness. (Real, lb-in, Default = 0.0)
DEL0	Initial pad angle for moment balance. (Real, rad, Default = 0.0)
HOTOVER	Percentage of hot oil available carried to the groove. (Real, Default = 100.0)
COLDOVER	Percentage of groove oil made up of inlet oil. (Real, Default = 0.0)
NCASE	Number of speed/load cases. (Integer > 0)
MAXC	Maximum number of position iterations. (Real, Default = 100)
IALPH	New stiffness smoothing factor. (Real)
RPMi	Speed of journal. (Real, rpm)
FXi	Applied force in the negative x-direction. (Real, lb)
FYi	Applied force in the negative y-direction. (Real, lb)
PGNUi	Whirl ratio of shaft. (Real)
RELAXi	Position iteration relaxation factor. (Real)
OFLOWi	Oil flow to bearing. (Real, cips)
DERELi	Relaxation factor for pivot deformation iteration. (Real)

Remarks:

1. This entry triggers the call to the THPAD service. The GROUP must match the GROUP field of the CONNECT SERVICE FMS entry.
2. On the FMS CONNECT entry, only the CONNECT SERVICE can be used with this entry.
3. PID must match an existing ELEMUDS and PBUSH2D PIDs.
4. All units must be in the English system.
5. Refer to <http://www.virginia.edu/romac/> for more information.

6. Use of this entry requires, that the user, has obtained the source for the THPAD routine from the *University of Virginia Rotating Machinery and Controls Laboratory (ROMAC)* and use the MSC Nastran Software Development Kit (SDK) to build it as a User Defined Service (UDS). SDK and UDS build instructions are available in the [MSC Nastran Rotordynamics User's Guide](#).

TIC**Transient Analysis Initial Condition**

Defines values for the initial conditions of variables used in structural transient analysis. Both displacement and velocity values may be specified at independent degrees-of-freedom (See Remark 6.). This entry may not be used for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TIC	SID	G	C	U0	V0				

Example:

TIC	100	10	3	0.1	0.5				
-----	-----	----	---	-----	-----	--	--	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
G	Grid, scalar, or extra point or modal coordinate identification number. (Integer > 0). See Remark 4.
C	Component numbers. (Any one of the integers 1 through 6 for grid points, integer zero or blank for scalar or extra points and -1 for modal coordinates.)
U0	Initial displacement. (Real)
V0	Initial velocity. (Real)

Remarks:

1. Transient analysis initial condition sets must be selected with the IC Case Control command. Note the use of IC in the Case Control command versus TIC on the Bulk Data entry. For heat transfer, the IC Case Control command selects TEMP or TEMPD entries for initial conditions and not the TIC entry.
2. If no TIC set is selected in the Case Control Section, all initial conditions are assumed to be zero.
3. Initial conditions for coordinates not specified on TIC entries will be assumed to be zero.
4. In direct transient analysis (SOL 109 and 129) as well as in modal transient analysis (SOL 112) wherein the TIC Bulk Data entry is selected by an IC or IC(PHYSICAL) Case Control command, G may reference only grid, scalar or extra points. In modal transient analysis (SOL 112) wherein the TIC Bulk Data entry is selected by an IC(MODAL) Case Control command, G may reference only modal coordinates or extra points.
5. The initial conditions for the independent degrees-of-freedom specified by this Bulk Data entry are distinct from, and may be used in conjunction with, the initial conditions for the enforced degrees-of-freedom specified by TLOAD1 and/or TLOAD2 Bulk Data entries.
6. SOL700 does not support U0 (initial displacement).

TICEL**Transient Initial Conditions of Elements**

Defines the initial values of element variables at the beginning of the analysis. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
TICEL	SID	SETID	NAME1	VALUE1	NAME2	VALUE2	-etc.-		

Example:

TICEL	3	40	DENSITY	100.	SIE	1.E5			
-------	---	----	---------	------	-----	------	--	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0; Required)
SETID	Number of a SET1 entry defining the elements to be initialized. (Integer > 0; Required)
NAMEi	Element variable to be initialized. See Remark 5. (Character; Required)
VALUEi	Value of the variable. (Real; Required)

Remarks:

- Initial conditions for elements that are not specified on TICEL entries are assumed to be zero except density, which is set to the reference density.
- Only initial conditions that are selected in the Case Control Section (IC = SID) will be activated by the solver.
- As many continuation lines as required can be used to specify all the variables being initialized. A blank field terminates the list.
- Element variables for Eulerian elements can be initialized with a TICEL or a TICEUL1 entry. The TICEL entry initializes a set of elements, while the TICEUL1 entry initializes either a set of elements or geometrical regions (sphere, cylinder,...). When a Euler element is part of both a TICEL and a TICEUL1 entry, the TICEL entry takes precedence, and overrules the TICEUL1 initialization for the element.
- The following variables for NAMEi can be used to initialize the Eulerian regions:

XVEL	x-velocity
YVEL	y-velocity
ZVEL	z-velocity
DENSITY	Density
SIE	Specific internal energy
Q	Artificial viscosity

DIV	Divergence
VOID	Void fraction
FMAT	Material fraction
XMOM	x-momentum
YMOM	y-momentum
ZMOM	z-momentum

6. To initialize the pressure use density. And depending on the equation of states also define the specific internal energy (SIE).
7. For the Euler solvers, you can, in addition to the “normal” element variables that the solver has defined, also define an initial radial velocity field. You have to enter the location of the center from where the radial emerges, the velocity to be applied to the element center and the decay coefficient for the velocity field. The center is defined by the keywords “X-CENTER, Y-CENTER, Z-CENTER”, the radial velocity by “R-VEL” and the decay coefficient by "DECAY". You have to input these keywords in the above order, and have every keyword followed by its value.

TICEUL1

Transient Initial Conditions of Eulerian Regions

Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
TICEUL1	SID	TSID							

Example:

TICEUL1	300	200							
---------	-----	-----	--	--	--	--	--	--	--

Descriptor	Meaning
SID	Unique TICEUL1 number referenced from a PEULER1 entry. (Integer > 0; Required)
TSID	Group of geometric region TICREG ID. (Integer > 0; Required)

Remarks:

1. Element variables for Eulerian elements can be initialized with a TICEL or a TICEUL1 entry. The TICEL entry initializes a set of elements, while the TICEUL1 entry initializes either a set of elements or geometrical regions (sphere, cylinder, ...). When a Euler element is part of both a TICEL and a TICEUL1 entry, the TICEL entry takes precedence and overrules the TICEUL1 initialization for the element.

TICEUDS

User-defined Transient Initial Conditions of Euler Elements or Lagrangian grid points.

Defines the initial values of element or grid point variables at the beginning of the analysis by a user written subroutine. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
TICEUDS	SID	GROUP	UNAME	SETID	COPT				

Example:

