

MATS3

Advanced Orthotropic, Nonlinear Elastic Materials for Axisymmetric Elements

Specifies NLELAST option for advanced orthotropic, nonlinear elastic materials at axisymmetric conditions in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATS3	MID	TEX	TETH	TEZ	TNUXTH	TNUTHZ	TNUZX	TRHO	
			TGZX	TAX	TATH	TAZ			

Example:

MATS3	33	56			67				
			12						

Descriptor	Meaning
MID	Identification number of a MAT3 entry. (Integer > 0; no Default)
TEX	ID of TABL3Di entry for EX. (Integer; no Default; leave blank if table is not required)
TETH	ID of TABL3Di entry for ETH. (Integer; no Default; leave blank if table is not required)
TEZ	ID of TABL3Di entry for EZ. (Integer; no Default; leave blank if table is not required)
TNUXTH	ID of TABL3Di entry for NUXTH. (Integer; no Default; leave blank if table is not required)
TNUTHZ	ID of TABL3Di entry for NUTHZ. (Integer; no Default; leave blank if table is not required)
TNUZX	ID of TABL3Di entry for NUZX. (Integer; no Default; leave blank if table is not required)
TRHO	ID of TABL3Di entry for RHO. (Integer; no Default; leave blank if table is not required)
TGZX	ID of TABL3Di entry for GZX. (Integer; no Default; leave blank if table is not required)
TAX	ID of TABL3Di entry for AX. (Integer; no Default; leave blank if table is not required)
TATH	ID of TABL3Di entry for ATH. (Integer; no Default; leave blank if table is not required)
TAZ	ID of TABL3Di entry for AZ. (Integer; no Default; leave blank if table is not required)

Remarks:

1. TABL3Di is a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc.

2. In SOL 400, MATS3 is used in conjunction with MAT3 and is only supported for axisymmetric elements with property extensions. This implies that for such elements, PLPLANE should be associated with PSHLN2. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATS3 data was not considered in the element's formulation. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

MATS8

Advanced Orthotropic, Nonlinear Elastic Material for Shell Elements

Specifies NLELAST option for advanced orthotropic, nonlinear elastic material for plane stress and shell elements in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATS8	MID	TE1	TE2	TNU12	TG12	TG1Z	TG2Z	TRHO	
	TA1	TA2							

Example:

MATS8	34	12							
	1								

Descriptor	Meaning
MID	Identification number of a MAT8 entry. (Integer > 0; no Default)
TE1	ID of TABL3Di entry for E1. (Integer; no Default; leave blank if table is not required)
TE2	ID of TABL3Di entry for E2. (Integer; no Default; leave blank if table is not required)
TNU12	ID of TABL3Di entry for NU12. (Integer; no Default; leave blank if table is not required)
TG12	ID of TABL3Di entry for G12. (Integer; no Default; leave blank if table is not required)
TG1Z	ID of TABL3Di entry for G1Z. (Integer; no Default; leave blank if table is not required)
TG2Z	ID of TABL3Di entry for G2Z. (Integer; no Default; leave blank if table is not required)
TRHO	ID of TABL3Di entry for RHO. (Integer; no Default; leave blank if table is not required)
TA1	ID of TABL3Di entry for A1. (Integer; no Default; leave blank if table is not required)
TA2	ID of TABL3Di entry for A2. (Integer; no Default; leave blank if table is not required)

Remarks:

1. TABL3Di is a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc. As an example, say the analysis required TE1 to be a function of strain. Then the corresponding TABL3D0 entry is TABL3D0, KIND=73 to specify *local strain component* as the independent variable. The dependent variable is *TE1*.
2. In SOL 400, MATS8 is used in conjunction with MAT8 and is only supported for shell elements with property extensions. This implies that for such elements, PSHELL should be associated with PSHLN1. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATS8 data was not considered in the element's formulation. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

MATSMA**Material Properties for Shape Memory Alloys**

For SOL 600 and SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATSMA	MID	MODEL	T_0	L					
	E_a	ν_a	α_a	σ_a	ρ_a	σ_s^A	σ_f^{AS}	C_a	
	E_m	ν_m	α_m	σ_m	ρ_m	σ_s^S	σ_f^{SA}	C_m	
	v^T	M_{frac}	σ_{eff}^g	σ_{max}^g					
	g_0	g_a	g_b	g_c	g_d	g_e	g_f	g_{max}	

Example:

MATSMA	1	2	77.	0.05					
	50000.	0.33	0.00001	1.0E+20		520.	600.	8.66	
	50000.	0.33	0.00001	1.0e+20		300	200.	6.66	
	0.0	0.0	100.	1.0e+20					
	300.	-4.	2.	0.	0.	0.	3.0	1.0	

Descriptor	Meaning
MID	Material ID. (Integer > 0)
MODEL	Flag to indicate the model being used. (Integer > 0; see Remark 1.): = 1 (Mechanical: Aruchchio's model) = 2 (Thermo-Mechanical: Asaro-Sayeedvafa model)
T_0	Reference temperature used to measure stresses. (Real > 0)
L	For the mechanical model, the parameter L represents maximum deformation, obtainable by detwinning of multiple-variant martensite. (Real > 0; for the thermo-mechanical model, see Remark 2.) (typically 0.06-0.104)

Austenite Properties (all Real):

E_a	Young's modulus of elasticity (typically 60-83 GPa).
ν_a	Poisson's ratio (typically 0.33).
α_a	Coefficient of thermal expansion (typically $3.67 \times 10^{-6}/^{\circ}\text{F}$).
σ_a	Equivalent von Mises stress (not used in mechanical model) (typically 195-690 MPa).
ρ_a	Mass density.
σ_s^{AS}	Material parameter representing start of Austenite to Martensite transformation. For thermo-mechanical model, the program calculates the Austenite start temperature in the stress-free configuration (A_s^0) related to σ_s^{AS} from relations shown in Table 9-26.

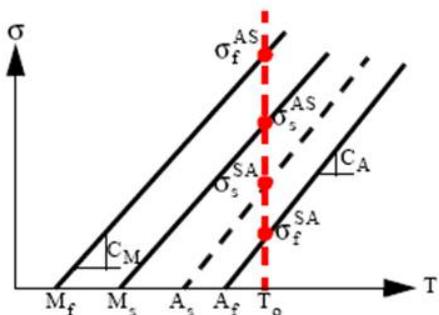
Descriptor	Meaning
σ_f^{AS}	Material parameter representing finish of Austenite to Martensite transformation. For thermo-mechanical model, the program calculates the Austenite finish temperature in the stress-free configuration (A_f^0) related to σ_f^{AS} from relations shown in Table 9-26 .
C_a	Slope of the stress-dependence of austenite start finish and start temperatures (typically 6-8 MPa).
Martensite Properties (all Real):	
E_m	Young's modulus of elasticity
v_m	Poisson's ratio
α_m	Coefficient of thermal expansion
σ_m	Equivalent von Mises stress (not used in mechanical model)
ρ_m	Density
σ_s^{SA}	Material parameter representing start of Martensite to Austenite transformation. For thermo-mechanical model, the program calculates the Martensite start temperature in the stress-free configuration (M_s^0) related to σ_s^{SA} from relations shown in Table 9-26 .
σ_f^{SA}	Material parameter representing finish of Martensite to Austenite transformation. For thermo-mechanical model, the program calculates the Martensite finish temperature in the stress-free configuration (M_f^0) related to σ_f^{SA} from relations shown in Table 9-26 .
C_m	Slope of the stress-dependence of austenite start finish and start temperatures. (typically 5-6 MPa)
The following quantities are applicable to only thermo-mechanical model (all Real):	
v^T	Equivalent volumetric transformation strain. (typically 0.0-0.003)
M_{frac}	Initial martensite volume fraction. (0.0-1.0)
σ_{eff}^g	Twinning stress (see Remark 2.). (typically 100-150 MPa)
σ_{max}^g	Stress at which the maximum value of $g = g_{max}$ is reached if a cut-off value is needed (normalized with g_0).
g_0	Stress level used to nondimensionalizing the stress in the function. (2.0-10.0* σ_{eff}^g)
g_a	g function coefficient (typically $g_a < 0.0$)
g_b	g function exponent (typically $g_b = 2.0$)
g_c	g function coefficient (typically $g_c \geq 0.0$)
g_d	g function exponent (typically $g_d = 2.25 \sim 2.75$)
g_e	g function coefficient (typically $g_e \leq 0.0$)
g_f	g function exponent (typically $g_f = 3.0$)
g_{max}	Maximum value of function g if a cut-off value is needed (typically $g_{max} = 1.0$).

Remarks:

1. The mechanical (Auricchio's) model can be obtained from the thermo-mechanical model by ignoring the last two rows in the input.
2. Twinning becomes active when the equivalent stress reaches twinning stress. For thermo-mechanical model, the “unstressed transformation temperatures” for Martensite and Austenite, M_s^0 , M_f^0 , A_s^0 , A_f^0 are calculated from the reference temperature, the material parameters representing the start and finish of the Austenite and Martensite transformations, i.e., σ_s^{SA} , σ_f^{SA} , σ_s^{AS} , and σ_f^{AS} as well as the coefficients, C_m , C_a that provide the stress dependence of the transformation temperatures as shown in [Table 9-26](#).

Table 9-26

The Relationship between Mechanical Model and Thermo-Mechanical Model
$\sigma_s^{AS} = (T_o - M_s^0)C_m$
$\sigma_f^{AS} = (T_o - M_f^0)C_m$
$\sigma_s^{SA} = (T_o - A_s^0)C_a$
$\sigma_f^{SA} = (T_o - A_f^0)C_a$

**Figure 9-113**

3. For the thermo-mechanical model, the equivalent deviatoric strain, (typically 0.05 - 0.085) is automatically calculated by the program as $eq^T = \sqrt{2/3} * L$ (since it is assumed that the input to this model is an extension of the mechanical model and conversion is done wherever applicable as in [Table 9-26](#). However, in the case where the thermal-mechanical model input parameters are directly used then one must enter a value of $\sqrt{2/3} * eq$ so that a correct value of eq is used in the calculations.

4. The Mechanical (Auricchio's) model is not available for 1-D or plane stress conditions. The thermo-mechanical model must be used.
5. In SOL 400, MATSMA is only supported for nonlinear elements with property extensions, PBARN1, PBEMN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MSC Nastran 2010, if the property extensions were missing, then the analysis would stop with an error. From MSC Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
6. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.
7. In SOL 400; remark 4. implies that the Mechanical model may not be used with PBAR, PBARL, PBEAM, PBEAML, PROD, PSHEAR, PSHELL, PCOMP(G). It can only be used with PLPLANE/PSHLN2 (without BEH=PSTRS) and PSOLID/PSLDN1.

MATSORT

Advanced Orthotropic, Nonlinear Elastic Material for Shell Elements

Specifies NLELAST option for advanced 3D orthotropic, nonlinear elastic materials in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATSORT	MID	TE1	TE2	TE3	TNU12	TNU23	TNU31	TRHO	
	TG12	TG23	TG31	TA1	TA2	TA3			

Example:

MATSORT	689	77	77	77					
				89	89	89			

Descriptor	Meaning
MID	Identification number of a MATORT entry. (Integer > 0; no Default)
TE1	ID of TABL3Di entry for E1. (Integer; no Default; leave blank if table is not required)
TE2	ID of TABL3Di entry for E2. (Integer; no Default; leave blank if table is not required)
TE3	ID of TABL3Di entry for E3. (Integer; no Default; leave blank if table is not required)
TNU12	ID of TABL3Di entry for NU12. (Integer; no Default; leave blank if table is not required)
TNU23	ID of TABL3Di entry for NU23. (Integer; no Default; leave blank if table is not required)
TNU31	ID of TABL3Di entry for NU31. (Integer; no Default; leave blank if table is not required)
TRHO	ID of TABL3Di entry for RHO. (Integer; no Default; leave blank if table is not required)
TG12	ID of TABL3Di entry for G12. (Integer; no Default; leave blank if table is not required)
TG23	ID of TABL3Di entry for G23. (Integer; no Default; leave blank if table is not required)
TG31	ID of TABL3Di entry for G31. (Integer; no Default; leave blank if table is not required)
TA1	ID of TABL3Di entry for A1. (Integer; no Default; leave blank if table is not required)
TA2	ID of TABL3Di entry for A2. (Integer; no Default; leave blank if table is not required)
TA3	ID of TABL3Di entry for A3. (Integer; no Default; leave blank if table is not required)

Remarks:

1. TABL3Di is a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc. If one of the independent variables is temperatures then you should not use MATTORT.

2. In SOL 400, MATSORT is used in conjunction with MATORT and is only supported for 2D and 3D elements with property extensions. This implies that for such elements, PCOMP / PCOMPG, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PSHLN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the analysis would stop with an error. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with default settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

MATT1**Isotropic Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT1 entry fields via TABLEMi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT1	MID	T(E)	T(G)	T(NU)	T(RHO)	T(A)		T(GE)	
	T(ST)	T(SC)	T(SS)						

Example:

MATT1	17	32	65	64		15			
	52								

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)
T(E)	Identification number of a TABLEMi entry for the Young's modulus. (Integer ≥ 0 or blank)
T(G)	Identification number of a TABLEMi entry for the shear modulus. (Integer ≥ 0 or blank)
T(NU)	Identification number of a TABLEMi entry for the Poisson's ratio. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for the mass density. (Integer ≥ 0 or blank)
T(A)	Identification number of a TABLEMi entry for the thermal expansion coefficient. See Remark 4. (Positive or negative integer, 0 or blank)
T(GE)	Identification number of a TABLEMi entry for the damping coefficient. (Integer ≥ 0 or blank)
T(ST)	Identification number of a TABLEMi entry for the tension stress limit. (Integer ≥ 0 or blank)
T(SC)	Identification number of a TABLEMi entry for the compression limit. (Integer ≥ 0 or blank)
T(SS)	Identification number of a TABLEMi entry for the shear limit. (Integer ≥ 0 or blank)

Remarks:

1. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT1 entry referenced in field 2. The value in a particular field of the MAT1 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E is modified by TABLEMi 32, G is modified by TABLEMi 65, NU is modified by TABLEMi 84, A is modified by TABLEMi 15, and ST is modified by TABLEMi 52. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT1 entry.
2. Any quantity modified by this entry must have a value on the MAT1 entry. Initial values of E, G, or NU may be supplied according to Remark 3 on the MAT1 entry.
3. Table references must be present for each item that is temperature dependent. If any one of Young's modulus, shear modulus or Poisson's ratio is temperature dependent, then T(E), T(G) and T(NU) should be specified together to get accurate results.
4. The T(A) table value can be either a positive or negative integer. For a nonlinear static analysis of a composite element with the PARAM,COMPMATT,YES, if the TABLEMi ID for the coefficient of thermal expansion is negative, the TABLEMi ordinate values will be interpreted as the thermal strain $\varepsilon(T)$ rather than the coefficient of thermal expansion ordinate values $\alpha(T)$. Internally to Nastran, a negative ID_{T(A)} value will be changed to |ID_{T(A)}| + 100000000 .
5. The continuation entry is not used by SOL 600. For SOL 600, see MATTEP.
6. T(G), T(RHO) and T(GE) are not used by SOL 600. T(E) and T(NU) should be used instead of T(E) and T(G).
7. This entry is not used by SOL 700.

MATT2**Anisotropic Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT2 entry fields via TABLEMj entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT2	MID	T(G11)	T(G12)	T(G13)	T(G22)	T(G23)	T(G33)	T(RHO)	
	T(A1)	T(A2)	T(A3)		T(GE)	T(ST)	T(SC)	T(SS)	
		T(GE11)	T(GE12)	T(GE13)	T(GE22)	T(GE23)	T(GE33)		

Example:

MATT2	17	32					15		
		62							

Descriptor	Meaning
MID	Material property identification number that matches the identification number on a MAT2 entry. (Integer > 0)
T(Gij)	Identification number of a TABLEMk entry for the terms in the material property matrix. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMk entry for the mass density. (Integer ≥ 0 or blank)
T(Ai)	Identification number of a TABLEMk entry for the thermal expansion coefficient. See Remark 3. (Positive or negative integer, 0 or blank)
T(GE)	Identification number of a TABLEMk entry for the damping coefficient. (Integer ≥ 0 or blank)
T(ST)	Identification number of a TABLEMk entry for the tension stress limit. (Integer ≥ 0 or blank)
T(SC)	Identification number of a TABLEMk entry for the tension compression limit. (Integer ≥ 0 or blank)
T(SS)	Identification number of a TABLEMk entry for the tension shear limit. (Integer ≥ 0 or blank)
T (GEij)	Identification number of a TABLEMk entry for the terms in the material structural damping property matrix. (Integer > 0 or blank)

Remarks:

- Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT2 entry referenced in field 2. The value in a particular field of the MAT2 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, G11 is modified by TABLEMk 32, G33 is modified by TABLEMk 15, and A1 is modified by TABLEMk 62. If Ri is zero or blank, then there is no temperature dependence of the field on the MAT2 entry.

2. Any quantity modified by this entry must have a value on the MAT2 entry.
3. Any of the T(A) table value can be either a positive or negative integer. For a nonlinear static analysis of a composite element with the PARAM,COMPMATT,YES, if the TABLEMk ID for the coefficient of thermal expansion is negative, the TABLEMk ordinate values will be interpreted as the thermal strain $\varepsilon(T)$ rather than the coefficient of thermal expansion ordinate values $\alpha(T)$. Internally to Nastran, a negative ID_{T(A)} value will be changed to |ID_{T(A)}| + 100000000

MATT3**MAT3 Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT3 entry fields via TABLEMi entries that are temperature dependent.

Format:

1	2	3	4	5	6	7	8	9	10
MATT3	MID	T(EX)	T(ETH)	T(EZ)	T(NUXTH)	T(NUTHZ)	T(NUZX)	T(RHO)	
				T(GZX)	T(AX)	T(ATH)	T(AZ)		T(GE)

Example:

MATT3	23	32		15					
				62					

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT3 entry. (Integer > 0)
T(EX)	Identification number of a TABLEMi entry for the Young's modulus in the x , θ , and z directions. (Integer ≥ 0 or blank)
T(ETH)	Identification number of a TABLEMi entry for the thermal expansion coefficients in the x , θ , and z directions. (Integer ≥ 0 or blank)
T(EZ)	Identification number of a TABLEMi entry for the Poisson's ratio in the $x\theta$, θz , and zx directions. (Integer ≥ 0 or blank)
T(GZX)	Identification number of a TABLEMi entry for the shear modulus. (Integer ≥ 0 or blank)
T(NUXTH)	Identification number of a TABLEMi entry for the mass density. (Integer ≥ 0 or blank)
T(NUTHZ)	Identification number of a TABLEMi entry for the damping coefficient. (Integer ≥ 0 or blank)
T(NUZX)	Identification number of a TABLEMi entry for the thermal expansion coefficients in the x , θ , and z directions. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for the thermal expansion coefficients in the x , θ , and z directions. (Integer ≥ 0 or blank)
T(AX)	Identification number of a TABLEMi entry for the Poisson's ratio in the $x\theta$, θz , and zx directions. (Integer ≥ 0 or blank)
T(ATH)	Identification number of a TABLEMi entry for the shear modulus. (Integer ≥ 0 or blank)
T(AZ)	Identification number of a TABLEMi entry for the shear modulus. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMi entry for the damping coefficient. (Integer ≥ 0 or blank)

Remarks:

- Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT3 entry referenced in field 2. The value recorded in a particular field of the MAT3 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, EX is modified by TABLEMi 32, EZ is modified by TABLEMi 15, and GZX is modified by TABLEMi 62. If Ri is zero or blank, then there is no temperature dependence of the field on the MAT3 entry.
- Any quantity modified by this entry must have a value on the MAT3 entry.

MATT4**Thermal Material Temperature Dependence**

Specifies table references for temperature-dependent MAT4 material properties.

Format:

1	2	3	4	5	6	7	8	9	10
MATT4	MID	T(K)	T(CP)		T(H)	T(μ)	T(HGEN)		

Example:

MATT4	17	102	103			105			
-------	----	-----	-----	--	--	-----	--	--	--

Descriptor	Meaning
MID	Identification number of a MAT4 entry that is temperature dependent. (Integer > 0)
T(K)	Identification number of a TABLEMj entry that gives the temperature dependence of the thermal conductivity. (Integer ≥ 0 or blank)
T(CP)	Identification number of a TABLEMj entry that gives the temperature dependence of the thermal heat capacity. (Integer ≥ 0 or blank)
T(H)	Identification number of a TABLEMj entry that gives the temperature dependence of the free convection heat transfer coefficient. (Integer ≥ 0 or blank)
T(μ)	Identification number of a TABLEMj entry that gives the temperature dependence of the dynamic viscosity. (Integer ≥ 0 or blank)
T(HGEN)	Identification number of a TABLEMj entry that gives the temperature dependence of the internal heat generation property for QVOL. (Integer ≥ 0 or blank)

Remarks:

1. The basic quantities on the MAT4 entry are always multiplied by the corresponding tabular function referenced by the MATT4 entry.
2. If the fields are blank or zero, then there is no temperature dependence of the referenced quantity on the MAT4 entry.

MATT5**Thermal Anisotropic Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT5 entry fields via TABLEMi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT5	MID	T(KXX)	T(KXY)	T(KXZ)	T(KYY)	T(KYZ)	T(KZZ)	T(CP)	
		T(HGEN)							

Example:

MATT5	24	73							
-------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning
MID	Identification number of a MAT5 entry that is to be temperature dependent. (Integer > 0)
T(Kij)	Identification number of a TABLEMi entry. The TABLEMi entry specifies temperature dependence of the matrix term. (Integer ≥ 0 or blank)
T(CP)	Identification number of a TABLEMi entry that specifies the temperature dependence of the thermal heat capacity. (Integer ≥ 0 or blank)
T(HGEN)	Identification number of a TABLEMi entry that gives the temperature dependence of the internal heat generation property for the QVOL entry. (Integer ≥ 0 or blank)

Remarks:

1. The basic quantities on the MAT5 entry are always multiplied by the tabular function referenced by the MATT5 entry.
2. If the fields are blank or zero, then there is no temperature dependence of the referenced quantity on the basic MAT5 entry.

MATT8**Shell Orthotropic Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT8 entry fields via TABLEMi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT8	MID	T(E1)	T(E2)	T(NU12)	T(G12)	T(G1Z)	T(G2Z)	T(RHO)	
	T(A1)	T(A2)		T(Xt)	T(Xc)	T(Yt)	T(Yc)	T(S)	
	T(GE)	T(F12)							

Example:

MATT8	17	32							
	15			52					

Descriptor	Meaning
MAT	Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)
T(E1)	Identification number of a TABLEMi entry for the Young's modulus 1. (Integer ≥ 0 or blank)
T(E2)	Identification number of a TABLEMi entry for the Young's modulus 2. (Integer ≥ 0 or blank)
T(NU12)	Identification number of a TABLEMi entry for Poisson's ratio 12. (Integer ≥ 0 or blank)
T(G12)	Identification number of a TABLEMi entry for shear modulus 12. (Integer ≥ 0 or blank)
T(G1Z)	Identification number of a TABLEMi entry for transverse shear modulus 1Z. (Integer ≥ 0 or blank)
T(G2Z)	Identification number of a TABLEMi entry for transverse shear modulus 2Z. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for mass density. (Integer ≥ 0 or blank)
T(Ai)	Identification number of a TABLEMi entry for thermal expansion coefficient 1. See Remark 3. (Positive or negative integer, 0 or blank)
T(A2)	Identification number of a TABLEMi entry for thermal expansion coefficient 2. See Remark 3. (Integer or blank)
T(Xt)	Identification number of a TABLEMi entry for tension stress/strain limit 1. (Integer ≥ 0 or blank)
T(Xc)	Identification number of a TABLEMi entry for compression stress/strain limit 1. (Integer ≥ 0 or blank)

Descriptor	Meaning
T(Yt)	Identification number of a TABLEMi entry for tension stress/strain limit 2. (Integer ≥ 0 or blank)
T(Yc)	Identification number of a TABLEMi entry for compression stress/strain limit 2. (Integer ≥ 0 or blank)
T(S)	Identification number of a TABLEMi entry for shear stress/strain limit. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMi entry for structural damping coefficient. (Integer ≥ 0 or blank)
T(F12)	Identification number of a TABLEMi entry for Tsai-Wu interaction term. (Integer ≥ 0 or blank)

Remarks:

1. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT8 entry referenced in field 2. The value in a particular field of the MAT8 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E1 is modified by TABLEMi 32, A1 is modified by TABLEMi 15, and Xt is modified by TABLEMi 52. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT8 entry.
2. Any quantity modified by this entry must have a value on the MAT8 entry.
3. Any of the T(Ai) table value can be either a positive or negative integer. For a nonlinear static analysis of a composite element with the PARAM,COMPMATT,YES, if the TABLEMi ID for the coefficient of thermal expansion is negative, the TABLEMi ordinate values will be interpreted as the thermal strain $\varepsilon(T)$ rather than the coefficient of thermal expansion ordinate values $\alpha(T)$. Internally to Nastran, a negative ID_{T(A)} value will be changed to |ID_{T(A)}| + 100000000.

MATT9**Solid Element Anisotropic Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT9 entry fields via TABLEMk entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT9	MID	T(G11)	T(G12)	T(G13)	T(G14)	T(G15)	T(G16)	T(G22)	
		T(G23)	T(G24)	T(G25)	T(G26)	T(G33)	T(G34)	T(G35)	T(G36)
		T(G44)	T(G45)	T(G46)	T(G55)	T(G56)	T(G66)	T(RHO)	T(A1)
		T(A2)	T(A3)	T(A4)	T(A5)	T(A6)		T(GE)	
		T(GE11)	T(GE12)	T(GE13)	T(GE14)	T(GE15)	T(GE16)	T(GE22)	T(GE23)
		T(GE24)	T(GE25)	T(GE26)	T(GE33)	T(GE34)	T(GE35)	T(GE36)	T(GE44)
		T(GE45)	T(GE46)	T(GE55)	T(GE56)	T(GE66)			

Example:

MATT9	17	32			18			17	
				12					
					5			10	

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT9 entry. (Integer > 0)
T(Gij)	Identification number of a TABLEMk entry for the terms in the material property matrix. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMk entry for the mass density. (Integer ≥ 0 or blank)
T(Ai)	Identification number of a TABLEMk entry for the thermal expansion coefficients. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMk entry for the damping coefficient. (Integer ≥ 0 or blank)
T(GEij)	Identification number of a TABLEMk entry for the terms in the material structural damping property matrix. (Integer > 0 or blank).

Remarks:

- Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT9 entry referenced in field 2. The value recorded in a particular field of the MAT9 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, G11 is modified by TABLEMj 32, G14 is modified by TABLEMj 18, etc. If the fields are zero or blank, then there is no temperature dependence of the fields on the MAT9 entry.
- Any quantity modified by this entry must have a value on the MAT9 entry.
- The continuation entries are optional.

MATTEP

Thermo-Elastic-Plastic Material Properties - SOL 400

Specifies temperature-dependent elasto-plastic material properties to be used for static, quasi static or transient dynamic analysis in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTEP	MID		T(Y0)	T(FID)				T(H)	
	N/A			N/A					
	"Chaboche"	R0	Rinf	B	C	Gam	Kap	N	
		Qm	μ	η					

Example:

MATTEP	100		20						
--------	-----	--	----	--	--	--	--	--	--

Descriptor	Meaning
MID	Identification number of MATEP entry. See Remark 1. (Integer > 0)
T(Y0)	Identification number of TABLEMi entry for thermo-elasto-plastic material. See Remarks 2. (Integer > 0 or blank)
T(FID)	Identification number of TABLEST entry for temperature-dependent stress-strain curves (Integer > 0 or blank). See Remark 4.
T(H)	Identification number of TABLEMi entry for temperature-dependent plasticity moduli in thermo-elasto-plastic material. See Remarks 3. (Integer > 0 or blank)
"Chaboche"	A keyword specifying the following data pertains to the Chaboche model.
R0	Identification number of TABLEMi entry for temperature dependent R0 for isotropic hardening (Integer > 0).
Rinf	Identification number of TABLEMi entry for temperature dependent Rinfinity for isotropic hardening (Integer > 0).
B	Identification number of TABLEMi entry for temperature dependent coefficient b for isotropic hardening (Integer > 0).
C	Identification number of TABLEMi entry for temperature dependent coefficient C for kinematic hardening (Integer > 0).
Kap	Identification number of TABLEMi entry for temperature dependent Kappa value for viscosity model (Integer > 0).
N	Identification number of TABLEMi entry for temperature dependent coefficient n for viscosity model (Integer).
Qm	Identification number of TABLEMi entry for temperature dependent coefficient Qm for isotropic hardening (Integer > 0)

Descriptor	Meaning
μ	Identification number of TABLEMi entry for temperature dependent coefficient μ for isotropic hardening (Integer > 0).
η	Identification number of TABLEMi entry for temperature dependent coefficient η to introduce progressive memory (Integer > 0).

Remarks:

1. The MATEP Bulk Data entry with the same MID must exist for MATTEP to be effective. All the fields defined in MATTEP correspond to the same fields of MATEP. The value in a particular field of the MATEP entry is replaced or modified by the table referenced in the corresponding field of this entry.
2. The table represents yield stresses as a function of temperature. Therefore, the curve should comprise the initial stress from Y0 or FID field on MATEP (most likely at room temperature). T(Y0) field accommodates FID field in case FID field defines the initial yield stress instead of Y0 field. In this case, the yield stresses at any plastic strain will be scaled by the same ratio as the initial yield stress at the same temperature.
3. The table represents a normalized plasticity moduli (work hardening slope) as a function of temperature.
4. Temperature dependent stress-strain curves may be entered in a general manner using the T(FID) option. The integer value entered in this field represents the ID of a TABLEST entry which provides IDs of TABLES1 stress-plastic strain curves vs. temperature. All such curves must be entered as stress vs. plastic strain. No curves should be referenced on the MATS1 entry. For this option T(Y0) and T(H) should be left blank and if entered, SOL 400 will re-set them to blank if T(FID) is a positive integer.
5. This entry must be used in conjunction with MAT1, MATEP and MATT1 all with the same MID. The MATT1 entry must have at least one non-blank entry in fields 3-7 of the primary MATT1 entry.

MATTEP**Thermo-Elastic-Plastic Material Properties - SOL 600**

Specifies temperature-dependent elasto-plastic material properties to be used for static, quasi static or transient dynamic analysis in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTEP	MID		T(Y0)	T(FID)				T(H)	
	N/A		T(yc10)	N/A					
	"Chaboche"	R0	Rinf	B	C	Gam	Kap	N	
		Qm	μ	η					
	"PwrLaw"	A	M	B	N	$\sigma_0 \varepsilon_0$			

Example:

MATTEP	100		20						
--------	-----	--	----	--	--	--	--	--	--

Descriptor	Meaning
MID	Identification number of MATEP entry. See Remark 1. (Integer > 0)
T(Y0)	Identification number of TABLEMi entry for thermo-elasto-plastic material. See Remarks 2. (Integer > 0 or blank)
T(FID)	Identification number of TABLEST entry for temperature-dependent stress-strain curves (Integer > 0 or blank). See Remark 4.
T(H)	Identification number of TABLEMi entry for temperature-dependent plasticity moduli in thermo-elasto-plastic material. See Remarks 3. (Integer > 0 or blank)
T(yc10)	Identification number of TABLEMi entry for equivalent 10th cycle tensile yield stress specified in the Yc10 field of MATEP entry. (Integer > 0 or blank).
"Chaboche"	A keyword specifying the following data pertains to the Chaboche model.
R0	Table for R0 for isotropic hardening. (Integer > 0)
Rinf	Table for Rinfinity for isotropic hardening (Integer > 0)
B	Table for b coefficient for isotropic hardening. (Integer > 0)
C	Table for C coefficient for kinematic hardening. (Integer > 0)
Kap	Table for Kappa value for viscosity model. (Integer > 0)
N	Table for n coefficient for viscosity model. (Integer > 0)
Qm	Table for Qm coefficient for isotropic hardening. (Integer > 0)
μ	Table for μ coefficient for isotropic hardening. (Integer > 0)
η	Table for η coefficient to introduce progressive memory. (Integer > 0)

Descriptor	Meaning
"PwrLaw"	A keyword specifying the following data pertains to the Power Law or Rate Power Law model (see Marc Vol C , ISOTROPIC option for more details.
A	Table for coefficient A. (Integer > 0)
M	Table for coefficient m. (Integer > 0)
B	Table for coefficient B. (Integer > 0)
N	Table for exponent n. (Integer > 0)
$\sigma_0\varepsilon_0$	Table for $\sigma_0\varepsilon_0$. (Integer > 0)

Remarks:

1. The MATEP Bulk Data entry with the same MID must exist for MATTEP to be effective. All the fields defined in MATTEP correspond to the same fields of MATEP. The value in a particular field of the MATEP entry is replaced or modified by the table referenced in the corresponding field of this entry.
2. The table represents yield stresses as a function of temperature. Therefore, the curve should comprise the initial stress from Y0 or FID field on MATEP (most likely at room temperature). T(Y0) field accommodates FID field in case FID field defines the initial yield stress instead of Y0 field. In this case, the yield stresses at any plastic strain will be scaled by the same ratio as the initial yield stress at the same temperature.
3. The table represents a normalized plasticity moduli (work hardening slope) as a function of temperature.
4. Temperature dependent stress-strain curves may be entered in a general manner using the T(FID) option. The integer value entered in this field represents the ID of a TABLEST entry which provides IDs of TABLES1 stress-plastic strain curves vs. temperature. All such curves must be entered as stress vs. plastic strain. No curves should be referenced on the MATS1 entry. For this option T(Y0) and T(H) should be left blank and if entered, MSC Nastran will re-set them to blank if T(FID) is a positive integer.
5. This entry must be used in conjunction with MAT1, MATEP and MATT1 all with the same MID. The MATT1 entry must have at least one non-blank entry in fields 3-7 of the primary MATT1 entry.

Note:

This Bulk Data entry accommodates Marc's input data under the model definition options TEMPERATURE EFFECTS.

MATTF**Material Failure Model Temperature Variation**

Describes the temperature, strain rate, or other type of variation of material failure properties used in conjunction with MATF in SOL 600 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATTF	MID		T(SB)						
	KIND	Criteria	T(Xt)	T(Xc)	T(Yt)	T(Yc)	T(Zt)	T(Zc)	1st
	T(Sxy)	T(Syz)	T(Szx)	T(Find)	T(Fxy)	T(Fyz)	T(Fzx)	T(Ext)	
	T(Exc)	T(Eyt)	T(Eyc)	T(Ezt)	T(Ezc)	T(Gxy)	T(Gyz)	T(Gzx)	
	KIND	Criteria	T(Xt)	T(Xc)	T(Yt)	T(Yc)	T(Zt)	T(Zc)	2nd
	T(Sxy)	T(Syz)	T(Szx)	F(Find)	F(Fxy)	T(Fyz)	T(Fzx)	T(Ext)	
	T(Exc)	T(Eyt)	T(Eyc)	T(Ezt)	T(Ezc)	T(Gxy)	T(Gyz)	T(Gzx)	
	KIND	Criteria	T(Xt)	T(Xc)	T(Yt)	T(Yc)	T(Zt)	T(Zc)	3rd
	T(Sxy)	T(Syz)	T(Szx)	T(Find)	T(Fxy)	T(Fyz)	T(Fzx)	T(Ext)	
	T(Exc)	T(Eyt)	T(Eyc)	T(Ezt)	T(Ezc)	T(Gxy)	T(Gyz)	T(Gzx)	

Example:

MATTF	100								
	12	1	51	52	53	54	55	56	
	61	62	63						
	12	2							2nd
	71	72	73	74	75	76	77	78	
	12	4	81	82	83	84	85	86	3rd
	91	92	93	94					

(Note: The 4th and 6th lines cannot be entirely blank and the last line of the 3rd criterion has been omitted.)

Descriptor	Meaning
MID	Identification number of a matching MATF entry. (Integer > 0; no Default; Required)
KIND	Enter the type of variation of the failure properties using the values listed below. (Integer; no Default) 12 Temperature

Descriptor	Meaning
Criteria	Enter same value as used by MATTF (reference only). (Integer) If more than one KIND is required, enter the three lines for each KIND and Criteria as many times are required. (Integer; must be same as used by companion MATTF, reference only)
T(SB)	Identification of a TABLEMi entry providing the variation of the allowable shear stress of the bonding material between layers (composites only) (Integer; Default = 0 meaning no variation)
T(Xt)	Identification of a TABLEMi entry providing the variation of the maximum tensile stress in x-direction (Integer; Default = 0 meaning no variation)
T(Xc)	Identification of a TABLEMi entry providing the variation of the maximum compressive stress (absolute value) in x-direction (Integer; Default = 0 meaning no variation)
T(Yt)	Identification of a TABLEMi entry providing the variation of the maximum tensile stress in y-direction (Integer; Default = 0 meaning no variation)
T(Yc)	Identification of a TABLEMi entry providing the variation of the maximum compressive stress (absolute value) in y-direction (Integer or blank)
T(Zt)	Identification of a TABLEMi entry providing the variation of the maximum tensile stress in z-direction (Integer; Default = 0 meaning no variation)
T(Zc)	Identification of a TABLEMi entry providing the variation of the maximum compressive stress (absolute value) in z-direction (Integer; Default = 0 meaning no variation)
T(Sxy)	Identification of a TABLEMi entry providing the variation of the maximum shear stress in xy-plane (Integer; Default = 0 meaning no variation)
T(Syz)	Identification of a TABLEMi entry providing the variation of the maximum shear stress in yz-plane (Integer; Default = 0 meaning no variation)
T(Szx)	Identification of a TABLEMi entry providing the variation of the maximum shear stress in zx-plane (Integer; Default = 0 meaning no variation)
T(Find)	Identification of a TABLEMi entry providing the variation of the Failure index (Real > 0., (Integer; Default = 0 meaning no variation)
T(Fxy)	Identification of a TABLEMi entry providing the variation of the interactive strength constant for xy-plane (Integer; Default = 0 meaning no variation)
T(Fyz)	Identification of a TABLEMi entry providing the variation of the interactive strength constant for yz-plane (Integer; Default = 0 meaning no variation)
T(Fzx)	Identification of a TABLEMi entry providing the variation of the interactive strength constant for zx-plane (Integer; Default = 0 meaning no variation)
T(Ext)	Identification of a TABLEMi entry providing the variation of the maximum tensile strain in x-direction (Integer; Default = 0 meaning no variation)

Descriptor	Meaning
T(Exc)	Identification of a TABLEMi entry providing the variation of the maximum compressive strain (absolute value) in x-direction (Integer; Default = 0 meaning no variation)
T(Yet)	Identification of a TABLEMi entry providing the variation of the maximum tensile strain in y-direction (Integer; Default = 0 meaning no variation)
T(Eyc)	Identification of a TABLEMi entry providing the variation of the maximum compressive strain (absolute value) in y-direction (Integer; Default = 0 meaning no variation)
T(Ezt)	Identification of a TABLEMi entry providing the variation of the maximum tensile strain in z-direction (Integer; Default = 0 meaning no variation)
T(Ezc)	Identification of a TABLEMi entry providing the variation of the maximum compressive strain (absolute value) in z-direction (Integer; Default = 0 meaning no variation)
T(Gxy)	Identification of a TABLEMi entry providing the variation of the maximum shear strain in xy-plane (Integer; Default = 0 meaning no variation)
T(Gyz)	Identification of a TABLEMi entry providing the variation of the maximum shear strain in yz-plane (Integer; Default = 0 meaning no variation)
T(Gzx)	Identification of a TABLEMi entry providing the variation of the maximum shear strain in zx-plane (Integer; Default = 0 meaning no variation)

Remarks:

1. See the MATTF entry for a complete description of the meaning of each of the variables for the various failure criterion.
2. If there is no variation of a particular entry, leave the field blank or enter zero.
3. Continuation entries except the last line are required. The last line is optional.

MATTG

Temperature Variation of Interlaminar Materials

Specifies gasket material property temperature variation to be used in SOL 600 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATTG	MID	IDYM	IDVM	IDDM	IDLD	IDU1	IDU2	IDU3	
	IDU4	IDU5	IDU6	IDU7	IDU8	IDU9	IDU10	IDYPR	
	IDEPL	IDGPL	IDGAP	N/A	N/A	N/A	N/A	N/A	

Example:

MATTG	100	10	20	1001	1002	1003			
								1010	
	1020	1030							

Descriptor	Meaning
MID	Material ID number that matches the material ID of a corresponding MATG material. (Integer > 0; Required)
IDYM	ID of TABLEMi entry that gives the temperature variation of Young's modulus for the membrane behavior of the material. (Integer ≥ 0 or blank)
IDVM	ID of TABLEMi entry that gives the temperature variation of Poisson's ratio for the membrane behavior of the material. (Integer ≥ 0 or blank)
IDDM	ID of TABLEMi entry that gives the temperature variation of the mass density for the membrane behavior of the material. (Integer ≥ 0 or blank)
IDLD	ID of TABLEMi entry that gives the temperature variation of the loading curve of the material. (Integer ≥ 0 or blank)
IDUi	ID of TABLEMi entry that gives the temperature variation of the unloading curve of the material. There can be up to 10 unloading curves and each can have a different temperature variation. If there is no unloading, there does not need to be any unloading curves. (Integer ≥ 0 or blank)
IDYPR	ID of TABLEMi entry that gives the temperature variation of the yield pressure for the out-of -plane behavior of the material. (Integer ≥ 0 or blank)
IDEPL	ID of TABLEMi entry that gives the temperature variation of the tensile modulus for the out-of -plane behavior of the material. (Integer ≥ 0 or blank)
IDGPL	ID of TABLEMi entry that gives the temperature variation of the transverse shear modulus for the out-of -plane behavior of the material. (Integer ≥ 0 or blank)
IDGAP	ID of TABLEMi entry that gives the temperature variation of the initial gap for the out- -plane behavior of the material. (Integer ≥ 0 or blank)

MATTHE**Hyperelastic Material - Temperature Dependence**

Specifies temperature-dependent properties of hyperelastic (rubber-like) materials (elastomers) for nonlinear (large strain and large rotation) analysis in SOLs 106, 129, 600 and SOL 400 only.

Format 1 (Default): Generalized Mooney-Rivlin model (Model=Mooney). The first two lines are required, the others may be omitted depending on how many terms are desired.)

1	2	3	4	5	6	7	8	9	10
MATTHE	MID	Model		T(K)	T(RHO)	T(Texp)		T(GE)	
	T(C10)	T(C01)	T(D1)						
	T(C20)	T(C11)	T(C02)	T(D2)					
	T(C30)	T(C21)	T(C12)	T(C03)	T(D3)				
	T(C40)	T(C31)	T(C22)	T(C13)	T(C04)	T(D4)			
	T(C50)	T(C41)	T(C32)	T(C23)	T(C14)	T(C05)	T(D5)		

Format 2: Ogden Model

1	2	3	4	5	6	7	8	9	10
MATTHE	MID	Model		T(K)	T(RHO)	T(Texp)		T(GE)	
	T(D1)	T(D2)	T(D3)	T(D4)	T(D5)				

Format 3: Arruda-Boyce model or Gent Model (Model = Aboyce or Gent)

1	2	3	4	5	6	7	8	9	10
MATTHE	MID	Model		T(K)	T(RHO)	T(Texp)		T(GE)	
	T(NKT)	T(N/E)	T(Im)						
	T(D1)	T(D2)	T(D3)	T(D4)	T(D5)				

Example - Format 1:

MATTHE	1	Monney							
	10	1	100						
	20	11	2	200					
	30	21	12	3	300				
	40	31	22	13	4	400			
	50	41	32	23	14	5	500		

Example - Format 2:

MATTHE	1	Ogden							
	100	200	300	400	500				

Example - Format 3:

MATTHE	1	Aboyce							
	1	2							
	100	200	300	400	500				

Descriptor	Meaning								
MID	Identification number of a MATHE entry. See Remark 1. (Integer > 0; no Default)								
Model	Select hyperelastic material model form. (Character; Default = Mooney)								
	Mooney	For generalized Mooney-Rivlin hyperelastic model							
	Ogden	For Ogden hyperelastic model							
	Foam	For Foam model							
	Aboyce	For Arruda-Boyce strain energy model							
	Gent	For Gent strain energy model							
T(Value)	Identification number of a TABLEMi entry for the temperature variation of the matching value from the MATHE entry - see the MATHE entry for the definition of each coefficient value. (Integer; Default = 0 which means no table variation for that particular value)								

Remarks:

1. The MATTHE entry must have the same ID as the corresponding MATHE entry. Each table ID on the MATTHE entry corresponds to a parameter on the MATHE entry.
2. If the thermal expansion coefficient is temperature-dependent, the thermal strain is computed as

$$\varepsilon_{\text{th}} = \bar{\alpha}(T - T_{\text{ref}}) - \bar{\alpha}_0(T_0 - T_{\text{ref}})$$

where T_{ref} is the reference temperature at which measurement of the thermal expansion coefficient is based, T_0 is an initial temperature, and $\bar{\alpha}_0$ is the thermal expansion coefficient at the initial temperature T_0 . If the thermal expansion is not temperature-dependent, the thermal strain expression is reduced to the usual expression:

$$\varepsilon_{\text{th}} = \bar{\alpha}(T - T_0)$$

3. If experimental data is provided, it is expected that the user has the data for multiple tests of the same type at different temperatures. The use of TABLEST's for this entry is not presently available. If experimental data is used to define the constants, use a GUI to define the constants at each temperature, enter the baseline values on the MATHE entry and the variation of each coefficient using TABLEMi.

MATTORT

Thermoelastic Orthotropic Material

Specifies temperature-dependent properties of elastic orthotropic materials for linear and nonlinear analyses used in SOL 600 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATTORT	MID	T(E1)	T(E2)	T(E3)	T(NU12)	T(NU23)	T(NU31)	N/A	
	T(G12)	T(G23)	T(G31)	T(A1)	T(A2)	T(A3)	T(SY)	T(WHS)	

Example:

MATTORT	100	5	6	7					
---------	-----	---	---	---	--	--	--	--	--

Descriptor	Meaning
MID	Identification number of a MATORT entry. See Remark 1. (Integer > 0; no Default)
T(Ei)	Identification number of a TABLEMi entry for the Young's modulus as a function of temperature in each respective direction. Remark 2. (Integer > 0 or blank)
T(Nuij)	Identification number of a TABLEMi entry for the Poisson's ratio as a function of temperature in each respective direction. (Integer > 0 or blank)
T(Gij)	Identification number of a TABLEMi entry for the shear modulus as a function of temperature in each respective direction. (Integer > 0 or blank)
T(Ai)	Identification number of a TABLEMi entry for the coefficient of thermal expansion as a function of temperature. (Integer > 0 or blank)
T(SY)	Identification number of a TABLEMi entry for the yield stress as a function of temperature. (Integer > 0 or blank)
T(WHS)	Identification number of a TABLEMi entry for the work hardening slope as a function of temperature. (Integer > 0 or blank)

Remarks:

1. The MATTORT entry must have the same ID as the corresponding MATORT entry. Each table ID on the MATTORT entry corresponds to a parameter on the MATORT entry.
2. The table represents material constants as a function of temperature. Therefore, the curve should comprise the original value specified in the MATORT entry (most likely at room temperature).
3. For SOL 600, the continuation line is required for versions prior to MD Nastran 2010 and MSC Nastran 2009.