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE initex 'SCAI.MDSolver.Obj.Uds.Dytran.InitOut'
```

In Bulk Data:

1	2	3	4	5	6	7	8	9	10
TICEUDS	12	initex	EXINIT	100	GRID				

Descriptor	Meaning	
SID	Unique output number. (Integer > 0; Required)	
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)	
UNAME	User subroutine name associated with the entry. (Character; default=EXINIT)	
SETID	Number of a SET1 entry defining the elements or grids to be initialized. (Integer > 0, Required)	
COPT	Flag for assigning initial velocities. (Character; default=Element)	
	Element	Apply the initial variables on elements
	GRID	Apply the initial variables on grids

Remarks:

1. Only can be used for SOL 700.
2. Initial conditions to be used must be selected in the Case Control Section (TIC = SID).
3. UNAME=EXINIT can only be used.

TICREG**Transient Initial Conditions of Eulerian Regions**

Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
TICREG	TRID	TSID	TYPE	VID	MID	TSID	LEVEL		

Example:

TICREG	300	200	SPHERE	400	100	3	4.0		
--------	-----	-----	--------	-----	-----	---	-----	--	--

Descriptor	Meaning	
TRID	Unique TRID number. (Integer > 0; Required)	
TSID	ID of group of Euler regions referenced from the TICEUL1 entry. (Integer > 0; Required)	
TYPE	The type of Eulerian region. (Character; Required)	
	SURF	Region inside or outside a surface.
	SPHERE	Region inside a sphere.
	CYLINDER	Region inside a cylinder.
	BOX	Region inside a box.
	ELEM	Region defined by list of Euler elements.
VID	ID of a geometric entity. (Integer > 0; Required)	
	Type:	Region:
	SURF	SURFINI
	SPHERE	SPHERE
	CYLINDER	CYLINDR
	BOX	BCBOX
	ELEM	SET1
MID	ID of a MATDEUL entry defining this material. (Integer ≥ 0 ; Default = 0)	
TSID	ID of a TICVAL entry containing a list of initial values for this material. (Integer ≥ 0 ; Default = 0)	
LEVEL	Level indicator for this material and initial values. (Real; Default = 0.0)	

Remarks:

1. A number of TICREG may exist in the input file with the same TSID. The TICEUL entry will combine all TICREGs with the same TSID into one initial definition for the Euler elements that are referenced from the same PEULER1 definition.
2. When the material number is left blank or zero, the Eulerian elements inside the region will be void. Note that this is not allowed in the Riemann solution-based Euler solvers, as they will not handle void elements. If you define void elements and select either the 1stOrder or 2ndOrder scheme, an error message will be issued and the analysis will stop.
3. All level indicators LEVEL of the same TSID group must have different values. The level indicator can be negative.
4. See also the parameter MICRO for the accuracy of the initial value generation.

TICVAL**Transient Initial Condition Set**

Defines the initial values of an Eulerian geometric region. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
TICVAL	TSID	METHOD	NAME1	VALUE1	NAME2	VALUE2	NAME3	VALUE3	
	NAMEi	VALUEi	-etc.-						

Example:

TICVAL	3		DENSITY	100.	YVEL	25.	SIE	3.7	
	XVEL	3.5							

Descriptor	Meaning
TSID	Unique TICVAL number referenced from a TICEUL entry. (Integer > 0; Required)
METHOD	Type of input definition. (Character; Default = NORMAL)
RADIAL	Initializes material with radial profiles. The entries VALUEi are interpreted as TABLED1 IDs. See Remarks 2., 3., and 4.
NORMAL	Normal initialization.
NAMEi	Variable to be initialized. See TICEL, 3251 . (Character; Required)
VALUEi	Value of the variable. (Real; Required)

Remarks:

- Element variables for Eulerian elements can be initialized with a TICEL or a TICEUL1 entry. The TICEL entry initializes a set of elements, while the TICEUL1 entry initializes either a set of elements or geometrical regions (sphere, cylinder, ...). When an Euler element is part of both a TICEL and a TICEUL1 entry, the TICEL entry takes precedence and overrules the TICEUL1 initialization for the element.
- METHOD = RADIAL allows to map results of a spherical symmetric 1-D solution onto a full 3-D model. For initialized variables SIE and DENSITY, a 1-D table has to be defined that specifies the variable value for a number of distances from the center. The center is by default (0,0,0) but can be changed by setting X-CENTER, Y-CENTER, Z-CENTER. The velocity is a radial velocity and has to be specified as R-VEL. Its values is also a TABLED1 ID.
- DYPARAM,SPHERSYM can be used to define a proper 1-D spherical mesh and speeds up the run by taking only the mesh-size in radial direction into account.

4. Radial initialization of JWL is supported. The entries DETSPH and the JWL entry from the 1-D spherical solution stage have to be included in the remap run. Alternatively, the 1-D solution may be run with JWL and the follow-up run with ideal gas, provided that all JWL material has fully ignited. Radial initialization of EOSIG is not supported. In the follow-up run, ideal gas material has to be used instead of IG material.

TIC3**Transient Analysis Initial Velocity with Increment Options**

Allows for the definition of a velocity field of grid points consisting of a rotation and a translation specification. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
TIC3	SID	G		SCALE					+
+	XVEL	YVEL	ZVEL	XROT	YROT	ZROT			+
+	G1	G2	THRU	G3	BY	G4			

Example:

TIC3	7	5		10.					+
+	100.0			5.0		-7.5			+
+	1	2	THRY	1000	BY	23			

Descriptor	Meaning
SID	Number of a set of loads. (Integer > 0; Required)
G	Number of a grid point at the center of rotation. (Integer > 0; Required)
SCALE	Scale factor of initial velocity. (Real, Default=1.0)
XVEL, YVEL, ZVEL	Initial translational velocity components. (Real; Default = 0.0)
XROT, YROT, ZROT	Initial rotational velocity components. (Real; Default = 0.0)
G1, G2, ...	Grid points to be initialized. THRU indicates a range of grid points. BY is the increment to be used within this range. (Integer > 0; Required)

Remarks:

1. Any number of TIC3 entries can be used.
2. The rotational velocity components are defined in radians per unit time.
3. For six degree of freedom grid points, the angular velocity components are also initialized.
4. Initial conditions for grid points that are not specified on TICn entries are assumed to be zero.
5. If the THRU specification is used, the grid points in the range definition are required to exist. The BY option enables grid points to be ignored in this range.
6. None of the fields in the list of grid points can be blank or zero, since this indicates the end of the list.
7. The initial conditions to be used in SOL 700 must be selected in the Case Control Section (IC = SID).

8. If grid points are part of a rigid body, it is recommended you enable double precision in SOL700. It is possible that a single precision SOL700 analysis will not assign the correct initial velocities.

TIM2PSD

FFT conversion tool for use in random vibration fatigue analysis

Fast Fourier Transformation (FFT) conversion tool for use in SOL 108 or SOL 111 random vibration fatigue analysis. This entry is used to convert time history data into power spectral density (PSD) functions directly input to the fatigue analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TIM2PSD	ID	SRATE	WINDOW	FORMAT	MEANS	NSKIP			
	“EVENT”	EVNTID1					T1	$\delta 1$	
		EVNTID2					T2	$\delta 2$	
		...							
		EVNTIDn					Tn	δn	
	“DELETE”	EVNTID1	Ti1_1	Tf1_1	Ti2_1	Tf2_1	Ti3_1	Tf3_1	
		EVNTID2	Ti1_2	Tf1_2	Ti2_2	Tf2_2	Ti3_2	Tf3_2	
		...							
		EVNTIDn	Ti1_n	Tf1_n	Ti2_n	Tf2_n	Ti3_n	Tf3_n	
	“MAP”	LCID1	CHAN1	LCID2	CHAN2	LCID3	CHAN3		
		LCID4	CHAN4	...	-etc.-				

Example:

TIM2PSD	42	512.0	HANNIN G	CSV	YES	1			
	EVENT	80001					2.0	1.0	
		80002					1.0	0.5	
		80003					2.0	1.0	
	DELETE	80001	1.1	1.9					
	MAP	1	3	2	5	3	7		

Descriptor	Meaning
ID	Unique identification number referenced by FATIGUE case control, otherwise TIM2PSD entry is ignored (Integer > 0).
SRATE	Number of samples per second (required) - thus dt = 1/SRATE (Real; No Default).
WINDOW	Window function to use. Choices are HANNING or NONE. This is applied to the "block" of data extracted from the total time signal. (Character; Default = HANNING).

Descriptor	Meaning
FORMAT	Format of time signal files (RPC or CSV) (Character; Default = CSV). RPC files are binary channel data usually with the .rsp file extension. CSV files are typical MS Excel comma separated files in which the channels are in the columns (i.e. three columns of data is three channels of data - y-values only. x-values assumed to be in ascending order with no regards to length of time between points).
MEANS	Used to decide if overall signal mean values are to be calculated (YES or NO) (Character; Default = NO). Currently this is only calculated and not applied in the analysis. Ignored if no mean stress correction specified on FTGPARM entry.
NSKIP	Number of header lines to skip if an FORMAT=CSV file (Integer>=0; Default=0). The next continuation lines are required and repeat for as many Events as necessary.
EVENT	Flag indicating that event parameters are to follow (one optional set for each event).
EVNTID _i	Number of this Event - must correspond to the TID of a FTGLOAD entry of TYPE=PSD that is referenced by an active FTGEVNT entry. (Note: it is NOT the actual FTGEVNT ID).
T _i	Length of window function in time for this Event (Real>0.0; no Default).
δ _i	Overlap or gap in time between windows for this Event (real) (+ means overlap) (Real>=0.0; Default = 0.0). The next continuation lines are optional and can repeat for as many Events as necessary.
DELETE	Flag indicating that event parameters are to follow for signal deletion. For any Event, time value pairs can be specified for deleting up to three (3) portions of the time signal for each Event.
EVNTID _i	Number of this Event - correspond to a previously defined EVNTID _i in the "EVENT" section above.
T _i ₁ _{_i}	Used to specify sections in an Event (defined by pairs of time values t ₁ -t ₂ , t ₃ -t ₄ , t ₅ -t ₆)
T _f ₁ _{_i}	to delete from Event files before FFT process is applied (Real>0; t ₁ <=t ₂ <=t ₃ <=t ₄ <=...; no Defaults; if none are specified, entire signal is used). Only three (3) delete pairs per event are currently supported. The next continuation lines are optional and only necessary if the SUBCASEs corresponding to the channel data is not one-to-one.
MAP	Flag used to map the channel data and load case (SUBCASE) IDs.
LCID _i	SUBCASE of transfer function (TF) corresponding to <i>i</i> th load event. SUBCASEs must be in ascending order in the Case Control! There cannot be more SUBCASEs listed than there are channels of data in the CSV/RPC files.
CHAN _i	Channel # corresponding to <i>i</i> th load event. Channels must be referenced in ascending order and cannot be repeated.

Remarks:

1. TIM2PSD bulk data entries are ignored if not selected by a FATIGUE case control.

2. The actual RPC/CSV files are specified with a UDNAME entry referenced by TID field on the FTGLOAD entry for each Event (FTGEVNT). This is a requirement in the presence of a TIM2PSD entry.
3. All events must be in the same format, use the same window function, have the same sample rate, and have the same number and order of channels.
4. Each event can be a different length and can have a different block length and gap.
5. If LOGLVL=1 or 2 on the FTGPARM entry, the input time histories and direct PSD computations are written to CSV files for possible plotting with MS Excel.
6. The TID on the FTGLOAD is also used as the SID of the auto-generated RANDPS entries in this process.
7. A file containing the auto-generated RANDPS and TABRND1 entries is created in the same directory as the referenced UDNAME filename. This file can then be used as an include file for subsequent analyses that use the same generated PSD data rather than using the TIM2PSD entry again in subsequent runs.
8. This process also determines the overall mean values of the time histories for each event. The means are normally thrown away in conversion to PSDs and cross-PSDs. Currently the mean effect is not taken into account in the subsequent fatigue analysis. These means are appended to the end of the file containing the auto-generated RANDPS and TABRND1 entries.
9. MAP flag is not necessary if all channels in RPC/CSV file are used in the exact same order as the SUBCASEs corresponding to the transfer functions. SUBCASE IDs must be in ascending order. Channels must be referenced in ascending order.

TLOAD1

Transient Response Dynamic Excitation, Form 1

Defines a time-dependent dynamic load or enforced motion of the form

$$\{P(t)\} = \{A\} \cdot F(t - \tau)$$

for use in transient response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	EXCITEID	DELAYI/ DELAYR	TYPE	TID/F	US0	VS0		

Example:

TLOAD1	5	7	15	LOAD	13				
--------	---	---	----	------	----	--	--	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0) See Remarks 1. and 5.
EXCITEID	Identification number of a static load set or a DAREA or SPCD entry set or a thermal load set (in heat transfer analysis) that defines $\{A\}$. See Remarks 2. and 3. (Integer > 0)
DELAYI	Identification number of DELAY Bulk Data entry that defines time delay τ . See Remark 8. and 13. (Integer > 0 or blank)
DELAYR	Value of time delay τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 8. and 13. (Real or blank)
TYPE	Defines the type of the dynamic excitation. See Remarks 2., 3. and 12. (Integer, character or blank; Default = 0)
TID/F	Identification number of TABLEDi entry that gives $F(t)$. (Integer > 0). Value of F to be used for all times. (Real, non-zero)
US0	Factor for initial displacements of the enforced degrees-of-freedom. See Remarks 9., 11. and 14. (Real; Default = 0.0)
VS0	Factor for initial velocities of the enforced degrees-of-freedom. See Remarks 10., 11. and 14. (Real; Default = 0.0)

Remarks:

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.

2. The type of the dynamic excitation is specified by TYPE (field 5) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS, or DISP	Enforced displacement using large mass or SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using large mass or SPC/SPCD data.
3, A, AC, ACC or ACCE	Enforced acceleration using large mass or SPC/SPCD data
4	FLOW boundary condition on the face of an Eulerian solid element (SOL 700 only).
12	Velocity of the center of gravity of a rigid body (SOL 700 only)
13	Force or moment on the center of gravity of a rigid body (SOL 700 only).

For enforced displacement, velocity and acceleration, the large mass method is not supported in SOL 400.

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 109, 112, 146, 200, and where applicable SOL 400.

3. TYPE (field 5) also determines the manner in which EXCITEID (field 3) is used by the program as described below

Excitation specified by TYPE is applied load

- There is no LOADSET request in Case Control

EXCITEID may reference DAREA, static and thermal load set entries

- There is a LOADSET request in Case Control

The program may reference DAREA entries as well as static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.

Excitation specified by TYPE is enforced motion

- There is no LOADSET request in Case Control

EXCITEID will reference SPCD entries. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference DAREA and static and thermal load set entries just as in the case of applied load excitation.

- There is a LOADSET request in Case Control

The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID, just as in the case of applied load excitation.

4. EXCITEID may reference sets containing QHBDY, QBDYi, QVECT, QVOL and TEMPBC entries when using the heat transfer option.
5. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
6. If the heat transfer option is used, the referenced QVECT entry may also contain references to functions of time, and therefore A may be a function of time.
7. If TLOADi entries are selected in SOL 111 or 146 then a Fourier analysis is used to transform the time-dependent loads on the TLOADi entries to the frequency domain and then combine them with loads from RLOADi entries. Then the analysis is performed as a frequency response analysis but the solution and the output are converted to and printed in the time domain. Please refer to [Fourier Transform](#) (Ch. 20) in the *MSC Nastran Dynamic Analysis User's Guide*.
8. If the DELAYI/DELAYR field is blank or zero, τ will be zero.
9. The USO field is used only when the dynamic excitation defined by the TYPE field is enforced velocity or enforced acceleration using SPC/SPCD specification. The initial displacements for the enforced degrees-of-freedom in this case are given by the product $\{A\}(US0)$ where $\{A\}$ is defined by the EXCITEID field.
10. The VS0 field is used only when the dynamic excitation defined by the TYPE field is enforced acceleration using SPC/SPCD specification. The initial velocities for the enforced degrees-of-freedom in this case are given by the product $\{A\}(VS0)$ where $\{A\}$ is defined by the EXCITEID field.
11. The initial conditions for the enforced degrees-of-freedom implied by the US0 and VS0 fields are distinct from, and may be used in conjunction with, the initial conditions for the independent degrees-of-freedom specified by a TIC Bulk Data entry (which, in turn, is selected by an IC Case Control command).
12. For TYPE=4, TID must be blank if it references a FLOW entry. Use the FLOWT entry to define a time dependent flow boundary on the face of an Eulerian element.
13. For RC network solver in thermal analysis, the DELAY1/DELAYR is ignored.
14. Fields US0 and VS0 are not supported in SOL 700.
15. DELAYI/DELAYR (Integer>0 or Real) is not supported for the follower force in SOL129 and SOL400. The corresponding loads are neglected in the analysis.
16. If Modules are present then this entry may only be specified in the main Bulk Data section.

TLOAD2**Transient Response Dynamic Excitation, Form 2**

Defines a time-dependent dynamic excitation or enforced motion of the form

$$\{P(t)\} = \begin{cases} 0 & , \quad t < (T1 + \tau) \text{ or } t > (T2 + \tau) \\ \{A\}\tilde{t}^B e^{\tilde{C}\tilde{t}} \cos(2\pi F\tilde{t} + P) & , \quad (T1 + \tau) \leq t \leq (T2 + \tau) \end{cases}$$

for use in a transient response problem, where $\tilde{t} = t - T1 - \tau$

Format:

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	EXCITEID	DELAYI/ DELAYR	TYPE	T1	T2	F	P	
	C	B	US0	VS0					

Example:

TLOAD2	4	10	5.0		2.1	4.7	12.0		
	2.0								

Descriptor	Meaning
SID	Set identification number. See Remarks 1. and 6. (Integer > 0)
EXCITEID	Identification number of a static load set or a DAREA or SPCD entry set or a thermal load set (in heat transfer analysis) that defines $\{A\}$. See Remarks 2. and 3. (Integer > 0)
DELAYI	Identification number of DELAY Bulk Data entry that defines time delay τ . See Remark 5. (Integer > 0 or blank)
DELAYR	Value of time delay τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 5. (Real or blank)
TYPE	Defines the type of the dynamic excitation. See Remarks 2. and 3. (Integer; character or blank; Default = 0)
T1	Time constant. (Real ≥ 0.0)
T2	Time constant. (Real; $T2 > T1$)
F	Frequency in cycles per unit time. (Real ≥ 0.0 ; Default = 0.0)
P	Phase angle in degrees. (Real; Default = 0.0)
C	Exponential coefficient. (Real; Default = 0.0)
B	Growth coefficient. (Real; Default = 0.0)

Descriptor	Meaning
US0	Factor for initial displacements of the enforced degrees-of-freedom. See Remarks 9., 11. and 16. (Real; Default = 0.0)
VSO	Factor for initial velocities of the enforced degrees-of-freedom. See Remarks 10., 11. and 16.(Real; Default = 0.0)

Remarks:

1. Dynamic excitation sets must be selected with the Case Control command with DLOAD=SID.
2. The type of the dynamic excitation is specified by TYPE (field 5) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS, or DISP	Enforced displacement using large mass or SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using large mass or SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration using large mass or SPC/SPCD data
12	Velocity of the center of gravity of a rigid body. (SOL 700 only)
13	Force or moment on the center of gravity of a rigid body. (SOL 700 only)

For enforced displacement, velocity and acceleration, the large mass method is not supported in SOL 400.

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 109, 112, 146, 200, and where applicable SOL 400.

3. TYPE (field 5) also determines the manner in which EXCITEID (field 3) is used by the program as described below

Excitation specified by TYPE is applied load

- There is no LOADSET request in Case Control

EXCITEID may reference DAREA, static and thermal load set entries

- There is a LOADSET request in Case Control

The program may reference DAREA entries as well as static and thermal load set entries specified by the LID or TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.

Excitation specified by TYPE is enforced motion

- There is no LOADSET request in Case Control

EXCITEID will reference SPCD entries. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference DAREA and static and thermal load set entries just as in the case of applied load excitation.

- There is a LOADSET request in Case Control

The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID, just as in the case of applied load excitation.

- EXCITEID (field 3) may reference sets containing QHBDY, QBDYi, QVECT, and QVOL and TEMPBC entries when using the heat transfer option.
- If the DELAYI/DELAYR field is blank or zero, τ will be zero.
- SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
- If the heat transfer option is used, the referenced QVECT entry may also contain references to functions of time, and therefore A may be a function of time.
- If TLOADi entries are selected in SOL 111 or 146 then a Fourier analysis is used to transform the time-dependent loads on the TLOADi entries to the frequency domain and then combine them with loads from RLOADi entries. Then the analysis is performed as a frequency response analysis but the solution and the output are converted to and printed in the time domain. In this case, B will be rounded to the nearest integer. Please refer to [Fourier Transform](#) (Ch. 20) in the *MSC Nastran Dynamic Analysis User's Guide*.
- The USO field is used only when the dynamic excitation defined by the TYPE field is enforced velocity or enforced acceleration using SPC/SPCD specification. The initial displacements for the enforced degrees-of-freedom in this case are given by the product $\{A\}(US0)$ where $\{A\}$ is defined by the EXCITEID field.
- The VS0 field is used only when the dynamic excitation defined by the TYPE field is enforced acceleration using SPC/SPCD specification. The initial velocities for the enforced degrees-of-freedom in this case are given by the product $\{A\}(VS0)$ where $\{A\}$ is defined by the EXCITEID field.
- The initial conditions for the enforced degrees-of-freedom implied by the US0 and VS0 fields are distinct from, and may be used in conjunction with, the initial conditions for the independent degrees-of-freedom specified by a TIC Bulk Data entry (which, in turn, is selected by an IC Case Control command).
- The continuation entry is optional.
- TYPE=4 (SOL 700) is not supported using TLOAD2.
- For SOL 700, TLOAD2 is converted to TLOAD1 and a TABLED1 is generated internally.
- RC network solver does not support TLOAD2 for thermal analysis.
- Fields US0 and VS0 are not supported in SOL 700.
- DELAYI/DELAYR (Integer>0 or Real) is not supported for the follower force in SOL129 and SOL400. The corresponding loads are neglected in the analysis.

18. If comparing results between pre-2005 versions and later versions, please note that results are shifted by one time step in later versions. The tload2 should be updated accordingly if there is a need to compare to these pre-2005 versions.
19. If Modules are present then this entry may only be specified in the main Bulk Data section.

TMPSET**Temperature Group Set Definition**

Define a time-dependent dynamic thermal load group for use in TTEMP Bulk Data entry in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
TMPSET	ID	G1	G2	G3	G4	G5	G6	G7	

Alternate Format:

TMPSET	ID	G1	"THRU"	G2	"BY"	INC			
--------	----	----	--------	----	------	-----	--	--	--

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	G8	G9	G10	G11	-etc.-				
--	----	----	-----	-----	--------	--	--	--	--

Continuation Entry Format 2:

	G8	"THRU"	G9	"BY"	INC				
--	----	--------	----	------	-----	--	--	--	--

Example:

TMPSET	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Descriptor	Meaning
ID	Temperature group identification number. (Integer > 0)
Gi	Grid point Identification numbers in the group. (Integer > 0)

Remarks:

1. This entry is used in SOL 400 only when ANALYSIS=NLTRAN (nonlinear transient analysis) and the temperature load is applied. It only applies to the nonlinear elements in the Residual (SEID=0).
2. GROUP_ID determines the group of a specified the time-dependent distribution of temperatures. It is used by the TTEMP Bulk Data entry to define the corresponding TABLEDi entry. GROUP_ID must be unique for all of the other TMPSET entries.
3. The temperature of grid point Gi must be defined using TEMP, TEMPD, TEMPP1, or TEMPRB Bulk Data entry. These bulk data entries must have the same SID as that referenced on the associated TTEMP Bulk Data entry.

TODYNA

Defines the Start of Direct Text to Dytran

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

Format:

1	2	3	4	5	6	7	8	9	10
TODYNA									

Example:

TODYNA									
MAT1	345	29.0E6		0.285	0.0004				
ENDDYN									

TOMVAR

Topometry Design Variable

Defines a design region for topometry optimization (element-by-element optimization).

Format:

1	2	3	4	5	6	7	8	9	10
TOMVAR	ID	TYPE	PID	PNAME/FID	XINIT	XLB	XUB	DELXV	
	“DLINK”	TID	C0	C1					
	“DDVAL”	DSVID							
	“STRESS”	STLIM							

Example:

Design all element's thickness referencing PSHELL ID = 5 with initial design = 10.0 ($t_0 = 10.0$ input element thickness), lower bound $0.5 \cdot t_0$ and upper bound $1.5 \cdot t_0$.

TOMVAR	1	PS1	PSHELL	5	T	10.0			
	STRESS	5.0							

Example:

Design all element's Young Modulus referred by PSHELL ID = 100 with initial design XINIT = 3.E+5, XLB=1.0, and XUB= 1.0E+6.

TOMVAR	10	PSHELL	100	E	3.E+5	1.0	1.E+6		
--------	----	--------	-----	---	-------	-----	-------	--	--

Descriptor	Meaning
ID	Unique topometry design region identification number. (Integer > 0)
TYPE	Property entry type. Used with PID to identify the elements to be designed. See Remark 2. (Character: “PBAR”, “PSHELL”, ‘PSOLID’, and “PCOMP”, etc.)
PID	Property entry identifier (Integer > 0). This PID must be unique for PIDs referenced by other TOPVAR, DVPREL1, DVPREL2, DVMREL1, and DVMREL2 entries. Topometry, topology, and sizing variables cannot share the same properties. (Integer > 0). Combined topometry, topology, topography, sizing, and shape variables are allowed.
PNAME/FID	Property name or property material name, such as “T”, “A”, “E”, and “GE”, or field position of the property entry or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referenced by field position. Only one property value for each property can be designed by TOMVAR in a job. See Remark 2. (Character or Integer > 0.)

Descriptor	Meaning
XINIT	Initial value. (Real or blank, no Default). Typically, XINIT is defined to match the mass target constraint (so the initial design does not have violated constraints) or the analysis model input property value.
XLB	Lower bound. (Real or blank; Default = blank) . The default is XLB=0.5*XINIT.
XUB	Upper bound . (Real or blank; Default = blank). The default is XLB=1.5*XINIT.
DELXV	Fractional change allowed for the design variable during approximate optimization. See Remark 6. (Real > 0.0; Default = 0.5)
“DLINK”	Indicates that this line relates a ply thickness to another ply thickness. See Remark 10.
TID	TOMVAR entry identifier. (Integer > 0)
C0	Constant term. (Real; Default = 0.0)
C1	Coefficient term. (Real; no Default)
“DDVAL”	Indicates that this line defines discrete TOMVAR variables.
DSVID	DDVAL entry identifier. (Integer > 0)
“STRESS”	Indicates that this line defines a stress limit.
STLIM	Von Mises stress upper bound. See Remark 11. (Real >0.0)

Remarks:

1. Multiple TOMVAR's are allowed in a single file.
2. Property name and FID > 0 can be used for element property values just like the DVPREL1 Bulk Data entry. Only property name can be used for material property values like DVMREL1.
3. Property name “A” is shared by material and property entries. If “A” is used for PNAME, it is a Material. For PROD, PBEAM and PBAR, the integer field ID must be used to point to the area of these entries. TOMVAR supports MAT1 only.
4. PBARL, PBEAML, PBRSECT and PBMSECT are not supported.
5. For TYPE = PSHELL and PNAME selecting a material property, all the MATi fields on the PSHELL must be the same.
6. Combined topometry, topography, topology, sizing, and shape optimization is supported in a single file. However, topometry and topology cannot reference the same property ID. It is possible to topometry certain elements while sizing others. It is allowed to simultaneously design the same elements with topometry and desvar (sizing and/or shape) variables but topometry and sizing cannot reference the same property name.
7. Topometry optimization with element response constraints is slow due to many design variables. In this case, fully stressed design (FSD) is an alternative for certain problems
8. Parameters DESPCH and DESPCH1 specify when the topometry optimized design values are written to the element result history file jobname.des that can be imported to Patran and other post-processor to view topometry optimized results.
9. The TOMVAR entry cannot be used with thermal loads.

10. The DLINK line can only be used when TYPE=PCOMP. The PID must be the same as the PID given on the TOMVAR referenced by TID and the PNAMES in the two TOMVAR entries must differ
11. “STRESS” limits can only be used for PTYPE=PSHELL and PNAME= T. The Von Mises stress (at element center) constraints apply to all shell elements in both designed and non-designed regions. All TOMVAR entries must have the same STLIM.

TOPVAR

Topological Design Variable

Define a topology design region for topology optimization.

Format:

1	2	3	4	5	6	7	8	9	10
TOPVAR	ID	LABEL	PTYPE	XINIT	XLB	DELXV	POWER	PID	
	“SYM”	CID	MS1	MS2	MS3	CS	NCS		
	“CAST”	CID	DD	DIE	ALIGN				
	“EXT”	CID	ED	ALIGN					
	“PRINT”	CID	PD						
	“TDMIN”	TVMIN	TVMAX						
	“STRESS”	STLIM							

Example:

TOPVAR	1	PS1	PSHELL	5	T	10.0	4	1	
	STRESS	5.0							

Descriptor	Meaning
ID	Unique topology design region identification number. (Integer > 0)
LABEL	User-supplied name for printing purpose. (Character)
PTYPE	Property entry name. Used with PID to identify the elements to be designed. (Character: “PBAR”, “PSHELL”, ‘PSOLID’, etc.)
XINIT	Initial value. (Blank or Real, XLB < XINIT ≤ 1.0; Default = Blank). Typically, XINIT is defined to match the mass constraint on DRESP1=FRMASS, so the initial design does not have violated constraints. In this case, the default is set to the constraint value. If the mass (DRESP1=FRMASS or WEIGHT) is the objective, the default is 0.9. The default of XINIT is 0.6 for the other cases.
XLB	Lower bound to prevent the singularity of the stiffness matrix. (Real; Default = 0.001)
DELXV	Fractional change allowed for the design variable during approximate optimization. See Remark 3. (Real > 0.0; Default = 0.2)
POWER	A penalty factor used in the relation between topology design variables and element Young’s modulus. (Real > 1.0; Default = 3.0, or 4.0 if symmetry constraints used). 2.0 ≤ POWER ≤ 5.0 is recommended.
PID	Property entry identifier. This PID must be unique for PIDs referenced by other TOPVAR, DVPREL1 and DVPREL2 entries. Topology and sizing variables cannot share the same properties. (Integer > 0)
“SYM”	Indicates that this line defines symmetry constraints.

Descriptor	Meaning
CID	Rectangular coordinate system ID used for specifying manufacturing constraints. See Remark 4. (Blank or Integer > 0; Default = blank)
MSi	Mirror symmetry plane. See Remark 5. & 7. (Character, 'XY', 'YZ', or 'ZX')
CS	Cyclic symmetry axis. (character X, Y, Z). See Remark 12.
NCS	Number of cyclic symmetric segments in 360 degrees (Integer > 0). See Remark 9.
"CAST"	Indicates that this line defines casting constraints (i.e., die draw direction constraints). See Remarks 6., 7., 8., and 10.
DD	Draw Direction. DDi=X, Y, Z or X-, Y-, Z- for a single die option (DIE=1) where X-, Y-, Z- indicates the opposite direction of X, Y, and Z respectively. DDi=X, Y, and Z for two die option (DIE =2) (Character)
DIE	Die Options. (Blank or integer 1 or 2; Default = 1) = 1 (or blank). A single die will be used and the die slides in the given draw direction (i.e., material grows from the bottom in the draw direction) = 2. Two dies will be used and the dies split apart along the draw direction (i.e., material grows from the splitting plane in opposite direction along the axis specified by the draw direction DDi. The splitting plane is determined by optimization)
ALIGN	Indicates whether the designed property finite element mesh is precisely aligned with the draw direction or extrusion direction. (Character: "YES" or "NO" or Blank; Default = blank = "NO") See Remark 10.
"EXT"	Indicates that this line defines extrusion constraints (i.e., enforce constant cross-section) See Remark 6. and 7.
ED	Extrusion direction. (Character, X, Y, or Z)
"PRINT"	Indicates that this line defines overhang constraints (maximum overhang angle is 45 degree). See Remark 12.
CID	Rectangular coordinate system ID used for specifying overhang constraints.
PD	PRINT Direction. PD =X, Y, Z or X-, Y-, and Z- where X-, Y-, Z- indicates the opposite direction of X, Y, and Z respectively (Character).
"TDMIN"	Indicates that this line defines a minimum and/or maximum member size., See remarks 11. and 12.
TVMIN	Minimum member size. See Remarks 11. and 12. (Real>=0.0 or blank)
TVMAX	Maximum member size. See Remarks 11. and 12. (Real > TVMIN or blank)
"STRESS"	Indicates that this line defines a stress limit.
STLIM	Von Mises stress upper bound. See Remark 13.. (Real >0.0)

Remarks:

1. The topologically designable element properties include PROD, PBAR, PBARL, PBEND, PBEAM, PBEAML, PSHELL, PSHEAR, PSOLID, and PWELD. Multiple TOPVAR's are allowed in a single file. Combined topology, topography (BEADVAR), topometry (TOMVAR) sizing, and shape optimization is supported in a single file. However, TOPVAR cannot be used with DVMREL1 and DVMREL2 entries.
2. All designed element properties must refer to a MAT1 entry or MAT9 entry (PSOLID only); therefore, a PCOMP/PCOMPG cannot be used as designed property in topology optimization. PCOMP/PCOMPG's can be used as non-designed properties in a topology optimization job.
3. If DELXV is blank, the default is taken from the specification of DELX parameter on the DOPTPRM entry.
4. Only CORD1R and CORD2R can be used as a referenced coordinate system to specify topology manufacturing constraints. Only one reference coordinate system CID is allowed for each TOPVAR entry.
5. One, two or three different mirror symmetry planes can present (such as MS1=XY, MS2=YZ, and MS3=ZX).
6. Casting ("CAST") and Extrusion ("EXT") manufacturability constraints can be applied to PTYPE="PSOLID" only. Casting constraints cannot be combined with extrusion constraints for the same TOPVAR entry.
7. Some symmetry constraint types can be combined with casting or extrusion constraints. The referenced coordinate system CID must be the same for the combined constraints. Some possible combinations are:
 - For "EXT" constraints, possible combinations are (ED=X, MSi=XY, and/or ZX or CS=X), (ED=Y, MSi=YZ, and/or XY or CS=Y), (ED=Z, MSi=ZX, and/or YZ or CS=Z).
 - For "CAST" constraints, possible combinations are (DD=X or X-, MSi=XY and/or ZX or CS=X), (DD=Y or Y-, MSi=YZ and/or XY or CS=Y), (DD=Z or Z-, MSi=ZX and/or YZ or CS=Z).
8. For two dies option (DIE=2), the splitting plane is optimized. For a single die DIE=1, the parting plane is the bottom surface of the designed part in the draw direction.
9. The first symmetry segment starts at the X-axis when CS = Z (at Z-axis when CS = Y, and at the Y-axis when CS = X). One cycle symmetry can be combined with one mirror symmetry constraint as long as the axis of cyclic symmetry is normal to the plane of mirror symmetry. For example, MSi = YZ and CS = X, MSi = XZ and CS = Y, and MSi = XY and CS = Z. This feature can also be used for < 360 degrees but NCS must be given in 360 degrees.
10. It is recommended to use aligned mesh for casting property due to smaller tolerance used.
11. Without a TDMIN continuation line, the minimum member size constraint is taken from the specification of TDMIN parameter on the DOPTPRM entry. This option is applied on 2 and 3 D elements only. Minimum member size constraints can be used with "SYM", "CAST", and "EXT" constraints.

12. TVMIN and TVMAX are dimensional quantities. A guideline is that TVMIN be at least three times a representative element dimension. TVMAX must be greater than TVMIN and it is recommended that it be twice as big. If TVMAX is blank, no maximum member size is imposed. It is recommended that TVMIN always be used when TVMAX is specified.
13. “STRESS” limits can only be used for PTYPE=PSOLID and PSHELL referencing MAT1 only. The Von Mises stress (at element center) apply to all solid and/or shell elements in both designed and non-designed regions. All TOPVAR entries must have the same STLIM.
14. The TOPVAR entry cannot be used with thermal loads.
15. For normal mode topology optimization, lower and higher mode may switch during optimization. This often occurs while maximizing or constraining the first eigenfrequency. This leads to a diverging solution. A workaround is using the mean value of a few of the lowest eigenfrequency (3-6) by DRESP2.
16. The CASI solver is strongly recommended for solid elements topology problems for efficiency.
17. “PRINT” (overhang constraints) can be used for PTYPE=PSOLID only and limited to one TOPVAR entry. The result is mesh dependent. If used with minimum member size, the location of reference coordinate system may have some influence on the optimal design.

TRIM**Trim Variable Constraint**

Specifies constraints for aeroelastic trim variables.

Format:

1	2	3	4	5	6	7	8	9	10
TRIM	SID	MACH	Q	LABEL1	UX1	LABEL2	UX2	AEQR	
	LABEL3	UX3	-etc.-						

Example:

TRIM	1	0.9	100.	URDD3	1.0	ANGLEA	7.0	0.0	
	ELEV	0.2							

Descriptor	Meaning
SID	Trim set identification number. (Integer > 0). See remarks 1 and 2.
MACH	Mach number. (Real ≥ 0.0 and $\neq 1.0$)
Q	Dynamic pressure. (Real > 0.0)
LABELi	The label identifying aerodynamic trim variables defined on an AESTAT or AESURF entry. (Character)
UXi	The magnitude of the aerodynamic extra point degree-of-freedom. (Real)
AEQR	Flag to request a rigid trim analysis (Real ≥ 0.0 and ≤ 1.0 ; Default =1.0). A value of 0.0 provides a rigid trim analysis, see Remark 5.

Remarks:

1. The TRIM entry must be selected with the Case Control command TRIM=SID.
2. The SID must be unique among all the TRIM and TRIM2 entries.
3. The selected TRIM entry specifies the constrained values of the listed extra point degrees-of-freedom (“trim variables”) for a particular loading condition. These variables are listed on AESTAT and/or AESURF entries.
4. If MACH is less than 1.0, then the Doublet-Lattice theory is used. If MACH is greater than 1.0, then the ZONA51 theory is used.
5. AEQR=0.0 can be used to perform a rigid trim analysis (ignoring the effects of structural deformation on the loading). AEQR=1.0 provides standard aeroelastic trim analysis. Intermediate values are permissible, but have no physical interpretation (they may be useful for model checkout).

TRIM2

Trim Variable Definition

Defines the state of the aerodynamic extra points for a trim analysis. All undefined extra points will be set to zero.

Format:

1	2	3	4	5	6	7	8	9	10
TRIM2	SID	MACH	Q					AEQR	
	LABEL1	VALUE1	LABEL2	VALUE2	-etc.-				

Example:

TRIM2	1	0.9	100.						
	URDD3	1.0	ANGLEA	FREE	ELEV	0.2			

Descriptor	Meaning
SID	Trim set identification number (Integer>0). See Remarks 1. and 2.
MACH	Mach number. (Real ≥ 0.0 and $\neq 1.0$). See Remark 5.
Q	Dynamic pressure. (Real > 0.0)
AEQR	Flag to request a rigid trim analysis (Real ≥ 0.0 and ≤ 1.0 ; Default = 1.0). A value of 0.0 provides a rigid trim analysis. See Remark 6.
LABELi	The label identifying aerodynamic trim variables defined on an AESTAT, AESURF, or AEPARM entry (Character)
VALUEi	The value assigned to LABELi. Either a real number that indicates the variable's fixed value, or one of the following words: FREE, LINKED, or SCHED. See Remarks 3. and 4.

Remarks:

1. The TRIM2 entry must be selected with the Case Control command TRIM=SID.
2. The SID must be unique among all TRIM and TRIM2 entries.
3. A value of FREE indicates that the controller value will be solved for by the trim process. A value of LINKED indicates that the controller value will be set by an AELINK entry. A value of SCHED indicates that the controller value will be set by a CSSCHD entry. The LINKED and SCHED inputs are optional and provided as a convenience to the user. Nastran will determine the linked and scheduled controller states from the AELINK and CSSCHD entries, respectively.
4. All aerodynamic extra points that have not been defined on a TRIM2, AELINK, or CSSHED entry will be fixed to a value of zero.
5. If MACH is less than 1.0, then the Doublet-Lattice theory is used. If MACH is greater than 1.0, then the ZONA51 theory is used.

6. AEQR=0.0 can be used to perform a rigid trim analysis (ignoring the effects of structural deformation on the loading). AEQR=1.0 provides standard aeroelastic trim analysis. Intermediate values are permissible, but have no physical interpretation (they may be useful for model checkout).

TRMCPL

Trim Component Interface Coupling Parameters

Defines the interface coupling parameters for computing interface coupling matrices.

Format:

1	2	3	4	5	6	7	8	9	10
TRMCPL	TID	CTYPE	PLTOL	GAPTOL1	GAPTOL2	GAPTOL3	GAPTOL4		

Example:

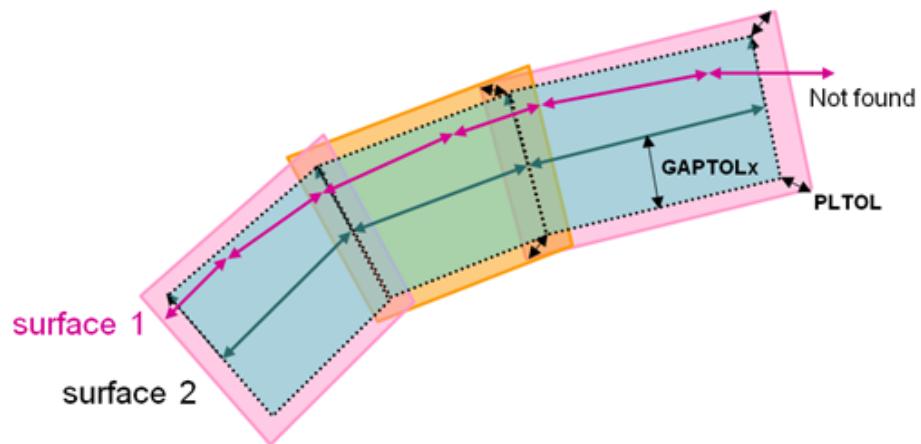
TRMCPL	1	SGLUED	0.12	5					
--------	---	--------	------	---	--	--	--	--	--

Descriptor	Meaning
TID	Identification number of trim component. (Integer>0; Required)
CTYPE	Interface coupling type; see Remark 2. (Character, "SGLUED", "SSLIDE", "SOPEN", "SIMPER" or "SAIRGAP"; Required)
PLTOL	Relative tolerance for in-plane/in-element/normal acceptance with respect to CTYPE. (Real>0.0; Default=0.1)
GAPTOLx	<ul style="list-style-type: none"> ■ GAPTOL1 Absolute tolerance for extrusion with respect to CTYPE. (Real>0.0; Default=0.01) ■ GAPTOL2 Second absolute tolerance for extrusion with respect to CTYPE. (Real>0.0; Default=GAPTOL1) ■ GAPTOL3 Third absolute tolerance for extrusion with respect to CTYPE. (Real>0.0; Default=GAPTOL2) ■ GAPTOL4 Fourth tolerance for extrusion with respect to CTYPE. (Real>0.0; Default=GAPTOL3)

Remarks:

1. For each interface coupling type, CTYPE, this entry is used to compute the interface matrix with incongruent meshes at the interface.
2. For the meanings of values on CTYPE, please see Bulk Data entry, ACPEMCP for the detailed descriptions.
3. TRMCPL is not a required entry for any trim component referenced by TRIMGRP and should be placed in the main bulk data section or under 'BEGIN BULK'.
4. GAPTOLx is the absolute tolerance for mapping surfaces of (structure, trim components) and (trim components, cavity). Grids on surfaces that are separated further than GAPTOLx will not be considered for coupling. GAPTOLx are utilized for successive projections. GAPTOL1 will be used for first projection. Then, GAPTOL2, GAPTOL3and GAPTOL4, if provided, will be used for subsequent projections for the remaining unprojected grids. GAPTOLx is in the same unit as the one used for length.

5. PLTOL is relative tolerance for computing volume contributions. PLTOL is in percentage.



TSTEP**Transient Time Step**

Defines time step intervals at which a solution will be generated and output in transient analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TSTEP	SID	N1	DT1	NO1					
		N2	DT2	NO2					
		-etc.-							

Example:

TSTEP	2	10	.001	5					
		9	0.01	1					

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
Ni	Number of time steps of value DTi. (Integer ≥ 1)
DTi	Time increment. (Real > 0.0)
NOi	Skip factor for output. Every NOi-th step will be saved for output. (Integer > 0 ; Default = 1)

Remarks:

1. TSTEP entries must be selected with the Case Control command TSTEP = SID.
2. Note that the entry permits changes in the size of the time step during the course of the solution. Thus, in the example shown, there are 10 time steps of value .001 followed by 9 time steps of value .01. Also, the user has requested that output be recorded for t = 0.0, .005, .01, .02, .03, etc.
3. See [Guidelines and Tools for Effective Dynamic Analysis](#) in *MSC Nastran Dynamic Analysis User's Guide* or a discussion of considerations leading to the selection of time steps.
4. In modal frequency response analysis (SOLs 111 and 146), this entry is required only when TLOADi is requested; i.e., when Fourier methods are selected.
5. The maximum and minimum displacement at each time step and the SIL numbers of these variables can be printed by altering DIAGON(30) before the transient module TRD1 and by altering DIAGOFF(30) after the module. This is useful for runs that terminate due to overflow or excessive run times.
6. For heat transfer analysis in SOL 159, use the entry.
7. If Modules are present then this entry may only be specified in the main Bulk Data section.

TSTEPNL**Parameters for Nonlinear Transient Analysis**

Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. is intended for SOLs 129, 159, 600 and SOLs 400 and 700.

Format:

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO	METHOD	KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS		
	MAXBIS	ADJUST	MSTEP	RB	MAXR	UTOL	RTOLB	MINITER	

Example:

TSTEPNL	250	100	.01	1	ADAPT	2	10	PW	
	1.E-2	1.E-3	1.E-6	2	10	2	.02		
	5	5	0	0.75	16.0	0.1	20.		

Descriptor	Meaning
ID	Identification number. (Integer > 0)
NDT	Number of time steps of value DT. See Remark 2. (Integer ≥ 3)
DT	Time increment. See Remark 2. (Real > 0.0)
NO	Time step interval for output. Every NO-th step will be saved for output. See Remark 3. (Integer $\neq 0$; Default = 1)
METHOD	Method for stiffness matrix update and the transient time integration schemes. For SOLs 129 and 159, only METHOD=ADAPT is allowed and it is also the default. For SOL 400, all options are allowed. The default is AUTO with non-contact analysis and FNT with contact analysis. See Remark 4. (Character: "AUTO", "ITER", "ADAPT", "SEMI", "FNT", or "PFNT")
KSTEP	KSTEP is the number of converged bisection solutions between stiffness updates for ADAPT method. (Default = 2). It is the number of iterations before stiffness updates for ITER method (Default = 10). See Remarks 18. and 19. (Integer > 0).
MAXITER	Limit on number of iterations for each time step. See Remarks 5., 6., and 18. (Integer $\neq 0$; Default = 10 for ADAPT method and 25 for non-ADAPT methods)
CONV	Flags to select convergence criteria. See Remarks 7. and 21. (Character; Default = "PW" for SOLs 129 and 159, "UPW" for SOL 400 with non-contact analysis, and "PV" for SOL 400 with contact analysis.)
EPSU	Error tolerance for displacement (U) criterion. See Remark 20. (Real $\neq 0.0$; Default = 1.0E-2 for all methods except PFTN. For PFTN, Default = -1.0E-2)
EPSP	Error tolerance for load (P) criterion. (Real > 0.0 ; Default = 1.0E-3 for SOLs 129 and 159, 1.0E-2 for SOL 400)

Descriptor	Meaning
EPSW	Error tolerance for work (W) or energy criterion. See Remark 20. (Real > 0.0; Default = 1.0E-6 for SOLs 129 and 159, 1.0E-2 for SOL 400 and non-PFNT methods, -1.0E-2 for SOL 400 and PFNT method)
MAXDIV	Limit on the number of diverging conditions for a time step before the solution is assumed to diverge. See Remark 8. (Integer ≠ 0; Default = 2)
MAXQN	Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark 9. (Integer ≥ 0; Default = 10 for all methods except PFNT. For PFNT, Default = 0)
MAXLS	Maximum number of line searches allowed per iteration. See Remark 9. (Integer ≥ 0; Default = 2 for all methods except PFNT. For PFNT, Default = 0)
FSTRESS	Fraction of effective stress ($\bar{\sigma}$) used to limit the subincrement size in the material routines. See Remark 10. (0.0 < Real < 1.0; Default = 0.2)
MAXBIS	Maximum number of bisections allowed for each time step. See Remark 11. and 12. (-9 ≤ Integer ≤ 9; Default = 5 except for MAXITER < 0 and SOL 400. For MAXITER < 0 and SOL 400, Default = 0)
ADJUST	Time step skip factor for automatic time step adjustment. See Remark 13. (Integer ≥ 0; Default = 5)
MSTEP	Number of steps to obtain the dominant period response. See Remark 14. (10 ≤ Integer ≤ 200; Default = variable between 20 and 40 for SOL 129 and between 10 and 20 for SOL 400.)
RB	Define bounds for maintaining the same time step for the stepping function during the adaptive process. See Remark 14. (0.1 ≤ Real ≤ 1.0; Default = 0.6)
MAXR	Maximum ratio for the adjusted incremental time relative to DT allowed for time step adjustment. See Remark 15. (Real ≥ 1.0; Default = 32.0)
UTOL	Tolerance on displacement or temperature increment below which a special provision is made for numerical stability. See Remark 16. (0.001 < Real ≤ 1.0; Default = 0.1)
RTOLB	Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 17. (Real > 2.0; Default = 20.0)
MINITER	Minimum number of iterations for a load increment, SOL 400 only. (Default = 1 except for the contact analysis. For contact analysis, Default = 2) It is recommended to set MINITER = 2 when high accuracy is expected)

Remarks:

- For SOL 129 and SOL 159, the Bulk Data entry is selected by the Case Control command = ID. Each residual structure subcase requires a entry and either applied loads via TLOADi data or initial values from a previous subcase. Multiple subcases are assumed to occur sequentially in time with the initial values of time and displacement conditions of each subcase defined by the end conditions of the previous subcase.

For SOL 400, the is selected in the STEP ID's listed within a SUBCASE ID. For SOL 400 each SUBCASE is independent of the previous statement.

2. NDT is used to define the total duration for analysis, which is NDT * DT. Since DT is adjusted during the analysis, the actual number of time steps, in general, will not be equal to NDT). Also, DT is used only as an initial value for the time increment.
3. For printing and plotting the solution with SOLs 129 and 159, data recovery is performed at time 0, NO*DT, 2*NO*DT, ..., and the last converged step, where DT is internally computed time increment and may change at every time step. For SOL 400 and NO > 0, data recovery is performed at time 0, NO*DTI, 2*NO*DTI, ..., and the last converged step, where DTI is the user input initial time increment and it is a constant. For SOL 400 thermal analysis and NO < 0, the SOL 159 scheme is used. For SOL 400 structural analysis and NO < 0, the SOL 129 scheme is used for SOL 400, i.e., data recovery is performed at time 0, |NO|*DT, 2*|NO|*DT, ..., and the converged step. See Remark 13. on how DT is changed.
4. The stiffness update strategy as well as the transient integration method is selected in the METHOD field.
 - a. METHOD=“ADAPT”: The program automatically adjusts the incremental time and uses bisection. During the bisection process, the stiffness matrix is updated every KSTEPth converged bisection solution. This is the **only** method available for SOLs 129 and 159 and is also their **default**.
 - b. METHOD=“AUTO”: The stiffness matrix is automatically updated to improve the convergence. Also, the program automatically adjusts the incremental time and uses bisection. The automatic time adjustment can be deselected by using ADJUST=0. KSTEP value is ignored during iteration. This is the **default** method for SOL 400.
 - c. METHOD = “ITER”: The stiffness is updated at every KSTEPth iterations. Also, the program automatically adjusts the incremental time and uses bisection. The automatic time adjustment can be deselected by using ADJUST=0.
 - d. METHOD=“SEMI”: Same as the AUTO method except that the stiffness updated at the first iteration, and then starts the AUTO iteration scheme.
 - e. METHOD = “FNT”: This is the Full Newton iteration method; the stiffness is updated at every iteration. In comparison with the PFNT method, the defaults for FNT are EPSU = 0.01, EPSW = 0.01 and MAXLS = 2. See Remark 19.
 - f. METHOD = “PFNT”: This is the Pure Full Newton iteration method. The PFNT method is the same as the FNT method except that the defaults for PFNT method are EPSU = -0.01, EPSW = -0.01, and MAXLS = 0. See Remark 19.
5. This remark applies to SOLs 129 or 159 only. The number of iterations for a time step is limited to MAXITER. If MAXITER is negative, the analysis is terminated when the divergence condition is encountered twice during the same time step or the solution diverges for five consecutive time steps. If MAXITER is positive, the program computes the best solution and continues the analysis until divergence occurs again. If the solution does not converge in MAXITER iterations, the process is treated as a divergent process. See Remark 8.

6. This remark applies to SOL 400 only. The number of iterations for a load increment is limited to $|MAXITER|$. If the solution does not converge in $|MAXITER|$ iterations, the load increment is bisected and the analysis is repeated. If the load increment cannot be bisected (i.e., MAXBIS is attained or $MAXBIS = 0$) and MAXDIV is positive, the best attainable solution is computed and the analysis is continued to the next time step. Best solutions for 4 time steps are computed. The analysis is terminated if the solution still diverges. If MAXDIV is negative, the analysis is terminated immediately.
- If MAXITER is negative, the solution is continued to the end of the current step, even if the solution is divergent. In this case, the best attainable solution is computed for each time step. The default for $MAXBIS = 1$, if $MAXITER < 0$. Also for SOL 400, the value of MAXITER for the AUTO method is an approximation. The program will try to obtain a converged solution if it senses the solution can converge.
7. The convergence test flags (U = displacement error test, P = load equilibrium error test, W = work error test, V = vector component method, N = Length method) and the error tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All requested criteria (combination of U, P, W, V, and/or N) are satisfied upon convergence. Note that at least two iterations are necessary to check the displacement convergence criterion. For SOL 400, if the U criterion is selected together with P or W, then for the first iteration of a load increment, the U criterion will not be checked. For V and N, see Remark 21.
8. MAXDIV provides control over diverging solutions. Depending on the rate of divergence, the number of diverging solutions (NDIV) is incremented by 1 or 2. The solution is assumed to diverge when NDIV reaches MAXDIV during the iteration. If the bisection option is used (allowed MAXBIS times) the time step is bisected upon divergence. Otherwise, the solution for the time step is repeated with a new stiffness based on the converged state at the beginning of the time step. If NDIV reaches MAXDIV again within the same time step, the analysis is terminated for SOLs 129 and 159. For SOL 400, the termination of analysis is dependent on the sign of MAXDIV. If MAXDIV is positive, the best attainable solution is computed and the analysis is continued to the next time step. Best solutions for 4 time steps are computed. The analysis is terminated if the solution is still diverges. If MAXDIV is negative, the analysis is terminated immediately. See Remark 6.
9. Nonzero values of MAXQN and MAXLS will activate the quasi-Newton update and the line search process, respectively.
10. The number of subincrements in the material routines is determined such that the subincrement size is approximately $FSTRESS \cdot \bar{\sigma}$. FSTRESS is also used to establish a tolerance for error correction in elastoplastic material, i.e., error in yield function $< FSTRESS \cdot \text{yield stress}$
- If the limit is exceeded at the converging state, the program will terminate with a fatal error message. Otherwise, the stress state is adjusted to the current yield surface, resulting in $\delta = 0$.
11. The remark applies to SOLs 129 or 159 only. The bisection process is activated when divergence occurs and $MAXBIS \neq 0$. The number of bisections for a time increment is limited to $|MAXBIS|$. If MAXBIS is positive and the solution does not converge after MAXBIS bisections, the best solution is computed and the analysis is continued to the next time step. If MAXBIS is negative and the solution does not converge in $|MAXBIS|$ bisection, the analysis is terminated.

12. This remark applies to SOL 400 only. The bisection process is activated when divergence occurs and $\text{MAXBIS} \neq 0$. The number of bisections for a time step is limited to the absolute value of MAXBIS . Different actions are taken when the solution diverges depending on the sign of MAXBIS . If MAXBIS is positive, the stiffness is updated on the first divergence, and the load is bisected on the second divergence. If MAXBIS is negative, the load is bisected every time the solution diverges until the limit on bisection is reached. If the solution does not converge after $|\text{MAXBIS}|$ bisections, the analysis is continued or terminated depended on the sign of MAXDIV . See Remark 8.
13. ADJUST controls the automatic time stepping. Since the automatic time step adjustment is based on the mode of response and not on the loading pattern, it may be necessary to limit the adjustable step size when the period of the forcing function is much shorter than the period of dominant response frequency of the structure. It is the user's responsibility to ensure that the loading history is properly traced with the ADJUST option. The ADJUST option should be suppressed for the duration of short pulse loading. In particular, for impact problems in SOL 400, it is recommended that the ADJUST option be suppressed since the short duration pulses during the impact may not be tracked well by the frequency-based algorithm. It should also be noted that the TZEROMAX process (where the step is repeated with the same/smaller time step after the first 2 increments) is not currently available for contact problems. If unsure, start with a value for DT that is much smaller than the pulse duration in order to properly represent the loading pattern.
 - If $\text{ADJUST} = 0$, then the automatic adjustment is deactivated. This is recommended when the loading consists of short duration pulses.
 - If $\text{ADJUST} > 0$, the time increment is continually adjusted for the first few steps until a good value of Δt is obtained. After this initial adjustment, the time increment is adjusted every ADJUST -th time step only.
 - For SOL 400, if $\text{ADJUST} > 0$ and $\text{NO} > 0$ (see Remark 3.), the analysis time step can reduce without limit, but it cannot increase more than $\text{DT} * \text{NO}$. This means that, if $\text{NO} = 1$, the analysis time step can only reduce, but cannot increase. The user should use $\text{NO} > 1$ to take the advantage of the range of automatic time stepping. If $\text{NO} < 0$, the time step can both increase and decrease without limit. But in this case, the user will not know the exact output locations.
 - If ADJUST is one order greater than NDT , then automatic adjustment is deactivated after the initial adjustment.
14. MSTEP and RB are used to adjust the time increment during analysis. The recommended value of MSTEP is 10 to 40. A larger value (e.g., 40) is required for highly nonlinear problems. By default, the program automatically computes the value of MSTEP based on the changes in the stiffness.

The time increment adjustment is based on the number of time steps desired to capture the dominant frequency response accurately. The time increment is adjusted as follows:

$$\Delta t_{n+1} = f(r)\Delta t_n$$

where

$$r = \frac{1}{\text{MSTEP}} \left(\frac{2\pi}{\omega_n} \right) \left(\frac{1}{\Delta t_n} \right)$$

where:

$f = 0.25$	for $r < 0.5 \cdot RB$
$f = 0.5$	for $0.5 \cdot RB \leq r < RB$
$f = 1.0$	for $RB \leq r < 2.0$
$f = 2.0$	for $2.0 \leq r < 3.0 / RB$
$f = 4.0$	for $r \geq 3.0 / RB$

15. MAXR is used to define the upper and lower bounds for adjusted time step size, i.e.,

$$\text{MIN}\left(\frac{DT}{2^{\text{MAXBIS}}}, \frac{DT}{\text{MAXR}}\right) \leq \Delta t \leq \text{MIN}(\text{MAXR} \cdot DT, NO \cdot DT)$$

16. UTOL is a tolerance used to filter undesirable time step adjustments; i.e.,

$$\frac{\|\dot{U}_n\|}{\|\dot{U}\|_{max}} < \text{UTOL}$$

Under this condition no time step adjustment is performed in a structural analysis (SOL 129). In a heat transfer analysis (SOL 159) the time step is doubled.

17. The bisection is activated if the incremental rotation for any degree-of-freedom ($\Delta\theta_x, \Delta\theta_y, \Delta\theta_z$) exceeds the value specified by RTOLB. This bisection strategy is based on the incremental rotation and controlled by MAXBIS.
18. For non-ADAPT (except FNT and PFNT) methods, the stiffness will be updated at convergence if the number of iterations since last iteration is equal or greater than KSTEP. In addition, for SOL 400 and ADAPT method, the stiffness will be updated if 3^*MAXITER iterations are performed. For SOL 400 and non-ADAPT methods, the stiffness will be updated if MAXITER iterations are performed.
19. For FNT and PFNT methods, whether the stiffness matrix will be updated between the convergence of a load increment and the start of the next load increment depends on the value of KSTEP. In this case, KSTEP = -1, 'BLANK', or 1. A user fatal error will be issued if other value is input. If KSTEP = 1, the stiffness matrix will not be updated. If KSTEP = 'BLANK', the program will decide whether to update depending element type. If KSTEP = -1, the stiffness matrix will be forced to be updated.
20. If EPSU > 0.0, the displacement error is computed file with respect to the total displacements. If EPSU < 0.0, the displacement error is computed with respect to the delta displacements of a load increment. If EPSW > 0.0, the energy error is computed with respect to the total energy. If EPSW < 0.0, the energy error is computed respect to the delta energy of a load increment. The options EPSU < 0.0 and EPSW < 0.0 are available for SOL 400 only.
21. V and N are additional methods for convergence checking using the displacement (U) and/or load (P) criteria. V stands for vector component checking. In this method, convergence checking is performed on the maximum vector component of all components in the model. N stands for length checking. In this method, the length of a vector at a grid point is first computed by the SRSS (square root of the sum of the squares) method. Then convergence checking is performed on the maximum length of all grid points in the model. For example, if CONV=UV, then V checking method will be performed with the U criteria, i.e., the maximum displacement component of all displacement components in the model is used for convergence checking. For V and N, the EPSU is always

negative, i.e., the error is computed with respect to the delta displacements of a load increment, even if positive value is requested by users. CONV=V is the same as CONV=UPV and CONV=D is the same as CONV=UPD. If both V and N are specified; V will take precedence over N. For example, CONV=VN is the same as CONV=V.

By default, for UPV or UPN, separate checks are made over force and moment vectors, and translation and rotation vectors. While the force/translation check is valid always, the moment or rotation check is only valid for 6 DOF elements (beams, shells, etc.). In certain cases (i.e., simply supported or hinged structures where moments are numerically small, small rotation problems), it may be beneficial to turn off the additional convergence testing done for moments and/or rotations.

22. For SOL 700, only fields ID, NDT, and DTare used.
23. In NLTRAN analysis, the output is also influenced by NLPACK. Please refer to NLPACK for an example.

TTEMP

Temperature Distribution of Transient Response

Defines a time-dependent temperature distribution for the nonlinear transient analysis in SOL 400.

$$\{T(t)\} = \{A(T(x)) \cdot F(t)\}$$

where $A(T(x))$ defines a spatial temperature distribution and $F(t)$ a time function. $T(t)$ is the time dependent temperature distribution for use in the nonlinear elements in nonlinear transient analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TTEMP	SID	GROUP_ID	TID						

Example:

TTEMP	11	101	31						
-------	----	-----	----	--	--	--	--	--	--

Descriptor	Meaning
SID	Temperature set identification number. (Integer > 0)
GROUP_ID	Temperature group identification number (Integer > 0 or ==1)
TID	Identification number of TABLEDi entry that gives $F(t)$. (Integer > 0)

Remarks:

1. The temperature distribution TTEMP must be selected by a Case Control command TEMP(LOAD)=SID in order to be used in the nonlinear transient analysis.
2. This command is used in SOL 400 only when ANALYSIS=NLTTRAN (nonlinear transient analysis) and the temperature load is applied. It only applies to the nonlinear elements in the Residual (SEID=0). There should be only one temperature set for each STEP.
3. GROUP_ID determines the distribution of temperatures. It references the TMPSET Bulk Data entry to define all grid points, which reference the same TABLEDi entry. Each grid point can have its own GROUP_ID if necessary. GROUP_ID==1 means all grid points are in one group and reference the same TTEMP Bulk Data entry.
4. If the TEMP(INIT) Case Control command references a TTEMP entry, then only the spatial distribution of the TTEMP will be used as the initial temperature distribution for the TEMP(INIT) command.
5. TTEMP may be used in nonlinear transient analysis in SOL 400. For pure linear analysis in SOL 400, DLOAD/TLOADi has to be used to define a time-dependent temperature distribution.

UDNAME**User Defined File Name**

Provides the name of a file that can be referenced from other bulk data entries such as FTGLOAD, TABLRPC, and MATDIGI.

Format:

1	2	3	4	5	6	7	8	9	10
UDNAME	UDID								
	NAME								

Example:

UDNAME	10								
	myfile.dat								

Descriptor	Meaning
UDID	Unique UDID (Integer>0). See Remark 1.
NAME	Name of a file (with or without path) such as the external loading time history in DAC, RPC, or S3T format or DIGIMAT material file (Character).

Remarks:

1. The UDID is referenced by FTGLOAD, TABLRPC, or MATDIGI entries
2. The NAME is limited to a maximum of 256 characters (corresponding to 4 lines of data in fields 2-9). For Fatigue analysis using .rsp (RPC) files, there is a 128 character limit (corresponding to 2 lines of data in fields 2-9). Embedded blanks are not supported.
3. If only a NAME with no path (e.g., *sine01.dac*) is supplied, the file is assumed to be located in the same directory as the Nastran input file. If an absolute or relative path is supplied (e.g., */local/user/fatigue/sine01.dac*), it will be used.

UDSESV

Define the number and names of user state variables for material user subroutines
(SOL 400 only)