MATTUSR**Specifies Table Variation of User Defined Generic Materials**

Specifies table variation of user defined generic materials in SOL 600 and SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTUSR	MID			T(RHO)	T(A1)	T(A2)	T(A3)		
	T(GE)	T(ST)	T(SC)	T(SS)					

Example:

MATTUSR	20			51	52				
---------	----	--	--	----	----	--	--	--	--

Descriptor	Meaning
MID	Identification number of MATUSR. (Integer > 0; no Default)
T(RHO)	Table ID of a TABLEM1 or TABL3Di for the mass density for structural analysis. (Integer, no Default)
T(Ai)	Table ID of a TABLEM1 or TABL3Di for the coefficient of thermal expansion. (Integer) If T(A1) is not zero and T(A2) and T(A3) are zero, then T(A1)=T(A2)=T(A3).
T(GE)	Identification number of a TABLEMk entry for the damping coefficient. (Integer ≥ 0 or blank)
T(ST)	Identification number of a TABLEMk entry for the tension stress limit. (Integer ≥ 0 or blank)
T(SC)	Identification number of a TABLEMk entry for the compression limit. (Integer ≥ 0 or blank)
T(SS)	Identification number of a TABLEMk entry for the tension shear limit. (Integer ≥ 0 or blank)

MATTVE**Thermo-Visco-Elastic Material Properties**

Specifies temperature-dependent visco-elastic material properties in terms of Thermo-Rheologically Simple behavior to be used for quasi-static or transient dynamic analysis in SOL 600 and SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTVE	MID	function	RT	ENER	FRACT	TDIF	TREF	NP	
	A1	A2							
	C0	C1	C2	C3	C4	etc.			
	W0	W1	W2	W3	W4	etc.			
	T0	T1	T2	T3	T4	etc.			

Example:

MATTVE	100	WLF	100.						
	0.8	1.2							
MATTVE	101	POWER	100.					11	
	0.0	0.0							
	1.0	.99	.98	.97	.96	.95	.94	.93	W0-7
		.92	.915	.914					W8-10
MATTVE	102	NARA		4.3E-8	.75	200.	400.	11	
	0.0	0.0							
	1.0	.99	.98	.97	.96	.95	.94	.93	W0-7
		.92	.915	.914					W8-10
	235.	234.	233.	232.	231.	230.	229.	228.	T0-7
		227.	226.	225.					T8-10
MATTVE	103	USER	345.					8	

Descriptor	Meaning
MID	Identification number. See Remark 1. (Integer > 0)
Function	Name of the shift function. See Remarks 2. and 3. (Character)
WLF	Williams-Landell-Ferry form, Requires A1 and A2. (Default)
POWER	power series form
NARA	Narayanaswamy model
USER	specify the shift function with a user subroutine
RT	Enter the reference or glass transition temperature. (Real; Default = 0.)

Descriptor	Meaning
ENER	Used for NARA model only, enter the activation energy divided by the gas constant Q/R. (Real; no Default)
FRACT	Used for NARA only, enter the fraction parameter. (Real; no Default)
TDIF	Used for NARA only, enter the temperature shift between your temperature and absolute temperature for calculating fictitious temperatures. (Real; no Default)
TREF	Used for NARA only, enter the reference temperature for structural relaxation. (Real; no Default)
NP	For POWER, enter the number of coefficients in the power series representation. For NARA, NP is the member of terms in the Proxy series.
A1, A2	For WLF model enter the constants A1 and A2. For other models, enter 0.0 for A1 and A2. Do not enter a blank line as it will be stripped out. (Real; no Default)
Ci	Coefficients of the shift function, enter NP values for POWER only. Do not enter Ci for NARA. (Real; no Defaults)
Wi	For NARA model only, enter the weighting factors in increasing order of subscript. Enter NP values. For WLF and POWER, skip these values. (Real; no Defaults)
Ti	For the NARA model only, enter the relaxation time values in increasing order of subscript. Enter NP values. For WLF and POWER, skip these values. (Real; no Defaults)

Remarks:

1. The MATVE Bulk Data entry with the same MID must exist for MATTVE to be effective.
2. The viscoelastic behavior is especially noticeable in the organic high polymers. There are many different kinds of such materials including various plastics, natural and synthetic rubbers. Their mechanical properties depend strongly on temperature, and these properties change drastically in the vicinity of a critical temperature called the glass-transition temperature T_g . The polymer well below T_g is an organic glass with a relatively high modulus. The viscoelastic behavior predominates in the transition range around T_g . The polymer above the transition region (but below the melting point) becomes a rubbery solid with a low modulus.

Polymers are broadly classified as amorphous polymers and polycrystalline polymers. Under stress-relaxation at a constant strain in the glass-transition region temperature, the amorphous polymer exhibits a phase change over time from the glassy state to the rubbery state. The response is manifested in the shear modulus as a function of time, in which initially high shear modulus changes into low shear modulus. The relaxation curve of the modulus in a log-log scale plot appears as a flat plateau of glassy modulus G_g shifting down to the equilibrium modulus G_e at the rubbery plateau. Such a relaxation behavior of the amorphous polymer is observed even when the temperature is well below T_g for a prolonged period of time in a very slow process. A similar behavior is found in the rubbery elastic region, but the process is faster.

Fortunately, the mechanical properties of amorphous polymers obey a time-temperature superposition principle, which allows the use of data obtained at different temperatures to extend the time scale at any given temperature. For such a behavior, the amorphous polymer is characterized as thermo-rheologically simple (TRS). Since the relaxation process extends several decades on the logarithmic time scale at lower temperatures, it is not feasible to determine the whole curve by a constant strain test at one temperature. Instead, the relaxation characteristics are measured at elevated temperatures in reduced time scale. Then the polymers exhibit a translational shift of all the characteristic responses with a change of temperature along the logarithmic time axis. This shift occurs parallel to the time axis without a change in properties: glassy and rubbery moduli. The modulus curve shifts towards shorter time with an increased temperature.

3. The reduced (ξ), or pseudo, time is related to the actual time (t) through a shift function which is a function of temperature, i.e.,

$$\xi(t) = \int_0^t \frac{ds}{A(T(s))}$$

where $A(T)$ is a shift function in terms of temperature T at time t . The shift function is a material property and must be determined experimentally. A shift function approximated by Williams-Landell-Ferry, known as WLF equation, has the form:

$$\log A = h(T) = -\frac{A_1(T - T_0)}{A_2 + (T - T_0)}$$

where T_0 is the reference temperature at which relaxation data are given and A_1, A_2 are calibration constants obtained at this temperature. Notice that $A = 1$ if the reduced time is the same as the actual time. If $T \leq T_0 - A_2$, the deformation will be elastic.

Another form of the shift function is available as a power series in $(T - T_0)$, i.e.,

$$\log A = h(T) = \sum_{i=0}^{10} C_i (T - T_0)^i$$

4. The WLF shift function requires A1 and A2. The power series can have a maximum 11 coefficients C0 through C10.

Note:

1. This entry matches the three options for SHIFT FUNCTION in Marc: William-Landell-Ferry, Power Series and Narayanaswamy models.
2. The user subroutine is TRSFAC in SOL 600; this is not supported in SOL 400.

MATUDS

Material User Defined Service or Subroutine

Allows the user to provide material routines for use with enhanced material models in SOL 400 only.

Format:

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

Example:

In FMS Section of the MSC Nastran input stream:

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

In Bulk Data:

MATUDS	17	MATEP	MATERIAL	UELASTOM					
MATUDS	17	MATUSR	MATERIAL	HYPELA2					
	REAL	.00134	1.467+4	.03					
	INT	8	3						

Descriptor Meaning

MID Identification number of a MAT1, MAT2, MAT3, MAT8, MAT9, MATHE, MATHP, MATORT, MATUSR, MCOHE, or MATD* entries.

MTYPE The name of the material entry. MAT1, MATEP, MATE, MATHE, MATORT, MATUSR, MATVE, or MATVP. (Character; no Default).

GROUP The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)

UNAME User subroutine name associated with the entry. See Remark 6.and 7.(Character).

“INT” Keyword indicating that the following data is integer. (Character)

IDATA_i Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)

“REAL” Keyword indicating that the following data is real. (Character)

RDATA_i Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)

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

Remarks:

1. This entry triggers the call to a user material subroutine for advanced nonlinear materials. The GROUP must match the GROUP field of the CONNECT SERVICE FMS statement.
2. On the FMS CONNECT statement, only the CONNECT SERVICE can be used with this entry.
3. The MID must match an existing MID.
4. A CDATAi entry cannot be the Character “REAL”, “INT”, or “CHAR”
5. Certain user subroutines, such as MATHE, require integer or real data input as specified in the User Defined Services document.
6. UNAME must be truncated to 8 characters in the bulk data field.
7. The following user subroutines are currently available for user convenience. See the User Defined Services document for details.

MTYPE	UNAME	SOL 400
MATHE	uelastomer	X
MATUSR	hypela2	X
MAT1	crplaw	X
MATF	ufail	X
MATF	uprogfail	X
MATORT	orient	X
MATUSR	umat	X
MCOHE	ucohes	X

8. In SOL 400, MATUDS is only supported for elements with property extensions. This implies that for such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. If the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
9. The UMAT and UCOHES user subroutines support user defined state variables, see UDSESV entry.
10. For hyper-elastic(or plastic) materials. the deformation gradient is available for bricks and shells in the user-defined material subroutines, the variable IHYPER on the material card should be set to 1. The deformation gradient components F_{11} , F_{21} , F_{31} , F_{12} , F_{22} , F_{32} , F_{13} , F_{23} and F_{33} can be found in the history variables array in positions IDATA2+1 to IDATA2+9, the positions coming right after the requested number of history variables. For shell elements, the components of the deformation

gradient are with respect to the co-rotational system for the element currently used. In this case the third row of the deformation gradient, the components F_{31} , F_{32} and F_{33} will not be properly updated when entering the user-defined material routine. These components depend on the thickness strain increment which in turn must be determined so that the normal stress in the shell vanishes. For a given thickness strain increment d_3 , these three components, f_{31} , f_{32} and f_{33} , can be determined by calling the subroutine. For hyper-elastic materials there are push forward operations that can be called from within the user defined subroutines too. If the local coordinate system option is invoked (IDATA3=1), then the deformation gradient is transformed to this local system prior to entering the user-defined material routine according to:

$$\bar{F}_{ij} = Q_{ki}^s F_{kj}$$

where Q_{ki}^s refers to a transformation between the current global and material frames. For IDATA equal to 3 one can choose to put IHYPER equal to -1 which results in that the deformation gradient is transformed according to:

$$\bar{F}_{ij} = F_{ik} Q_{kj}^r$$

where Q_{kj}^r is the transformation between the reference global and material frames. For this latter option the spatial frame remains the global one so the stresses should be expressed in this frame of reference upon exiting the user defined routines. The suitable choice of IHYPER depends on the formulation of the material model. For shells, there is also the option of setting IHYPER=3 which will make the deformation gradient computed from the nodal coordinates and in the global coordinate system. With this option the user must compute the stress in the local system of interest, whence a transformation matrix between the global and this local system is passed to the user material routines. The columns in this matrix correspond to local basis vectors expressed in global coordinates, and this is the system that stress needs to be computed in. The user must be aware that since the deformation gradient is calculated directly from the element deformation it may not be consistent with the theory of the element that is used for the material. Also, the thickness used in the calculations is constant and there is currently no thickness change treatment for this option. In the following, a Neo-Hookean material is used as an example of the usage of the deformation gradient in user-defined materials. With λ and μ being the Lame parameters in the linearized theory, the strain energy density for this material is given by:

$$\psi = \frac{1}{2} \lambda [\ln(\det F)]^2 - \mu \ln(\det F) + \frac{1}{2} \mu [\text{tr}(F^T F) - 3]$$

meaning that the Cauchy stress can be expressed as:

$$\sigma = \frac{1}{\det F} [\lambda \ln(\det F) I + \mu (F^T F - 1)]$$

MATUDS**Material User Defined Service or Subroutine - SOL 600**

Allows the user to provide material routines for use with enhanced material models in SOL 600.

Format:

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

Examples:

MATUDS	17	MATEP		uelastomer	2				
MATUDS	33	MATHYP		hypela2	13				
	REAL	1.0E-4	2.0E-6	4.8E-6	38.6		1.2E-4		
	INT	1	3	51	52	53	54		
		55	56						

Descriptor	Meaning
MID	Material property identification number that matches the identification number on a MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, or MATHE, primary material entry or a MATVE, MATVP, MATEP, MATF, associated material entry for structural analysis. MAT4 and MAT5 material entries for heat analysis. COHESIV for Cohesive zone modeling. (Integer > 0)
MTYPE	The name of the material entry. MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHE, MATG, MATVE, MATVP, MATEP, MATF, MATS1, MATS3, MATS8, MAT4, MAT5, COHESIV. (Character; no Default)
UNAME	A primary or secondary name to identify the user subroutine. See Remark 6. (Character; no Default)
IOPT	A user subroutine option flag. See Remark 9. (Character; Default blank)
“INT”	Keyword indicating that the following data is integer. (Character)
IDATA _i	Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)
“REAL”	Keyword indicating that the following data is real. (Character)
RDATA _i	Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)

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

Remarks:

1. This entry triggers the call to a user material subroutine for advanced nonlinear materials.
2. If the matching material entry is not present, the RDATA and IDATA fields may be used to construct the applicable material entry in which case the real and integer field values are in the order shown for the material entry (MATxxx and MATTxxx). The applicable material entries will be internally constructed from the MID, RDATA and IDATA fields.
3. Certain user subroutines, such as MATHE, require integer or real data input as specified in the User Defined Services document.
4. This entry triggers a call to a user subroutine.
5. The following user subroutines are currently available for user convenience. See the User Defined Services document for details.

MTYPE	EVAL	SOL 600
MATHE	uelastomer	X
MATVP	crplaw	X
	vswell	X
	crpvis	X
	ucrplw	X
	uvscpl	X

Material-related structural and heat transfer user subroutines documented in Marc Volume D “User Subroutines and Special Routines” may be entered using MATUDS in addition to those tabulated above. They may alternatively be entered using Bulk Data entry [USRSUB6, 3305](#).

6. If IDATAi values are defined they are stored in common block /userii-mid/ ivals(1000) Up to 1000 integer values may be specified. The userii common block would then be placed in the user subroutine along with other coding.
7. If RDATAi values are defined they are stored in common block /userr8-mid/ rvals(1000) Up to 1000 real*8 values may be specified. The userr8 common block would then be placed in the user subroutine along with other coding.
8. The MATUDS entry may be used instead of the USRSUB6 entry. Both entries should not be used in the same run. When USERSUB is used, flags such as IUSER on the MATVP entry or parameters that are needed will be filled in automatically although they may also be entered if desired. EVAL will be stored as a character*16 name in common block /userch/
9. This name identifies the user subroutine name to be called.

10. IOPT is a user subroutine option flag required by certain MATxxx entries. See the applicable MATxxx entry for values IOPT may use.

MATUSR

Defines Generic Material Properties for Hypoelastic or User Material Model

Specifies user-defined, generic material properties for hypoelastic material models in SOL 600 and user defined material models in SOL 400 only. This entry is used in conjunction with the MATUDS option to activate user subroutine md_hypela2 (SOL 400) or hypela2 (SOL 600) for planar, shell or solid elements. For nonintegrated beam elements it is used in conjunction with user subroutine ubeam (SOL 600).

Format:

1	2	3	4	5	6	7	8	9	10
MATUSR	MID	IPREF	IKINEM	RHO	A1	A2	A3	TREF	
	GE	ST	SC	SS					

Example:

MATUSR	20	1	2	1.0E-4	2.0E-6				
--------	----	---	---	--------	--------	--	--	--	--

Descriptor	Meaning
MID	Identification number. (Integer > 0; no Default)
IPREF	Flag to indicate that user defined material is given with respect to user defined coordinate system. (Integer; Default = 0; see Remark 14.) 0 User stress-strain law is isotropic (Default) 1 User stress strain is with respect to user defined coordinate system
IKINEM	Flag to indicate which additional kinematic quantity is passed into user subroutine. Incremental and total strains are always passed into user subroutine. (Integer; Default = 0 , see Remark 3.) 0 Pass in incremental strain $\Delta\epsilon$ only.. 1 Pass in $\Delta\epsilon$, deformation gradient (F) and rotation (R) 2 Pass in $\Delta\epsilon$, deformation gradient (F) and stretch ratios (λ) 3 Pass in $\Delta\epsilon$, F, R and λ 13 Pass in $\Delta\epsilon$ F, R and λ . Kinematic quantities are calculated at the mid increment if large strain is requested resulting in logarithmic strains. 23 Pass in $\Delta\epsilon$ F, R and λ . Kinematic quantities are calculated at the end of the increment if large strain is requested.
RHO	Mass density for structural analysis (Real \geq 0; no Default)
Ai	Coefficient of thermal expansion. (Real; no Default)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal coefficient. See Remarks 6., 8. and 15. (Real; Default = 0.0)

Descriptor	Meaning
GE	Structural element damping coefficient. See Remarks 7., 8., 9. and 15.
ST,SC,SS	Stress limits for tension, compression and shear are optionally supplied (these are used only to compute margins of safety in certain elements) and have no effect on the computational procedures. See Remark 15. (Real; Default = 0.0)

Remarks:

1. This option is used together with MATUDS to define a generic material model. MATUDS is not required for SOL 600.
2. Only enter those fields necessary for the analysis being performed.
3. For SOL 400, IKINEM should be greater than 0 and user subroutine HYPELA2 is used. For SOL 600, if IKINEM=0, then subroutine HYPELA is used, else HYPELA2.
4. The generic material model is able to represent a nonlinear material behavior. For this constitutive theory, it is assumed that

$$\dot{\sigma}_{ij} = L_{ijkl} \dot{\varepsilon}_{kl} + g_{ij}$$

where L is a function of the mechanical strain and g is a function of the temperature.

The stress and strains are true stresses and logarithmic strains, respectively, when used in conjunction with large strain requests.

When used in conjunction with PARAM,LGDISP,1 (without large strain) the above equation is expressed as

$$\dot{S}_{ij} = L_{ijkl} \dot{E}_{kl} + g_{ij}$$

where E and S are the Green-Lagrange strain and second Piola-Kirchhoff stress, respectively.

5. For the MD_HYPELA2 user subroutine, in order to provide an accurate solution, a tangent stiffness should be evaluated at each iteration. In addition, the total stress should be defined as its exact value at the end of the increment. This allows the residual load correction to work effectively. In MD_HYPELA2, additional information is available regarding the kinematics of deformation. In particular, the deformation gradient (F), rotation tensor (R), and the eigenvalues (λ) and eigenvectors (N) to form the stretch tensor (U) are also provided. This information is available only for the continuum elements namely: plane strain, generalized plane strain (SOL 600 only), plane stress, axisymmetric, axisymmetric with twist, and three-dimensional cases.
6. TREF is used as the reference temperature for the calculation of the thermal loads. TEMPERATURE (INITIAL) may be used for this purpose, in which case TREF should be blank
7. Note that the structural damping will change with time, to obtain the damping coefficient, multiply the critical damping ratio C/C_0 by 2.
8. TREF and GE are ignored if this entry is referenced by a PCOMP or a PCOMPG entry.
9. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See [Parameter Descriptions, 784](#).

10. This material model should not be used with the PSHELL option or with PBARN1, PBEAMN1 or PCOMP(G) and smeared section integration.
11. For SOL 400 the user subroutines md_hypela2 or md_ubeam are used. For SOL 600 user subroutine hypela or hypela2 and ubeam are used, unless a MATUDS option is also included.
12. For SOL 400 this option automatically invokes the PSHLN1, PSHLN2, PSLDN1 element formulation
13. In a thermal-mechanically coupled simulation, MAT4 or MAT5 should be used to define the material behavior.
14. For SOL 600, IPREF = 0 means user subroutines ANEXP and ORIENT are not called. IPREF = 1 means the two routines are called.
15. TREF, GE, ST, SC and SS are ignored in SOL 600.
16. If A1 is given and A2, A3 are not then isotropic thermal expansion and A1 = A2 = A3.
17. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.

MATVE

Visco-Elastic Material Properties

Specifies visco-elastic material properties to be used for quasi-static or dynamic analysis in SOL 600 and SOL 400.

Format for Model = Iso:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model	Alphas	Alpha1	G1	Td1	K1	Tv1	
	G2	Td2	G3	Td3	G4	Td4	G5	Td5	
	K2	Tv2	K3	Tv3	K4	Tv4	K5	Tv5	

Format for Model = Iso1:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model	Alphas	Alphal					
	G1	Td1	K1	Tv1					
	G2	Td2	K2	Tv2					
	G3	Td3	K3	Tv3					
	G4	Td4	K4	Tv4					
	G5	Td5	K5	Tv5					
	Gi	Tdi	Ki	Tvi					

Format for Model = Mooney, Ogden and Foam:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model	Alphas	Alphal	Wd1	Td1	Wv1	Tv1	
	Wd2	Td2	Wd3	Td3	Wd4	Td4	Wd5	Td5	
	Wv2	Tv2	Wv3	Tv3	Wv4	Tv4	Wv5	Tv5	

Format for Model = Mooney1, Ogden1 and Foam1:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model	Alphas	Alphal					
	Wd1	Td1	Wv1	Tv1					
	Wd2	Td2	Wv2	Tv2					
	Wd3	Td3	Wv3	Tv3					
	Wd4	Td4	Wv4	Tv4					
	Wd5	Td5	Wv5	Tv5					
	Wdi	Tdi	Wvi	Tvi					

Format for ORTHO:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	ORTHO							
	Td1	Exx1	Eyy1	Ezz1	Vxy1	Vyz1	Vzx1		
	Gxy1	Gyz1	Gzx1						
	Td2	Exx2	Eyy2	Ezz2	Vxy2	Vyz2	Vzx2		
	Gxy2	Gyz2	Gzx2						
	Td3	Exx3	Eyy3	Ezz3	Vxy3	Vyz3	Vzx3		
	Gxy3	Gyz3	Gzx3						
	Td4	Exx4	Eyy4	Ezz4	Vxy4	Vyz4	Vzx4		
	Gxy4	Gyz4	Gzx4						
	Td5	Exx5	Eyy5	Ezz5	Vxy5	Vyz5	Vzx5		
	Gxy5	Gyz5	Gzx5						
	Tdi	Exxi	Eyyi	Ezzi	Vxyi	Vysi	Vzxi		
	Gxyi	Gyzi	Gzxi						

Example

1	2	3	4	5	6	7	8	9	10
MATVE	1	ISO			3030	0.98			

1	2	3	4	5	6	7	8	9	10
MATVE	1	ISO1							
	3030	0.98							

Descriptor	Meaning
MID	Identification number of primary material (MAT1 or MAT3 or MAT8 or MATROT or MATHE) entry. (Integer > 0). See Remark 2.
Model	Selects a visco-elastic model defining time-dependent deformation behavior (Character): ISO1 for isotropic materials referenced by MAT1 (Default) (an alternate name is Linear). ORTHO if referenced by MAT3, MAT8 and MATORT. MOONEY1 for Mooney-Rivlin model if referenced by MATHE. OGDEN1 for Ogden model if referenced by MATHE. FOAM1 for foam model if referenced by MATHE (SOL 400 only). See Remarks 2, 3, 4, 6, and 8.
Alphas	Solid coefficient of thermal expansion (Real; Default = 0). See Remark 6.
Alphal	Liquid coefficient of thermal expansion (Real; Default = 0). See Remark 6.
Gi	Shear modulus for i^{th} term in Prony series (Real > 0.; Default = 0).
Ki	Bulk modulus for i^{th} term in Prony series (Real > 0.; Default = 0).
Wdi	Multiplier (scale factor) for i^{th} term deviatoric behavior in Prony series (Real \geq 0.; Default = 0). for Hyperelastic Material.
Wvi	Multiplier (scale factor) for i^{th} term volumetric behavior in Prony series (Real \geq 0.; Default = 0.). See Remark 7.
Tdi	Defines time constants for deviatoric behavior in Prony series (Real \geq 0.; Default = 0). See Remark 7.
Tvi	Defines time constants for volumetric behavior in Prony series (Real \geq 0.; Default = 0). See Remark 7.