Define the number and names of the user state variables. Each state variable has default nominal name if a user name is not given. This is a global entry.

Format:

1	2	3	4	5	6	7	8	9	10
UDSESV		NSTATS							
	SV2	SV2_NAME	SV3	SV3_NAME	SV4	SV4_NAME	SV5	SV5_NAME	
	SV6	SV6_NAME	.ETC						

Example:

UDSESV		3							
	SV2	VAR2	SV3	VAR3					

Descriptor	Meaning
NSTATS	The number of user defined state variables. (Integer ≥ 1)
SVi	The default nominal name of state variable (CHARACTER, $i \geq 1$, where i is the index number of the state variable)
SVi_NAME	The state variable name defined by user (CHARACTER, Default = SVi, where i is the index number of the state variable)

Remarks:

1. This is a global entry that defines user state variables for material user subroutines. The temperature will always be passed to material use subroutine as the first state variable; its name should not be redefined in this entry.
2. If a state variable is not given a name, SVi will be used as its name. The number i is the index number of the state variable.
3. For output, either state variables names given in UDSESV or default SVi names can be used in NLOUT entry in case control. The state variables names will be used as keywords for output selection.
4. The 1st state variable is always temperature. The remaining user defined state variables are defined and used only by user, Nastran will not use them.
5. This entry only supports UMAT and UCOHES user subroutines, see UNAME in MATUDS
6. This entry is required when MATDIGI is defined to define the number of state variables.
7. The number of state variables is limited to 100 state variables (including temperature) for SOL400.

3296

UNBALNC

Specifies an Unbalanced Load for Rotordynamic Transient or Frequency Response Analysis

UNBALNC

Specifies an Unbalanced Load for Rotordynamic Transient or Frequency Response Analysis

Used in rotordynamic analyses to specify a rotating unbalance load in terms of a cylindrical system with the rotor rotation as the z-axis.

Format:

1	2	3	4	5	6	7	8	9	10
UNBALNC	SID	MASS	GRID	X1	X2	X3			
	ROFFSET	THETA	ZOFFSET	T/F _{on}	T/F _{OFF}	CFLAG			
	UFT1	UFT2	UFT3	UFR1	UFR2	UFR3			
	MCT1	MCT2	MCT3	MCR1	MCR2	MCR3			
	SCR1	SCR2	SCR3						

Example:

UNBALNC	100	0.1	1001	0.0	1.0	0.0			
	0.02	30.0	0.5			MASS			
	1001	1002	1003	1004	1005	1006			
	2001	2002	2003	2004	2005	2006			
	3001	3003	3004						

Descriptor	Meaning
SID	Set identification number. (Integer; Required; no Default)
MASS	Mass of imbalance. See Remark 4. (Real or Integer; if integer must be > 0; Required; no Default)
GRID	Grid identification number of applying imbalance. (Integer; Required; no Default)
X1, X2, X3	Components of the vector, from GRID, in the displacement coordinate system of GRID, which is used to define a cylindrical coordinate system centered at GRID; see Remark 5. (Real; Required; no Default)
ROFFSET	Offset of mass in the radial direction of the unbalance coordinate system. See Remark 4. (Real or Integer; if integer, must be > 0; Default = 1.0)
THETA	Angular position, in degrees, of the mass in the unbalance coordinate system. (Real; Default = 0.0)
ZOFFSET	Offset of mass in the z-direction of the unbalance coordinate system. See Remark 4. (Real or Integer; if integer, must be > 0; Default = 0.0)
T/F _{ON}	Start time or frequency for applying imbalance load. See Remark 6. (Real ≥ 0.0 ; Default = 0.0)
T/F _{OFF}	Stop time or frequency for terminating imbalance load. See Remark 6. (Real > 0.0 ; Default = 999999.0)

Descriptor	Meaning
UFT1-3	EPOINTS to output the unbalanced forces in the T1, T2, and T3 directions. See Remark 6. (Integer > 0)
UFR1-3	EPOINTS to output the unbalanced forces in the R1, R2, and R3 directions. See Remark 6. (Integer > 0)
MCT1-3	EPOINTS to output the mass-correction forces in the T1, T2, and T3 directions. See Remark 6. (Integer > 0)
MCR1-3	EPOINTS to output the mass-correction forces in the R1, T2, and R3 directions. See Remark 6. (Integer > 0)
SCR1-3	EPOINTS to output the speed-correction forces in the R1, R2, and R3 directions. (Integer > 0)
CFLAG	Correct flag to specify whether 1) the mass will be used to modify the total mass in the transient response calculations, 2) the effect of the rotor spin rate change will be included in the transient response calculation, or 3) both; see Remark 7. (Character: NONE, MASS, SPEED, or BOTH, Default = NONE).

Remarks:

1. Multiple UNBALNC entries with the same SID value are allowed.
2. For transient response, UNBALNC is specified by the RGYRO Case Control command. For frequency response, it is specified by the DLOAD Case Control command.
3. The imbalance load will be generated based on the mass value, offset values, and the rotor spin speed.
4. If the entry is a real number, the value is considered constant. If the entry is an integer number, the value references a TABLEDi entry that specifies the value as a function of time for transient response or frequency for frequency response.
5. A cylindrical coordinate system is used to determine the initial position and rotation direction of the mass unbalance. Theta is measured from the plane defined by the z-axis and the user specified vector (X1, X2, X3). Theta = 0.0 is in the direction of the user-specified vector (X1, X2, X3). Unbalance rotation is in the positive z-direction.
6. For frequency response analysis, the EPOINTS and the continuation cards are ignored.
7. If the mass loss (increase) is relatively small, the correction may safely be ignored. (CFLAG = NONE). These error corrections terms are applied for analysis in fixed reference frame only.
8. The mass specified on UNBALNC is scaled with PARAM, WTMASS factor for both transient and frequency response analysis.

Equations in fixed reference frame:

- a. For Frequency response analysis

$$\begin{Bmatrix} F_x(\omega) \\ F_y(\omega) \end{Bmatrix} = WTMASS \cdot \begin{Bmatrix} mr\omega^2(\cos\theta_k + i\sin\theta_k) \\ mr\omega^2(\sin\theta_k + i\cos\theta_k) \end{Bmatrix}$$

b. For transient analysis

$$\begin{Bmatrix} F_x(\omega) \\ F_y(\omega) \end{Bmatrix} = WTMASS \cdot \begin{Bmatrix} m(t)r(t)(\Omega_j^2(t)\cos\theta_{jk}(t) + \dot{\Omega}_j(t)\sin\theta_{jk}(t)) \\ m(t)r(t)(\Omega_j^2(t)\sin\theta_{jk}(t) - \dot{\Omega}_j(t)\cos\theta_{jk}(t)) \end{Bmatrix}$$

where,

- $m(t)$ Specified by the user (MASS field in UNBALNC)
- $r(t)$ Specified by the user (ROFFSET field in UNBALNC)
- $\Omega_j(t)$ Specified by the user through RSINT/RSPINR entry
- $\dot{\Omega}_j(t)$ Derived from $\Omega_j(t)$
- $\theta_{jk}(t)$ Instantaneous angular location, measured from axis 1 to axis 2 ($= \int_0^t \omega_j(t)dt + \theta_k$)
- θ_k Specified by the user (THETA field in UNBALNC)
- j Rotor on which the grid is located

UNGLUE

Defines Grids that Should Be Eliminated From Glued Contact

This entry may be used in SOLs 101 or 400 but is only necessary if glued contact has been specified and some of the grids should use standard contact instead of glued contact. This option is normally used for crack analysis where the grids along the crack are not glued but all other grids on a contact body have glued contact.

Format:

1	2	3	4	5	6	7	8	9	10
UNGLUE	ID	BID	ID1	THRU	ID2	BY	N		
	ID3	THRU	ID4	ID5	ID6	-etc.-			

Example:

UNGLUE	10	101	20	THRU	300	BY	2		
		3457	8456	4712	1000	THRU	2000		

Descriptor	Meaning
ID	Identification number referenced by a SUBCASE or STEP Case Control BCONTACT or UNGLUE command. See Remark 1. (Integer; no Default)
BID	Identification of the specified BCBODY. (Integer > 0; no Default).
IDi	ID list of grids. (Integer > 0; no Default).
THRU	Enter THRU if a range of grids is required. (Character)
BY	Enter BY if the range of grids is not consecutive. (Character)
N	BY increment. (Integer > 0).

Remarks:

1. To place an entry in loadcase 0, set ID=0, which does not need any corresponding Case Control command BCONTACT=0 or UNGLUE=0 and is always automatically executed. To place an entry in any physical loadcase (SUBCASE or STEP), ID must be selected by the Case Control command BCONTACT=ID or UNGLUE=ID. Note that UNGLUE Case Control will take precedence over the BCONTACT Case Control for this entry with same ID. UNGLUE is ignored by Permanent contact.
2. This entry may be repeated as many times as necessary to define all subcases and bodies with grids that should not have glued contact.

UNGLUE Defines Grids that Should Be Eliminated From Glued Contact in SOL 600

This entry is only necessary if glued contact has been specified and some of the grids should use standard contact instead of glued contact. This option is normally used for crack analysis where the grids along the crack are not glued but all other grids on a contact body have glued contact. SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
UNGLUE	IDBC	IBOD	D1	THRU	D2	BY	N3		
			D4	THRU	D5	BY	N6		

Example:

UNGLUE	1	1	1	THRU	100				
--------	---	---	---	------	-----	--	--	--	--

Descriptor	Meaning
IDBC	Identification number of a Case Control BC CONTACT command defining the subcase to which these items should be available. Enter 0 if this entry should apply to Marc's increment zero. Enter -9999 if it should apply to all subcases. (Integer; no Default)
IBOD	Identification number of a matching BCBODY Bulk Data entry of a flexible surface defining the body with GRIDS to be removed from glued contact. The BCBODY may include additional grids (not included in this entry) that remain glued. (Integer; Default = 1) See Remarks 4. and 5. (3,1)
IDi	ID list of grids. (Integer > 0; no Default).
THRU	Enter THRU if a range of grids is required. (Character)
BY	Enter BY if the range of grids is not consecutive. (Character)
N	BY increment. (Integer > 0).

Remarks:

1. Repeat the continuation entry until all grids for the particular body are described.
2. This entry may be repeated as many times as necessary to define all subcases and bodies with grids that should not have glued contact.
3. This entry corresponds to Marc's DEACT GLUE option. Items (i,j) indicate the corresponding data block and field.
4. In certain models, there are no BCBODY entries (for example; self contact) because the entire model comprises one body. For such models IBOD may be left blank.

5. If IBOD is a positive number, it will be converted to the body number using the BCBODY entries. To override this conversion and use the body number directly, enter IBOD as a negative number whose magnitude is equal to the body number desired.
6. Multiple UNGLUE entries with the same IDBC are not allowed.

USET**Degree-of-Freedom Set Definition**

Defines a degree-of-freedom set.

Format:

1	2	3	4	5	6	7	8	9	10
USET	SNAME	ID1	C1	ID2	C2	ID3	C3		

Example:

USET	U4	333	26	17	0				
------	----	-----	----	----	---	--	--	--	--

Descriptor	Meaning
SNAME	Set name. (One to four characters, or the string “ZERO” followed by the set name.)
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets, 1111](#) or their new names on the DEFUSET entry. However, it is recommended that SNAME refer only to the set names U1 through U6 or their new names on the DEFUSET entry. If set names a through v are used then the degrees-of-freedom may also have to be defined in the applicable super sets on other USETi entries.
2. If SNAME = “ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.
3. A maximum of 18 degrees-of-freedom may be designated on a single entry.
4. If degrees-of-freedom defined by USET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the USET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.
5. The USET entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if USET entries are used.
6. If a USETi Bulk Data entry lists a standard degree-of-freedom set, such as S or M, the program may fail in the PARTN module with the message “SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE”. This entry should only reference new sets defined on DEFUSET Bulk Data entries.

USET1

Degree-of-Freedom Set Definition, Alternate Form

Defines a degrees-of-freedom set.

Format:

1	2	3	4	5	6	7	8	9	10
USET1	SNAME	C	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	ID8	-etc.-						

Example:

USET1	SB	345	2	1	36	5	9	7	
	40								

Alternate Format and Example:

USET1	SNAME	C	ID1	“THRU”	ID2				
USET1	SB	123	170	THRU	180				

Descriptor	Meaning
SNAME	Set name. (One to four characters or the word “ZERO” followed by the set name.)
C	Component numbers. (Integer zero or blank for scalar points or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification number. (Integer > 0; for “THRU” option, ID1 < ID2.)

Remarks:

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets, 1111](#) or their new names on the DEFUSET entry. However, it is recommended that SNAME refer only to the set names U1 through U6 or their new names on the DEFUSET entry. If set names a through v are used then the degrees-of-freedom may also have to be defined in the applicable super sets on other USETi entries.
2. If SNAME=“ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.
3. If the alternate format is used, all of the points ID1 through ID2 are assigned to the set.
4. If degrees-of-freedom defined by USET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the USET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.
5. The USET1 entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if USET entries are used.

6. If a USETi Bulk Data entry lists a standard degree-of-freedom set, such as S or M, the program may fail in the PARTN module with the message “SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE”. This entry should only reference new sets defined on DEFUSET Bulk Data entries.

USRSUB6

Defines User Subroutines for SOL 600

Defines user subroutines used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
USRSUB6	U1	U2	U3	U4	U5	U6	U7	U8	
	U9	U10							

Examples:

USRSUB6	UDAMAG	uvoid	TENSOF						
USRSUB6*	SEPFORBB C								

Descriptor	Meaning
Ui	Name of user subroutine(s) to be included. See Marc Volume D for list of available user subroutines. Do not include the .f extension on this entry, however, the actual file on the disk must have the .f extension. If any user subroutine exceeds 8 characters, use the wide field format for the primary line and all continuation lines. (Character; no Default)

Notes:

1. All user subroutines must reside in the directory where the Nastran input file resides.
2. All user subroutines on disk must be in lower case and have an extension of .f. The names entered in the bulk data entry may be in upper or lower case. They will be converted to lower case.
3. SOL 600 combines all user subroutines into one large subroutine named u600.f and u600.f is passed to the Marc command line when spawned from Nastran.
4. If only one user subroutine is required, an alternate is to use PARAM,MARCUSUB,name.

UXVEC**Control Parameter State**

Specification of a vector of aerodynamic control point (extra point) values. These data define the control positions corresponding to user defined nonlinear control forces that have been defined by AEDW, AEPRESS and AEFORCE entries. Only nonzero values need to be defined.

Format:

1	2	3	4	5	6	7	8	9	10
UXVEC	ID								
	LABEL1	UX1	LABEL2	UX2	-etc.-				

Example:

UXVEC	1001								
	THRUST	1.E4	ANGLEA	.015					

Descriptor	Meaning
ID	Control vector identification number, see Remark 1. (Integer > 0)
LABELi	Controller name. This must correspond to an existing AESURF, AESTAT or AEPARM label or INTERCPT (Character). See Remark 4.
UXi	The magnitude of the aerodynamic extra point degree-of-freedom (Real)

Remarks:

1. The ID is referenced by the AEUXREF Case Control command and/or by AEDW, AEPRESS, and/or AEFORCE entries.
2. The units of the user defined AEPARM controllers are implied by their use on this entry and the corresponding values on the force vector definition. The user must be self-consistent in all uses of the same controller. AESURF controllers are expressed in radians as are the rigid body angles ANGLEA and BETA. The rigid body rates, ROLL, PITCH and YAW are nondimensional rates $pb/2V$, $qc/2V$, $rb/2V$; respectively. V is the velocity and b and c are the reference span and chord lengths, respectively.
3. LABELs that are part of the UX vector that are omitted in the UXVEC specification are assigned a value of 0.0.
4. INTERCPT = 1.0 is implied on all UXVEC input.
INTERCPT = 0.0 indicates that associated load is a perturbation.

VCCT

Virtual Crack Closure Technique - SOL 400

Format:

1	2	3	4	5	6	7	8	9	10
VCCT	ID	IDCR	ITYPE	IGROW	INCM	METHOD			
	CGI	GC					GC-II	GC-III	
	TABCGI	TABGC					TABGC-II	TABGC-III	
	G1	G2	G3	G4	G5	etc.			

Example:

VCCT	101	1	2	2	2	1		1	
		2000.							
		0							
	51	52							

Descriptor	Meaning
ID	Identification of a matching Case Control VCCT entry. See Remark 2. (Integer; no Default)
IDCR	Identification of this particular crack. IDCR must be unique among all VCCT cracks with the same ID but can replace a crack with the same IDCR. (Integer > 0; Default = 1)
ITYPE	Type of crack propagation. (Integer; Default = 0) <ul style="list-style-type: none"> 0 No crack propagation 2 Direct crack propagation
IGROW	Specifies how the crack grows. (Integer; Default = 2) <ul style="list-style-type: none"> 1 Uses remeshing (not presently available) 2 Release user tyings or glued contact
INCM	Specifies the crack growth increment (Integer; Default = 1) <ul style="list-style-type: none"> 1 Uses a fixed increment or a user subroutine
METHOD	Specifies the method used for the estimated crack growth direction (Integer; Default=1) <ul style="list-style-type: none"> 1 Maximum hoop stress criterion (Default)
CGI	Crack growth increment (Real; Default = 0.0). If the option of releasing tyings or glued contact is used, the length of the released element edge is used. Leave blank for fatigue growth defined by the Paris law. (Not presently used)
GC	Crack growth resistance. (Real; Default = 0.0) Ignored for fatigue growth. If different crack growth resistance values are needed from modes I, II, III, this is the mode I value and modes II and III are entered in fields 8 and 9. For SOL 600, if it is desired that the mode I crack resistance be zero, enter GC as a negative value.

Descriptor	Meaning
GC-II	Crack growth resistance, Mode II (Real; Default = 0.0) Ignored for fatigue growth.
GC-III	Crack growth resistance, Mode III (Real; Default = 0.0) Ignored for fatigue growth.
TABC _{GI}	TABLEM _i or TABL3D _i for CGI (crack growth increment). (Integer; Default = 0)
TABG _C	TABLEM _i or TABL3D _i for scaling GC (crack growth resistance). (Integer; Default = 0)
TABG _{C-II}	TABLEM _i or TABL3D _i for scaling GC-II (Integer; Default = 0)
TABG _{C-III}	TABLEM _i or TABL3D _i for scaling GC-III (Integer; Default = 0)
G _i	Grids along the crack front – for a shell there is only one node. See Remark 6. (Integer or “THRU” or “BY”; no Default; at least one value, G ₁ , must be provided.)

Remarks:

1. The grids entered on this entry MUST be associated with SOL400 elements that have had their capabilities extended by use of either a PSNLN1, PSNLN2, PSLDN1, PLCOMP or PCOMPLS or a combination of these entries.
2. ID corresponds to a Case Control VCCT entry.
3. If tables are not required, enter at least one field with a zero value. Do not enter a blank line.
4. This entry may be repeated as many times as necessary to describe all the cracks in the model.
5. The 4th line may be repeated as many times as necessary to describe all grids on the crack front
6. If G₁ is negative, the absolute value of G₁ is used as the ID of a SET3 entry providing the list of grids. For this case, G₂, G₃, etc. must be blank (only one SET3 ID per VCCT entry is allowed).
7. The values of the entries on the second line that are not needed should be set to 0.0 or blank. If no tables are required to specify the variation with time, temperature or some other variable, one or all of the table entries on the third line can be set to zero or blank.
8. If TABLEM1 is used, accumulated crack growth will be used for the X coordinates instead of the usual value of temperature.

VCCT**Virtual Crack Closure Technique in SOL 600**

Provides data to simulate crack growth using virtual crack closure technology methods in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
VCCT	ID	IDCR	ITYPE	IGROW	INCM	METHOD	TIME	IACT	
	CGI	GC	GTH	C	M	GMIN	GC-II	GC-III	
	TABCGI	TABGC	TABGTH	TABC	TABM	TABGMIN	TABGC-II	TABGC-III	
	G1	G2	G3	G4	G5	etc.			

Alternate Format:

VCCT	ID	IDCR	ITYPE	IGROW	INCM	METHOD	TIME	IACT	
	CGI	GC	GTH	C	M	GMIN	GC-II	GC-III	
	TABCGI	TABGC	TABGTH	TABC	TABM	TABGMIN	TABGC-II	TABGC-III	
	G1	THRU	G2	BY	G3				

Example:

VCCT	101	1	2	2	2	1		1	
		2000.	12.	4.	2.0				
		0	0	0	0				
	51	52							

Descriptor	Meaning
ID	Identification of a matching Case Control VCCT entry. See Remark 2. (Integer; no Default)
IDCR	Identification of this particular crack. IDCR must be unique among all VCCT cracks with the same ID but can replace a crack with the same IDCR and a different ID using the IACT field. (Integer > 0; Default = 1)
ITYPE (6-1)	Type of crack propagation. (Integer; Default = 2)
	0 No crack propagation
	1 Fatigue type crack propagation
	2 Direct crack propagation
IGROW (6-2)	How crack grows (Integer; Default = 2)
	1 Uses remeshing (not presently available)

Descriptor	Meaning
	2 Release user tyings or glued contact
INCM (6-3)	Specifies the crack growth increment (Integer; Default = 2)
	1 Uses a fixed increment or a user subroutine
	2 Uses Paris law
METHOD (6-4)	Specifies the method used for the estimated crack growth direction (Integer; Default=1)
	1 Maximum hoop stress criterion (Default)
TIME (6-5)	Time period for fatigue load sequence. (Real; no Default) Only enter if ITYPE=1
IACT (3-1)	Flag for activating or deactivating an existing crack (Integer; Default = 0)
	0 Leave as is
	1 Activate
	2 Deactivate
CGI (7-1)	Crack growth increment (Real; Default = 0.0). If the option of releasing tyings or glued contact is used, the length of the released element edge is used. Leave blank for fatigue growth defined by the Paris law.
GC (7-2)	Crack growth resistance. (Real; Default = 0.0) Ignored for fatigue growth. If different crack growth resistance values are needed from modes I, II, III, this is the mode I value and modes II and III are entered in fields 8 and 9.
GTH (7-3)	Paris law energy release rate threshold. (Real; Default = 0.0)
C (7-4)	Paris law parameter C. (Real; Default = 0.0) Only enter if INCM=2
M (7-5)	Paris law parameter m. (Real; Default = 0.0) Only enter if INCM=2
GMIN (7-6)	Minimum growth increment. (Real; Default = 0.0) Only enter if INCM=2
GC-II (7-4)	Crack growth resistance, Mode II (Real; Default = 0.0) Ignored for fatigue growth.
GC-III	Crack growth resistance, Mode III (Real; Default = 0.0) Ignored for fatigue growth.
TABCGI (8-1)	TABLEMi or TABL3Di for CGI (crack growth increment). (Integer; Default = 0)
TABGC (8-2)	TABLEMi or TABL3Di for scaling CG (fracture toughness). (Integer; Default = 0)

Descriptor	Meaning
TABGTH (8-3)	TABLEMi or TABL3Di for scaling GTH (Paris law energy release rate) (Integer; Default = 0)
TABC (8-4)	TABLEMi or TABL3Di for scaling C (Paris law parameter C). (Integer; Default = 0)
TABM (8-5)	TABLEMi or TABL3Di for scaling M (Paris law parameter m). (Integer; Default = 0)
TABGMIN (8-6)	TABLEMi or TABL3Di for scaling GMIN (Minimum growth increment). (Integer; Default = 0)
TAGC-II	TABLEMi or TABL3Di for scaling GC-II (Integer; Default = 0)
TAGC-III	TABLEMi or TABL3Di for scaling GC-III (Integer; Default = 0)
Gi (5-i)	Grids along the crack front – for a shell there is only one node. (Integer; no Default; at least one value, G1, must be provided. See Remark 6.)

Remarks:

1. This entry corresponds to Marc's VCCT model definition and history definition options.
2. (i,j) corresponds to Marc Vol C VCCT entry ith datablock jth field
3. ID corresponds to a Case Control VCCT command. Set ID=0 to enter VCCT entries into Marc's model definition.
4. If tables are not required, enter at least one field with a zero value. Do not enter a blank line.
5. This entry may be repeated as many times as necessary to describe all the cracks in the model.
6. The 4th line may be repeated as many times as necessary to describe all grids on the crack front
7. If the Alternate Format is used, entries may only be made in the fields indicated, however this line may be repeated as many times as necessary to describe all grids on the crack front.
8. If G1 is negative, the absolute value of G1 is used as the ID of a SET3 entry providing the list of grids. For this case, G2, G3, etc. must be blank (only one SET3 ID per VCCT entry is allowed).
9. Most analyses do not use all of the entries on the second line, however a meaningful analysis will specify one or more of these values. Those not needed should be set to 0.0 or blank. If no tables are required to specify the variation with time, temperature or some other variable, one or all of the table entries on the third line can be set to zero or blank.
10. If TABLEM1 is used, accumulated crack growth will be used for the X coordinates instead of the usual value of temperature.

VIEW**View Factor Definition**

Defines radiation cavity and shadowing for radiation view factor calculations.

Format:

1	2	3	4	5	6	7	8	9	10
VIEW	IVIEW	ICAVITY	SHADE	NB	NG	DISLIN			

Example:

VIEW	1	1	BOTH	2	3	0.25			
------	---	---	------	---	---	------	--	--	--

Descriptor	Meaning
IVIEW	Identification number. (Integer > 0)
ICAVITY	Cavity identification number for grouping the radiant exchange faces of CHBDYi elements. (Integer > 0)
SHADE	Shadowing flag for the face of CHBDYi element. (Character; Default = "BOTH") NONE means the face can neither shade nor be shaded by other faces. KSHD means the face can shade other faces. KBSHD means the face can be shaded by other faces. BOTH means the face can both shade and be shaded by other faces. (Default)
NB	Subelement mesh size in the beta direction. (Integer > 0; Default = 1)
NG	Subelement mesh size in the gamma direction. (Integer > 0; Default = 1)
DISLIN	The displacement of a surface perpendicular to the surface. See Figure 9-168 . (Real; Default = 0.0)

Remarks:

1. VIEW must be referenced by CHBDYE, CHBDYG, or CHBDYP elements to be used.
2. ICAVITY references the cavity to which the face of the CHBDYi element belongs; a zero or blank value indicates this face does not participate in a cavity.
3. NB, NG, and DISLIN are used in the calculation of view factors by finite difference or contour integration techniques. They are not used with the VIEW3D entry.
4. A summary of the shadowing conditions can be requested by the PARAM,MESH,YES Bulk Data entry.
5. SHADE references shadowing for CHBDYi elements participating in a radiation cavity, the VIEW calculation can involve shadowing.

6. DISLIN should only be used with LINE type CHBDYE and CHBDYP surface elements. DISLIN > 0.0 means into the cavity. See [Figure 9-168](#).

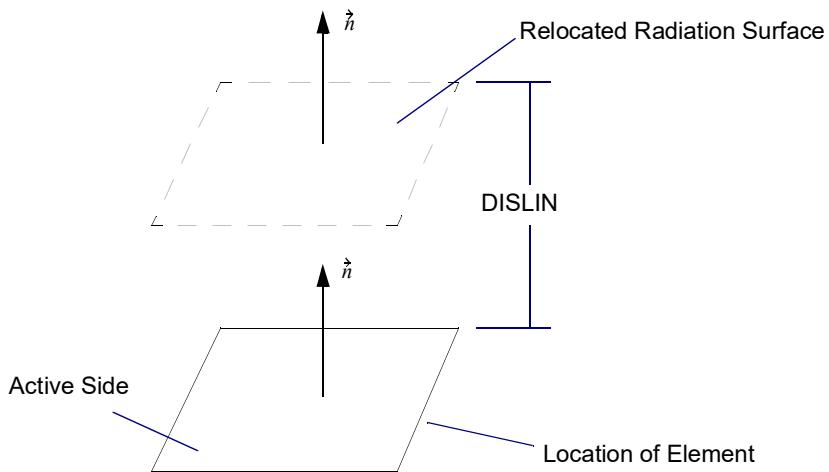


Figure 9-168 DISLIN Convention

7. NB and NG define the subelement mesh refinement when using the VIEW module (as opposed to the VIEW3D module) for the calculation of view factors.

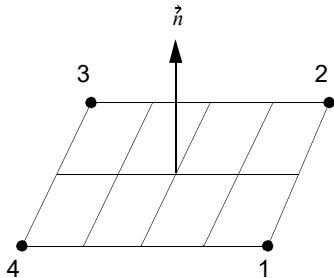


Figure 9-169 Typical AREA4 surface element where NB=2 and NG=4

8. For RC network solver in thermal analysis, the SHADE, NB, NG and DISLIN are ignored.

VIEW3D**View Factor Definition - Gaussian Integration Method**

Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.

Format:

1	2	3	4	5	6	7	8	9	10
VIEW3D	ICAVITY	GITB	GIPS	CIER	ETOL	ZTOL	WTOL	RADCHK	

Example:

VIEW3D	1	2	2	4		1.0E-6			
--------	---	---	---	---	--	--------	--	--	--

Descriptor	Meaning
ICAVITY	Radiant cavity identification number on RADCAV entry. (Integer > 0)
GITB	Gaussian integration order to be implemented in calculating net effective view factors in the presence of third-body shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default = 4)
GIPS	Gaussian integration order to be implemented in calculating net effective view factors in the presence of self-shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default = 4)
CIER	Discretization level used in the semi-analytic contour integration method. (1 ≤ Integer ≤ 20; Default = 4)
ETOL	Error estimate above which a corrected view factor is calculated using the semi-analytic contour integration method. (Real ≥ 0.0; Default = 0.1)
ZTOL	Assumed level of calculation below which the numbers are considered to be zero. (Real ≥ 0.0; Default = 1.E-10)
WTOL	Assumed degree of warpage above which the actual value of F_{ii} will be calculated. (0.0 ≤ Real ≤ 1.0; Default = 0.01)
RADCHK	Type of diagnostic output desired for the radiation exchange surfaces. (Integer; Default = 3) <ul style="list-style-type: none"> -1 No diagnostic output requested 1 Grid table and element connectivity 2 Surface Diagnostics - Surface type, area, skewness, taper, warpage, grid point sequencing, aspect ratio, and shading flags. 3 Area, view factor, area-view factor product with error estimate, existence flags for partial self-shadowing, third-body shadowing with error estimate, and enclosure summations for view factor. (Default) 0 Same as RADCHK = 1, 2, and 3 12 Same as RADCHK = 1 and 2

Descriptor	Meaning
13	Same as RADCHK = 1 and 3
23	Same as RADCHK = 2 and 3

Remarks:

1. For ETOL, when the error estimate exceeds the value input for the ETOL entry, the contour method is employed to develop an improved view factor.
2. For ZTOL, the use of a geometry scale that results in small numerical values of $A_i F_{ij}$ should be avoided.
3. When WTOL is exceeded, the actual value of F_{ii} will be calculated when using the adaptive view module. Warpage will not be considered in the calculation of F_{ij} .
4. For axisymmetric analysis, RADCHK = -1 or 3 only.

VIEWEX

Advanced RC Radiation Solver Parameters - SOL 400

Defines the radiation solver and correlating solver parameters for radiation calculations in RC heat transfer.
 Note: You must have a copy of the external radiation code to use it.

Format: (NEVADA)

1	2	3	4	5	6	7	8	9	10
VIEWEX	ICAVITY	Run Interactively	RADK Distro Method	Orbital	Re-use existing results				
	"NEVADA"	RENO Reflection	Restart	Reno Ray count	Vegas Ray count	Energy Cutoff	Confidence	GRID closure	
		GRID iterations	Time Scale	RADK cutoff					

Example:

VIEWEX	2	T	FULL	T	F				
	NEVADA	T	T	5000	5000	-3	99.	0.001	
		300	1.0	1.0-8					

Format: (TSS)

1	2	3	4	5	6	7	8	9	10
VIEWEX	ICAVITY	Run Interactively	RADK Distro Method	Orbital	Re-use existing results				
	"TSS"								

Example:

VIEWEX	3	T	FULL	T	F				
	TSS								

Format: (THERMICA)

1	2	3	4	5	6	7	8	9	10
VIEWEX	ICAVITY	Run Interactively	RADK Distro Method	Orbital	Re-use existing results				
	"THERMIC A"	Solar Flux	Planet Albedo	Planet BlackBody	Restart	Suppress VF Articulation	Radiation ray count	Orbital flux ray count	
		Confidence	Time scale	RADK cutoff					

Example:

VIEWEX	4	T	FULL	T	F				
--------	---	---	------	---	---	--	--	--	--

	THERMICA	1380.0	0.3	-19.0	T	F	5000	5000	
		99.0	1.0	1.0-8					

Format: (TRASYS)

1	2	3	4	5	6	7	8	9	10
VIEWEX	ICAVITY	Run Interactively	RADK Distro Method	Orbital	Re-use existing results				
	"TRASYS"	Axi Radial mesh	Axi Axial mesh	Axi Angular mesh	Time scale	RADK cutoff			

Example:

VIEWEX	5	T	FULL	T	F				
	TRASYS	1	1	4	1.0	1.0-8			

Format: (SRR)

1	2	3	4	5	6	7	8	9	10
VIEWEX	ICAVITY	Run Interactively	RADK Distro Method	Orbital	Re-use existing results				
	"SRR"	Gebhart Solver	Convergence Tol	Max Iter	Fij smoothing method	Fij Filter cutoff	Fij Smoothing Tol	Fij Smooth Iter	
		Bij smoothing method	Bij Filter cutoff	Bij Smoothing Tol	Bij Max Iter				

Example:

VIEWEX	6	T	FULL	T	F				
	SRR	GS	1.0-5	50	CROP	1.0-8	1.0-4	50	
		CROP	1.0-8	1.0-4	50				

Format: (SRQ)

1	2	3	4	5	6	7	8	9	10
VIEWEX	ICAVITY	Run Interactively	RADK Distro Method	Orbital	Re-use existing results				
	"SRQ"	Flux solver	Convergence Tol	Max Iter	Fij smoothing method	Filter cutoff	Fij Smoothing Tol	Fij Smooth Iter	

Example:

VIEWEX	7	T	FULL	T	F				
	SRQ	GS	1.0-5	50	CROP	1.0-8	1.0-4	50	

Descriptor	Meaning
ICAVITY	Cavity identification number (Integer > 0; Required)
Run Interactively	Run the radiation code interactively (Character; “T” or “F”, Default “F”) Do not currently have batch mode for Thermica or TSS Do not have interactive mode for TRASYS
RADK Distro Method	How to redistribute RADK onto elements. (Character; “FULL”, “AREA”, or “DIRECT”; Default “FULL”)
Orbital	Use orbital analysis for radiation (Character; “T” or “F”; Default “F”) Not supported in SindaRad
Re-use existing results	Re-use previous radiation results (Character; “T” or “F”; Default “F”)
“NEVADA”	Identification that NEVADA will be used (Character)
RENO Reflection	Use reflection method of ray tracing for RADK (Character; “T” or “F”; Default “T”)
Restart	Use Restart (Character; “T” or “F”; Default “F”)
Reno Ray count	Number of rays cast for Reno module (Integer > 0; Default 5000)
Vegas Ray count	Number of rays cast for Vegas module (Integer > 0; Default 5000)
Energy Cutoff	Energy cutoff level (Integer; Default -3)
Confidence	Confidence Level % (Real > 0.0; Default 99.0)
GRID closure	GRID closure tolerance (Real > 0.0; Default 0.001)
GRID iterations	Maximum GRID iterations (Integer > 0; Default 300)
Time Scale	Orbital time scale factor, number of time units in an hour. Ex. If using seconds, value would be 3600.0. (Real > 0.0; Default 1.0)
RADK cutoff	RADK filter smallest element (Real \geq 0.0; Default 1.0e-8)
“TSS”	Identification that TSS will be used (Character)
“THERMICA”	Identification that THERMICA will be used (Character)
Solar Flux	Quantity of solar flux (Real > 0.0; Default 1380.0 W/m ²)
Planet Albedo	Planetary Albedo (Real; Default 0.3; assumes Earth orbit)
Planet BlackBody	Planet Blackbody (Real; Default -19.0; assumes Earth orbit)
Restart	Use Restart option (Character, “T” or “F”; Default “F”)
Suppress VF Articulation	Suppress view factor articulation (Character; “T” or “F”; Default “T”)
Radiation ray count	Number of rays cast for radiation calculation (Integer > 0; Default 5000)
Orbital flux ray count	Number of rays cast for orbital flux (Integer > 0; Default 5000)
Confidence	Confidence Level % (Real > 0.0; Default 99.0)
Time Scale	Orbital time scale factor, number of time units in an hour. Ex. If using seconds, value would be 3600.0. (Real > 0.0; Default 1.0)

Descriptor	Meaning
RADK cutoff	RADK filter smallest element (Real ≥ 0.0 ; Default 1.0e-8)
“TRASYS”	Identification that TRASYS will be used (Character)
Axi Radial mesh	Axisymmetric element mesh in radial direction (Integer > 0 ; Default 1)
Axi Axial mesh	Axisymmetric element mesh in axial direction (Integer > 0 ; Default 1)
Axi Angular mesh	Axisymmetric element mesh in angular direction (Integer > 0 ; Default 4)
Time Scale	Orbital time scale factor, number of time units in an hour. Ex. If using seconds, value would be 3600.0. (Real > 0.0 ; Default 1.0)
RADK cutoff	RADK filter smallest element (Real ≥ 0.0 ; Default 1.0e-8)
“SRR”	Identification that the SindaRad RADK method will be used (Character)
Gebhart Solver	Which RADK solver to use (Character; “GS” or “FGS”; Default “GS”)
Convergence Tol	Tolerance for convergence of RADK calculation (Real ≥ 0.0 ; Default 1.0e-5)
Max Iter	Maximum allowable iterations to converge (Integer > 0 ; Default 50)
Fij smoothing method	How to filter view factors (Character; “CROP” or “HIGH”; Default “CROP”)
Fij Filter cutoff	Parameter for filter (Real > 0.0 ; Default 1.0e-8)
Fij Smoothing Tol	Tolerance for smoothing (Real ≥ 0 ; Default 1.0e-4)
Fij Smooth Iter	Maximum allowable iterations to smoothing (Integer > 0 ; Default 50)
Bij smoothing method	How to filter conductors (Character; “CROP” or “HIGH”; Default “CROP”)
Bij Filter cutoff	Parameter for filter (Real > 0.0 ; Default 1.0e-8)
Bij Smoothing Tol	Tolerance for smoothing (Real ≥ 0 ; Default 1.0e-4)
Bij Max Iter	Maximum allowable iterations to smoothing (Integer > 0 ; Default 50)
“SRQ”	Identification that the SindaRad QRad method will be used (Character)
Flux Solver	Which QRad solver to use (Character, “GS” or “CG”; Default “GS”)
Convergence Tol	Tolerance for convergence of QRad calculation (Real ≥ 0.0 ; Default 1.0e-5)
Max Iter	Maximum allowable iterations to converge (Integer > 0 ; Default 50)
Fij smoothing method	How to filter view factors (Character, “CROP” or “HIGH”; Default “CROP”)
Fij Filter cutoff	Parameter for filter (Real > 0.0 ; Default 1.0e-8)
Fij Smoothing Tol	Tolerance for smoothing (Real ≥ 0 ; Default 1.0e-4)
Fij Smooth Iter	Maximum allowable iterations to smoothing (Integer > 0 ; Default 50)

Remarks:

1. This entry is for RC Network solver only.
2. Each entry type is designed for one specific radiation solver, except the very last two types, which are for SindaRad’s two options.

NEVADA

TSS

TERMICA

TRASYS

SINDARad RADK method

SINDARad Q method

3. For more details about the parameters in the entry, please reference SINDA for *Patran User's Guide* and the *SINDARad User's Guide*.
4. MSC provides these radiation programs on Windows only.

WALL**Rigid Wall**

Defines a rigid plane through which specified Lagrangian grid points cannot penetrate. Finite or infinite size (FINITE), Orthotropic friction (ORTHO) a mass and an initial velocity (MOVING) and Force output option (FORCES) can be defined. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
WALL	ID	XP	YP	ZP	NX	NY	NZ	BCGRID	
	METHOD	FS	FK	EXP					

Example:

WALL	17						1.0	21	
	PENALTY	0.2							

Descriptor	Meaning	
ID	Unique rigid-wall number. (Integer > 0; Required)	
XP, YP, ZP	Coordinates of the origin of the wall. (Real; Default = 0.0)	
NX, NY, NZ	A vector normal to the wall pointing towards the model. (Real; Default = 0.0)	
BCGRID	Number of a BCGRID entry listing the points that cannot penetrate the wall. (Integer > 0; Required)	
METHOD	Algorithm for contact processing. (Character; Default = PENALTY)	
	PENALTY	Penalty method, allowing for extra boundary conditions, friction and output.
	KINMATIC	Kinematic method, only included for compatibility reasons with older Dytran versions. This method allows no extra boundary conditions, no friction and no output.
FS	Static coefficient of friction. See Remark 5. (Real ≥ 0 ; Default = 0.0)	
FK	Kinetic coefficient of friction. See Remark 5. (Real ≥ 0 ; Default = 0.0)	
EXP	Exponential decay coefficient. See Remark 5. (Real ≥ 0 ; Default = 0.0)	

Remarks

1. A rigid plane of infinite size is generated that the grid points cannot penetrate. The plane is fixed in space.
2. The grid points can slide on the wall and separate from it.
3. A (moving) rigid plane of finite size can be modeled by using a rigid surface and a master-slave contact.

4. For the wall definition using penalty method, output can be requested by referencing it in a SET command in the Case Control Section. The keywords for output are WALLS and WALLOUT. Please check \$S700.
5. The coefficient of friction is given by:

$$\mu = \mu_k + (\mu_s - \mu_k)e^{-\beta v}$$

where

μ_s = Static coefficient of friction FS.

μ_k = Kinetic coefficient of friction FK.

β = Exponential decay coefficient EXP.

v = Relative sliding velocity at the point of contact.

WEAR Specifies Values for Modeling Mechanical Wear in Deformable Contact Bodies

Specifies values for modeling mechanical wear in deformable contact bodies for SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
WEAR	BID	Model	IUPD						
	COEF	HARD	SEXP	VEXP	B	SFGU			
	T(COEF)	T(HARD)	T(SEXP)	T(VEXP)	T(B)				

Example:

WEAR	3	2	1						
	3.6	200.	1.1	1.2	.087	1.0			
	22	23	24	25	26				

Descriptor	Meaning
BID	BCBODY ID for which this wear entry applies. (Integer > 0; no Default; see Remark 2)
MODEL	Wear model. (Integer > 0; Default = 1, see Remark 1)
IUPD	Geometry update due to wear flag. (Integer; Default = 0) <ul style="list-style-type: none"> 0 Do not update the geometry 1 Update the geometry
COEF	Wear coefficient. (Real; no Default)
HARD	Material hardness. (Real; no Default)
SEXP	Stress exponent. (Real; Default = 1.0)
VEXP	Velocity exponent. (Real; Default = 1.0)
B	Thermal activation constant. (Real; Default = 1.0)
SFGU	Scale factor for geometry update. (Real; Default = 1.0)
T(COEF)	TABL3D ID for the wear coefficient. (Integer; Default = 0, meaning no table variation)
T(HARD)	TABL3D ID for the material hardness. (Integer; Default = 0, meaning no table variation)
T(SEXP)	TABL3D ID for the stress exponent. (Integer; Default = 0; meaning no table variation)
T(VEXP)	TABL3D ID for the velocity exponent. (Integer; Default = 0, meaning no table variation)
T(B)	TABL3D ID for the thermal activation constant. (Integer; Default = 0, meaning no table variation)

Remarks:

1. The following wear models are available

$$\dot{w} = \frac{K}{H} \sigma v_{rel} \quad \text{Archard base model}$$

$$\dot{w} = \frac{K}{H} \sigma^m v_{rel}^n \quad \text{Archard model; Bayer exponential form}$$

$$\dot{w} = \frac{K}{H} \sigma^m v_{rel}^n e^{\frac{-B}{T}} \quad \text{Archard model; exponential form with thermal activation}$$

2. BID may also point the ID in field 2 of BCPROP, BCMATL or BCBOX.

WETELME

Wetted element in OpenFSI by side in SOL 400

Defines a wetted element used in OpenFSI fluid structure interaction simulations in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
WETELME	WEID	EID	SIDE						

Example:

WETELME	10001	34	3						
---------	-------	----	---	--	--	--	--	--	--

Descriptor	Meaning
WEID	Wetted element identification number. (Integer > 0; no Default)
EID	Structural element identification number, which corresponds to a surface element CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR; or a solid element CTETRA, CPENTA, or CHEXA. (Integer > 0; no Default)
SIDE	Side identification number of element EID. ($1 \leq \text{Integer} \leq 6$; no Default)

WETELMG

Wetted element in OpenFSI by type in SOL 400

Defines a wetted element used in OpenFSI fluid structure interaction simulations in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
WETELMG	WEID	TYPE							
	G1	G2	G3	G4	G5	G6	G7	G8	

Example:

WETELMG	10001	QUAD4							
	23	35	124	28					

Descriptor	Meaning
WEID	Wetted element identification number. (Integer > 0; no Default)
TYPE	Wetted element type, which can be any of TRIA3, TRIA6, QUAD4, QUAD8, LINE2 or LINE3. (Character; no Default)
G1, ..., G8	Grid point identification numbers for the wetted surface element WEID. (Integer > 0; no Default)

WETLOAD

Load set and associated wetted surface for OpenFSI in SOL 400

Defines the load set and associated wetted surface for OpenFSI fluid structure interaction simulations in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
WETLOAD	WLID	WSID	SERV_ID						

Example:

WETLOAD	1	1001	scafsi						
---------	---	------	--------	--	--	--	--	--	--

Descriptor	Meaning
WLID	Load set ID, referenced by the EXCITEID field in the TLOAD entry. (Integer > 0; no Default)
WSID	Wetted surface identification number. The wetted surface must be defined in the WETSURF Bulk Data entry. (Integer > 0; no Default)
SERV_ID	OpenFSI SCA service name associated with the wetted surface loads. The OpenFSI SCA service is defined using the CONNECT SERVICE FMS entry. (Character; no Default)

Remarks:

1. To activate WETLOAD, the following entries must be present and referenced:
 - a. The load set ID WLID must be referenced by a TLOAD1 entry.
 - b. The TLOAD1 entry must be referenced by a DLOAD Case Control command or a DLOAD Bulk Data entry, which in turn is referenced by a DLOAD Case Control command.
 - c. The TLOAD1 entry must reference a TABLED1 entry.
 - d. The TABLED1 entry must have constant values and be the same for all TLOAD1 entries referenced by the WETLOAD entries used within the same service.

Procedures a ~ d are required in the Nastran load generator processes. Their values do not really influence the load calculation of WETLOAD in OpenFSI. Please refer to the remarks below to apply WETLOAD(s) to OpenFSI.

2. The SERV_ID string must be 8 characters or less.
3. For multiple WETLOAD entries with same SERV_ID, the wet surfaces in WSID will be merged together for the service and the WLID in the first WETLOAD entry will replace other entries WLID.
4. In the current implementation, all OpenFSI services in a model are active and participate in the calculation. The TLOAD1 and DLOAD selection in case control are not applied on OpenFSI services.
5. For OpenFSI services, the scale factors in DLOAD are always taken to be constant 1.0.

6. In the FMS Section, the following statement should be included: CONNECT SERVICE SERV_ID ExternalCodeVendor.OpenFSI

WETSURF

Wetted surface used in OpenFSI in SOL 400

Defines a wetted surface used in OpenFSI fluid structure interaction simulations in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
WETSURF	WSID	WTAG							
	WEID1	WEID2	WEID3	WEID4	WEID5	WEID6	WEID7	WEID8	
	WEID9	WEID10	-etc.-						

Alternate Format:

WETSURF	WSID	WTAG							
	WEID1	"THRU"	WEID2	"BY"	INC				

Example:

WETSURF	10001	wall1							
	5	THRU	21	BY	4				
	27	30	33						
	35	THRU	44						
	67	68	70	72	77	82	86	79	
	89	THRU	110	BY	3				

Descriptor	Meaning
WSID	Wetted surface identification number. (Integer > 0; no Default)
WTAG	Wetted surface tag name exported to an external code using the OpenFSI SCA interface. (Character; no Default)
WEID1, WEID2, ...	Wetted element identification numbers defined using the WETELMG or WETELME Bulk Data entries. (Integer > 0; no Default)
THRU, BY	Keywords to specify a range of wetted elements. (Character; no Default)
INC	Increment to use with the "THRU" and "BY" keywords. (Integer; Default = 1)

YLDHY

Hydrodynamic Yield Model

Defines a yield model with zero yield stress. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLDHY	YID								

Example:

YLDHY	200								
-------	-----	--	--	--	--	--	--	--	--

Descriptor	Meaning
YID	Unique yield-model number referenced from a MATDEUL entry. (Integer > 0; Required)

Remark:

1. This yield model should be used for fluids that have no shear strength.
2. YID must unique among all YLDxx entries in one model.

YLDJC**Johnson-Cook Yield Model**

Defines a Johnson-Cook yield model where the yield stress is a function of effective plastic strain, strain rate, and temperature. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLDJC	YID	A	B	n	C	m	EPS0	CP	
	100	200E6	50.E6	0.1	.15	.95	1.	285.	

Example:

YLDJC	TMELT	TROOM							
	1500.	273.							

Descriptor	Meaning
YID	Unique yield-model number referenced from a MATDEUL entry. (Integer > 0; Required)
A	Static yield stress. (Real ≥ 0.0 ; Required)
B	Hardening parameter. (Real; Default = 0.0)
n	Hardening exponent. (Real; Default = 1.0)
C	Strain-rate parameter. (Real; Default = 0.0)
m	Temperature exponent. (Real; Default = 1.0)
EPS0	Reference strain rate. (Real > 0.0 ; Default = 1.0)
CP	Specific heat. (Real > 0.0 ; Default = 1.E20)
TMELT	Melt temperature. (Real; Default = 1.E20)
TROOM	Room temperature. (Real; Default = 293.0)

Remark:

1. The yield stress is computed from

$$\sigma_y = (A + B\dot{\varepsilon}_p^n) \left(1 + C \ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right) (1 - T^*^m) \right)$$

where

$$\dot{\varepsilon}_p = \text{effective plastic strain}$$

$$T^* = \frac{(T - T_r)}{(T_m - T_r)}$$

$\dot{\epsilon}$	=	effective strain rate
$\dot{\epsilon}_0$	=	referenced strain rate
T	=	temperature
T_r	=	room temperature
T_m	=	melt temperature

and A , B , n , C , and m are constants.

2. The reference strain rate is per unit time.
3. YID must unique among all YLDxx entries in one model.

YLDMC**Mohr-Coulomb Yield Model**

Defines a Mohr-Coulomb yield model. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLDMC	YID	Y1	Y2	Y3					

Example:

YLDMC	1	10.E5	20.E5	1.E4					
-------	---	-------	-------	------	--	--	--	--	--

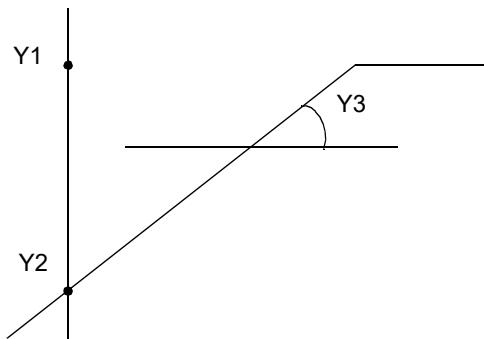
Descriptor	Meaning
YID	Unique yield-model number referenced from MATDEUL for Eulerian elements with shear strength. (Integer > 0; Required)
Y1	Cap yield stress. (Real; Required)
Y2	Cohesion. (Real; Required)
Y3	Internal friction angle. (Real; Required)

Remarks:

1. The yield stress depends on the pressure as

$$\sigma_y = \text{MIN}(Y1, (Y2 + Y3 \cdot P))$$

where Y1, Y2, Y3 are constants and P is the pressure.



2. This yield model is applicable only for Eulerian materials with shear strength.
3. YID must unique among all YLDxx entries in one model.

YLDMSS**Multi-Surface Yield Model for Snow**

Defines the yield model for snow material. This entry must be used in combination with MATDEUL, EOSPOL and SHREL. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLDMSS	YID	KC	T	CC	AC	BC	FC	FTU	
	7	0.149	82	1E-5	0.09	0.2	0.99	82	

Example:

YLDMSS	ALP0	DS							
	-0.37	0.0							

Descriptor	Meaning
YID	Unique yield model number referenced from a MATDEUL entry. (Integer > 0; Required)
KC	Parameter related to the angle of friction. (Real > 0; Required)
T	Equivalent value of the snow cohesion. see Remark 5. (Real > 0; Required)
CC	Shape of the yield surface. See Remark 4. (Real > 0; Required)
AC	Hardening parameter for compression. See Remark 4. (Real > 0; Required)
BC	Hardening parameter for compression. See Remark 4. (Real > 0; Required)
FC	Factor to avoid singularity. See Remark 4. (0 < Real < 1; Default = 0.99)
FTU	Hydrostatic tensile strength. See Remark 6. (Real > 0; Default = T/3)
ALP0	Initial compressive volumetric plasticity strain. See Remark 4. (Real < 0; Required)
DS	Softening modulus. See Remark. (Real \geq 0; Default = 0.0)

Remarks:

1. This material model can be used to model Snow material.
2. In addition to deviatoric plastic strain there is also volumetric plastic strain. This volumetric strain is stored in the output variable VOLPLS. The deviatoric strain is stored in the variable EFFPLS.
3. For detail description of each parameter in this model, the user should refer to the Theory Manual in which the mechanical properties of snow are described.
4. If CC is set to zero then the material behaves as a Drucker-Prager model. Ac, Bc, Fc^{*} and ALP0 will be ignored.

5. The T value must be consistently converted from the cohesion data, C_{DP} , of the Drucker-Prager model as follows:

$$T = C_{DP}/K_C$$

6. FTU, hydrostatic tensile strength, may not be greater than T divided by 3. Otherwise it will be set to that value.
7. The softening modulus is used to update the hardening parameter q_i , see Theory Manual. It can be requested as output using FTU variable. The corresponding accumulated-plastic-volumetric-tensile-strain variable is SOFTE.
8. This material model is valid for the Euler with Strength solver and the Multi-material Euler with Strength solver.
9. YID must unique among all YLDxx entries in one model.

YLDPOL**Polynomial Yield Model**

Defines a polynomial yield model where the yield stress is a function of effective plastic strain. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLDPOL	YID	A	B	C	D	E	F	SMAX	

Example:

YLDPOL	7	180.E6							
--------	---	--------	--	--	--	--	--	--	--

Descriptor	Meaning
YID	Unique yield model number referenced from MATDEUL. (Integer > 0; Required)
A	Initial yield parameters. (Real > 0; Required)
B	Coefficient B. (Real; Default = 0.0)
C	Coefficient C. (Real; Default = 0.0)
D	Coefficient D. (Real; Default = 0.0)
E	Coefficient E. (Real; Default = 0.0)
F	Coefficient F. (Real; Default = 0.0)
SMAX	Maximum yield stress. (Real; Default = 1.E20)

Remarks:

1. The yield stress is computed from

$$\sigma_y = \text{MIN}(\sigma_{\max}, A + B\varepsilon_p + C\varepsilon_p^2 + D\varepsilon_p^3 + E\varepsilon_p^4 + F\varepsilon_p^5)$$

where

ε_p = effective plastic strain

σ_{\max} = maximum yield stress

and A, B, C, D, E and F are constants.

2. YID must unique among all YLDxx entries in one model.

YLDRPL**Rate Power Law Yield Model**

Defines a rate power law yield model where the yield stress is a function of effective plastic strain and strain rate. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLDRPL	YID	A	B	n	m	C			

Example:

YLDRPL	7	180.E6							
--------	---	--------	--	--	--	--	--	--	--

Descriptor	Meaning
YID	Unique yield model number referenced from MATDEUL. (Integer > 0; Required)
A	Initial yield parameters. (Real > 0; Required)
B	Hardening parameter. (Real; Default = 0.0)
N	Hardening exponent. (Real; Default = 1.0)
M	Strain rate exponent. (Real; Default = 1.0)
C	Minimum yield stress. (Real; Default = 1.E20)

Remarks:

1. The yield stress is computed from

$$\sigma_y = \text{MAX}(C, A + B \varepsilon_p^{n \cdot m})$$

where

ε_p = effective plastic strain

$\dot{\varepsilon}$ = effective strain rate

and A , B , n , m and C are constants.

2. YID must unique among all YLDxx entries in one model.

YLD SG

Steinberg-Guinan Yield Model

Defines the Steinberg-Guinan yield model where the yield stress is a function of effective plastic strain, pressure and temperature. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLD SG	YID	A1	A2	A3	A4	H	B	CP	
	T MELT	T ROOM							

Example:

YLD SG	7	8E+6	100E+6	110	0.5				
	1500	273							

Descriptor	Meaning
YID	Unique yield model number referenced from a MATDEUL entry. (Integer > 0; Required)
A1-A4	Yield parameters. (Real > 0; Required)
H, B	Yield parameters. See Remark 4. (Real; Default = 0.0)
CP	Specific heat. (Real > 0; Default = 1.E20)
T MELT	Melt temperature. (Real; Default = 1.E20)
T ROOM	Room temperature. (Real; Default = 293.0)

Remarks:

1. This material model can be used to model metals for a wide range of strain rates.
2. The yield stress is computed from

$$A_T = A_1(1 + A_3 \varepsilon_p)^A$$

$$\sigma_y = \min(A_2, A_T) \left[1 - H(T - T_r) + B p \left(\frac{\rho}{\rho_{ref}} \right)^{\frac{1}{3}} \right] \quad T < T_m$$

$$\sigma_y = 0, \quad T \geq T_m$$

And A_1, \dots, A_4, H and B are constants.

3. The reference and quasi-static strain rate are per unit time.
4. YID must unique among all YLDxx entries in one model.

YLDTM**Tanimura-Mimura Yield Model**

Defines the Tanimura-Mimura yield model where the yield stress is a function of effective plastic strain, strain rate and temperature. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLDTM	YID	A	B	C	D	M	EPSM	CP	
	TMELT	TROOM	SCR	E	K	EPS0			

Example:

YLDTM	7	45.6E6	19.5E6			1.0	0.001		
			4000.e6	2100.	0.5	1.0			

Descriptor	Meaning
YID	Unique yield model number referenced from MATDEUL. (Integer > 0; Required)
A	Static yield parameter. (Real > 0; Required)
B	Hardening parameter. (Real; Default = 0.0)
C	Strain rate parameter C. (Real; Default = 0.0)
D	Strain rate parameter D. (Real; Default = 0.0)
M	Temperature exponent. (Real; Default = 0.0)
EPSM	Quasi-static strain rate. (Real > 0; Default = 1.0)
CP	Specific heat. (Real > 0; Default = 1.E20)
TMELT	Melt temperature. (Real; Default = 1.E20)
TROOM	Room temperature. (Real; Default = 293.0)
SCR	Critical yield stress. (Real > 0; Default = 1.0)
E	Strain rate parameter E. (Real; Default = 0.0)
K	Strain rate exponent. (Real; Default = 1.0)
EPS0	Reference strain rate. (Real > 0; Default = 1.0)

Remarks:

1. This material model can be used to model metals for a wide range of strain rates.
2. The yield stress is computed from

$$\sigma_Y = \left[A + B\varepsilon_P + (C + D\varepsilon_P) \left(1 - \frac{A + B\varepsilon_P}{\sigma_{cr}} \right) \ln \left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_s} \right) \right] (1 - T^m) + E \left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \right)^k$$

where

ε_p	=	effective plastic strain
σ_{cr}	=	critical yield stress
$\dot{\varepsilon}$	=	effective strain rate
$\dot{\varepsilon}_s$	=	quasi-static strain rate
$\dot{\varepsilon}_0$	=	reference strain rate
T^*	=	$(T - T_r)/(T_m - T_r)$
T	=	temperature
T_r	=	room temperature
T_m	=	melt temperature

and A, B, C, D, m, E and k are constants.

3. The reference and quasi-static strain rate are per unit time.
4. YID must unique among all YLDxx entries in one model.

YLDUDS

User-defined Yield Model for Elements.

Specifies that a user subroutine is being used to define a simple yield model. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLDUDS	YLD	GROUP	UNAME						

Example:

In FMS Section of the MSC Nastran input stream:

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

In Bulk Data:

1	2	3	4	5	6	7	8	9	10
YLDUDS	12	myyld	EXYLD						

Descriptor	Meaning
YLD	Unique yield model ID. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. (Character; default=EXYLD)

Remarks:

1. Only can be used for SOL 700.
2. The YLD must be referenced by a MATDEUL or MAT1 entry.
3. This yield model is applicable only for Euler elements with shear strength
4. UNAME can be:

Subroutine Name	Function
EXYLD	Standard user defined yield model

5. The yield strength can depend on the amount of failure or damage of the Euler element and on the VOLPLS and SOFTE element variables. This amount of damage can be either specified by the damage variable of the FAILJC entry or by a more general failure estimate by using the FAILUDS entry with UNAME=EXFAIL2. The VOLPLS and SOFTE variables can also be filled by the YLDUDS entry and FAILUDS entry with UNAME=EXFAIL2.

6. The damage variable is determined by either FAILJC or FAILUDS entry with UNAME=EXFAIL2. In UNAME=EXYLD, the yield stress can be reduced depending on the magnitude of the damage variable. In that case the NOFAIL option should be set on the FAILJC or FAILUDS entry with UNAME=EXFAIL2.

YLDVM

von Mises Yield Model

Defines a bilinear or piecewise-linear yield model with isotropic hardening, using the von Mises yield criterion. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLDVM	YID	YIELD	EH						
	TABLE	TYPE	TABY	D	P				

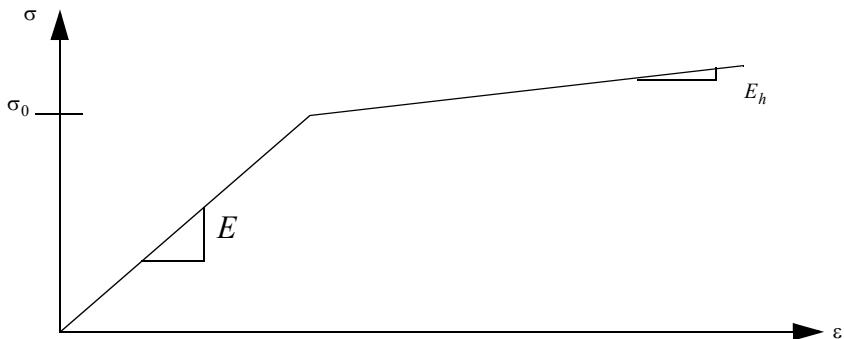
Example:

YLDVM	32	250.E6	2000.E6						

Descriptor	Meaning	
YID	Unique yield-model number referenced from MATDEUL. (Integer > 0; Required)	
YIELD	Yield stress. (Real; Required)	
EH	Hardening modulus. (Real; Default = 0.0)	
TABLE	Number of a TABLED1 entry giving the variation of effective stress (y-value) with effective strain (x-value). See Remark 4. (Integer > 0)	
TYPE	The type of stress and strain defined in TABLED1. (Character; Default = TRUE)	
	ENG	Engineering stress and strain.
	TRUE	True stress and strain.
	PLAST	True stress and plastic strain.
	PMOD	Plastic modulus and true stress.
TABY	Number of TABLED1 entry giving the variation of the scale factor for the yield stress (y-value) with the strain rate (x-value). Strain-rate effects are also specified using the Cowper-Symonds relation (see input parameters D and P). See Remark 6. (Integer > 0)	
D	Factor D in the Cowper-Symonds rate enhancement equation. See Remark 6. (Real ≥ 0)	
P	Factor P in the Cowper-Symonds rate enhancement equation. See Remark 6. (Real ≥ 0)	

Remarks:

1. A bilinear stress-strain characteristic is used by specifying YIELD and EH:



where the yield stress σ_y is given by

$$\sigma_y = \sigma_0 + \frac{EE_h}{E - E_h} \varepsilon_p$$

where

σ_0	= yield stress specified in the YIELD field
E	= Young's modulus
E_h	= hardening modulus specified in the EH field
ε_p	= equivalent plastic strain
σ_y	= yield stress

2. A piecewise linear, stress-strain characteristic is used by specifying TABLE and TYPE (beams and shells only)

$$\sigma_{ij} = [(\sigma_i - \sigma_{i-1})(\varepsilon - \varepsilon_{i-1}) / (\varepsilon_i - \varepsilon_{i-1})] + \sigma_{i+1}$$

The stress-strain characteristic used internally in the solver is in terms of true stress and equivalent plastic strain. However, for convenience, the stress-strain characteristic can be input in any of the following ways:

True stress/true strain (TYPE = TRUE)

Engineering stress/engineering strain (TYPE = ENG)

True stress/plastic strain (TYPE = PLAST)

Plastic modulus/true stress (TYPE = PMOD)

3. With Lagrangian and Eulerian solid elements, only an elastic perfectly plastic yield model is currently used. Only the YIELD field is used.
4. If TABLE is blank or zero, a bilinear stress-strain curve is assumed. If TABLE has a value, it refers to a TABLED1 entry giving the stress-strain curve for the material.
5. If TABLE is defined, the value of YIELD is left blank, since it is determined from the stress-strain curve.
6. If TABY is blank or zero and D and P are blank or zero, the yield stress does not vary with strain rate. If TABY has a value, then it references a TABLED1 entry, which gives the variation of the scale factor applied to the yield stress with strain rate. (D and P must be blank or zero.)
If TABY is blank or zero and D and P are defined, the enhancement of the yield stress with strain rate is calculated as

$$\frac{\sigma_d}{\sigma_y} = 1 + \left(\frac{\dot{\varepsilon}_p}{D} \right)^{1/p}$$

Where σ_d is the dynamic stress, σ_y is the static yield stress (YIELD), and $\dot{\varepsilon}_p$ is the equivalent plastic strain rate.

7. If TYPE is PLAST or PMOD, Young's modulus must be defined. If TYPE is ENG or TRUE and Young's modulus is defined it will override the value calculated from the stress-strain curve.
8. Note that for values exceeding the maximum x-value of either of the TABLED1 entries (see TABLE and TABY fields), linear extrapolation is used based upon the last two points specified in the TABLED1.
9. YID must unique among all YLDxx entries in one model.

YLDZA**Zerilli-Armstrong Yield Model**

Defines the Zerilli-Armstrong yield model where the yield stress is a function of effective plastic strain, strain rate and temperature. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
YLDZA	YID	A	B	n	C	m	EPS0		
	D								

Example:

YLDZA	7	200.E6	50.E6	0.1					
	0.0								

Descriptor	Meaning
YID	Unique yield model number referenced from a MATDEUL entry. (Integer > 0; Required)
A	Static yield parameters. (Real > 0; Required)
B	Hardening parameters. (Real ≥ 0 ; Default = 0.0)
n	Hardening exponent. (Real; Default = 0.0)
C	Strain rate parameter. (Real; Default = 1.0)
m	Temperature exponent. (Real; Default = 1.0)
EPS0	Reference strain rate. (Real > 0; Default = 1.0)
CP	Specific heat. (Real > 0; Default = 1.E20)
D	Bcc parameter. See Remark 4. (Real; Default = blank)

Remarks:

1. This material model can be used to model Fcc (iron and steels) and Bcc (aluminum and alloys) metals.
2. The yield stress is computed from
for Fcc metals:

$$\sigma_y = (A + B\dot{\varepsilon}_p^n)\varepsilon^{\left[-mT + CT\ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\right]} \text{ for Fcc metals}$$

$$\sigma_y = (A + B\dot{\varepsilon}_p^n) + De^{\left[-mT + CT\ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\right]} \text{ for Bcc metals}$$

where

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.

3. The reference strain rate are per unit time.
4. In case the Bcc parameter D is not supplied, it is assumed that a Fcc metal is defined.

A

Configuring the Runtime Environment

- Specifying Parameters
- User-Defined Keywords
- Resolving Duplicate Parameter Specifications
- Customizing Command Initialization and Runtime Configuration Files
- Symbolic Substitution

Specifying Parameters

MSC Nastran execution is controlled by a variety of parameters, either keywords or special Nastran statements, both required and optional. The purpose of this section is to describe how and where these parameters may be specified, not to describe these parameters in detail. This is done in subsequent sections. The MSC Nastran parameters may be specified on the command line, in a command initialization (INI) file, in runtime configuration (RC) files and, for some parameters, from environment variables. The information from these sources is consolidated at execution time into a single set of values. Much of this information is passed to analysis processing in a "control file", built using the templates ([Customizing the Templates, 51](#)). (The records in this control file are echoed to the .log file.) Examples of INI and RC files are given in the [User-Defined Keywords, 3355](#) and [Customizing Command Initialization and Runtime Configuration Files, 3363](#).

Command Initialization and Runtime Configuration Files

Although the purposes of the INI and RC files are somewhat different, the format of each file is the same. All INI and RC files are processed twice, once (the "first" pass) to extract parameters (keywords and other information) that are to be used for all MSC Nastran jobs, and once (the "second" pass) to extract parameters specific to a particular job. This is accomplished by separating the INI and RC files into a series of "sections" identified by a "section header" and "subsections" within sections, identified by a subsection "header." There are two types of sections: "unconditional" and "conditional." Subsections are always "conditional."

- An unconditional section is one that starts with the name of the section enclosed in square brackets ("[", "]"). Section names may not contain any embedded blanks but may be separated from the square brackets by any number of blanks. As currently implemented, there are three valid unconditional names: "General", "Solver" and "Nastran". (These section names are case-insensitive.) In addition, there is an implicit "unnamed" section that consists of all parameters in the INI or RC file that appear before the first named section or subsection. There is no special meaning assigned to any of the unconditional sections. Their use is optional; the section names are intended to be used for descriptive purposes.
- A conditional section or subsection is one that starts with an expression in the form:

`<keyword><operator><value>`

enclosed in section header identification characters. For a conditional section, the section header identification characters are square brackets ("[", "]"), just as for unconditional sections. For a subsection, the section header identification characters are "less than" and "greater than" ("<", ">") characters. Keywords and values may not contain any embedded blanks but may be separated from each other and from the enclosing section header identification characters (the square brackets or "less than"- "greater than" characters) by any number of blanks. In the expression:

- | | |
|-------------------------------|--|
| <code><keyword></code> | represents any valid internal keyword (see Keywords, 182) or user-defined keyword (see User-Defined Keywords, 3355). |
| <code><operator></code> | specifies the comparison to be performed between <code><keyword></code> and <code><value></code> as follows: |
| | = equal (either string or numeric) |

!	not equal (either string or numeric)
!=	not equal (either string or numeric)
<	numerically less than
<=	numerically less than or equal
>	numerically greater than
>=	numerically greater than or equal
<value>	specifies the appropriate keyword value to be used in the comparison.
	Keywords and values may be specified in any case.

Parameters in unconditional sections, but not in subsections (which are always conditional) within unconditional sections, are processed on the first pass through an INI or RC file. On the second pass, these parameters are ignored (they are not reprocessed). Parameters in conditional sections and subsections are ignored on the first pass. Parameters in conditional sections and subsections whose expressions evaluate to "true" are processed on the second pass through an INI or RC file, thus allowing conditional expressions to reference all of the valid keywords. Note that for subsections within conditional sections, both the conditional expression for the section *and* the conditional expression for the subsection must evaluate to "true" before parameters in the subsection are processed.

Parameter specifications in, either unconditional or conditional sections, may be continued, if necessary, by specifying a backslash ("\") character as the last non-blank character of the line. Note for Windows users, if the parameter value itself ends with a backslash, the statement must have additional characters, such as a comment, after the value specification. For example, a specification such as:

```
sdir=e:\
```

will not work properly. Instead, write the statement as:

```
sdir=e:\ $ Specify the scratch directory
```

In addition to parameters, INI and RC files may contain "comment" records. There are two types of comment records: ignored and printed.

- Ignored comments are records that start with a semi-colon (";") or pound sign ("#"). These records are completely ignored. When running in Windows, there is a special form of ignored comments that may be specified in an INI file (but not in RC files). These are records that start with "REM", short for "REMARK". The test for "REM" is case-insensitive.
- Printed comments are records that start with the currency symbol ("\$"). These records are passed on as part of the analysis information but are otherwise ignored.

Note: Although sectioning within INI and RC files was first introduced in MSC Nastran 2004, valid INI and RC files from prior versions of MSC Nastran are fully compatible with this new format. Since sections were not supported in previous versions (except for INI files on Windows, which allowed unconditional sections), all parameters will be in the "unnamed" implicit section (or, on Windows, in named unconditional sections) and will be processed on the first pass through the file. No information will be extracted from these files on the second pass.

The list below specifies the INI and RC files that MSC Nastran uses. Any or all of these files may be omitted. [Table 1-1](#) lists the keywords that are generally set in the unconditional sections of the command initialization file. [Table 1-2](#) lists the keywords that are generally set in RC files.

■ Command Initialization (INI) File

This file is used to define keywords that are to be set whenever the nastran command is executed. Typical keywords in the unconditional sections include the installation base directory and the version of MSC Nastran. Conditional sections and subsections might include keywords such as "rcmd" and "rsdirectory" in sections that are conditional upon the value of the "node" keyword.

LINUX: *install_dir/prod_ver/arch/nastran.ini*

Windows: *install_dir\prod_ver\arch\nastran.ini*

Starting with MSC Nastran 2011, there are two possible RC files that may be defined in each of the locations that are searched for RC files. The first name is a version independent name and the second name is a version dependent name, where the version number is indicated by *<vernum>* in the file name and the version number for MSC Nastran is 20200.

■ System RC Files

These files are used to define parameters that are applied to all MSC Nastran jobs using this installation structure. Many of the parameters that might be specified in the INI file could, alternatively, be specified in this file.

LINUX: *install_dir/conf/nastranrc* and

install_dir/conf/nast<vernum>.rc

Windows: *install_dir\conf\nastran.rcf* and

install_dir\conf\nast<vernum>.rcf

■ Architecture RC Files

This files are used to define parameters that are applied to MSC Nastran jobs using this architecture.

LINUX: *install_dir/conf/arch/nastranrc* and

install_dir/conf/arch/nast<vernum>.rc

Windows: *install_dir\conf\arch\nastran.rcf* and
install_dir\conf\arch\nast<vernum>.rcf

■ Node RC Files

These files are used to define parameters that are applied to MSC Nastran jobs running on this node. Alternatively, the parameters in this file could be specified in a conditional section in one of the previous files, using *nodename* as the value of the "s.hostname" keyword in the conditional expression.

LINUX: *install_dir\conf\net\nodename\nastranrc* and
install_dir\conf\net\nodename\nast<vernum>rc

Windows: *install_dir\conf\net\nodename\nastran.rcf* and
install_dir\conf\net\nodename\nast<vernum>.rcf

■ User RC Files

These files are used to define parameters that are applied to MSC Nastran jobs run by an individual user.

LINUX: \$HOME/.nastranrc and
\$HOME/.nast<vernum>rc

Windows: %HOMEDRIVE%&%HOMEPATH%\nastran.rcf and
%HOMEDRIVE%&%HOMEPATH%\nnast<vernum>.rcf

■ Local RC Files

These files should be used to define parameters that are applied to MSC Nastran jobs that reside in the input data file's directory. This RC file is in the same directory as the input data file. If the "rcf" keyword ([page 214](#)) is used, this local file is ignored.

LINUX: .nastranrc and
.nast<vernum>rc

Windows: nastran.rcf and
nast<vernum>.rcf

Please note that the LINUX shorthand "~", to refer to your or another user's home directory, cannot be used in an RC file. In addition, environment variables are only recognized within the context of a logical symbol definition.

Also, note that, on LINUX systems, the leading period (".") on the User RC Files and Local RC Files file names cannot be deleted even if alternate names are specified using the "a.rc" and "a.urcb" keywords as described below.

The file names listed above may be changed by the user using the "a.rc", "a.rcb", "a.urc" and "a.urcb" keywords, noting that the directories in which the files are located may not be changed.

- The “a.rc” keyword can be used to change the names of the version dependent RC file names for the System RC Files, the Architecture RC Files and the Note RC File. The default for this keyword is “nast<vernum>rc” for LINUX and “nast<vernum>.rcf” for Windows.
- The “a.rcb” keyword can be used to change the names of the version-independent RC file names for the System RC Files, the Architecture RC Files and the Node RC Files. The default for this keyword is “nastranrc” for LINUX and “nastran.rcf” for Windows.
- The “a.urc” keyword can be used to change the names of the version dependent RC file names for the User RC Files and the Local RC Files. For LINUX, the default for this keyword is the value of the “a.rc” keyword with a leading period (“.”) added. For Windows, the default for this keyword is the value of the “a.rc” keyword.
- The “a.urcb” keyword can be used to change the names of the version-independent RC file names for the User RC Files and the Local RC Files. For LINUX, the default for this keyword is the value of the “a.rcb” keyword with the leading period (“.”) added. For Windows, the default for this keyword is the value of the “a.rcb” keyword.

In addition to keyword specifications, the following MSC Nastran statements (from the NASTRAN and FMS Sections) may appear in RC files and conditional sections in an INI file: NASTRAN, ACQUIRE, ASSIGN, CONNECT, DBCLEAN, DBDICT, DBFIX, DBLOAD, DBLOCATE, DBSETDEL, DBUNLOAD, DBUPDATE, DEFINE, ECHOOFF, ECHOON, ENDJOB, EXPAND, INCLUDE, INIT, PROJ, RESTART and RFINCLUDE. Except for minimal checking of the NASTRAN and PARAM statements, the syntax of these statements is not validated. These records are simply passed on for use in MSC Nastran analysis processing.

INI files and RC files also may contain PARAM statements that specify values that affect MSC Nastran analysis processing. The values associated with PARAM names may be specified using PARAM statements in INI files and RC files or by using PARAM keywords, defined using the PARAM keywords feature as described in [User-Defined Keywords, 3355](#). PARAM statements must be specified in "free-field format", i.e., in the Case Control PARAM format (PARAM,name,value), not in Bulk Data fixed-field format. Please see [Parameters \(Ch. 6\)](#) in the *MSC Nastran Quick Reference Guide* for more information on PARAM names and statements and their usage.

Environment Variables

Several keywords may have their values set from associated environment variables. When this is the case, the environment variable takes precedence over any INI or RC file keyword specification. A command-line specification will over-ride the environment variable specified value. This same precedence rule applies to user-defined keywords that may have their initial values taken from environment variables, as described in the next section. A list of the keywords and their associated environment variables, along with a description of each keyword, may be obtained by using the following command:

`prod_ver nastran help env`

User-Defined Keywords

In addition to the internally defined keywords (see [Keywords, 182](#)), MSC Nastran allows users to define their own keywords. There are two classes of user-defined keywords:

- General keywords. These are intended for use in INI file or RC file conditional section clauses, in user modifications to the run template files (nastran.dmp, nastran.lcl, nastran.rmt or nastran.srv) and, for LINUX, in customized queue commands ("submit" keyword).
- PARAM keywords. These are keywords associated with a PARAM name. Using descriptive keywords to set a PARAM value may be more convenient than specifying the PARAM statement in an RC file. Also, keywords are not limited to a maximum of eight characters, as PARAM names are, and may be more descriptive of the action being affected or requested.

User-defined keywords are supported by the "help" and "whence" functions.

General Keywords

These keywords are defined in the file specified by the "0.kwds" keyword. While the file is not delivered, an administrator could create the file. The default file names are:

LINUX: *install_dir/prod_ver/arch/nastran.kwds*
Windows: *install_dir\prod_ver\arch\nastran.kwds* or
 install_dir\bin\nast20200.kwds
The file used is the *first* one found.

The records in this file consist of:

- Comment records. These are records that start with a comment character (hash, '#', semi-colon, ';' or currency symbol, '\$') and are completely ignored.
- Blank or null records. These records are ignored.
- Keyword records. These records consist of the keyword name along with an optional value descriptor and comment in the form:

```
keyword_name[, attributes] : value_descriptor comment
```

where:

<code>keyword_name</code>	is the name to be assigned to the user keyword. This name may not contain any embedded blanks and may not be the same as any internal keyword or previously specified user-defined keyword. It is also case-insensitive except in the case when its initial value may be set from an environment variable with the same name.
<code>attributes</code>	specifies optional attributes to be assigned to the keyword defined by <code>keyword-name</code> . Currently, the only defined attribute is: <code>argv</code> keyword and its value is to be added to the “ <code>r.argv</code> ” keyword value Any number of blanks may separate <code>keyword_name</code> , the separating command and the attributes specification.
<code>value_descriptor</code>	is optional. If specified, it should be as described in Value Descriptors, 3357 and may not contain any embedded blanks. If this field is not present, the separating colon may be omitted.. The default value descriptor is “ <code>string</code> ”. This field may also specify that the initial value of this keyword be taken from an environment variable with the same name.
<code>comment</code>	is an optional comment field. If present, it must be separated from <code>value_descriptor</code> or <code>keyword_name</code> by blanks or must begin with a comment character.

There may be any number of leading blanks in the record and before and after the separating colon.

General keywords and the values assigned to them only affect MSC Nastran processing if:

- there are customized INI and RC files that have conditional sections, using these keywords in expressions, that specify other keywords and statements (e.g., NASTRAN and PARAM statements) that modify MSC Nastran processing to meet the requirements of a user's site and installation.
- they are used in customized templates ([Customizing the Templates, 51](#)).
- for LINUX systems, they are used in customized queue commands defined using the "submit" keyword ([Customizing Queue Commands \(LINUX\), 49](#)).

PARAM Keywords

These keywords are defined in the file specified by the "0.params" keyword. The default file names are:

LINUX: `install_dir\prod_ver\arch\nastran.params`

Windows: `install_dir\prod_ver\arch\nastran.params`

The records in this file consist of:

- Comment records. These are records that start with a comment character (hash, '#', semi-colon, ';' or currency symbol, '\$') and are completely ignored.
- Blank or null records. These records are ignored.
- Keyword-name records. These records consist of the keyword name, the associated PARAM name, along with an optional value descriptor and comment in the form:

keyword_name : param_name : value_descriptor comment

where:

keyword_name	is the name to be assigned to the PARAM keyword. This name is case-insensitive, may not contain any embedded blanks and may not be the same as any internal keyword, general user-defined keyword or previously specified PARAM keyword.
param_name	is the PARAM name to be associated with keyword_name. This name is case-insensitive, may be a maximum of eight characters, must begin with an alphabetic character and may not contain any embedded blanks. Also, it may not be the same as any previously specified PARAM name.
value_descriptor	is optional. If specified, it should be as described in Value Descriptors and may not contain any embedded blanks. If this field is not present, the separating colon may be omitted. The default value descriptor is "string".
comment	is an optional comment field. If present, it must be separated from value_descriptor or param_name by blanks or must begin with a comment character.

There may be any number of leading blanks in the record and before and after the separating colons.

Keyword names that are the same as PARAM names are allowed, as long as the keyword name is not an internal or general user-defined keyword name.

Values associated with PARAM names, whether set using PARAM keywords or set using PARAM statements (statements having the form PARAM,*name,value*), directly affect MSC Nastran analysis processing.

Value Descriptors

Value descriptors enable limited syntax checking for values assigned to general and PARAM user-defined keywords. For general keywords, they may also specify that the initial value of the keyword be set from the value associated with the environment variable having the same name as the keyword. There are two types of syntax checking available: value must be one of a list of entries or value must be numeric. Also, the two forms can be combined. These are specified as follows:

List: { "val1", "val2", ..., "valn" }

That is, the acceptable values are enclosed in double quotes ("") and separated from each other by commas. The specification, including the various acceptable values, may not contain any embedded blanks. Values are case-insensitive and any partial specification is acceptable and will be replaced by the full value. For example, if a keyword may only have the values "preliminary", "check" and "final", the value descriptor would be:

```
{"Preliminary","Check","final"}
```

and a value specification of "Ch" would be accepted and replaced by "check".

Numeric: number

Values will be checked to see if they are valid numbers, either integer or floating point. For example, valid keyword value specifications could be: "1", "-3.247", "4.e-5", "3.75.4", "4.24x" and "-4-5" are invalid specifications.

Note: This checking does *not* support the NASTRAN "nnnseee" numeric format, where the 'e' between the number and the signed exponent ("seee") is missing.

Complex value: number, number

This format is only supported for PARAM keyword value descriptors. Values will be checked to see if they consist of two valid numeric values, separated by a comma.

Combined: {"val1", "val2", ..., "valn", number}

Note: This "combined" format does not support complex numbers.

In addition, for general keywords, if the value descriptor starts or ends with the string "env", specified in any case and separated from the rest of the value descriptor with a comma (unless the value descriptor is only "env"), the keyword value will be set using the value associated with the environment variable having the same name as the keyword. The environment value will be subjected to the same syntax-checking rules that an INI file, RC file or command line specification would be, with a warning message generated if syntax checking fails. This occurs even if the keyword is specified on the command line. Note that, for LINUX systems, since environment variable names are case-sensitive, the keyword name must be specified exactly the same as the environment variable name. This is the only time that the keyword name is case-sensitive. For Windows systems, since environment variable names are not case-sensitive, this restriction does not apply. Keyword values set from environment variables over-ride keyword values set in INI or RC files but do not over-ride keyword values set on the command line.

If a value descriptor is omitted or is not one of these formats, no syntax checking will be performed.

Examples:

1. The following value descriptor would accept a value of "test", "final" or a number:

```
{"Test","Final",Number}
```

Acceptable values would be: `te` (replaced by test), `FIN` (replaced by final), `7`, `14.5`, `3.e-4`, `-5`

2. The following value descriptor would accept only the strings "abc", "def", "ghi" and "glm":

```
{"abc","def","ghi","glm"}
```

Acceptable values would be: `g` (replaced by ghi), `aB` (replaced by abc), `g1` (replaced by glm), `D` (replaced by def)

3. The following value descriptor, only valid for a PARAM keyword, would only accept a complex number specification:

```
number,number
```

Acceptable values would be: `1,2`, `7.54`, `3.14`

4. The following value descriptors, only valid for a general keyword, would accept only the strings "qrs", "test", and "xyz". In addition, the value descriptor requests that the keyword value be set from the environment.

```
env,{"qrs","test","xyz"}
```

or

```
{"qrs","test","xyz"},Env
```

Acceptable values would be: `q` (replaced by qrs), `xY` (replaced by xyz), `T` (replaced by test)

Resolving Duplicate Parameter Specifications

MSC Nastran processing information is obtained by scanning the various INI and RC files, the system environment, and the Nastran command line in the following order:

1. Nastran command line, first pass. Only "program options", i.e., "-x" options, are processed during this command line scan. For example, this is when the "-i ini_file_name" program option is processed.
2. Environment variables, first pass. During this pass, the only keywords whose values are set are those that may only be specified as environment variables. This includes keywords such as HOME (for LINUX), HOMEDRIVE and HOME PATH (for WINDOWS) and PWD.
3. INI file, first pass, if this file exists. During this pass, only unconditional sections are processed. Generally, the only keywords processed in this pass are: 0.kwds, 0.params, accmd, acvalid, rcmd, rsdirectory, sysmsg and version (although rcmd and rsdirectory probably should be in conditional sections scanned during the second pass).
4. Environment variables, second pass. During this pass, only those keywords that may only be set in global sections of the INI file or as environment variables are processed. This includes keywords such as MSC_ARCH, MSC_BASE and MSC_VERSD.
5. Nastran command line, second pass. The only general use keywords processed during this command line scan are: dmparallel, jid, jidpath, jidtype, node, pause, rcf, username, version and whence. The processing of other command line keywords is deferred until later command line scans.

This is the time that the user-defined keyword definition files (for both general use and PARAM keywords), if any, are processed and the keyword specifications defined by these files are added to the keywords tables. The keywords defined in these files may be used just as internal keywords are used. (See [User-Defined Keywords, 3355](#).)

6. System RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
7. Architecture RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
8. Node RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
9. User RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
10. Local RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
11. Environment variables, third pass. During this pass, only "general" user-defined keywords that have been flagged to be set from environment variables are processed. (This pass will be skipped if there are no "general" user-defined keywords.)
12. Nastran command line, third pass. Only "general" user-defined keywords are processed during this command line scan. (This pass will be skipped if there are no "general" user-defined keywords.)

At this point, all keyword values that can be used in conditional section expressions are known.

13. INI file, second pass, if this file exists and has conditional sections. During this pass, only the conditional sections are processed.
14. System RC files, second pass, if these files exist and have conditional sections. During this pass, only the conditional sections are processed.
15. Architecture RC files, second pass, if these files exist and have conditional sections. During this pass, only the conditional sections are processed.
16. Node RC files, second pass, if these files exist and have conditional sections. During this pass, only the conditional sections are processed.
17. User RC files, second pass, if these files exists and have conditional sections. During this pass, only the conditional sections are processed.
18. Local RC files, second pass, if these files exist and have conditional sections and if they are not ignored. During this pass, only the conditional sections are processed.
19. Environment variables, fourth pass. During this pass, all keywords that may be set from environment variables and that have not been processed previously are now processed.
20. Nastran command line, fourth pass. All keywords not processed during the previous passes are now processed. For example, this is when user-defined PARAM keyword specifications are processed.
At this point, all information necessary to generate the "control file" has been collected. This file is generated when the "script templates" (see [Customizing the Templates, 51](#)) are processed.
21. NASTRAN, FMS and PARAM statements in the input file.

If duplicate keywords are encountered, the *last* specification found is the one used. That is, the above list specifies the precedence order, from lowest precedence (number 1) to highest (number 21). The only case in which the last keyword specification is not used is when keywords are "locked", i.e., when a specification of the form

```
lock=keyword
```

is processed. After this "lock" request is processed, any requests to set *keyword*, whether from INI files, RC files, environment variables or command line arguments, are quietly ignored. That is, processing proceeds as if any *keyword* specifications specified after the "lock=keyword" request do not exist. Once a keyword has been "locked," there is no way to "unlock" it. (Note that it is valid to "lock" the lock keyword itself.)

If duplicate NASTRAN and FMS statements are encountered, they are simply passed on for use in MSC Nastran analysis processing in the order in which they were encountered.

Thus, the general rule for resolution is:

- Information specified in NASTRAN input data files always takes precedence over any other values.
- Command line parameters have the next highest precedence.
- Environment variables associated with keywords and that have non-null values are next.
- RC file parameter specifications are next.
- INI file parameter specifications are last.

Generally, the only exceptions to this precedence ordering are "general" user-defined keyword specifications. The command line values take precedence over values specified in unconditional INI file and RC file sections but have lower precedence than values specified in conditional INI file and RC file sections. Because the primary purpose for general user-defined keywords is for conditional section selection, changing a general user-defined keyword in a conditional section *may* lead to unexpected results. Such specifications should be used with care. Also, because user-defined PARAM keywords on the command line are not processed until the last command line scan, PARAM keywords should not be used in INI file and RC file conditional section expressions since command line specified values will not be in effect when these expressions are evaluated.

Because PARAM values may be specified either using PARAM statements or using PARAM keywords, they require further explanation. PARAM statements and PARAM keywords referring to the same PARAM name are considered equivalent definitions for the PARAM name. As such, the last specification, regardless of whether it was a PARAM statement or a PARAM keyword, is the one that is used to establish the value associated with the PARAM name.

Customizing Command Initialization and Runtime Configuration Files

Table 1-1 lists the keywords that are generally set in the unconditional sections of the command initialization file.

Table 1-1 Command Initialization File Keywords

Keyword	Purpose
0.kwds	Alternate name for user-defined keywords definition file.
0.params	Alternate name for PARAM keywords definition file
acct	Enables job accounting, see Enabling Account ID and Accounting Data, 39 .
acvalid	Activates account ID validation, see Enabling Account ID Validation, 39 .
MSC_BASE	Defines the installation base directory. Normally this is defined as an environment variable by the prod_ver command.
version	Specifies the default version of MSC Nastran to be run.

Most of the command line keywords can be set in any of the RC files. **Table 1-2** lists keywords that are generally set in the system, architecture, or node RC files:

Table 1-2 RC File Keywords

Keyword	Preferred RC File	Purpose
accmd	System	Command line to invoke accounting logger program.
acct	System	Enables job accounting.
acvalid	System	Enables account ID (acid) validation.
authorize	System	Specifies the licensing method.
buffsize	System	Set the default buffsize. Suggested values are in Table 4-5 .
lock	Any	Prevent further changes to a keyword's value.
memory	Node	Specifies a default memory allocation
memorymaximum	Node	Specifies a maximum "memory" request. May be specified as a percentage of RAM e.g. memorymax=0.5xPhysical
ncmd	Architecture	Specifies the notify command when "notify=yes" is set.
news	System	Controls the display of the news file at the beginning of the .f06 file.

Table 1-2 RC File Keywords (continued)

Keyword	Preferred RC File	Purpose
post	Architecture	LINUX: Specifies commands to be run after each job is completed.
pcedelta	Architecture	LINUX: Specifies the value that is subtracted from the "CPU" keyword value to determine the NQS per-process CPU time limit.
ppmdelta	Architecture	LINUX: Specifies the value that is added to the "memory" keyword value to determine the NQS per-process memory limit.
pre	Architecture	LINUX: Specifies commands to be run before each job begins.
prmdelta	Architecture	LINUX: Specifies the value that is added to the "ppm" value to determine the NQS per-request (per-job) memory limit.
qoption	Architecture	LINUX: Specifies a string of additional queuing options to be set in the queue submittal command.
rcmd	Any	Specifies the remote Nastran command to be used when "node" is specified. Should be in a conditional section using "node" in the conditional expression.
real	Node	Specifies the "REAL" parameter to limit virtual memory usage.
rsdirectory	Any	Specifies the scratch directory to be used when "node" is specified. Should be in a conditional section using "node" in the conditional expression.
scratch	Any	Specifies the default job status as scratch or permanent.
sdirectory	Node	Specifies a default scratch directory.
submit	Architecture	LINUX: Defines queues and their associated submittal commands.
sysn	Any	Specifies system cells. Can also be specified using the synonym keywords, e.g., bufsize is equivalent to sys1.

Examples

The following (relatively simplistic) examples illustrate how unconditional and conditional sections could be used.

Example 1:

Assumptions: There are three computer nodes, sysnode1, sysnode2 and sysnode3, that may be accessed.

On sysnode1:

- MSC Nastran 2014 and MSC Nastran 2020 are installed:
 - MSC Nastran 2014 is accessed using "/local/msc/bin/nast2014"
 - MSC Nastran 2020 is accessed using "/local/msc/bin/nast20200"
 - The scratch directory is /local/temp

On sysnode2:

- Only MSC Nastran 2014 is installed and is accessed using "/local1/msc/bin/nast2014"
- The scratch directory is /local1/temp

On sysnode3:

- MSC Nastran 2014 and MSC Nastran 2020 are installed:
 - MSC Nastran 2014 is accessed using "/local2/msc/bin/nast2014"
 - MSC Nastran 2020 is accessed using "/local2/msc/bin/nast20200"
- The scratch directory is /local2/temp

All of this information could be specified in an INI file, identical on all three nodes, as follows:

```
;  
; This is the MSC Nastran Command Initialization File  
; The default version is to be set to 2020.0  
;  
version=2020.0  
; Define conditional sections giving the appropriate sdir  
; values when MSC Nastran is run locally.  
  
[ s.hostname = sysnode1 ]  
sdir=/local/temp  
[ s.hostname = sysnode2 ]  
sdir=/local1/temp  
[ s.hostname = sysnode3 ]  
sdir=/local2/temp  
  
; Define conditional sections giving the appropriate  
; remote access keywords when a "node" value,  
; requesting remote execution, is specified.  
;  
[ node = sysnode1 ]  
rsdir=/local/temp  
< version = 2014.0 >
```

```
rcmd=/local/msc/bin/nast2014
< version = 2020.0 >
rcmd=/local/msc/bin/nast2000

[ node = sysnode2 ]
rsdir=/local1/temp
< version = 2014.0 >
rcmd=/local1/msc/bin/nast2014

[ node = sysnode3 ]
rsdir=/local2/temp
< version = 2014.0 >
rcmd = /local2/msc/bin/nast2014
< version = 2020.0 >
rcmd=/local2/msc/bin/nast20200

;

; This is the end of the Command Initialization file
;
```

Alternatively, the information could be split between an INI file and a system RC file, identical on all three nodes, as follows:

In the INI file:

```
;

; This is the MSC Nastran Command Initialization File
; The default version is to be set to 2020.0
;
version=2020.0

; Define conditional sections giving the appropriate
; remote access keywords when a "node" value,
; requesting remote execution, is specified.
;
[ node = sysnode1 ]
rsdir=/local/temp
< version = 2014.0 >
rcmd=/local/msc/bin/nast2014
< version = 2020.0 >
rcmd=/local/msc/bin/nast20200

[ node = sysnode2 ]
rsdir=/local1/temp
< version = 2014.0 >
rcmd=/local1/msc/bin/nast2014

[ node = sysnode3 ]
rsdir=/local2/temp
< version = 2014.0 >
rcmd = /local2/msc/bin/nast2014
< version = 2020.0 >
rcmd=/local2/msc/bin/nast20200
```

```
;  
; This is the end of the Command Initialization file;
```

In the system RC file, identical on all three nodes:

```
;  
; This is the MSC Nastran system RC file.  
;  
; Define conditional sections giving the appropriate sdir  
; values when MSC Nastran is run locally.  
  