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.
2. The time-dependent behavior in the viscoelastic material is modeled by a Prony series expression for both small and large strain problems. The stress relaxation behavior can be modeled by relaxation functions of the shear modulus and bulk modulus and total strain energy in terms of a series of exponential decay terms, which is known as the Prony series. This is equivalent to the generalized Maxwell model, which consists of many Maxwell models connected in parallel along with an elastic spring representing a long-term behavior. The constitutive behavior of viscoelasticity depends not only on the current state of stress and strain, but also on the entire history of the development of these states. Such a behavior is most readily expressed by hereditary or Duhamel integral. The Prony series is a discrete form of this hereditary integral.
3. ISO1 supports both volumetric and deviatoric viscoelastic behavior by allowing deviatoric terms (Gi and Tdi) and volumetric terms (Ki and Tvi) in the Prony series. The shear and bulk moduli for isotropic material (ISO1) can be expressed in a Prony series as

$$G(t) = G^\infty + \sum_{i=1} G_i e^{-t/T_{di}}$$

$$K(t) = K^\infty + \sum_{i=1} K_i e^{-t/T_{di}}$$

where instantaneous values are given by:

$$G^0 = G^\infty + \sum_{i=1} G_i$$

$$K^0 = K^\infty + \sum_{i=1} K_i$$

In terms of instantaneous values, the shear and the bulk modulus relaxation can also be written as.

$$G(t) = G^0 - \sum_{i=1} G_i (1 - e^{-t/T_{di}})$$

$$K(t) = K^0 - \sum_{i=1} K_i (1 - e^{-t/T_{di}})$$

If a Prony series is selected, at least one set of modulus and time constant must be provided i.e., (G1, Td1) and /or (K1, Tv1).

4. ORTHO supports only deviatoric viscoelastic behavior by allowing deviatoric terms (G_{ij} and T_{di}) in the Prony series. For orthotropic material behavior, the relaxation coefficients are applied to the shear modulus as

$$G_{ij}(t) = G_{ij}^0 - \sum_{n=1} G_{ij}^n (1 - e^{-t/T_{di}})$$

where G_{ij}^n is the shear modulus for n^{th} term is Prony series (Real > 0; Default = 0) for Orthotropic Material. (Here G_{ij} imply G_{xy} , G_{yz} and G_{zx} as mentioned in ORTHO table).

5. In case of a viscous hyperelastic material, total strain energy be expressed as a Prony series expansion in terms of energy functional

$$\psi = \psi^\infty + \sum_{n=1} \psi_n e^{-t/T_{di}}$$

Above equation can also be expressed as a function of the elastic strain energy density for instantaneous deformations ψ° .

$$\psi = \psi^\circ + \sum_{n=1} W_n \psi^0 e^{-t/T_i}$$

where W_i are weighting factors ψ_i/ψ^0 and T_i are time constants. Two Prony series can be formed based on deviatoric (ψ^d) and volumetric strain energy (ψ^v). If a viscous hyperelastic Prony series is selected, at least one pair of weighting factor and time constant must be provided: (Wd1, Td1) and /or (Wv1, Tv1). OGDEN support both deviatoric (Wdi, Tdi) and volumetric strain energy (Wvi, Tvi) Prony series. However, Mooney supports only deviatoric strain energy (Wdi, Tdi) Prony series. Foam is defined in total strain energy density function and specified through W_i and T_i .

6. If ALPHAS or ALPHAL is specified, the thermal expansion coefficient specified in MAT1, MATT1, MATHE, MATTHE, MATORT or MATTORT will be ignored. Use of ALPHAS or ALPHAL requires the MATTVE Narayanaswami model.
7. If the weighting factor is left blank, the relaxation function corresponding to that coefficient is omitted.
8. In SOL 400, MATVE is used only supported for elements with property extensions. This implies that for such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MSC Nastran 2010, if the property extensions were missing, then the MATVE data was not considered in the element's formulation. If the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS, SPROPMAP, -1 in the bulk data.
9. FOAM (FOAM1) no difference between deviatoric and volumetric multipliers and relaxation time. Program will use deviatoric input to apply on entire strain energy function.

Notes:

1. The Prony series uses values of G_i or K_i for Isotropic and Orthotropic materials. It uses weighting (W_i) functions for Mooney, Ogden and Foam materials.
2. The instantaneous material properties are specified on primary material entries (i.e., for MAT1 or MAT3 or MAT8 or MATORT or MATHE)
3. In the MATVE entries, material properties with the Prony series parameters are specified.
4. MATVE with "ISO", "MOONEY", "OGDEN", "FOAM" models are similar to the corresponding "ISO1", "MOONEY1", "OGDEN1", "FOAM1" models with the only exception being that the former models are limited to 5 terms in the Prony series. With the latter models, there is no limit to the number of terms in the Prony series.

MATVE

Visco-Elastic Material Properties- SOL700

Specifies isotropic visco-elastic material properties to be used for quasi-static or dynamic analysis in SOL 700.
Used in SOL 700 only.

Format (for types Iso, Mooney, Ogden and Foam):

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model			G0	GI	BETA	VISC	

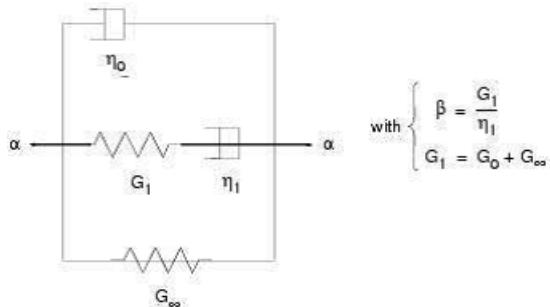
Example:

MATVE	1	ISOLIN			8.E7	1.E7	0.1		
-------	---	--------	--	--	------	------	-----	--	--

Descriptor	Meaning
MID	Identification number of MAT1 or MATHE entry (Integer > 0; required).
Model	Selects a visco-elastic model defining time-dependent deformation behavior (Character): ISOLIN for linear isotropic materials referenced by MAT1
G0	Short-time shear modulus. (Real; default=0.0)
GI	Long-time shear modulus. (Real; default=0.0)
BETA	Decay constant. (Real; default=0.0)
VISC	Shear viscosity constant. (Real; default=0.0)

Remarks:

1. The spring-damper analog of this model is:



2. The deviatoric stress is given by

$$\sigma'_{ij}(t) = 2G_1\varepsilon'_{ij}(t) + 2 \int_0^t G(t-\tau) + \frac{\partial \varepsilon'_{ij}(\tau)}{\partial \tau} d\tau + 2\eta_0 \frac{\partial \varepsilon'_{ij}(t)}{\partial t}$$

$$G(t - \tau) = G(G_0 - G_I)e^{-\beta(t - \tau)}$$

The above equation for the deviatoric stress is the integral form of the differential equation

$$\dot{\sigma}'_{ij} + \beta\sigma'_{ij}t = 2\eta_0\ddot{\varepsilon}'_{ij} + (2\eta_0\beta + 2G_0)\dot{\varepsilon}'_{ij} + 2G_I\beta\varepsilon'_{ij}$$

A special case is for which is often written $\eta_0 = G_I = 0$

$$\dot{\varepsilon}'_{ij} = \dot{\varepsilon}'_{ij\text{elastic}} + \dot{\varepsilon}'_{ij\text{viscous}} = \frac{\dot{\sigma}'_{ij}}{2G_0} + \frac{\beta}{2G_0}\sigma'_{ij}$$

MATVP**Viscoplastic or Creep Material Properties**

Specifies viscoplastic or creep material properties to be used for quasi-static analysis in SOL 600 and SOL 400 only.

Primary Format: (for SOL 600, also enter Bulk Data parameter, PARAM,MARCMATT,1)

1	2	3	4	5	6	7	8	9	10
MATVP	MID	A	IT3D	M	N	P	Q	IUSER	

Primary Format: (for SOL 400 only, using ANAND solder creep model)

1	2	3	4	5	6	7	8	9	10
MATVP	MID	ANAND	PREXF	ACTEN	MULST	STNRT	SATCO	STNSA	
	HRCN	STNHR	DEFRS						

Alternate Format: (SOL 600 only, leave out PARAM,MARCMATT or enter PARAM,MARCMATT,-1))

1	2	3	4	5	6	7	8	9	10
MATVP	MID	Form	Coeff	Stress	Strain	Temp	Time	IUSER	

Example 1: (Primary Format, A is built into TABL3Di with IT3D=20)

MATVP	10	1.0	20	1.5	1.1	1.0	1.0		
-------	----	-----	----	-----	-----	-----	-----	--	--

Example 2: (Primary Format, A is provided along with exponents for stress, strain, temperature and time)

MATVP	10	.00375	0	1.5	1.1	1.0	1.0		
-------	----	--------	---	-----	-----	-----	-----	--	--

Example 3: (Primary Format, user subroutines crplaw.f and vswell.f are used)

MATVP	10	0.0							
-------	----	-----	--	--	--	--	--	--	--

Example 4: (Primary Format, ANAND solder creep model is used)

MATVP	2	ANAND	2.23e4	15583	7	0.143	72.73	0.00437	
	1787.02	3.73	15.09						

Example 5: (Alternate Format with 4 table inputs, one each for stress, strain, temperature and time)

MATVP	100	TABLE	3.5E-15	101	102	103	104		
-------	-----	-------	---------	-----	-----	-----	-----	--	--

Example 6: (Alternate Format, user subroutine ucrplw.f is used)

MATVP	100	USER							
-------	-----	------	--	--	--	--	--	--	--

Descriptor	Meaning
MID	Identification number of MAT1, MAT2, MATORT or MAT9 entry. See Remark 1. (Integer > 0)
A	Enter the coefficient A in for the equations of Remark 2. (this value could alternatively be built into the table referenced using IT3D in which case A should be set to 1.0. If A=0.0, user subroutine(s) designated by the flag in field 9 may be used to define the creep law and fields 4-8 will be ignored if entered. (Real; no Default)
IT3D	ID of a TABL3Di entry that defines the variation with respect to stress, strain, temperature and/or time per the equations discussed in Remark 2. The exponents are provided in the next 4 fields. (Integer > 0; Default = 0)
M	Exponent m for stress in equation in Remark 2. (Real; Default = 0.0)
N	Exponent n for strain in equation in Remark 2. (Real; Default = 0.0)
P	Exponent p for temperature in equation in Remark 2. (Real; Default = 0.0)
Q	Exponent q for time in equation in Remark 2. (Real; Default = 0.0)
IUSER	Packed list designating which user subroutines (if any) apply to this analysis. The user must confirm that the selected user subroutines are appropriate for the current analysis and are consistent with other entries in the model. (Integer; Default = 0) <ul style="list-style-type: none"> 1 crplaw.f 2 vswell.f (only available in SOL 600) 3 crpvis.f (only available in SOL 600) 4 ucrplw.f (only available in SOL 600) 5 uvscpl.f (only available in SOL 600) (examples, if crplaw.f is used, enter 1 if both crplaw.f and vswell.f are required, enter 12)
Form	(Alternate format only) Selects a creep data input form defining creep strain rate from the options listed below. (Character; no Default): <ul style="list-style-type: none"> "POWER" for exponent input in power law form "TABLE" for piece-wise linear curve input in TABLEM1 entry.

Descriptor	Meaning
"USER"	user subroutine crplaw.f will be used instead of the values/table ID's on the entry (if this form is used, fields 4-9 will be ignored if entered).
	See Remark 2.
Coeff	Specifies the coefficient value, A in equation. (Real > 0)
Stress	Identification number of TABLEM1 (Integer > 0) for the function f or exponent m (Real > 0.) for an effective stress function, depending on the Form field.
Strain	Identification number of TABLEM1 (Integer > 0) for the function g or exponent n (Real; Default = 0.) for an equivalent creep strain function, depending on the Form field.
Temp	Identification number of TABLEM1 (Integer > 0) for the function h or exponent p (Real; Default = 0) for a temperature function, depending on the Form field.
Time	Identification number of TABLEM1 (Integer > 0) for the function K or exponent q (Real > 0; Default = 1) for a time function, depending on the Form field.
ANAND	A keyword specifying the Anand solder creep model. The Anand solder creep model consists of a simple set of constitutive equations for large, isotropic, viscoplastic problems. If this key word is used, next 9 fields for parameters of Material properties must be filled with values. Please note that Anand solder creep model is only for isotropic materials. See Remark 12. and 13..
PREXF	Pre-exponential factor. $A, (s^{-1})$
ACTEN	Activation Energy. Q
MULST	Multiplier of stress. $?$
STNRT	Strain rate sensitivity of stress. M
SATCO	Deformation resistance saturation coefficient. $\hat{s}, (MPa)$
STNSA	Strain rate sensitivity of saturation. n
HRCN	Hardening constant. $h_0, (MPa)$
STNHR	Stain rate sensitivity of hardening. A
DEFRS	Initial value of deformation resistance. $s_0, (MPa)$

Remarks:

1. This Bulk Data entry is activated if a MAT1, MAT2, MATORT, or MAT9 entry with the same MID is specified in an implicit nonlinear analysis. This creep capability is available for isotropic, orthotropic, and anisotropic elasticity, which can be coupled with plasticity using MATEP entry. Coupling with plasticity is allowed only for selected plasticity models, which include von Mises, Hill's anisotropy (creep stays isotropic), and Mohr-Coulomb models. However, viscoelasticity (MATVE) cannot be combined with viscoplasticity (MATVP).

Please note that ANAND solder creep model may be associated with isotropic material, i.e., MAT1, only.

2. The creep behavior of the material is expressed in terms of creep strain rate as a product of a number of terms (functions of effective stress, equivalent creep strain, temperature, and time) in either piece-wise linear curves or exponential form, i.e., $\dot{\varepsilon}^c = A \cdot \bar{\sigma}^m \cdot (\bar{\varepsilon}^c)^n \cdot T^p \cdot (qt^{q-1})$

$$\text{or } \dot{\varepsilon}^c = A \cdot f(\bar{\sigma}) \cdot g(\bar{\varepsilon}^c) \cdot h(T) \cdot \frac{dK(t)}{dt}$$

Power-Law creep (exponential form) can be used with either explicit or implicit creep methods. The choice of explicit or implicit creep is made via the NLMOPTS option.

- a. Explicit creep can be combined with other plasticity options defined through the MATEP card. The supported plasticity models include Von Mises, Hill's Anisotropy (creep remains isotropic) and Mohr-Coulomb models. The creep and plastic strains are treated separately with the algorithm handling the creep in an explicit manner and handling the plasticity in an implicit manner.
- b. Implicit power law creep can be combined with the Von Mises yield criterion defined through the MATEP card. Both the creep and plasticity are handled simultaneously through an implicit viscoplasticity algorithm and a combined inelastic strain is computed.
- c. Note that for the explicit creep strain-rate evaluation, N should only be specified if some creep strain is already present in the model. Otherwise, N should be zero (blank), or implicit creep should be selected in CREEP of NLMOPTS.
- d. The dependence of time (Q) is specified as function of total equivalent creep strain i.e., $e_c = A t^Q$ (i.e., $\text{dot}(e_c) = A Q t^{Q-1}$).

The functions f, g, h and K are specified as piece-wise linear functions in a tabular form using TABLEM1 entry, if the Table Form is selected. Notice that the last term in time shows function K for the equivalent creep strain in terms of time, instead of creep strain rate.

The creep strain from the creep material is a permanent strain unlike the creep strain for materials using the CREEP Bulk Data entry. As such, this creep material may be classified as viscoplastic material. This creep capability is provided for the primary and the secondary creep behavior, because the tertiary creep involves material instability such as necking.

3. For SOL 600, see associated Bulk Data entry, MPCREEP.
4. For SOL 400 see associated Bulk Data entry, NLMOPTS.
5. The alternate format is determined by field 3 with a character string POWER, TABLE, or USER. The alternative format is currently only available for SOL 600 not SOL 400.
6. There must be a non-blank entry in field 3 for either format.
7. In SOL 400, MATVP can be used with NLPARM only when total time (NINC * DT = 1.0). It is recommended that MATVP be used in conjunction with NLSTEP.
8. Furthermore, for NLSTEP adaptive stepping, use of TABSCTL to specify user criteria on creep strains and stresses is also recommended.

9. For SOL 600, MATVP is used in conjunction with NLPARM with no restrictions on the value of the total time (NINC * DT). NLAUTO may be added to specify under stress, strain or other criteria and/or for advanced convergence controls not available on the first line of NLPARM. Additional convergence and solver controls are available using NLSTRAT.
10. In SOL 400, MATVP is used only supported for elements with property extensions. This implies that for such elements, PBAR / PPARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATVP data was not considered in the element's formulation. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
11. When used with PBAR, PBEAM, PBEAML, PCOMP, PCOMPG, PROD, PSHEAR, PSHELL or PLPLANE (with BEH=PSTRS) the explicit formulation must be used, set VALC3=0 on NLMOPTS.
12. To activating creep analysis with ANAND creep model, the following parameters are mandatory:
 - a. In Bulk Data Card NLMOPTS: keyword "CREEP" and "LRGSTRN" must be selected with value:
"LRGSTRN" with value 1 for updated Lagrange formulation, Hypo-elasticity and additive plasticity with mean normal value.
"CREEP" with valc3=1, Implicit Creep on; and valc4=0/1/2, for elastic tangent as default (Blank or 0), 1 for secant tangent, and 2 for radial setting, respectively.
 - b. In Bulk Data Card MATEP: keyword "IMPCREEP" is required for ANAND model.
 - c. In Bulk Data Card NLSTEP: CREEP in "GENERAL" is required for ANAND creep model.
 - d. In Bulk Data Card MATVP: material parameters of ANAND creep model are defined in MATVP card with keyword ANAND" in the 3rd field and following 9 fields.
13. NLSTRESS in Case Control will output the equivalent creep strain on grid point. To output the component of the creep strain, NLOUT in NLSTRESS Case Control command must be set and request Creep output in Bulk Data Card.

In case control section:

```
NLSTRESS (NLOUT=num) = ALL
```

In bulk data section:

```
NLOUT, num, TOTTEMP, CCRPSTRN
```

Note:

1. This Bulk Data entry accommodates Marc's input data under the model definition option CREEP.

MAUXCMD**Auxiliary Command to Spawn one Nastran Process from Another - SOL 600**

Defines auxiliary command to spawn on Nastran process from another Nastran process in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
MAUXCMD	Command	Cont							

Example:

MAUXCMD	/mscinst/nast2008/bin/mdnast2008	
---------	----------------------------------	--

Descriptor	Meaning
Command	Enter the command you normally use to execute Nastran from the command line (Character; no default; limit of 64 characters)

Remark:

1. At present, this entry may only be used for CINTC when PARAM,MBLADCMD,10 is set.

MBOLT Defines a Bolt For Use in SOL 600 in Countries Outside the USA

Defines a bolt for use in SOL 600 in countries outside the USA. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MBOLT	ID	GRIDC	V1	V2	V3				
	GRIDS	G1	G2	G3	G4	G5	G6	G7	
		G8	G9	Etc.					
	ELEMS	E1	E2	E3	E4	E5	E6	E7	
		E8	E9	Etc.					

Example:

MBOLT	100	1025	0.0	1.0	0.0				
	GRIDS	101	102	103	104	105			
	ELEMS	1	2	3	4	5	6	7	
		10							

Descriptor	Meaning
ID	Element ID of the bolt. (Integer; Required; no Default)
GRIDC	Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)
V1	First component of vector normal to the bolt cross section in basic coordinate system. (Real; Default = 0.0)
V2	Second component of vector normal to the bolt cross section in basic coordinate system. (Real; Default = 0.0)
V3	Third component of vector normal to the bolt cross section in basic coordinate system. (Real; Default = 0.0)
GRIDS	Enter the character string GRIDS to define the start of the entry that defines all of the grids at the bolt intersection cross section (do not enter the ID for GRIDC). (Integer; no Default)
G1, G2, etc.	Grid IDs of the grid points at the bolt intersection. (Integer; no Default)
ELEMS	Enter the character string ELEMS to define the start of the entry that defines all of the elements at the bolt intersection cross section lying on the side of the cross section corresponding to the negative normal direction. (Integer; no Default)
E1, E2, etc.	Element IDs of the grid points at the bolt intersection. (Integer; no Default)

Remarks:

1. Enter as many GRIDS and ELEMS lines as necessary to define all the grid and element IDs in the cross section.
2. All GRIDS must precede all ELEMS.
3. The bolt itself is not actually modeled, just the intersecting surfaces. The nodes and elements where the bolt goes through the intersecting surfaces are described by this entry.
4. Specify a different MBOLT entry for each individual bolt.
5. This entry can only be used with Marc 2003 or later outside the USA.
6. For more information, please consult the Marc Theoretical Manual (Volume A of the Marc documentation).
7. This entry maps to Marc's CROSS SECTION entry.
8. GRIDC must already exist. It is not generated by the MBOLT entry. It typically would not be used by any other element, MPC, etc.

MBOLTUS

Defines a Bolt for Use in SOL 600 in the USA

Defines a bolt for use only in SOL 600 and only in the USA.

Format:

1	2	3	4	5	6	7	8	9	10
MBOLTUS	ID	GRIDC							
	TOP	GT1	GT2	GT3	GT4	GT5	GT6	GT7	
		GT8	GT9	Etc.					
	BOTTOM	GB1	BG2	BG3	BG4	GB5	GB6	GB7	
		GB8	GB9	Etc.					

Example:

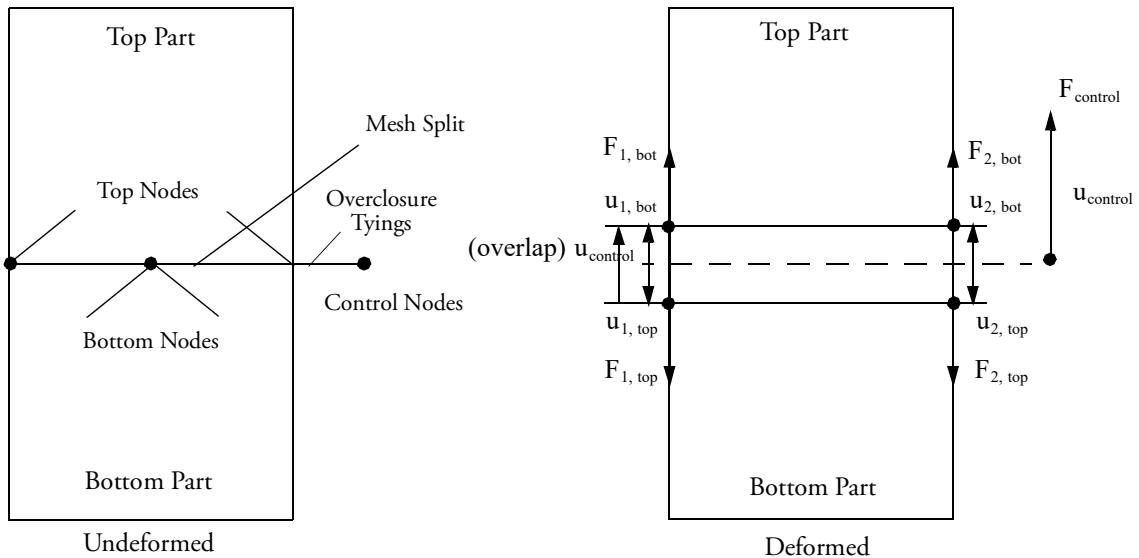
MBOLTUS	100	1025		1.0					
	TOP	101	102	103	104	105			
	BOTTOM	1	2	3	4	5			

Descriptor	Meaning
ID	Element ID of the bolt. (Integer; Required; no Default)
GRIDC	Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)
TOP	Enter the character string TOP to define the start of the entry that defines all of the grids at the “top” of the bolt intersection with the structure (do not enter the ID for GRIDC). (Integer; no Default)
GT1, GT2, etc.	Grid IDs of the grid points at the top of the bolt intersection. (Integer; no Default)
BOTTOM	Enter the character string BOTTOM to define the start of the entry that defines all of the grids at the “bottom” of the bolt intersection with the structure (do not enter the ID for GRIDC) (Integer; no Default)
GB1, GB2, etc.	Grid IDs of the grid points at the bottom of the bolt intersection (Integer; no Default)

Remarks:

1. Enter as many GRIDS as necessary (up to the limit of 999 top grids and 999 bottom grids) to define all the grids at the “top” and “bottom” of the bolt intersection with the structure.
2. The bolt itself is not actually modeled, just the intersecting surfaces. The nodes and elements where the bolt goes through the intersection surfaces are described by the entry.
3. Specify a different MBOLTUS entry for each individual bolt.
4. This entry can only be used with Marc 2003 or later.

5. For more information, please consult the Marc Theoretical Manual (Volume A of the Marc documentation)
6. This entry maps to Marc's TYING type 69.
7. GRIDC must already exist. It is not generated by the MBOLTUS entry. It typically would not be used by any other element, MPC, etc.
8. The following figure indicates the required modeling and data input.



MCHSTAT

Option to Change State Variables for SOL 600

This option provides various ways of changing state variables throughout the model. It is required if Bulk Data entry, MTCREEP is used. It may also be used to enter temperatures calculated from a previous heat transfer analysis and saved on a t16 or t19 file in which case MINSTAT is used to define the initial stress-free temperatures and MCHSTAT is used to define the temperatures that cause thermal strains.

Format:

1	2	3	4	5	6	7	8	9	10
MCHSTAT	ID	IDV	IOPT		INCR	NSET	IFORM	IPRT	
		NPST							
	“FILE”	Name							
	“ELEM”	ELE1	ELE2	INT1	INT2	LAY1	LAY2	VAL	
	“STATE”	NS	IS1	IS2	IS4	IS5	IS6		
		IS7	IS8	IS9	etc.				

Example:

MCHSTAT	0	1	3		1		0		
		9							
	FILE	change_state_example01							

Descriptor	Meaning
ID	ID of a matching Case Control MCHSTAT command. If ID=0 this entry is in the Marc Model definition, otherwise it is in Marc's History Definition for the applicable subcase. (Integer; no Default)
IDV	State variable identifier (1=temperature). (Integer; Default = 1) (2,1) If more than one state variable is required, enter -1.
IOPT	Option of how to enter the data. (Integer; Default = 3) (2,2) <ul style="list-style-type: none"> 1 Use the “ELEM” continuation line for as many elements as necessary 2 Enter the data using user subroutine NEWSV 3 Read the data from a t16 or t19 file (see IFORM)
INCR	Increment number on t16 or t19 file defining the new state values if IOPT=3. (Integer; no Default) (2,5)
NSET	Number of sets to be read to define the temperature history if Marc's iteration method is controlled using MTHERM or MTCREEP. (Integer; no Default) (2,6)
IFORM	Designates whether a binary (t16) or formatted (t19) post file is used if IOPT=3. (Integer; Default = 0) (2,7) <ul style="list-style-type: none"> 0 Use binary (t16) file

Descriptor	Meaning
	1 Use formatted (t19) file
IPRT	Enter a value of 1 to suppress printing of state variable values defined in user subroutine NEWSV (only applicable if IOPT=2). (Integer; Default = 0) (2,8)
NPST	Post Code ID to be read into this state variable. (Integer; Default = 9 [temperature]) (2,9) See MARCOUT for a list of the post codes.
FILE	Enter the character string FILE if IOPT=3. (Character, no Default; Required if IOPT=3)
NAME	Enter the file root name of the previous heat transfer job without any extension. For example if the previous heat transfer job was heat32.dat or heat32.bdf, enter heat32. The file name must be entirely in lower case for case-sensitive computer systems and is limited to 56 characters. This file must be in the same directory as the Nastran input file. (Character, no Default)
ELEM	Enter the character string ELEM to start a list of elements and associated values if IOPT=1 (Character)
ELE1	First element with value VAL. (Integer; no Default; Required) (3,1)
ELE2	Last element with value VAL. (Integer; Default = ELE1) (3,2)
INT1	First integration point with value VAL. (Integer; no Default; Required) (3,3)
INT2	Last integration point with value VAL. (Integer; Default = INT1) (3,4)
LAY1	First cross-section layer with value VAL. (Integer; no Default; Required) (3,5)
LAY2	Last cross-section layer with value VAL. (Integer; Default = LAY1) (3,6)
VAL	New state value for these elements. (Real; no Default; Required) (4,1)
STATE	Enter the character string STATE to start a list of state variables. (Character)
NS	Number of state variables to be defined. (Integer; no Default; limited to 16 maximum)
ISi	State variable post codes. (Integer; no Default) (9,i) See MARCOUT entry for applicable post codes.

Remarks:

1. All MCHSTAT ID's must be unique.
2. This entry maps to Marc's CHANGE STATE entry.
3. This entry must be entered if the MTCREEP (Marc's AUTO THERM CREEP) entry is used.
4. (i,j) refer to Marc's CHANGE STATE (data block, field).
5. MCHSTAT (and/or MINSTAT) cannot be the only applied "loads". At least one standard load such as FORCE, PLOAD4 or a standard TEMP entry must be entered with a Case Control LOAD command that references the standard load(s). If there are no standard loads, please enter a dummy load with a very small magnitude and a Case Control LOAD command to reference it.

MCOHE**Interface Cohesive Zone Modeling Element Material Properties**

Specifies material cohesive properties for a fully nonlinear element used to simulate the onset and progress of delamination in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
MCOHE	MID	MODEL		TID					
	COHE	CRTOD	MAXOD	SNSR	EXP	VED	RRRD	SFC	

Example:

MCOHE	701	2		357					
	136.5	0.0	0.0	1.0	0.02	0.0		1.0	

Descriptor	Meaning
MID	Identification number of a MCOHE entry. (Integer > 0) (See Remark 1.)
MODEL	(Integer > 0; Default = 1) (See Remark 3.)
	1 Bilinear model
	2 Exponential model
	3 Linear-exponential model
	-1 User Defined Subroutine (See Remark 2.)
TID	Table identifier for a combination of TABLES1/TABLEST for cohesive energy vs temperature. (Integer ≥ 0 ; Default = 0)
COHE	Cohesive energy. (Real ≥ 0.0)
CRTOD	Critical opening distance. (Real ≥ 0.0)
MAXOD	Maximum opening displacement (bilinear model only). (Real ≥ 0.0)
SNSR	Shear Normal Stress Ratio. (Real > 0.0 ; Default = 1.0)
EXP	Exponential decay factor (linear-exponential model only). (Real > 0.0 ; Default = 1.0)
VED	Factor for viscous energy dissipation. (Real ≥ 0.0 ; Default = 0.0)
RRRD	Reference rate of relative displacement. Used only if VED $\neq 0.0$. A value of 0.0 implies that the reference rate will be automatically calculated. (Real ≥ 0.0 ; Default = 0.0)
SFC	Stiffening factor in compression. (Real ≥ 0.0 ; Default = 1.0)
SNER	Shear Normal Energy Ratio. (Real > 0.0 ; Default = 1.0)

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.
2. User subroutine UCOHES is used. User must also use the MATUDS bulk data entry.
3. For details on model see Nonlinear User's Guide (SOL 400).

MDBCNCT

Defines the Touching and Touched Contact Bodies in Different Modules

Defines the touching and touched contact and used in SOLs 101, 103, 105, 107, 108, 109, 110, 111, and 112 for general contact, glued, step glue or permanent glue between contact bodies residing in different Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDBCNCT	ID	BCGPID	BCPPID	MODS	IDSLAVE	MODM	IDMASTER		
	“SLAVES”	MODS1	IDSL1	MODS2	IDSL2	MODS3	IDSL3		
		MODS4	IDSL4	-etc.-					
	“MASTERS”	MODM1	IDMA1	MODM2	IDMA2	MODM3	IDMA3		
		MODM4	IDMA4	-etc.-					

Example:

MDBCNCT	57	306		101	2	201	1002		
---------	----	-----	--	-----	---	-----	------	--	--

MDBCNCT	9		108						
	SLAVES	101	30	201	26				
	MASTERS	101	294	201	135	301	528		

Descriptor	Meaning
ID	Unique identification number referenced by a MDBCTB1 entry (Integer ≥ 0). See Remark 5.
BCGPID	Parameter identification number of a BCONPRG entry (Integer ≥ 0 or blank). See Remark 6.
BCPPID	Parameter identification number of a BCONPRP entry (Integer ≥ 0 or blank). See Remark 6.
MODS	Identification number of a Module that defines IDSLAVE (Integer ≥ 0 or blank)
IDSLAVE	Identification number of BCBODY1 entry defining the touching body (Integer ≥ 0 or blank). See Remarks 7. and 8.
MODM	Identification number of a Module that defines IDMASTER (Integer ≥ 0 or blank)
IDMASTER	Identification number of BCBODY1 entry defining the touched body (Integer ≥ 0 or blank). See Remarks 9. and 10.
“SLAVES”	Indicates the start of the list of the touching bodies. See Remark 8.
MODSi	Identification number of a Module that defines IDSLi (Integer ≥ 0 or blank)
IDSLi	Identification number of BCBODY1 entry defining touching bodies (Integer ≥ 0 or blank,). See Remarks 7. and 8.

Descriptor	Meaning
"MASTERS"	Indicates the start of the list of bodies touched by touching bodies. See Remark 10.
MODMi	Identification number of a Module that defines IDMAi (Integer ≥ 0 or blank)
IDMAi	Identification number of BCBODY1 entry defining touched bodies (Integer ≥ 0 or blank).

Remarks:

1. Both BCTABL1 and MDBCTB1 may only be specified in the main Bulk Data section.
2. BCTABL1 BCID must be unique with respect to all other BCTABL1 and MDBCTB1.
3. BCONTACT Case Control command may only select BCTABL1 or MDBCTB1 but not both. If BCTABL1 is selected, then all bodies matching IDi in any and all Modules will be activated.
4. Only MDBCTB1 can select MDBCNCT.
5. MDBCNCT can be selected by the Case Control command BCONTACT=ID to define surface contact if MDBCTB1 entry does not exist. See Remarks 2 and 3 of MDBCTB1 entry.
6. If BCGPID or BCPPID field is blank, then default values are set for the parameters of touching bodies.
7. A short input to define a single touching body exists if the user provides IDSLAVE. On the other hand, if the user leaves IDSLAVE blank, then "SLAVES" descriptor is required and IDSL1 must be specified..
8. "SLAVES" and IDSLi fields will be ignored if IDSLAVE exists. If IDSLAVE field is blank, then "SLAVES" and IDSLi must be specified. In this case, each IDSLi will be processed separately.
9. A short input to define a single touched body exists if the user provides IDMASTER. On the other hand, if the user leaves IDMASTER blank, then "MASTERS" descriptor is required and IDMA1 must be specified.
10. "MASTERS" and IDMAi fields will be ignored if IDMASTER exists. If IDMASTER field is blank, then "MASTERS" and IDMAi must be specified.
11. The concept of Slave and Master relation is important to the node-to-segment contact but not relevant for segment-to-segment contact. In segment-to-segment contact, they are mainly used to define the contact pair(s).
12. If all the BCONPRG that are referenced by a MDBCNCT (which is referenced by MDBCTB1) have a value of IGLUE > 0, and this MDBCNCT is referenced in the 1st Loadcase (SOL 100*) then the connections are considered to be permanent and do not change (unless a BCPARA is used to deactivate the permanent glue).

MDBCTB1

Defines a Contact Table for Bodies in Different Modules

MDBCTB1 specifies a list of contact pairs (for bodies residing in different Modules) through the MDBCNCT option for SOL 101. This contact table is activated in the BCONTACT Case Control command.

Format:

1	2	3	4	5	6	7	8	9	10
MDBCTB1	BCID	MOD1	ID1	MOD2	ID2	MOD3	ID3	MOD4	
	ID4	MOD5	ID5	-etc.-					

Examples:

MDBCTB1	2	101	198	201	62	301	75	401	
	501	8	601	159	701	31	801	44	
MDBCTB1	0	1001	23	1002	56				

Descriptor	Meaning
BCID	Unique identification number referenced by a BCONTACT Case Control command. See Remark 3.(Integer ≥ 0 ; Required)
MODi	Identification number of module that defines BCNECT IDi (Integer ≥ 0 or blank,)
IDi	Identification number of BCNECT entry (Integer > 0). See Remark 4. and 5.

Remarks:

1. MDBCTB1 defines surface contact. If MDBCTB1 does not exist, the Case Control command BCONTACT=BCID may refer to the MDBCNCT Bulk Data entry directly.
2. If neither MDBCTB1 nor MDBCNCT is given, the default for contact analysis is assumed, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If MDBCTB1 or MDBCNCT is given, the default for everybody is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. MDBCTB1 or MDBCNCT is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.
3. For SOL 101, the MDBCTB1 or MDBCNCT with ID=0 will be used in loadcase 0 automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable BCBODY1's. To place an entry in

any physical loadcase (SUBCASE or STEP), the BCID must be selected by the Case Control command BCONTACT=BCID or MDBCNCT=BCID. When BCONTACT=ALLBODY, there is no BCID of MDBCTB1 or MDBCNCT specified; therefore, the default values of all entries of BCONPRG and BCONPRP are assumed. Case Control command BCONTACT=ALLBODY cannot be used for permanently glued contact.

4. The “THRU” option is not supported.
5. Intervening blank fields are not allowed.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

MDBNDRY**Module to Module Boundary Point Definitions**

Defines a list of grid points in a module for the automatic boundary search between a specified Module or between all other Modules. In other words, the automatic boundary search will only search for connections in this list.

Format:

1	2	3	4	5	6	7	8	9	10
MDBNDRY	MIDA	MIDB	GIDA1	GIDA2	GIDA3	GIDA4	GIDA5	GIDA6	
	GIDA7	GIDA8	etc.						

Examples 1:

MDBNDRY	400	4	10	20	30	40			
---------	-----	---	----	----	----	----	--	--	--

Examples 2:

MDBNDRY	400	ALL	10	20	30	THRU	35		
---------	-----	-----	----	----	----	------	----	--	--

Descriptor	Meaning
MIDA	Module identification number. (Integer ≥ 0 , Default=0)
MIDB	Reference Module identification number. (Integer ≥ 0 or Character “ALL”; Default=ALL)
GIDAi	Identification number of a boundary grid point in the Module MIDA. (Integer > 0 or “THRU”; for “THRU” option, GID1 < GID2.)

Remarks:

1. MDBNDRY may only be specified in the Main Bulk Data Section and cannot be specified in other BEGIN MODULE sections.
2. This entry is valid only if Modules exist.

MDBOLT

Defines the Multi-Point Constraints for a Bolt Between Two Modules

Defines a rigid bolt between two Modules by a set of MPC constraints.

Format:

1	2	3	4	5	6	7	8	9	10
MDBOLT	ID	GRIDC	MODC	MODT	MODB				
	TOP	GT1	GT2	GT3	GT4	GT5	GT6		
		GT8	GT9	etc.					
	BOTTOM	GB1	GB2	GB3	BG4	GB5	GB6		
		GB8	GB9	etc.					

Example:

MDBOLT	100	1025	11	11	30				
	TOP	101	102	103	104	105			
	BOTTOM	1	2	3	4	5			

Descriptor	Meaning
ID	Element ID of the bolt. (Integer; Required; no Default)
GRIDC	Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)
TOP	Enter the character string TOP to define the start of the entry that defines all of the grids at the “top” of the bolt intersection with the structure. (Integer; no Default)
MODC	Module containing the GRID entry for GRIDC. (Integer; Default=0, If MODC>0 then must be same as MODT or MODB)
MODT	Module containing the GRID entries for GTi. (Integer; Default=0)
MODB	Module containing the GRID entries for GBi. (Integer; Default=0)
GT1, GT2, etc.	Grid IDs of the grid points at the top of the bolt intersection. (Integer; no Default)
BOTTOM	Enter the character string BOTTOM to define the start of the entry that defines all of the grids at the “bottom” of the bolt intersection with the structure (do not enter the ID for GRIDC). (Integer; no Default)
GB1, GB2, etc.	Grid IDs of the grid points at the bottom of the bolt intersection. (Integer; no Default)

Remarks:

1. The MDBOLT entry must be defined in the main Bulk Data section only (Module 0).

2. See all remarks and figures under the BOLT entry description. All remarks and figures under BOLT apply to MDBOLT.

MDBULK**Module Type Definitions**

Defines module search options and typing such as repeated or mirrored.

Format:

1	2	3	4	5	6	7	8	9	10
MDBULK	MODID	TYPE	RMODID	METHOD	TOL				

Example:

MDBULK	14			AUTO	1.0e-3				
--------	----	--	--	------	--------	--	--	--	--

Descriptor	Meaning
MODID	Module identification number. (Integer > 0 or Character = "ALL")
TYPE	Module type (Character; default = "PRIMARY") <ul style="list-style-type: none"> PRIMARY Module with at least one GRID entry in its BEGIN MODULE section. REPEAT Copied Module via reposition or mirroring of a primary Module. See Remarks 9 and 10. MIRROR Copied Module via mirroring of a primary Module. See Remarks 9 and 10. MOVE Copied or primary Module repositioning via MDMOVE entry. See Remarks 8 and 10.
RMODID	Identification number of the reference Module, used only if making a copy; i.e., RMODID>0 and RMODID?MODID and TYPE= "REPEAT", "MIRROR" or "MOVE". (Integer>0; Default=0). See Remarks 9 and 10.
METHOD	Method to be used when searching for boundary grid points. (Character = "AUTO" or "MANUAL"; Default= "MANUAL")
TOL	Location tolerance to be used when search for boundary grid points. (Real; Default = 1.0e-5)

Remarks:

1. METHOD="MANUAL" requires MDCONCT entries. MDBNDRY and MDEXCLD, that reference MODID, will produce fatal messages.
2. MDCONCT, MDBNDRY and MDEXCLD entries can be used to augment the search procedure and/or override the global tolerance.
3. For combined automatic and manual boundary search, the METHOD="AUTO" should be specified and connections should be specified on a MDCONCT entry.
4. TOL is the default value that can be modified between two Modules by providing the required tolerance on the MDCONCT entry.

5. MDBULK may only be specified in the Main Bulk Data Section and cannot be placed in other BEGIN MODULE sections
6. This entry is only valid if Modules exist.
7. With MDBULK,ALL the TYPE and RMODID fields must be blank and MDBULK,ALL means that the MDBULK options METHOD and TOL will be applied to all Modules which are not already defined on an MDBULK entry.
8. TYPE="MOVE" refers to the enhanced interface. If TYPE="MOVE" then an MDMOVE Bulk Data entry must be defined. The MDMOVE entry references MDTRAN, MDROTi, and MDMIRi entries.
 - a. If TYPE="MOVE" is specified for a given Module, then MDLOC and MDMPLN cannot be specified and vice-versa.
 - b. If MDMIRi is referenced by MDMOVE then it must be the first MVID and cannot be specified as the second or subsequent MVIDs.
 - c. If MODID defines a secondary Module (RMODID>0) then it will automatically inherit all of the Primary Module's Bulk Data. The inheritance of a particular entry may be overridden by specifying new Bulk Data entries or ignored via the EXCLUDE Bulk Data entry in the secondary Module's Bulk Data section.
 - d. A primary Module (RMODID=0) may be re-positioned as indicated in Remark 10.
9. TYPE="REPEAT" and "MIRROR" refer to the classic interface. These types have many common features, but they also have some important differences. These are described below:
 - a. The following comments apply to both TYPE="REPEAT" and "MIRROR":
 - A Module (MODID) whose TYPE is "REPEAT" or "MIRROR" and RMODID is greater than 0 is referred to as a secondary Module. The RMODID specified in this case is regarded as the primary Module.
 - The primary Module may be re-positioned as indicated in Remark 10.
 - If MODID defines a secondary Module then it will automatically inherit all of the Primary Module's Bulk Data. The inheritance of a particular entry may be overridden by specifying new Bulk Data entries or ignored via the EXCLUDE Bulk Data entry in the secondary Module's Bulk Data section.
 - If both an MDLOC and MDMPLN entry are specified for a secondary Module, then a mirror image copy of the primary Module will be created by first using the plane defined by the MDMPLN entry and then repositioned at the location implied by the MDLOC entry.
 - b. The following comments apply only to TYPE= "REPEAT":
 - The secondary Module in this case may reference an MDLOC entry, an MDMPLN entry or both or none.
 - If it references an MDLOC entry, then an identical copy of its primary Module will be positioned at the location implied by the MDLOC entry.
 - If it references an MDMPLN entry, then a mirror image copy of the primary Module will be positioned using the plane defined by the MDMPLN entry.

- If it references neither an MDLOC entry nor an MDMPLN entry, then the secondary Module will merely be a duplicate of the primary Module positioned at the same location as the primary Module. This usage is extremely uncommon. Hence the program cautions the user about this usage by issuing a user warning message
 - c. The following comments apply only to TYPE= "MIRROR":
 - The secondary Module in this case must reference an MDMPLN. (Otherwise, the program will terminate the execution with an appropriate user fatal message.)
 - If the secondary Module references only an MDMPLN entry, then a mirror image copy of the primary Module will be positioned using the plane defined by the MDMPLN entry.
10. A Module whose TYPE is "PRIMARY" (or TYPE="MOVE" and RMODID=0) may itself be repositioned by the use of MDLOC or MDMPLN entry or both (or MDTRAN, MDROTi or MDIRi or any combination).
- a. If it references an MDLOC (or MDTRAN, MDROTi) entry, then the primary Module will be positioned at the location implied by the MDLOC (or MDTRAN, MDROTi) entry.
 - b. If it references an MDMPLN (or MDMIRi) entry, then a mirror image of the primary Module will be repositioned using the plane defined by the MDMPLN (or MDMIRi) entry.
 - c. If it references both an MDLOC (or MDTRAN, MDROTi) entry and an MDMPLN (or MDMIRi) entry, then a mirror image of the primary Module will first be created using the plane defined by the MDMPLN (or MDMIRi) entry and then repositioned at the location specified by the MDLOC (or MDTRAN, MDROTi) entry.

MDCONCT**Module Boundary Point Connections**

Explicitly defines grid and scalar connection procedures for a module to module operation.

Format:

1	2	3	4	5	6	7	8	9	10
MDCONCT	BID	TYPE	TOL	X	Y	Z	MODID	CID	
	MID1	GID1	MID2	GID2	MID3	GID3	MID4	GID4	
	MID5	GID5	etc.						

Alternate Format:

1	2	3	4	5	6	7	8	9	10
MDCONCT	BID	TYPE	TOL	GRID			MODID		
	MID1	GID1	MID2	GID2	MID3	GID3	MID4	GID4	
	MID5	GID5	etc.						