[ s.hostname = sysnode1 ]  
sdir=/local/temp  
[ s.hostname = sysnode2 ]  
sdir=/local1/temp  
[ s.hostname = sysnode3 ]  
sdir=/local2/temp  
  
;  
; This is the end of the system RC file  
;
```

Example 2:

Assumptions: User keywords defining "run type" and "data complexity" are needed and AUTOSPC, AUTOSPCR, BAILOUT and ERROR PARAM values are to be set based on these keywords.

The nastran.kwds file could be:

```
; User Keywords  
Runtype:{"prelim","development","final"};Analysis stage  
    Level : number          # Data complexity level  
    ;
```

The nastran.params file could be:

```
; PARAM keywords  
  
Set_AutoSPC : AutoSPC : {"Yes","No"}  
Set_AutoSP_Cr : AUTOSPCR : {"yes","no"}  
Bailout_Value : bailout : number  
Set_Error : Error : number  
;
```

Then, the system RC file could contain:

```
; RC file  
[ runtype = prelim ]  
set_automspc = yes  
bailout_value = -1  
set_error = 0  
set_automspc_cr = yes  
  
[ runtype = development ]  
set_automspc=yes  
bailout_value=0
```



```
set_error=-1

[runtype=final]
set_automspc=no
param,bailout,0
param,error,-1
param,autospcr,no

[level < 3]
; basic data complexity parameters
[level >= 3]
<level>8>
; advanced data complexity parameters

<level<=8>
; intermediate data complexity parameters

; End of RC file
```

Symbolic Substitution

Introduction

Symbolic Substitution is a capability added to MSC Nastran that allows a user to effectively modify a Nastran data file using command line and RC file keyword specifications without actually editing the file. This capability is very similar to “environment variable” expansion that happens in various command prompt shells such as the Linux Bourne, Korn and C shells and the Windows Command Prompt shell when scripts are processed. It is also analogous in some ways to the capabilities provided by programming language preprocessors, for example, the CPP preprocessor used by the various C/C++ compilers. The key feature of symbolic substitution is that these modifications do *not* affect the actual data file but present the data read from the data file to the processing program as if it was the modified data that was being processed.

Generally, symbolic substitution means that a data record is scanned to see if it contains special data strings (that identify the “symbolic” variables) that specify “symbolic substitution” requests. If such strings are found, the record is modified to replace the special data strings with user-defined substitution (replacement) strings (the values currently associated with the “symbolic” variables, i.e., the variable “values”) and it is this modified record that is actually processed. This symbolic substitution happens before any other processing of the record occurs, thus making it transparent to the rest of the program processing the data record. In the case of MSC Nastran, this symbolic substitution processing will happen immediately after a record is read from the Nastran data file and before any other processing (with the possible exception of special processing required to satisfy licensing requirements) is performed.

Simple Examples

Two very simple examples illustrate how this capability could be used in Nastran data files. Note that the details of the syntax are completely described in the following sections and may be ignored for now. Also note that the examples do not deal with things such as managing the output from multiple Nastran runs. These issues, involving, among other techniques, using command line or RC file keywords such as "out=", "append=" and "old=yes", are beyond the scope of this document.

Example 1:

Suppose you want to make several tests where the thickness of a PSHELL element is to be varied. You could do this by defining the thickness of the PSHELL element as a “symbolic variable” (identified using the string "%thickness%"), setting a default value (using the "%defrepsym" statement) and specifying the desired thickness on the command line (using the "REPSYM=" keyword). A very simple data file (sym.dat) could be (where most of the BULK entries are in an include file named "model.bdf", not shown here):

```
%defrepsym thickness=5.0
SOL 103
CEND
TITLE = 1st perturbation, t = %thickness%
ECHO = NONE
SUBCASE 1
METHOD = 100
```

```

SPC = 1
DISP = ALL
BEGIN BULK
EIGRL,100,,,6
PARAM,POST,0
PARAM,GRDPNT,0
$PBEAML Properties
PBEAML   2      1      I
          70.0    60.0    60.0    3.3     5.     5.
$
$PSHELL Properties
$
pshell,1,1,%thickness%,1,,1
$
include 'model.bdf'
enddata

```

If the test is run using the following command line:

```
nast20200 sym repsym=thickness=1.0 ...
```

the test will run as if the "TITLE" and "pshell" records are:

```
TITLE = 1st perturbation, t = 1.0
```

and

```
pshell,1,1,1.0,1,,1
```

If the test is run using the following command line:

```
nast20200 sym repsym=thickness=3.5 ...
```

the test will run as if the "TITLE" and "pshell" records are:

```
TITLE = 1st perturbation, t = 3.5
```

and

```
pshell,1,1,3.5,1,,1
```

If the test is run without specifying any REPSYM setting for "thickness", e.g., using the following command line:

```
nast20200 sym ...
```

the test will run as if the "TITLE" and "pshell" records are:

```
TITLE = 1st perturbation, t = 5.0
```

and

```
pshell,1,1,5.0,1,,1
```

Example 2:

Suppose you have a test that contains two superelements, where the only difference between the data for each superelement is the area of a PBAR element. Instead of having two different definitions, you could have a single definition of the data in an include file, where the area of the PBAR is specified as a symbolic variable. The include file (called "bar.bdf") could be:

```
%defrepsym area=1.  
grid,2,,1.0,0.0,0.0  
grid,3,,2.0,0.0,0.0  
grid,4,,3.0,0.0,0.0,,123456  
cbar,2,2,2,3,0.,1.,0.  
cbar,3,2,3,4,0.,1.,0.  
pbar,2,2,%area%,1.,1.,1.  
mat1,2,1.e7,,,3
```

and the actual input file could be:

```
sol 101  
cend  
title=simple part se  
echo=both  
subcase 1  
load=1  
disp=all  
elforce=all  
begin bulk  
grid,1,,0.0,0.0,0.0  
grid,2,,1.0,0.0,0.0  
cbar,1,1,1,2,0.,1.,0.  
pbar,1,1,1.,1.,1.,1.  
mat1,1,1.e7,,,3  
force,1,1,,1.,1.,1.,1.  
$  
begin super=1  
%setrepsym area=1.  
include 'bar.bdf'  
$  
begin super=2  
%setrepsym area=2.  
include 'bar.bdf'  
enddata
```

The first "include 'bar.bdf'" statement will be processed as if the pbar record is

```
pbar,2,2,1.,1.,1.,1.
```

and the second "include 'bar.bdf'" statement will be processed as if the pbar record is

```
pbar,2,2,2.,1.,1.,1.
```

Detailed Specifications

The use of the Symbolic Substitution capability is defined by a number of “rules”. These “rules” are given in the following sections and provide the complete specification. Following the rules, there is information about requesting report information and about error handling. Finally, there are some (again simple) examples showing usage.

Symbolic Substitution Rules

The following rules define the symbolic substitution user interface. The descriptions start with the rules for variable naming, followed by the rules for defining the replacement width information, followed by the various keywords and statements used to control symbolic substitution.

Variable Naming

The rules for naming symbolic substitution variables are:

- Symbolic variable names are not case-sensitive, are a maximum of 32 characters long and may not contain leading, trailing or embedded blanks or special characters including ("_"). Variable names must start with an alphabetic character followed by zero or more alphabetic or numeric characters.
For example:
 - The variable name "VaRiaBLe1" is the same as "VARIABLE1" and "variable1"
 - The following variable names are valid:
 - abcdef
 - abc123
 - Name1
 - The following variable names are not valid:
 - 123abc Does not start with an alphabetic character
 - a bcd Contains an embedded blank
 - abc& Contains an invalid character ('&')
 - /def Does not start with an alphabetic character
 - _abc123 Uses an underscore in the name.
- Unless symbolic variable values are quoted, they are not case-sensitive and may not contain leading, trailing or embedded blanks or percent ('%') characters. The quoting rules are given below.

Substitution Field Width Specification

The ability to control the appearance of any symbolic substitution is an important requirement when generating data for a program such as MSC Nastran. The result of a symbolic substitution request is identified as a *field*. Substitution field width information can be taken by default, specified in the data file or specified using command line and/or RC file keywords. These methods are explained below.

The rules for defining substitution field width information are:

- Symbolic variable substitution is, by default, *exact*. That is, the number of characters occupied by the symbolic symbol replacement is exactly the same as the replacement value. However, this default replacement processing can be controlled by specifying the substituted field *width*, the field *precision* and the *justification* within the field. This information is specified using the syntax

-w.p

where the ‘-‘, ‘w’ and ‘p’ are all optional and have the following meanings.

- The field width specification (w) defines the *minimum* number of characters the field is to have as a decimal integer value. If the replacement value has fewer characters than the field width, it will be padded with spaces on the left (by default) or on the right (if the left justification flag is specified). If the replacement value has more characters than the field width and if no precision value was specified, the entire replacement string will be used. A field width value of 0 (zero) is equivalent to omitting the width specification. Note that a negative width value will be processed as if the “left-justification” flag was specified (see below) since a negative field width is meaningless.
 - The field precision specification (p) defines the *maximum* number of characters the field is to have. The format is a period (.) followed by a decimal integer value. If the replacement value length exceeds the precision value, only the last p (by default) or the first p (if the left justification flag is set) characters of the replacement value will be used. A field precision value of 0 (zero) (or a negative value) is equivalent to omitting the precision specification.
 - If both field width and field precision are specified and are positive, the precision value cannot be less than the width value. If it is, it will be reset to the field width.
 - The ‘-‘ character is the “left-justification” flag and specifies that the replacement value is to be left-justified within the field. If this character is omitted, the replacement value will be right-justified within the field.
- For example, the width, precision and justification of a typical field in the Bulk Data portion of a Nastran data file is:

-8 . 8

meaning that the field is exactly eight characters wide and that data is to be left-justified within the field. For a wide-format Bulk Data record, this specification would be:

-16 . 16

The specification for an exact replacement, i.e., where the replaced field is exactly the size of the replacement value, is:

0 . 0

- To simplify width specification for Nastran widths, the following (case-insensitive) synonyms for common widths are available and may be used wherever a width specification can be used:

exact	is equivalent to 0 . 0
bulk	is equivalent to -8 . 8
wide	is equivalent to -16 . 16

It is very important to note that there are two distinct portions to a Nastran data file, that part that is before the first BEGIN statement and that has “free format”, and that part that is after the first BEGIN statement (the Bulk Data Section) and often has fixed format fields. Because of this, two different sets of field width information are maintained for use when field width information is not explicitly specified as part of a symbolic substitution request, one for use before the first BEGIN statement and one for use after the first BEGIN statement.

Defining Variable Values and Width Information

Symbol names and associated values and symbol width specifications may be set using keywords on the command line or in RC files and may be set using special statements in the Nastran data file itself. Each keyword and statement is explained in detail.

Using Command Line or RC File Keywords

Setting Variable Value Using REPSYM

Symbolic variables and associated values may be set on the Nastran command line or in RC files using the keyword

```
repsym=<varname>=<varvalue>
```

where <varname> specifies the name of the symbolic variable and <varvalue> specifies the value to be associated with the variable name. For example,

```
repsym=abc=1.23e-5
```

Setting Variable Width Information Using REPWIDTH

Symbolic variable substitution default width information may be set on the Nastran command line or in RC files using the keyword

```
repwidth=<widthinfo1>,<widthinfo2>
```

where <widthinfo1> specifies the default width information for the portion of the Nastran data file before the BEGIN statement and <widthinfo2> specifies the default width information for the portion of the Nastran data file after the BEGIN statement. Each is specified using a -w.p specification or as one of the synonyms, as described previously. If either width specification is omitted, the current default for that section is not changed. Note that the separating comma is required if the Bulk Data Section width value is to be set, i.e., if <widthinfo2> is specified. For example,

```
repwidth=12,bulk
```

specifies that symbolic substitution default width is to be 12.0 before the BEGIN statement is encountered and -8.8 after the BEGIN statement is encountered and

```
repwidth=,bulk
```

specifies that symbolic substitution default width is to be EXACT (or 0.0, the default) before the BEGIN statement is encountered and -8.8 after the BEGIN statement is encountered.

Just as with other Nastran command line or RC file keywords, the REPSYM and REPWIDTH keywords are not case-sensitive.

Using Special Statements in a Nastran Data File

Setting Values Using setrepsym

Symbolic variables and associated values may be set in a Nastran data file using the following statement:

```
%setrepsym <varname>=<varvalue>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following <varvalue>, where the start of the comment is indicated by a ' \$' (blank,

currency symbol). The `%setrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. For example,

```
%setrepsym abc=1.23e-5
```

Clearing ("Unsetting") Values Using `unsetrepsym`

A symbolic variable value set using the `%setrepsym` statement may cleared ("unset") in a Nastran data file using the following statement:

```
%unsetrepsym <varname>
```

where the ' %' character *must* be in column 1 and nothing else may appear in the record except for optional comments following `<varname>`, where the start of the comment is indicated by a ' \$. The `unsetrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. For example, to clear the variable `abc`, use

```
%unsetrepsym abc
```

Setting Default Values Using `defrepsym`

Default variable values can be set in a Nastran data file using the following statement:

```
%defrepsym <varname>=<varvalue>
```

where the ' %' character *must* be in column 1 and nothing else may appear in the record except for optional comments following `<varvalue>`, where the start of the comment is indicated by a ' \$. The `defrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. The specified value will be used *only* if a value for `<varname>` was not previously set, i.e., by a `repsym` keyword on the command line or in an RC file or by a `%setrepsym` statement previously specified in the data file that has not been unset by a `%unsetrepsym` statement. For example,

```
%defrepsym abc=2.46e+2
```

Clearing ("Unsetting") Default Values Using `undefrepsym`

The default value for a symbolic variable may cleared ("unset") in a Nastran data file using the following statement:

```
%undefrepsym <varname>
```

where the ' %' character *must* be in column 1 and nothing else may appear in the record except for optional comments following `<varname>`, where the start of the comment is indicated by a ' \$. The `undefrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. For example, to clear the default value associated with variable `abc`, use

```
%undefrepsym abc
```

Setting Width Information Using `setrepwidth`

Symbolic variable substitution default width information may be set in a Nastran data file using the following statement:

```
%setrepwidth <widthinfo1>,<widthinfo2>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following <widthinfo2>, where the start of the comment is indicated by a ' \$'. The setrepwidth string is not case-sensitive and at least one blank must separate this string from the width specifications. There may not be any blanks within the width specifications. <widthinfo1> specifies the width information for the portion of the Nastran data file before the BEGIN statement and <widthinfo2> specifies the width information for the portion of the Nastran data file after the BEGIN statement. Each is specified using a -w.p specification or as one of the synonyms, as described above. If either width specification is omitted, the current width information for that section is not changed. Note that the separating comma is required if the Bulk Data Section width value is to be set, i.e., if <widthinfo2> is specified. For example,

```
%setrepwidth 0.0,wide
```

specifies that the symbolic substitution width specification is to be 0.0 before the BEGIN statement and is to be -16.16 after the BEGIN statement.

Clearing ("Unsetting") Width Information Using unsetrepwidth

Symbolic variable substitution width information set using the %setrepwidth statement may be cleared in a Nastran data file using the following statement:

```
%unsetrepwidth
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following the unsetrepwidth string, where the start of the comment is indicated by a ' \$. The unsetrepwidth string is not case-sensitive and must be followed by at least one blank. This statement does not have any arguments and clears both width specifications.

Setting Default Width Information Using defrepwidth

Default symbolic variable substitution width information may be set in a Nastran data file using the following statement:

```
%defrepwidth <widthinfo1>,<widthinfo2>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record (except for optional comments following <widthinfo2>, where the start of the comment is indicated by a ' \$. The defrepwidth string is not case-sensitive and at least one blank must separate this string from the width specifications. There may not be any blanks within the width specifications. <widthinfo1> specifies the default width information for the portion of the Nastran data file before the BEGIN statement and <widthinfo2> specifies the default width information for the portion of the Nastran data file after the BEGIN statement. Each is specified using a -w.p specification or as one of the synonyms, as described above. If either width specification is omitted, the current width information for that section is not changed. Note that the separating comma is required if the Bulk Data Section width value is to be set, i.e., if <widthinfo2> is specified. For example,

```
%defrepwidth 0.0,wide
```

specifies that default symbolic substitution is to be 0.0 before the BEGIN statement and is to be -16.16 after the BEGIN statement.

Clearing ("Unsetting") Default Width Information Using undefrepwidth

Default symbolic variable substitution width information may be cleared in a Nastran data file using the following statement:

```
%undefrepwidth
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following the `undefrepwidth` string, where the start of the comment is indicated by a ' \$'. The `undefrepwidth` string is not case-sensitive and must be followed by at least one blank. This statement does not have any arguments and clears both default width specifications.

General Information For Special Statements

The `%setrepsym`, `%unsetrepsym`, `%defrepsym`, `%undefrepsym`, `%setrepwidth`, `%unsetrepwidth`, `%defrepwidth` and `%undefrepwidth` statements are deleted, logically, from the data file and will never be processed by the rest of Nastran unless an error is encountered while they are being processed. This is discussed in the [Error Handling, 3380](#).

Requesting Symbolic Substitution

Symbolic variable substitution will occur when a string having the form

```
%<varname>,<widthinfo>:<varvalue>%
```

is found anywhere within a Nastran data file, except that this string may *not* span records, i.e., the substitution request must be on a single record (line). The leading and trailing '%' characters are required as is the `<varname>` field. The `<widthinfo>` field is optional. If it is omitted, the comma (,) separating it from the `<varname>` field may be omitted and the rules for determining what width specification will be used are discussed below. The `<varvalue>` field is optional and provides a way of specifying a default value, i.e., the "local default value", as described below. If it is omitted, the colon (:) separating it from the `<varname>` (or `<widthinfo>`) field may be omitted. The rules for determining what symbolic value will be used as the substitution value are discussed below. For example, if the symbolic variable abc is to be replaced by its current value with no special processing (or if default width processing is to be used), the substitution request would be:

```
%abc%
```

If the symbolic variable is to be replaced by its current value, with the minimum field width to be 12 characters and with the value always to be left-justified, the substitution request would be:

```
%abc,-12%
```

Quoting Rules For Symbolic Variable Values

- If a symbolic variable value is case-sensitive, if it contains leading, trailing or embedded blanks or if it contains percent characters, tab characters or other special characters, it must be quoted. (Note that "escape" sequences such as '\t' or '\n' are not given any special treatment; that is, they are left as is.)
 - If the value is part of a `repsym` keyword command-line specification, the quoting rules of the command shell being used apply.
 - If the value is part of a `repsym` keyword specified in an RC file, it must be enclosed in single quotes (').

- If the value is part of a %setrepsym or %defrepsym record or if it specified as the “local default value” in a symbolic substitution request, quoting a symbolic variable value means enclosing the value in one of the following pairs of characters:

Starting Quote Character	Ending Quote Character
"	"
'	'
/	/
\	\
[]
{	}
()

If the first non-blank character encountered in a variable value specification is one of the starting quote characters, the variable value *must* be ended by the associated ending quote character. The actual variable value will be the (possibly null) string between (but not including) the starting and ending quote characters. If the variable value starts with one of the starting quote characters, it must be quoted using an alternate quote character.

General Rules For Symbolic Variable Substitution

- Nested symbolic substitution is not supported. Even if the value associated with a symbolic variable name is, itself, in the format of a symbolic variable substitution request, that request will be ignored. That is, after symbolic variable substitution has occurred, the substituted string is *not* re-scanned.
- Determining what symbolic variable value will be used when a variable substitution request is encountered depends on where the variable value associated with the specified variable name was set. The *first* value encountered in the following hierarchy is the value that will be used:
 - A value specified in the Nastran data file using the %setrepsym statement, if there is one active, i.e., if it has not been deactivated by a %unsetrepsym statement.
 - A value specified on the Nastran command line or in RC files using the repsym keyword.
 - As part of the variable symbol substitution request, using the local default value, if there is one.
 - A value specified in the Nastran data file using the %defrepsym statement, if there is one active, i.e., if it has not been deactivated by a %undefrepsym statement.

This precedence follows normal MSC Nastran ordering, i.e., "the data file wins," while still providing great flexibility. Also, the ordering of the last two items in this hierarchy allows a user to set all defaults except for special cases and follows the idea that the specification "closest" to the use is the one used. If no replacement value is found, the substitution request will be ignored and the record will be unchanged.

- Determining what symbolic width specification will be used when a variable substitution request is encountered depends on where the width information has been specified and on the part of the Nastran data file that is being processed, i.e., is the variable substitution request before or after the first BEGIN statement. The *first* width specification value encountered in the following hierarchy is the specification that will be used:
 - A value specified in the symbolic substitution request itself, i.e., if a <widthinfo> entry was specified as part of the symbolic substitution request.
 - A value specified on a %setrepwidth statement corresponding to the current section in the Nastran data file, if there is one active, i.e., if it has not been deactivated by an %unsetrepwidth statement.
 - A value specified on the Nastran command line or in RC files using the repwidth keyword corresponding to the current section in the Nastran data file.
 - A value specified in the Nastran data file using the %defrepwidth statement corresponding to the current section in the Nastran data file, if there is one active, i.e., if it has not been deactivated by a %undefrepwidth statement.
 - The program default value of exact (0 . 0).

This precedence also follows normal Nastran ordering, i.e., "the record wins followed by the data file wins," while still providing great flexibility.

- When running in licensing "Interlock" mode, i.e., in CRC validation mode, the following restrictions will be in effect. If a restriction is violated, Nastran processing will be terminated.
 - The %setrepsym, %unsetrepsym, %defrepsym and %undefrepsym statements are not allowed. Also, specifying a default value within the symbolic substitution request is not allowed. That is, symbolic variable values may only be set using the repsym keyword on the command line or in an RC file. Note that the %setrepwidth, %unsetrepwidth, %defrepwidth and %undefrepwidth statements *are* allowed.
 - A maximum of two symbolic substitution specifications are allowed per record and a maximum of ten symbolic substitution requests are allowed in the entire input data file.
 - Interlock CRC calculations will be made on the input record *before* symbolic substitution occurs. Note that any alterations to the record made as part of the CRC calculation processing will not affect symbolic substitution processing.

Requesting Symbolic Substitution Replacement Information Using REPINFO

- A report of what symbolic substitutions were made is generated at the end of Nastran processing, with the level of detail in the report controlled by an "information level" flag set using the `repinfo=n` keyword, where *n* is an integer number that specifies the level of detail desired. The meanings the various values for *n* are as follows:
 - 0 suppress the report altogether
 - 1 report the various values assigned using the repsym keyword

- 2 same as 1 except add the various values assigned using the `setrepssym` statement
- 3 same as 2 except add the various values assigned using the `defrepssym` statement
- 4 same as 3 except add the various values assigned as local default values
- 5 same as 1 except add all locations where the specified `repssym` value was used
- 6 same as 2 and 5 except add all locations where the specified `setrepssym` value was used
- 7 same as 3 and 6 except add all locations where the specified `defrepssym` value was used
- 8 same as 4 and 7 except add all locations where local default values were used.

The report is written to the .f06 file. If there is not enough dynamic memory available to save the report information, the `repinfo` level may be reduced. When running in MSC Nastran, the default is `repinfo=1`. Otherwise, `repinfo=0` will be forced.

- Just as with other Nastran command line or RC file keywords, the `REPINFO` keyword is not case-sensitive.

Error Handling

If an error is encountered processing a `setrepssym`, `unsetrepssym`, `defrepssym`, `undefrepssym`, `setrepwidth`, `unsetrepwidth`, `defrepwidth` or `undefrepwidth` statement, a comment string will be added to the record giving the error information and the record will be passed to Nastran (or the application reading the data file) as if the record was a normal Nastran data record. If an error is encountered in a record containing a symbolic substitution request, the symbolic substitution request will not be processed and, if `repinfo=1` or greater is in effect, a message giving information about the error will be written to the .log file. It is expected that the statements in error will not be valid Nastran statements and so will be flagged as an error.

Examples

1. The value on an “OPTION” statement is to be settable using the command line, taking a default value of “OPT1val” (case-sensitive) if no command line value is set. The OPTION statement could be

```
OPTION=%Option:'OPT1val'%
```

and the command line parameter that would be used to set OPTION to a different value, OP2VAL (not case-sensitive), would be

```
RepSym=Option=op2val
```

2. An INCLUDE file contains records that are to be used four times in the Bulk Data Section of a Nastran data file, with the only difference being the value in Field 3 of one record. The first time the file is used, this field must contain the value 1.234, the second time this field must contain the value 4.567 and the last two times this field must contain the value -12.578. In all cases, the replacement field must be eight characters wide and the data must be left-justified in the field. Assuming that the symbolic variable is DATFL3 and that the include file name is incl.data, this could be done as follows:

In the include file, specify the following statements before the record to be modified:

```
%DefRepSym datfl3=-12.578
```

then the record to be modified could be specified as follows:

```
FL1      FL2      %datfl3%FL4      FL5      FL6
```

and, for completeness, specify the following record after the record to be modified:

```
%UndefrepSym datfl3
```

Then the data file would contain:

```
....  
%setrepSym DATFL3=1.234
```

```
%DefRepWidth ,bulk
```

```
include 'incl.data'
```

```
....  
%setrepSym DATFL3=4.567
```

```
include 'incl.data'
```

```
%UnsetrepSym datfl3
```

```
....  
include 'incl.data'
```

```
....  
include 'incl.data'
```