Example 1:

MDCONCT	123		0.4	0.5	74.12	100.3			
	10	423	20	123					

Example 2:

MDCONCT	123	RRBE2	0.4	777					
	10		20						

Descriptor	Meaning
BID	Unique Boundary identification number. (Integer > 0)
TYPE	Connection Option. (Character: “MERGE”, “RIGID”, “RRBE2”, “MRBE2” or Blank, Default=“MRBE2”).
TOL	Location tolerance to be used when searching or checking for boundary grid points. (Real; Default=1.0e-5)
X,Y,Z	Connection (Search) location. (Real, Default=0.0)
GRID	Grid Identification for Connection (Search). (Integer > 0) See Remark 11.
MODID	Module Identification number for the Location Reference. (Integer ≥ 0, Default=0)
CID	Coordinate System in MODID to be applied to Location. (Integer ≥ 0, Default=0)

Descriptor	Meaning
MIDI	Module Identification number for grid/scalar reference. (Integer ≥ 0)
GIDi	Identification number of a grid or scalar point in the reference module MIDI, which will be verified to be in TOL of the Location. (Integer ≥ 0)

Remarks:

1. MDCONCT may only be specified in the Main Bulk Data Section and cannot be placed in other BEGIN MODULE sections.
2. TOL can be used to override the default value specified on the MDBULK entries.
3. The continuation entry is optional when used in automatic searching.
4. The GIDi selections must be all of the same class, grids or scalar points.
5. All six degrees-of-freedom of grids will be defined as boundary degrees-of-freedom.
6. This entry is only valid if Modules exist.
7. When GIDi is blank or zero, then all grids of the module will be tested against the location and tolerance.
8. MIDI values can be specified more than once.
9. For TYPE=“MERGE”, the boundary grids will be moved to the location specified by MODID and connected by RBAR when only two boundary nodes are specified or RBE2 when more than two boundary nodes are specified.
10. For TYPE=“MRBE2”, the boundary grids will be moved to the location specified by MODID and only those grids that are specified as the independent grids on RBE2 elements will be used to connect the Modules. If any of the GIDi grids are not specified as independent grids on RBE2 elements or there are no RBE2 elements then the run will terminate with User Fatal Message 6717.
11. For TYPE=“RIGID”, the boundary grids will not be moved and will be connected via RBE2 elements.
12. For TYPE=“RRBE2”, the boundary grids will not be moved and only those grids that are specified as the independent grids on RBE2 elements will be used to connect the Modules. If any of the GIDi grids are not specified as independent grids on RBE2 elements or there are no RBE2 elements then the run will terminate with User Fatal Message 6717.
13. When GRID is used, then this identification/location will be used for the independent point of the RBE2 element.
14. When scalar points are specified for GIDi, then GRID, X, Y, Z, MODID and CID should not be specified.
15. If any GIDi are not connected due to exceeding TOL then User Fatal Message 6783 will be issued.

MDDMIG**Direct Matrix Input at Points Defined in Two or More Modules**

Defines direct input matrices related to grid, extra, and/or scalar points defined in two or more Modules. The matrix is defined by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
MDDMIG	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

MDDMIG	NAME	MODJ	GJ	CJ					
		MOD1	G1	C1	A1	B1			
		MOD2	G2	C2	A2	B2	-etc.-		

Example:

MDDMIG	STIF	0	1	3	4				
MDDMIG	STIF	11	27	1					
		20	2	3	3.+5	3+3			
		20	2	4	2.5+10	0.			
		45	50		1.0	0.			

Descriptor	Meaning
NAME	Name of the matrix. See Remark 1 under DMIG. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix input. IFO = 6 must be specified for matrices selected by the K2GG, M2GG, and B2GG Case Control commands. (Integer)
	1 Square
	2 or 9 Rectangular
	6 Symmetric
TIN	Type of matrix being input: (Integer)
	1 Real, single precision (One field is used per element.)
	2 Real, double precision (One field is used per element.)
	3 Complex, single precision (Two fields are used per element.)
	4 Complex, double precision (Two fields are used per element.)
TOUT	Type of matrix that will be created: (Integer)

Descriptor	Meaning
0	Set by precision system cell (Default)
1	Real, single precision
2	Real, double precision
3	Complex, single precision
4	Complex, double precision
POLAR	Input format of Ai, Bi. (Integer=blank or 0 indicates real, imaginary format; Integer > 0 indicates amplitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. See Remarks 5 and 6 under DMIG. (Integer > 0)
MODJ	Module IDs that contain the GRID entry for GJ. (Integer ≥ 0).
GJ	Grid, scalar or extra point identification number in Module MODJ for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer < 6; blank or zero if GJ is a scalar or extra point.)
MODi	Module IDs that contain the GRID entry for Gi. (Integer ≥ 0).
Gi	Grid, scalar, or extra point identification number in Module MODi for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ < 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

Remarks:

1. The MDDMIG entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the DMIG entry description. All remarks and figures under DMIG apply to MDDMIG.
3. When IFO=2 or 9, MODJ indicates the column number and not a module ID. GJ and CJ are ignored.

MDELAM

Delamination materials in SOL 600

Defines materials for which delamination may occur in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MDELAM		MID1	MID2	ITYPE	STRN	EXPN	STRT	EXPT	
	ITSTRN	ITEXPN	ITSTRT	ITEXPT					

Example:

MDELAM		4	5	2	35000.	1.	20000.	1.	
--------	--	---	---	---	--------	----	--------	----	--

Descriptor	Meaning
MID1	First material ID. (Integer; no Default; required field)
MID2	Second material ID. If MID2=MID1 delamination occurs within the material. If MID2 is not the same as MID1, delamination may occur at the interfaces between the materials. (Integer; Default = MID1)
ITYPE	Delamination type (Integer; Default = 1) <ul style="list-style-type: none"> 1 Determine if delamination occurs but do not actually allow delamination 2 Split the mesh when delamination occurs
STRN	Allowable normal stress (Real; no Default; required value)
EXPN	Exponent for normal stress (Real; Default = 1.0)
STRT	Allowable tangential stress (Real; no Default; required value)
EXPT	Exponent for tangential stress (Real; Default = 1.0)
ITSTRN	ID of a TABL3D entry providing the variation of STRN vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)
ITEXPN	ID of a TABL3D entry providing the variation of EXPN vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)
ITSTRT	ID of a TABL3D entry providing the variation of STRT vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)
ITEXPT	ID of a TABL3D entry providing the variation of EXPT vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)

Remarks:

1. The delamination criteria is based upon $(S_n/STRN)^{**EXPB} + (S_t/STRT)^{**EXPT} > 1.0$
Where S_n and S_t are the normal and tangential stresses
2. If no tables are required, the continuation entry may be omitted.

MDEXCLD**Module to Module Excluded Boundary Point Definitions**

Defines a list of grid points in a Module that will be excluded during the attachment to another Module.

Format:

1	2	3	4	5	6	7	8	9	10
MDEXCLD	MIDA	MIDB	GIDA1	GIDA2	GIDA3	GIDA4	GIDA5	GIDA6	
	GIDA7	GIDA8	etc.						

Example 1:

MDEXCLD	400	4	10	20	30	40			
---------	-----	---	----	----	----	----	--	--	--

Example 2:

MDEXCLD	400	ALL	10	20	30	THRU	35		
---------	-----	-----	----	----	----	------	----	--	--

Descriptor	Meaning
MIDA	Module identification number. (Integer ≥ 0 , Default=0)
MIDB	Reference Module identification number. (Integer ≥ 0 or Character "ALL"; Default=ALL)
GIDAi	Identification number of a boundary grid point in the module MIDA to be excluded from connection to module MIDB. (Integer > 0 or "THRU"; for "THRU" option, GID1 < GID2.)

Remarks:

1. This entry is only valid if Modules exist.
2. MDEXCLD may only be specified in the Main Bulk Data Section and cannot be placed in other BEGIN MODULE sections.

MDFAST

A Shell Patch Fastener Connection Between Two Modules

Defines a fastener with material orientation connecting two surface patches defined in two Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDFAST	EID	PID	TYPE	IDA	IDB	GS	GA	GB	
	XS	YS	ZS			MODA	MODB		

Example using PROP:

MDFAST	3	20	PROP	21	24	206			
							10	20	

Example using ELEM:

MDFAST	7	70	ELEM	27	74	707			
							11	12	

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PFAST entry. (Integer > 0; Default = EID)
TYPE	Specifies the surface patch definition: (Character) If TYPE = 'PROP', the surface patch connectivity between patch A and patch B is defined with two PSHELL (or PCOMP) properties with property ids given by IDA and IDB. Under CFAST see Remark 1. and Figure 8-28.
	If TYPE = 'ELEM', the surface patch connectivity between patch A and patch B is defined with two shell element ids given by IDA and IDB. Under CFAST see Remark 1. and Figure 8-28.
IDA, IDB	Property id (for PROP option) or Element id (for ELEM option) defining patches A and B in Modules MODA and MODB, respectively. IDA≠IDB (Integer > 0)
GS	Grid point defining the location of the fastener. Its GRID entry must be defined in the main Bulk Data section (Module 0). See also Remark 2. under CFAST. (Integer > 0 or blank)
GA, GB	Grid ids of piercing points on patches A and B. Their GRID entries must be defined in the main Bulk Data section (Module 0). See also Remark 2 under CFAST. (Integer > 0 or blank)

Descriptor	Meaning
XS,YS,ZS	Location of the fastener in basic. Required if neither GS nor GA is defined. See also Remark 2. under CFAST. (Real or blank)
MODA,M ODB	Module IDs that contain the surface patches defined by property or element entries IDA and IDB (Integer ≥ 0).

Remarks:

1. The MDFAST entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the CFAST entry description. All remarks and figures under CFAST apply to MDFAST.

MDLABEL**Module Output Label**

Defines a label or name to be printed in the Module's results output page headings.

Format:

1	2	3	4	5	6	7	8	9	10
MDLABEL	MID			LABEL					

Example:

MDLABEL	400	LEFT REAR FENDER, MODEL XYZ2000	
---------	-----	---------------------------------	--

Descriptor	Meaning
MID	Module identification number. (Integer ≥ 0 , Default=0)
LABEL	Label associated with Module MID for results output page headings. (Character)

Remarks:

1. MDLABEL can only be specified in the main Bulk Data Section and is ignored in-between the BEGIN MODULE=n and ENDMODULE commands.
2. Only one MDLABEL per Module may be specified.
3. The label will appear in all Module's results output page headings. However, in some headings the label may be truncated.
4. This entry is valid only if Modules exist.

MDLOC**Module Reposition by Translation and/or Rotation**

Defines reposition of a Module by listing three non-collinear points in the Module and three corresponding points in Module 0.

Format:

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

Example:

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

Descriptor	Meaning
MODID	Module identification number of primary, repeated, or mirrored module (Integer>0).
PAi	GRID or POINT identification numbers of three non-collinear points in (Integer>0): <ul style="list-style-type: none"> ■ "RMODID's Bulk Data section. RMODID is referenced on MDBULK if this MODID is a secondary (copied) Module ■ "Module MODID's Bulk Data section if this is a primary Module.
PBi	GRID or POINT identification numbers of three non-collinear points in Module 0 (main Bulk Data section). Must have the same relative locations as PAi. (Integer>0).

Remarks:

1. MDLOC can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. The Module will be rotated and translated for alignment of the PAi and PBi locations.
3. The PAi and PBi can be either GRIDs or POINTs.
4. PA1, PA2, and PA3 must be contained in the BEGIN Module=MODID Bulk Data section.
5. PB1, PB2, and PB3 must be specified in the main Bulk Data Section or Module 0. If they belong to a Module that is also relocated, then the original (unmoved) positions of PB1, PB2, and PB3 are used.
6. PB1, PB2, and PB3 must have the same relative locations as PA1, PA2, and PA3.
7. Three GRIDs or POINTs are required even if the Module connects to only one or two boundary grids.
8. Coordinate systems, global displacement directions, and element coordinate systems for the Module will rotated and translated.

9. The global coordinate directions of the grid points in the Module will be transformed internally to the global coordinate directions of the grid points in the Module 0. For displacement data recovery, the output will be in the original global coordinate system.
10. The translation and rotation of the Module to the new position is accomplished by defining local rectangular coordinate systems based on the specified grid locations:
 - The local systems have their origin at PX1 and the x-axis points from PX1 to PX2.
 - The y-axis lies in the plane containing PX1, PX2, and PX3, is perpendicular to the x-axis, and points toward PX3.
 - The z-axis is defined by the cross product of the x-axis into the y-axis.
 - The rotation and translation transformation aligns the local system defined by the Module grids with the local system defined by the main Bulk Data Section grids
11. This entry will only work if Modules (BEGIN MODULE) exist.

MDLPRM**Model Parameters**

Specifies parameters which affect the solution of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
MDLPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	-etc.-		

Example:

MDLPRM	QR6ROT	2	QRSHEAR	1					
--------	--------	---	---------	---	--	--	--	--	--

Descriptor	Meaning
PARAMi	Name of the parameter. Allowable names are given in Table 24 . (Character)
VALi	Value of the parameter. (Real or Integer; see Table 24)

Remark:

1. Multiple entries of MDLPRM are allowed in the Bulk Data Section. However, multiple entries of a particular parameter PARAMi are illegal.
2. This entry is not supported in SOL 600.
3. If Modules are present then this entry may only be specified in the main Bulk Data section.

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
BRTOBM	Flag to determine whether to convert a CBAR element to a CBEAM element for the nonlinear analysis.	
	0	Do not convert CBAR to CBEAM. (Default)
	1	Convert CBAR to CBEAM.
	-1	Same as 1, but print the converted Bulk Data entries on f06 file.
BUSHRT	Flag to control large rotation effects for CBUSH or CFAST elements in nonlinear analysis (SOL400). For CBUSH, this flag is overridden by the LRGR field on the element PBUSHT entry.	
	0	Large rotation effects are included. Element coordinate system is rotated with the rotation of grid A. (Default)
	1	No large rotation effects. Small rotation is used.
	2	Large rotation effects are included. A mid-increment method is used to rotate the element system. This is recommended for large rotation analysis. For the CBUSH element, CID must equal zero.

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
COMP1	Control if stresses and strains of advanced composite elements use MSC Nastran conventional layer format	
	NO(default)	Don't use
	YES	Use conventional layer format
DBCTOLE	Tolerance for DBC date conversion, real(default=1.e-4) >= 0.0	
DCFLTEXP	Determines the exponent of the tolerance value used to filter out small entries from the coefficient matrix before a matrix decomposition. In each column, matrix entries which are DCFLTEXP orders of magnitude smaller than the diagonal entry of the column are filtered out. If DCFLTEXP=0, then the coefficient matrix is not altered. (Integer; Default = 0)	
DELELAS	Randomly delete c*100% of CELASi elements in a job. (0.0 ≤ c ≤ 1.0; Default: c = 0.02)	
DELFAST	Randomly delete c*100% of CFAST elements in a job. (0.0 ≤ c ≤ 1.0; Default: c = 0.02)	
DELMASS	Randomly delete c*100% of CMASSi, CONM1, and CONM2 elements in a job. (0.0 ≤ c ≤ 1.0; Default: c = 0.02)	
DELSEAM	Randomly delete c*100% of CSEAM elements in a job. (0.0 ≤ c ≤ 1.0; Default: c = 0.02)	
DELWELD	Randomly delete c*100% of CWELD elements in a job. (0.0 ≤ c ≤ 1.0; Default: c = 0.02)	
GEV1417	See Remark 9 of PBUSH	
	0	Use GE default rules on PBUSH. (Default)
	1	Use Nastran Version 2014-2017 incompatible rule for GE defaults.
GNLSTN	Strain formulation flag for QUADR/TRIAR elements used in geometric nonlinear analysis. (Parameter LGDISP=1)	
	0	Small strain. (Default)
	1	Green strain.
HDF5	Parameter to create NH5RDB database	
	-1	Do not create NH5RDB(default)
	0	Create NH5RDB without compression
	1	Create NH5RDB with compression
	2	Create uncompressed NH5RDB database without input data.
	3	Create compressed NH5RDB database without input data.

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
H5GM34	Write GEOM3 and GEOM4 data in NH5RDB	
	1	Write GEOM3 and GEOM4 data in NH5RDB.
	0	Do not write GEOM3 and GEOM4 data in NH5RDB.
	-1	Use OP2GM34 setting for GEOM3 and GEOM4 output (Default).
H5INFO	Write job run information in NH5RDB or not.	
	1	Yes (Default)
	0	No
H5MDL	Write model input data in separate file	
	1	Write model input data into a separate file
	0	Do not write model input data into a separate file (Default)
H5MTX	Write matrix data in separate file	
	1	Write matrix data into a separate file
	0	Do not write matrix data into a separate file (Default)
H5NORDOF	Parameter to suppress output rotational components to NH5RDB database	
	0	Output both translational and rotational components to NH5RDB database (Default).
	1	Do not output rotational components to NH5DB database. Only used when HDF5 is 0 or 1.
H5XHH	Write BHH, MHH and KHH matrices in NH5RDB	
	0	Do not write BHH, MHH and KHH in NH5RDB (Default)
	1	Write BHH, MHH and KHH in NH5RDB
IGNSHBDN	Check adjacent shell elements have opposite direction of normal vector, if found	
	0(default)	Issue a user fatal message
	1	Issue a user warning message
INTOUT	Flag to control FORCE/STRESS/STRAIN OUTPUT location for QUADR/TRIAR elements	
	0	Corner output. (Default)
	1	Integration point output.

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
	Integer	Meaning
LMT2MPC, INTEGER INPUT	Parameter to enable the use of elimination method to process Lagrange Multipliers.	
	0	Do not force the use of elimination method. (Default)
	1	Use elimination method to process Lagrange Multipliers using unsymmetric method.
	2	Use elimination method to process Lagrange Multipliers using symmetric method (may be faster).
MLTSPLIN	Parameter to specify whether an aerodynamic grid can be splined more than once.	
	0	References on separate splines to the same aero grid are not allowed. (Default)
	1	Aero grids can be referenced on multiple spline entries.
MPCF129	Request the calculation of MPC forces in SOL129. The forces include only the contributions from linear elements.	
	0	Don not compute MPC Forces (default)
	1	Compute MPC Forces
NLDIFF	Flag to determine whether the differential stiffness matrix and follower force stiffness are to be computed for nonlinear elements with geometric nonlinear analysis (parameter LGDISP=1) in SOL 400. Options 3, 4, and 5 are available only for elements with PSHNL1, PSHNL2, PSLN1, or PSHEARN Bulk Data entry.	
	0	Compute. (Default except for CTRLDEF=SEVERELY of NLSTEP and not 3D contact analysis)
	1	Do not compute
	2	The differential stiffness matrix and follower force stiffness will not be computed if the tangential stiffness matrix is negative definite (Default for CTRLDEF=SEVERELY of NLSTEP and not 3D contact analysis).
	3	Include only the deviatoric part of the differential (or initial stress or geometric) stiffness
	4	Include only the tensile part of the differential (or initial stress or geometric) stiffness
	5	Include the stress at the beginning of the increment for the differential (or initial stress or geometric) stiffness

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
NONUPIV	Parameter to select the numeric compute kernel and pivoting methods in MSCLDL and MSCLU sparse direct solvers.	
	0	Use the native Bunch-Kauffman threshold pivoting in MSCLDL, and the native threshold partial pivoting in MSCLU (Default).
	1	Use no numeric pivoting in MSCLDL and MSCLU. BLAS3 TRSMs are called to compute the pivot column update to improve performance. Ill-conditioned models may die of “singular matrix” during sparse factorization.
	2	Not documented.
	3	LAPACK SYTRFs with Bunch-Kaufman pivoting and GETRFs with partial pivoting are called to perform factorizations, and BLAS3 TRSMs are called to compute pivot column update to improve performance.
NSGRDS4	Number of structural grids to be used in dividing a SPLINE4 using the RIS spline method. The spline will be divided into NGRIDS/NSGRDS4 regions, where NGRIDS is the number of grids listed on the associated SET1 entry. (Integer > 0, default=0. If NGRIDS < NSGRDS4, or NSGRDS4 is not specified, no divisions will occur.)	
OFFDEF	Element offset definition. A flag to determine how shell elements and bar and beam elements behave when the user supplies ZOFF values on the shell connection entries (CQUAD4, CQUADR, CTRIA3, CTRIAR, CQUAD8, and CTRIA6) and WiA and WiB on CBAR, CBEAM, and CBEAM3 connection entries. (Character)	
	ELMOFF	Standard Nastran offset method. The ZOFF rotate with the shell element. The WiA and WiB offsets for beams are fixed. MD Nastran R3 and earlier. (Character, Default)
	LROFF	Large rotation offsets. The shell normal directions are used to define the offset direction at each shell grid for CQUAD4, CTRIA3, CQUADR, CTRIA4, CQUAD8 and CQUAD8 elements. For beams, WiA and Wib defines the offset direction. This method allows for thermal load effects on ZOFF for shells and WiA and WiB for beams. Thermal load effect for offset is computed based on the grid point or element temperature, and thermal coefficient of the element (see NOTHRM). The mass moment of inertia is computed for the offset due to the grid point location change introduced by offset. Differential stiffness is computed for the offset using the same method as that of the Lagrange formulation of the RBAR. Not supported in SOL106, SOL129 and SOL 200. (Character)

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
NODIF	LROFF is used but the differential stiffness effect is turned off. (Character)	
NOTHRM	LROFF is used but the thermal load effects are turned off. The thermal load has two effects: 1) the location of thermal load changes due to offset and 2) the length of offset changes due to thermal load. Effect (1) is computed for all solution sequences and Effect (2) is computed for SOL 400 only. Both effects are turned off by NOTHRM. (Character)	
NODT	LROFF is used but the differential stiffness and thermal load effect are turned off. (Character)	
ELMZ	LROFF is used but the element z-direction is used for the offset direction. IF PARAM, SNORM, 0.0 or the computed value for SNORM is greater than the PARAM,SNORM,value, then the LROFF option will revert to this method for CQUAD4, CTRIA3, CQUADR, CTRIAR, CQUAD8 and CTRIA6. This option is not applicable for beams. (Character)	
NOMASS	LROFF is used but the no mass effects are included. (Character)	
NDMTZ	LROFF is used but the element z-direction is used for the offset direction and the differential stiffness, the thermal load effects, and the mass effects are turned off. For CQUAD4 and CTRIA3 elements this method should get similar results to the standard ELMOFF method. (Character)	
	Note: This entry only effects ZOFF calculations for ZOFF specified on the shell connection entries. For Z0 specified on the PCOMP or PCOMPG entries, the standard ELMOFF method will be used.	
	If the computed value for SNORM is greater than the PARAM,SNORM,value and the user wishes not to change the parameter value, the Bulk Data entry SNORM can be used to override the shell normal.	
	Solution sequences affected: For linear - all solution sequences. For nonlinear - SOL 400 only. <i>The LROFF method is not implemented into SOLs 106 and 129.</i>	
PEXTS4	Used in conjunction with NSGRDS4. After partitioning the spline, each of the smaller splines will be extended by PEXTS4 in each direction (top,bottom, left and right). The value is expressed in percent so that PEXTS4=10.0 would extend the four boundaries by 10%. Real (0.0<PEXTS4<100.0, default=10.0)	

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
PIVTHRSH	Parameter to set the threshold ‘uu’ for numeric pivot selection in the MSCLU and Intel MKL Paradiso solvers.	
	0 (Default)	uu is set to <ul style="list-style-type: none"> ■ MSCLU: 10^{-9} ■ MKL Pardiso: 10^{-13} for non-symmetric matrices and 10^{-8} for symmetric indefinite matrices. For more general information, see Intel MKL Pardiso iparm Parameter Table, iparm(10).
	-i	uu is set to 10^{-i} for both MSCLU and MKL Pardiso (where i is between 1 and 20)
PRDIDPVT	Control of pivoting in Intel MKL Pardiso, when solving symmetric indefinite matrices. For more information, see Intel MKL Pardiso iparm Parameter Table , iparm(21).	
	0	1x1 diagonal pivoting
	1	1x1 and 2x2 Bunch-Kaufman pivoting (Default)
	2	1x1 diagonal pivoting without automatic iterative refinement
	3	1x1 and 2x2 Bunch-Kaufman pivoting without automatic iterative refinement
PRDITRFN	Control of iterative refinement step for Intel MKL Pardiso. For more information, see Intel MKL Pardiso iparm Parameter Table , iparm(8).	
	0	Solver automatically performs two steps of iterative refinement (Default)
	>0	Maximum number of iterative refinement steps, up to a maximum of nine.
	<0	Same as above, but residual is computed using extended precision. Maximum of nine steps.

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
PRDMTYPE	Control of matrix type for Intel MKL Pardiso. For more information, see Intel MKL Paridso documentation , Input Parameters.	
	1	real structurally symmetric
	2	real symmetric positive definite (SOL 101 default)
	-2	real symmetric indefinite (SOL 400 default)
	3	complex structurally symmetric
	4	complex Hermitian positive definite
	-4	complex Hermitian indefinite
	6	complex symmetric (SOL 108 & 111 default)
	11	real unsymmetric
	13	complex unsymmetric (SOL 107 default)
PRDOOC	Control of out-of-core (OOC) solution algorithm in Intel MKL Pardiso. For more information, see Intel MKL Paridso iparm Parameter Table , iparm(60).	
	0	In-core mode
	1	Choose between in-core and OOC mode based on memory (opencore) size
	2	Out-of-core mode
PRDWMTCH	Control of weighted matching algorithm in Intel MKL Pardiso. For more information, see Intel MKL Paridso iparm Parameter Table , iparm(13).	
	0	Disable matching (SOL 101 default)
	1	Enable matching (SOL 107, 108, 111, and SOL 400 default)
PRTELAS	Print list of ID's of CELASI elements that are deleted.	
	NO (or blank)	Turn off the print. (Default)
	YES	Turn on the print.
PRTFAST	Print list of ID's of CFAST elements that are deleted.	
	NO (or blank)	Turn off the print. (Default)
	YES	Turn on the print.
PRTMASS	Print list of ID's of CMASSi, CONM1 and CONM2 elements that are deleted.	
	NO (or blank)	Turn off the print. (Default)
	YES	Turn on the print.

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
PRTSEAM	Print list of ID's CSEAM elements that are deleted.	
	NO (or blank)	Turn off the print. (Default)
	YES	Turn on the print.
PRTWELD	Print list of ID's of CWELD elements that are deleted.	
	NO (or blank)	Turn off the print. (Default)
	YES	Turn on the print.
QR6ROT	Parameter to determine whether the drilling degrees-of-freedom are to be deactivated for QUADR/TRIAR elements. If the drilling degrees-of-freedom are deactivated, the QUADR/TRIAR become elements similar to QUAD4/TRIA3. QR6ROT has the following values:	
	0	The drilling degrees-of-freedom are active. (Default)
	1	The drilling degrees-of-freedom are deactivated for all QUADR/TRIAR element in the model.
	2	The drilling degrees-of-freedom are deactivated for those QUADR/TRIAR which have membrane stiffness only (MID2 and MID3 are blank on the PSHELL entry)
QRSHEAR	Parameter to select the off-plane shear formulation for the QUADR element. There are two types of off-plane shear formulations: the stiffness method and the flexibility method. The stiffness method is a new method implemented in QUADR. The flexibility method was the method implemented in the QUAD4 element. Therefore, if the flexibility method is selected, the solution results of QUADR are closer to those of QUAD4. QRSHEAR has the following values:	
	0	Use stiffness method if MID3 ≠ 0 on the PSHELL Bulk Data entry. Use the flexibility method if MID3 = 0. (Default)
	1	Use flexibility method.
	2	Use the stiffness method.

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
RDBOTH	Parameter to select Rayleigh damping approach for rotordynamics (compatibility with MSC Nastran 2005) implementation, Integer. A cumulative sum can be provided in case multiple features are desired in the analysis.	
0	Uses implementation for Rayleigh Damping as described in RSPINR/RSPINT entry description (Default)	
1	Switch to V2005 implementation of Rayleigh damping where damping coefficients specified in the model through "PARAM, ALPHA1" and "PARAM, ALPHA2" are applied to the complete model and Rayleigh damping specified through "ALPHAR1" and "ALPHAR2" in RSPINR/RSPINT is set to 0.0.	
2	Ignore circulation effects in rotodynamic analysis.	
4	Include effect of stress stiffening using method = 1 (see RFORCE entry)	
8	Include effect of stress stiffening using method = 2 (see RFORCE entry)	
16	This option treats structural damping for rotors using complex stiffness matrix for complex eigenvalue analysis. Here, circulation terms are ignored in the analysis.	
RELAXF	If there are SPRELAX entries, RELAXF=1 will result in the GI module outputting the GPGK datablock without relaxation while the GDGK datablock will include the relaxation effects. =0 (Default) Relaxation is applied to all splines =1 Relaxation is only applied to the splining of displacements.	
REUPSE	Choose method for handling rigid elements in superelements when RIGID=LAGRAN or LGELIM	
0	(Default) Automatic selection: a. If ALPHA is zero for ALL r-elements in ALL superelements (SEID>0) then keep of these r-elements in the superelements with RIGID=LINEAR formulation. b. If ALPHA is not zero for ANY r-element in ANY superelement (SEID>0) then transfer ALL r-elements in ALL superelements to the residual structure.	
1	Transfer all r-elements in the superelements to the residual structure.	

Table 24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
RMRBE3RT	Parameter to control if remove rotational dof from REFC of any RBE3 which reference grid point(REFG) is used by solid elements only and REFC has rotational dof..	
	0(default)	Don't remove.
	1	Remove rotational dof.
RSTIGNDP	A flag to determine if ignoring duplicate PARAM entries between a restart run and its initial run.	
	0(default)	Don't ignore duplicating.
	>0	Ignore duplicating.
SHEARP	Parameter to select the new HARDER shear panel over the classic GARVEY panel. The HARDER panel through testing appears to be superior to the GARVEY panel especially in cases of thermal loading. (Character-all required)	
	GARVEY	GARVEY SHEAR PANEL (Default)
	HARDER	HARDER SHEAR PANEL - NEW
SHRTOQ4	Flag to determine whether to convert a CSHEAR element to a CQUAD4 element for nonlinear analysis. Cannot be used with the PSHEARN entry.	
	0	Do not convert CSHEAR to CQUAD4. (Default)
	1	Convert CSHEAR to CQUAD4.
	2	Do converting and print out converted Bulk Data entries on f06 file.
SPBLNDX	Factor to be applied to D1 and D2 blend depths on the SPBLND1 and SPBLND2 bulk data entries for determining the structural grids in the blended region. (Real, ≥ 1.0 , default = 1.0)	
STREQCNT	Flag to control calculation method for equivalent von Mises stress/strain at element center for advanced nonlinear solid elements in SOL 400.	
	0	Average integration point equivalent stress/strain (Default).
	1	Average integration point stress/strain components then compute equivalent value at element center.
TWBRBML	Parameter to select method for computing properties of PBARL/PBEAML.	
	0	Select Finite Element Method. (Default)
	1	Select Beam Library Equations.

MDMIAUX**Matrices from Marc - SOL 600**

Specifies the DOMAINsolver command to be used in conjunction with secondary spawned jobs when MDMIOUT is used. SOL 600 only. (See the MDMIOUT Bulk Data entry.)

Format:

1	2	3	4	5	6	7	8	9	10
MDMIAUX	ID								
	“FMS”	STRF1							
		STRF2							
	“EXEC”	STRE1							
		STRE2							

Example:

MDMIAUX	2								
	FMS		ASSIGN OUTPUT2=MODEL1.OP2,UNIT=50,DELETE						
	EXEC		DOMAINsolver ACMS (PRINT=YES,UPFACT=3.5)						

Descriptor	Meaning
ID	ID of a matching IAUX field on the corresponding MDMIOUT entry. (Integer; no Default)
“FMS”	Enter the string FMS to start a series of FMS strings that should appear before the SOL entry for the secondary spawned job. (Character; no Default)
STRFi	Series of FMS strings, entry as many as desired. (Character; no Default)
“EXEC”	Enter the string EXEC to start a series of strings that should appear after the SOL entry for the secondary spawned job. (Character; no Default)
STREi	Series of Executive Control strings, enter as many as desired. (Character; no Default)

Remarks:

1. This entry is not active unless a matching MDMIOUT entry is found in the input.
2. STRFi and STREi may occupy fields 3-9. (columns 9 to 72).
3. Standard FMS and Executive Control continuation rules apply to the strings.

MDMOUT

Matrices from Marc - SOL 600

Defines full or reduced stiffness and mass matrices to be output from the Marc portion of SOL 600. This entry may be used to generate External Superelements using DMIG Matrices or an MSC Adams MNF File from the Marc portion of a SOL 600 analysis. SOL 600 only. (See the MNF600 and DMIGOUT Bulk Data entries.)

Format:

1	2	3	4	5	6	7	8	9	10
MDMOUT	ID	IDOF	G1	THRU	G2	ITYPE	NAME	IAUX	
	IDOF2	G3	THRU	G4	IDOF3	G5	THRU	G6	
	etc.								

Example:

MDMOUT	100	123456	1	THRU	5456	1	m343	1	
--------	-----	--------	---	------	------	---	------	---	--

Descriptor	Meaning
ID	Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example, if the case control contains SUBCASE 20, ID would be 20. (Integer; Default = 1)
IDOF, IDOF2	List of DOF's to be output (any or all of the integers 1-6 are acceptable). (Integer; Default = 123456)
G1, G3	Starting grid ID for reduced matrices. (Integer; Required; no Default)
G2, G4	Ending grid ID for reduced matrices. (Integer; Required; no Default)
ITYPE	Type of file to generate. (Integer; Default = 1) <ul style="list-style-type: none"> 1 DMIG Matrices 2 Adams MNF
NAME	Name of file containing the output (Character; no Default; limited to 8 characters, should be in lower case, the extension .dmi is added automatically)
IAUX	ID of a MDMIAUX entry if the secondary run requires the use of file management entries or additional Executive Control statements such as the DOMAININSOLVER option for parallel processing. (Integer; Default = 0) A value of zero means that no auxiliary entries are required.
ISOL	Solution sequence to run using the DMIG matrices. To speed up the solution, use DOMAININSOLVER ACMS (PARTOP=DOF) for eigenvalues and set ISOL to the negative value of the solution sequence desired (-103, -111 or -112). (Integer absolute value > 100; Default = 0 which means do not run any solution sequence using the DMIG's created by Marc in this execution)

Remarks:

1. The continuation line(s) are not required.
2. This entry corresponds to Marc's entry, SUPERELEM with a value of 1 in the second line 4th field and produces DMIG's or an MDF file for the initial geometry prior to any nonlinear iterations.
3. DMIG output will be in jid.marc_dmigst_0001.
4. The reduced matrices may be used in the Nastran analysis for eigenvalue extraction or any other purpose by invoking the CONTINUE=5 option on the SOL 600 entry.
5. If the SOL 600 CONTINUE options is invoked, case control commands and a bulk data entry include statements to receive the matrices will be automatically added to the original input data file. A second Nastran execution will be spawned from the original Nastran execution after completion of the Marc execution.
6. ID must be 106 or 129 in the Executive Control statement, SOL 600.ID.
7. Only one MDMOUT entry should be entered per run. If more are entered, only the first will be used.
8. MNF controls for other solution sequences are ignored for SOL 600.
9. For the case where DMIG's are generated and a continuation option is used, the following Bulk Data parameters are usually required in addition to the MDMOUT entry:
\$2345678x234567890123456x34567890123456
param marcfile nastb.rc
param marcfil1 m3356
10. For a more general form of the DMIG output, see Bulk Data entry, DMIGOUT.

MDMIR1

Module Reposition by Mirroring - Option 1

Defines reposition of a Module with a mirror plane defined by three non-collinear points in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDMIR1	MVID	P1	P2	P3					

Example:

MDMIR1	110	12	45	1125					
--------	-----	----	----	------	--	--	--	--	--

Descriptor	Meaning
MVID	Move identification number to be referenced on an MDMOVE entry. (Integer>0).
Pi	GRID or POINT identification numbers of three non-collinear points in Module 0. (Integer>0).

Remarks:

1. MDMIR1 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. MDMIR1 must be referenced by an MDMOVE entry in order to reposition a Module.
3. GRIDs or POINTs referenced on this entry must be defined in the main Bulk Data Section (Module 0).

MDMIR2**Module Reposition by Mirroring - Option 2**

Defines a Module mirror by specifying a pair of coordinate system axes on the mirror plane in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDMIR2	MVID	CID	RID	AXES					

Example:

MDMIR2	110		45	XZ					
--------	-----	--	----	----	--	--	--	--	--

Descriptor	Meaning
MVID	Move identification number to be referenced on an MDMOVE entry. (Integer>0).
CID	Coordinate system identification number of a CORDij entry. Blank or zero means basic coordinate system. Module will be mirrored about CID's origin if RID is blank. (Integer ≥ 0 , default=0)
RID	GRID or POINT identification number or coordinates of reference point through which mirror plane will be applied. If RID is blank, then the origin of CID will be used (Integer ≥ 0 , default=0)
AXES	Coordinate system axes pair: "XY", "XZ", "YZ", "YX", "ZX", or "ZY" (Character).

Remarks:

1. MDMIR2 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. MDMIR2 must be referenced by an MDMOVE entry in order to reposition a Module.
3. CORDij, GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

MDMOVE

Defines a Module repositioning sequence

Defines a Module repositioning sequence by referencing MDMIRi, MDROTi, and MDTRAN entries.

Format:

1	2	3	4	5	6	7	8	9	10
MDMOVE	MODID	MVID1	MVID2	MVID3	-etc.-				

Example:

MDMOVE	40	10	11	12					
--------	----	----	----	----	--	--	--	--	--

Descriptor	Meaning
MODID	Module identification number of a primary or copied module (Integer>0)
MVIDi	Move identification number of an MDTRAN, MDROTi or MDMIRi Bulk Data entry (Integer>0).

Remarks:

1. MDMOVE can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. In order to use MDMOVE, an MDBULK entry with the same MODID must be specified with TYPE="MOVE".
3. If MDMIRi is referenced on an MDMOVE then its MVID must be specified in the MVID1 field and not in the second or subsequent MVIDi fields.

MDMPC**Multipoint Constraint Between Two or More Modules**

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

where u_j represents degree-of-freedom Cj at grid or scalar point Gj. Gj may be contained in different Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDMPC	SID	MOD1	G1	C1	A1				
	MOD2	G2	C2	A2	MOD3	G3	C3	A3	
	MOD4	G4	C4	A4	-etc-				

Example:

MDMPC	3	10	28	3	6.2				
	11	2		4.29	21	1	4	-2.91	

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
MODj	Module IDs that contain the GRID entries for Gj. (Integer ≥ 0).
Gj	Identification number of grid or scalar point. (Integer > 0)
Cj	Component number. (Any one of the Integers 1 through 6 for grid points; blank, zero or 1 for scalar points.)
Aj	Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)

Remarks:

1. The MDMPC entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the MPC entry description. All remarks and figures under MPC apply to MDMPC.

MDMPLN**Module Reposition by Mirroring**

Defines reposition of a Module with a mirror plane defined by three non-collinear points in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDMPLN	MODID	P1	P2	P3					

Example:

MDMPLN	110	12	45	1125					
--------	-----	----	----	------	--	--	--	--	--

Descriptor	Meaning
MODID	Module identification number of a primary or secondary (copied) module (Integer>0).
Pi	GRID or POINT identification numbers of three non-collinear points in Module 0 (Integer>0).

Remarks:

1. MDMPLN can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MDMPLN=n command.
2. GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

MDRBE2**Rigid Body Element Between Two or More Modules, Form 2**

Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points. All grid points may be defined in different Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDRBE2	EID	MODN	GN	CM	MOD1	GM1	MOD2	GM2	
	MOD3	GM3	MOD4	GM4	MOD5	GM5	MOD6	GM6	
	MOD7	GM7	MOD8	GM8	-etc.-	ALPHA			

Example:

MDRBE2	9	101	8	12	201	10	202	12	
	202	14	203	15	203	16	6.5-6		

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$).
MODN	Module ID that contains the GRID entry for GN. ($\text{Integer} \geq 0$).
GN	Identification number of grid point to which all six independent degrees-of-freedom for the element are assigned. ($\text{Integer} > 0$)
CM	Component numbers of the dependent degrees-of-freedom in the global coordinate system at grid points GMi. See Remark 12. under RBE2. (Integers 1 through 6 with no embedded blanks.)
MODi	Module IDs that contain the GRID entries for GMj. ($\text{Integer} \geq 0$).
GMi	Grid point identification numbers at which dependent degrees-of-freedom are assigned. ($\text{Integer} > 0$)
ALPHA	Thermal expansion coefficient. See Remark 11. under RBE2. (Real or blank)

Remarks:

1. The MDRBE2 entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RBE2 entry description. All remarks and figures under RBE2 apply to MDRBE2.

MDRBE3**Interpolation Constraint Element Between Two or More Modules**

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

Format:

1	2	3	4	5	6	7	8	9	10
MDRBE3	EID	REFMOD	REFGRID	REFC	WT1	C1	MOD1,1	G1,1	
	MOD1,2	G1,2	MOD1,3	G1,3	WT2	C2	MOD2,1	G2,1	
	MOD2,2	G2,2	-etc.-	WT3	C3	MOD3,1	G3,1	MOD3,2	
	G3,2	-etc.-	WT4	C4	MOD4,1	G4,1	MOD4,2	G4,2	
	-etc.-								
	"UM"	MOD1	GM1	CM1	MOD2	GM2	CM2		
		MOD3	GM3	CM3	-etc.-				
	"ALPHA"	ALPHA							

Example:

MDRBE3	14	1000	100	1234	1.0	123	101	1	
	102	3	103	5	4.7	1	101	2	
	102	4	103	6	5.2	2	101	7	
	102	8	103	9	5.1	1	104	15	
	105	16							
	UM	101	100	14	102	5	3		
		103	7	2					
	ALPHA	6.5-6							

Descriptor	Meaning
EID	Element identification number. Unique with respect to all elements. (0 < Integer < 100,000,000)
REFMOD	Module IDs that contain the GRID entry for REFGRID. (Integer ≥ 0).
REFGRID	Reference grid point identification number. (Integer > 0)
REFC	Component numbers at the reference grid point. (Any of the integers 1 through 6 with no embedded blanks.)
WTi	Weighting factor for components of motion on the following entry at grid points Gi,j. (Real)
Ci	Component numbers with weighting factor WTi at grid points Gi,j. (Any of the integers 1 through 6 with no embedded blanks.)
MODi,j	Module IDs that contain the GRID entries for Gi,j. (Integer ≥ 0).

Gi,j	Grid points with components Ci that have weighting factor WTi in the averaging equations. (Integer > 0)
"UM"	Indicates the start of the degrees-of-freedom belonging to the dependent degrees-of-freedom. The default action is to assign only the components in REFC to the dependent degrees-of-freedom. (Character)
MODk	Module IDs that contain the GRID entries for GMk. (Integer ≥ 0).
GMi	Identification numbers of grid points with degrees-of-freedom in the m-set. (Integer > 0)
CMi	Component numbers of GMi to be assigned to the m-set. (Any of the Integers 1 through 6 with no embedded blanks.)
"ALPHA"	Indicates that the next number is the coefficient of thermal expansion. (Character)
ALPHA	Thermal expansion coefficient. See Remark 14. under RBE3. (Real or blank)

Remarks:

1. The MDRBE3 entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RBE3 entry description. All remarks and figures under RBE3 apply to MDRBE3.

MDRJNT**Rigid Joint Between Two Modules**

Defines a rigid joint element connecting two coinciding grid points in two different Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDRJNT	EID	MODA	GA	MODB	GB	CB			

Example:

MDRJNT	5	11	1	21	2	12345		1	
--------	---	----	---	----	---	-------	--	---	--

Descriptor	Meaning
EID	Element identification number. (Integer > 0)
MODA,MODB	Module IDs that contain the GRID entries for GA and GB in Modules MODA and MODB, respectively. (Integer ≥ 0).
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB. These degrees-of-freedom are constrained to move with the same degrees-of-freedom at GA. See Remarks 4. and 5. under RJOINT. (Integers 1 through 6 with no embedded or blank.)

Remarks:

1. The MDRJNT entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RJOINT entry description. All remarks and figures under RJOINT apply to MDRJNT.

MDROT1**Module Reposition by Rotation - Option 1**

Defines a Module rotation by specifying a rotation vector and reference point in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDROT1	MVID	RID	IDS	IDE	MAG				

Example:

MDROT1	110		45	72	90.				
--------	-----	--	----	----	-----	--	--	--	--

Descriptor	Meaning
MVID	Move identification number to be referenced on an MDMOVE entry. (Integer>0).
RID	GRID or POINT identification number of a reference point at which rotation will occur. If blank or 0 then Module will be rotated about rotation vector defined from IDS to IDE. (Integer ≥ 0 , Default=0).
IDS	GRID or POINT identification number of rotation vector's starting point. (Integer>0)
IDE	GRID or POINT identification number of rotation vector's end point. (Integer>0)
MAG	Magnitude of rotation in degrees. (Real $\neq 0.0$).

Remarks:

1. MDROT1 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. MDROT1 must be referenced by an MDMOVE entry in order to reposition a Module.
3. GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

MDROT2**Module Reposition by Rotation - Option 2**

Defines a Module rotation by specifying a coordinate system axis for the rotation vector.

Format:

1	2	3	4	5	6	7	8	9	10
MDROT2	MVID	CID	RIDS	AXIS	MAG				

Example:

MDROT2	40		45	Y	90.				
--------	----	--	----	---	-----	--	--	--	--

Descriptor	Meaning
MVID	Move identification number to be referenced on an MDMOVE entry. (Integer>0).
CID	Coordinate system identification number of a CORDij entry. Blank or zero means basic coordinate system. Module will be rotated about CID's origin if RID is blank. (Integer ≥ 0 , default=0).
RID	GRID or POINT identification number of a reference point at which rotation will occur. If RID is blank or 0 then the origin of CID will be used. (Integer ≥ 0 , Default=0)
AXIS	Coordinate system axis: "X", "Y", "Z", "MX", "MY", or "MZ". The latter three values indicate the negative side of the axis. (Character)
MAG	Magnitude of rotation in degrees. (Real $\neq 0.0$).

Remarks:

1. MDROT2 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE = n command.
2. MDROT2 must be referenced by an MDMOVE entry in order to reposition a Module.
3. CORDij, GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

MDRROD**Rigid Pin-Ended Element Connection Between Two Modules**

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

Format:

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

Example:

MDRROD	14	11	1	21	2	2		6.5-6	
--------	----	----	---	----	---	---	--	-------	--

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

Remarks:

1. The MDRROD entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RROD entry description. All remarks and figures under RROD apply to MDRROD.

MDSEAM**A Shell Patch SEAM Connection Between Two Modules**

Defines a SEAM connecting two surface patches in two Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDSEAM	EID	PID	SMLN	CTYPE	IDAS	IDBS	IDAE	IDBE	
	GS	GE					MODA	MODB	

Alternate Format:

1	2	3	4	5	6	7	8	9	10
MDSEAM	EID	PID		CTYPE	IDAS	IDBS	IDAE	IDBE	
	XS	YS	ZS	XE	YE	ZE	MODA	MODB	

Example:

MDSEAM	552	297			43	48			
	30422	77987					15	32	

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSEAM entry. (Integer > 0)
SMLN	SEAM line identification. See Remark 2. under CSEAM. (CHAR or blank)
CTYPE	Connectivity search type. (Character)
	If CTYPE = “PSHELL”, IDAS and IDBS are property identification numbers of PSHELL’s. (Default)
	If CTYPE = “ELEM”, IDAS and IDBS are element identification numbers.
IDAS, IDBS	Used to define patch A and B in Modules MODA and MODB or the start of patch A or B for a tailored blank in Modules MODA and MODB. See Remark 2. under CSEAM (Integer > 0)
	If CTYPE = “PSHELL”, required property id defining patches A and B in Modules MODA and MODB. If CTYPE = “PSHELL” and IDAS = IDBS or IDBS = blank the patch will be considered as two-sided and the property identification numbers of PSHELL’s will be the same for both the top and bottom. See Remark 6. under CSEAM.
	If CTYPE = “ELEM”, required element id defining patches A and B in Modules MODA and MODB. IDAS ≠ IDBS.

Descriptor	Meaning
IDAE, IDBE	Used to define the end of patch A and the end of patch B for a tailored blank in Modules MODA and MODB. See Remark 4. under CSEAM. (Integer > 0 or blank) If CTYPE = "PSHELL", property id defining patches A and B. If CTYPE = 'PSHELL' and IDAE = IDBE or IDBE=blank the patch will be considered as two-sided and the property identification numbers of PSHELL's will be the same for both the top and bottom. If CTYPE = "ELEM", element id defining patches A and B in Modules MODA and MODB. IDAE ≠ IDBE.
GS, GE	Grid ids of piercing points on patches A and B of the Start and End of the SEAM. Their GRID entries must be defined in the main Bulk Data section (Module 0). (Integer > 0)
XS,YS,ZS	Location of the SEAM Start. (Real or blank)
XE,YE,ZE	Location of the SEAM End. (Real or blank)
MODA,MODB	Module IDs that contain the surface patches defined by property or element entries IDAS/IDAE and IDBS/IDBE, respectively (Integer ≥ 0).

Remarks:

1. The MDSEAM entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the CSEAM entry description. All remarks and figures under CSEAM apply to MDSEAM.

MDTRAN**Module Reposition by Translation**

Defines a Module translation by specifying a vector in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDTRAN	MVID	IDS	IDE	MAG					

Example:

MDTRAN	110	72	45						
--------	-----	----	----	--	--	--	--	--	--

Descriptor	Meaning
MVID	Move identification number to be referenced on an MDMOVE entry. (Integer>0).
IDS	GRID or POINT identification number of translation vector's starting point. (Integer>0).
IDE	GRID or POINT identification number of translation vector's end point. (Integer>0).
MAG	Magnitude of translation. If blank or zero, then the length of the vector will be used. (Real, Default=0.0)

Remarks:

1. MDTRAN can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE = n command.
2. MDTRAN must be referenced by an MDMOVE entry in order to reposition a Module.
3. GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

MDWELD**Weld or Fastener Element Connection Between Two Modules**

Defines a weld or fastener connecting two surface patches or points between two Modules.

Format PARTPAT:

1	2	3	4	5	6	7	8	9	10
MDWELD	EWID	PWID	GS	"PARTPAT"	GA	GB		MCID	
	PIDA	PIDB			MODA	MODB			
	XS	YS	ZS						

Example:

MDWELD	101	8	203	PARTPAT					
	21	33			11	21			

Alternate formats and examples:

Format ELPAT:

1	2	3	4	5	6	7	8	9	10
MDWELD	EWID	PWID	GS	"ELPAT"	GA	GB		MCID	
	SHIDA	SHIDB			MODA	MODB			
	XS	YS	ZS						

Example:

MDWELD	103	5	403	ELPAT					
	309	511			11	21			

Format ELEMID:

MDWELD	EWID	PWID	GS	"ELEMID"	GA	GB		MCID	
	SHIDA	SHIDB	MODS		MODA	MODB			

Example:

MDWELD	103	5	403	ELEMID					
	309	511			11	12			

Format GRIDID:

MDWELD	EWID	PWID	GS	"GRIDID"	GA	GB	SPTYP	MCID	
					MODA	MODB			
	GA1	GA2	GA3	GA4	GA5	GA6	GA7	GA8	
	GB1	GB2	GB3	GB4	GB5	GB6	GB7	GB8	

Example:

MDWELD	7	29	233	GRIDID			QT		
					11	21			
	15	28	31	35	46	51	55	60	
	3	5	8						

Format ALIGN:

MDWELD	EWID	PWID		"ALIGN"	GA	GB		MCID	
					MODA	MODB			

Example:

MDWELD	7	29		ALIGN	103	259			
					11	21			

Descriptor	Meaning	Type	Default
EWID	MDWELD element identification number. See Remark 1 under CWELD.	0 < Integer < 100,000,000	Required
PWID	Property identification number of a PWELD entry.	Integer > 0	Required
GS	Identification number of a grid point which defines the location of the connector. GRID entry for GS must be defined in main Bulk Data section only (Module 0) except for a point-to-patch connection with OPTION="ELEMID" in which case GS must be defined in Module MODS. See Remarks 2. and 3. under CWELD.	Integer > 0 or blank	

Descriptor	Meaning	Type	Default
	Character string indicating the type of connection. The format of the subsequent entries depends on the type. "PARTPAT", for example, indicates that the connectivity of surface patch A to surface patch B is defined with two property identification numbers of PSHELL entries, PIDA in Module MODA and PIDB in Module MODB, respectively. The "PARTPAT" format connects up to 3x3 elements per patch. See Remark 4. under CWELD.	Character	Required
GA, GB	Grid point identification numbers of piercing points on surface A and surface B, respectively. GRID entries for GA and GB must be defined in the main Bulk Data section only (Module 0) except for OPTION="ALIGN". See Remark 5. under CWELD.	Integer > 0 or blank	Blank
MCID	Specifies the element stiffness coordinate system. See Remark 16. under CWELD	Integer > -1 or blank	Default = -1
MODS	Module ID that contains GRID entry for GS. Only used for point-to-patch connection with OPTION='ELEMID'.	Integer ≥ 0	Default = 0
MODA, MODB	Module IDs that contain the surface patches defined by (1) property entries PIDA and PIDB, (2) element entries SHIDA and SHIDB or (3) GRID entries GAI and GBi, respectively.	Integer ≥ 0	Default = 0
PIDA, PIDB	Property identification numbers of PSHELL entries defining surface A in Module MODA and B in Module MODB, respectively.	Integer > 0	Required for "PARTPAT"
XS, YS, ZS	Coordinates of spot weld location in basic. See Remark 2. under CWELD.	Real	Required if GS and GA are not defined.

For the alternate formats, the describer meaning are described below:

Describer	Meaning	Type	Default
"ELPAT"	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA in Module MODA and SHIDB in Module MODB, respectively. The "ELPAT" format connects up to 3x3 elements per patch. See Remark 6. under CWELD.	Character	Required
SHIDA, SHIDB	Shell element identification numbers of elements on patch A in Module MODA and B in Module MODB, respectively.	Integer > 0	Required for "ELPAT"
"ELEMID"	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA in Module MODA and SHIDB in Module MODB, respectively. The "ELEMID" format connects one shell element per patch. See Remark 7. under CWELD.	Character	Required
SHIDA, SHIDB	Shell element identification numbers of elements on patch A in Module MODA and B in Module MODB, respectively.	Integer > 0	Required for "ELEMID"
"GRIDID"	Character string indicating that the connectivity of surface patch A in Module MODA to surface patch B in Module MODB is defined with two sequences of grid point identification numbers, GAi in Module MODA and GBi in Module B, respectively. The "GRIDID" format connects the surface of any element. See Remark 8. under CWELD.	Character	Required
SPTYP	Character string indicating types of surface patches A and B. SPTYP = "QQ", "TT", "QT", "TQ", "Q" or "T". See Remark 9. under CWELD.	Character	Required for "GRIDID"

Descriptor	Meaning	Type	Default
GAi	Grid identification numbers of surface patch A. GA1 to GA3 are required. Their GRID entries must be defined in Module MODA only. See Remark 10. under CWELD.	Integer > 0	Required for "GRIDID"
GBi	Grid identification numbers of surface patch B. Their GRID entries must be defined Module MODB only. See Remark 10. under CWELD.	Integer > 0	
"ALIGN"	Character string indicating that the connectivity of surface A to surface B is defined with two shell vertex grid points GA in Module MODA and GB in Module MODB, respectively. See Remark 11. under CWELD.	Character	Required
GA, GB	Vertex grid identification number of shell A and B, respectively. GA and GB GRID entries must be defined in Module MODA and Module MODB, respectively.	Integer > 0	Required for "ALIGN"

Remarks:

1. The MDWELD entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the CWELD entry description. All remarks and figures under CWELD apply to MDWELD.

MESH

Mesh Generator

Defines a mesh. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MESH	MID	TYPE	DXEL	DYEL	DZEL	XREF	YREF	ZREF	
	X0	Y0	Z0	DX	DY	DZ			
	NX	NY	NZ	SUBMESH	NSTGP	NSTEL	PROP	PID	
	RESIZE	TID-X	TID-Y	TID-Z	METHOD				
	BIAS	GROWX	GROWY	GROWZ	IBIDX	IBIDY	IBIDZ		
	XOBX	YOBX	ZOBY	DXBX	DYBX	DZBX			
	CID								
	NELCUBE	NBX	NBY	NBZ	PROCDIR	NPX	NPY	NPZ	

Example:

Data									
MESH	1	ADAPT	0.1	0.2	0.3				
	SCALE	101			ALL				
	CENTER	1.2	1.2	1.2					

Descriptor	Meaning
MID	Unique MESH number. See Remark 1. (Integer > 0; no Default)
TYPE	Type of mesh generation: See Remark 1. (Character; Required)
ADAPT	An Euler mesh will be created around a coupling surface. This option is only valid for PROP=EULER, and requires that the MID of the MESH is referenced from the MESHID of a COUPLE card. During the simulation, when the coupling surface moves or deforms, the Euler mesh will adapt itself by adding and removing elements. The adapt algorithm ensures that the coupling surface is contained inside the Euler mesh at all times with the minimum amount of elements. The Euler elements are aligned with the basic coordinate.
BOX	A rectangular mesh will be created, that is aligned with the basic coordinate system. The mesh will use CHEXA elements.
DXEL,DYEL, DZEL	Euler element sizes. See Remark 1. (Real)

Descriptor	Meaning
XREF,YREF,ZREF	Coordinates of reference point. For TYPE=ADAPT, these coordinates provide control over the location of the Euler mesh, to avoid that faces of the Euler mesh are initially at the same location as faces of the coupling surface. For TYPE=BOX, these coordinates will be used as the origin of the mesh. They are the default setting for (X0, Y0, Z0). (Real; Default = -1e-6)
X0,Y0,Z0	Coordinates of point of origin. (Real, XREF, YREF, ZREF) Not used for TYPE=ADAPT
DX,DY,DZ	Width of mesh in different directions. For TYPE=ADAPT, these values will only be used if (DXEL, DYEL, DZEL) are left blank. See Remark 1. for more detail. (Real)
NX,NY,NZ	Number of elements in the different directions. For TYPE=ADAPT, these values will only be used if (DXEL, DYEL, DZEL) are left blank. See Remark 1. for more detail. (Integer > 0)
SUBMESH	Allows using smaller mesh sizes for a part of the mesh. SUBMESH is the MESH ID of a finer mesh that is to replace part of the mesh. See Remark 8. (Integer ≥ 0 , default=0)
NSTGP	Starting grid-point number. Not used for TYPE=ADAPT. If there are multiple couple surfaces then the starting grid-point number can only be specified if param,flow-method,facet has been activated. See Remark 2. (Integer > 0)
NSTEL	Starting element number. Not used for TYPE=ADAPT. If there are multiple couple surfaces then the starting element number can only be specified if param,flow-method,facet has been activated. See Remark 2. (Integer > 0)
PROP	Property type: (Character; default=EULER) EULER An Eulerian mesh will be created.
PID	ID of a PEULER or PEULER1 entry. (Integer > 0, Required.)
RESIZE	Only valid for TYPE=ADAPT. See Remark 6. (Character, NONE) Option to change the element size during the simulation: NONE No resizing of DX,DY,DZ during simulation. SCALE The elements are resized by a scale-factor as a function of time. LENGTH The elements are resized by specifying the length as a function of time.
TID-X	ID of a TABLED1. See RESIZE for contents of table. It must define a step function. See Remarks 4. and 5. (Integer > 0; Blank)
TID-Y	ID of a TABLED1. See RESIZE for contents of table. It must define a step function. See Remarks 4. and 5. (Integer > 0; TID-X)
YID-Z	ID of a TABLED1. See RESIZE for contents of table. It must define a step function. See Remarks 4. and 5. (Integer > 0; TID-X)
METHOD	Method for determining when to create Euler elements: (Character; ALL)

Descriptor	Meaning
ALL	Always re-mesh any existing Euler element. Maintains existing void regions. Only used for resizing.
MATERIAL	Only re-mesh those Euler elements that contain material. Removes void regions. See Remark 7.
BIAS	Adds bias to the mesh. (Character; Blank)
CENTER	Starting at the center of the BOX the mesh size gradually changes such that the mesh size at the boundaries of the BOX is GROWX times the mesh size at the center.
REF	Starting at the reference point the mesh size gradually changes such that the mesh size at the boundaries of the BOX is GROWX times the mesh size at the center
GROWX, GROWY, GROWZ	Total grow factor. Is the ratio between finest and coarsest element size. (Real > 0; Only required if BIAS is not blank)
IBIDX, IBIDY, IBIDZ	BIAS ID reference. See Remarks 9. and 10. (Integer; Default = 0)
X0BX,Y0BX,Z 0BX,DXBX,D YBX,DZBX	Definition of an auxiliary box for output purposes. By defining an auxiliary box all adaptive elements that are within the box for one of the cycles requested are stored in the archive. This allows multiple cycles in one Euler archive. This box should be sufficiently large such that it contains all elements. The fields X0BX, Y0BX and Z0BX specify the start point and DXBX, DYBX and DZBX specify the width of box. If the there are adaptive elements outside the box the run is terminated and a larger box needs to be specified. Only used for TYPE=ADAPT. (Real; Blank) See Remark 14.
CID	ID - number of a local coordinate system. (Integer ≥ 0 ; Default=0)
NELCUBE	The number of elements per cube. This number is used as a guideline. The actual number used per cube can differ and can be found in the OUT file. See Remark 12. (Integer ≥ 1 ; Default=1)
NBX	Overrules NELCUBE. The number of cubes in the x-direction. (Integer ≥ 0 ; Default=1)
NBY	The number of cubes in the y-direction.(Integer ≥ 0 ; Default=NBX)
NBZ	The number of cubes in the z-direction. (Integer ≥ 0 ; Default=NBX)
PROCDIR	This directive controls the way cubes are distributed across processors. The effect can be checked by checking the Eulerian output variable PARTITION. (Char, Default=X) X Partition in global X direction first. Y Partition in global Y direction first. Z Partition in global Z direction first. USER Define user defined partitioning.

Descriptor	Meaning
	SIMPLE Partition Euler cubes in a simple pattern.
NPX	The number of cubes in the x-direction. Required for PROCDIR=USER. (Integer, Default=1)
NPY	The number of cubes in the y-direction. (Integer, Default=NPX)
NPZ	The number of cubes in the z-direction. (Integer, Default=NPX)

Remarks:

1. The grid-points of the mesh are generated at following locations:

Type=ADAPT: $(x,y,z) = (XREF+i^*DXEL, YREF+j^*DYEL, ZREF+k^*DZEL)$

Grid-points and elements located a certain distance outside the coupling surface will not be created. This saves memory and CPU time.

When $(XREF, YREF, ZREF)$ are outside the coupling surface, no grid-point will be created at this location, but the mesh will be shifted appropriately.

Type=BOX: $(x,y,z) = (X0+i^*DXEL, Y0+j^*DYEL, Z0+k^*DZEL)$

Nodes and Elements will always be created, even if the MESH is referenced from the MESHID of a COUPLE entry.

One of the following input combinations is required:

TYPE=ADAPT

a. (DXEL, DYEL, DZEL)

or

b. 1

c. (DX, DY, DZ) and (NX, NY, NZ)

→ DXEL = DX/NX ; DYEL=DY/NY ; DZEL=DZ/NZ

TYPE=BOX

a. (DXEL, DYEL, DZEL) and (NX, NY, NZ)

or

b. (DX, DY, DZ) and (NX, NY, NZ)

→ DXEL = DX/NX ; DYEL=DY/NY ; DZEL=DZ/NZ

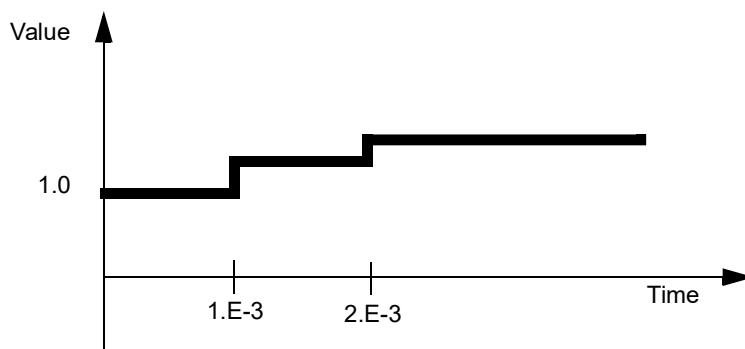
2. When the starting grid point and/or element number is left blank, then the default start number for the elements and grid-points is equal to the maximum number used +1. For simulations with multiple coupling surfaces two methods of treating transport between the Euler meshes are available. One method supports meshes of TYPE = ADAPT, but does not allow the specification of starting element or starting grid-point number.

3. The PID should refer to an existing property id, which can handle the property type given by PROP.

4. To avoid that the Euler mesh will be resized every time-step, the functions defined by TID-X, TID-Y, TID-Z must describe a ‘step-function’, like in this example:

- i. TABLED1,1,,,,,+
- ii. +,0.0,1.0,,,,,+
- iii. +,1.E-3,1.0,,,,,+
- iv. +,1.E-3,1.1,,,,,+
- v. +,2.E-3,1.1,,,,,+
- vi. +,2.E-3,1.2

Which specifies following function:



5. Care must be taken when refining the Euler mesh. To avoid instabilities, it is advised to stay within the following guidelines:
 - a. Each refining step, use a scale factor larger than 0.5
 - b. Allow the solution to become smooth again after each refining step. For airbag simulations, use an interval larger than $5 * \text{diameter_airbag} / \text{soundspeed}$
6. Resizing is not available for the Multi-material solver.
7. In most cases METHOD = ALL is the preferred method. Using METHOD=MATERIAL may be helpful in case of instabilities due to presence of void regions.
8. SUBMESH glues a fine mesh into a coarse mesh and uses the same gluing functionality as PARAM, GRADEDMESH. If the fine mesh is completely contained inside the coarse mesh no restrictions apply. Then to avoid any restrictions the grid points of the fine mesh are slightly displaced. But if parts of the fine mesh are outside the coarse mesh a restriction applies. In that case an Euler element of the coarse mesh has to be fully active or fully inactive. This means that the coarse element should not intersect elements of the fine mesh or it should be fully covered by the fine elements. Fine elements are not allowed to cover any part of the coarse elements. In practice, this means that the fine mesh has to fit nicely in the coarse mesh. For details refer to the section "Graded meshes" in the MSC Nastran Explicit Nonlinear User's Manual.

When running on one cpu the elements of the Euler mesh and the submesh will be put into one euler archive. But when running with multiple cpus, the mesh and sub mesh will be put in different euler archives. To distinguish the archives, the name _FVX is added to the Euler archive names. Here X is the MESH-ID MID. The Euler archive of the mesh and the submesh can be read simultaneously into Patran.

9. A biased mesh has nonconstant element sizes in selected directories. Neighbor element size can have a constant ratio or have identical size. In literature, this type of mesh is also referred to as a nonuniform mesh or a locally refined mesh.
10. A block mesh consists of a number of planes in all three directions. For a nonbiased mesh, these planes are at fixed distance from each other. In a biased mesh, the distance between subsequent planes can differ. The varying element size is determined by:
 - a. IBIDX
 - b. GROWX
 - c. The constant step size specified by X0, NX, DX.

Here, IBIDX overrules GROWX and GROWX overrules the X0, NX DX specification. Likewise, for the other directions. The locations of the planes are written out in the OUT file. Intersecting an x-plane with a y-plane and z-plane will give a grid point. By carrying out all intersections, the grid points are constructed.

11. Defining the CID allows for positioning the mesh box arbitrarily in space. When active, the following restrictions apply:
 - MESH,ADAPT cannot be used
 - The use of single material Euler with strength elements is not supported

12. Setting NELCUBE, NBX, NBY, NBZ allow to divide the Euler mesh into a number of cubes. By setting NELCUBE equal to 2000, optimal use is made of memory caching during Euler computation. This can give a speedup of 1.5. If NBX is defined, also NBY and NBZ need to be defined. Defining NBX overrules the definition of NELCUBE.

When using multiple Euler cubes the BARRIER and FLOW definitions only support geometric conditions like boundary face direction and square definition.

13. There are several ways to distribute cubes across processors. Some ways may lead to bad load balancing. To avoid this it is possible to control the way Euler cubes are distributed across processors by defining PROCDIR.

When option PROCDIR=USER, the values for NBX, NBY, and NBZ must be such that NBX is equal or a multiple of NPX, NBY is equal or a multiple of NPY and NBZ is equal or a multiple of NPZ. Also for this option, NPX*NPY*NPZ must be equal to the number of processors used in the cluster.

For option PROCDIR=SIMPLE, the values NBX, NBY, and NBZ on DYPARAM,EULERCB must be such that NBX*NBY*NBZ is equal or a multiple of the number of processors used. For instance, if the number of processors in the cluster is 4, NBX*NBY*NBZ must be equal to either 4 or 8 or 12, etc. Otherwise, the calculation will terminate prematurely with an error message.

When using these options the Barrier and FLOW definitions only support geometric conditions like boundary face direction and a square definition

14. To determine a suitable size for the static output box, the simulation can be run first without the fields X0BX, Y0BX, Z0BX, DXBX, DYBX, and DZBX set. In the ,OUT file each summary of adaptive meshing gives the smallest box surrounding all adaptive elements so far. The last summary then yields the dimension of the static output box. To accommodate for elements that are not completely inside the box, the actual static output box is automatically extended a little. The actual dimensions are written in the out file after the first adaptive meshing summary. Is not needed to set the XREF, YREF, and ZREF option. If they are set the static output box will be compatible with the defined reference point.

MESUPER

Defines External Superelement DMIG Input for SOL 600 Residual Analyses

Superelement DMIG matrices are created by Nastran and used when Marc is spawned from Nastran, SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MESUPER	ID				Fname				

Example:

MESUPER	1	super1.pch	
MESUPER	2	super2.pch	

Descriptor	Meaning
ID	Superelement ID. (Integer; no Default)
Fname	Filename containing external superelement data from the creation run (using Case Control EXTSEOUT (ASMBULK, DMIGPCH,EXTID=ID) (Character; no Default). Left justify in field, see Remark 8.

Remarks:

1. Enter as many MESUPER lines as necessary to define all external superelements.
2. This entry can presently only be used with SOL 600,106, SOL 600,101, SOL 600,nlstatic, or SOL 600,sestatic. External Superelements are not presently available for other types of SOL 600 analyses such as nonlinear (or linear) transient dynamics, eigenvalue analysis or buckling.
3. Fname is limited to 56 characters.
4. Include entries with the same Fname must be specified as include files in the Nastran input file. The include specifications must appear at the end of the Bulk Data portion of the file.
5. External superelement creation runs should use the Case Control command:
EXTSEOUT(ASMBULK,DMIGPCH,EXTID=N)
where N is the external superelement ID number. All creation runs must have the same number of subcases and use the same subcase IDs.
6. The SOL 600 residual input file must have the same number of subcases and subcase numbers as the creation runs.
7. This entry acts like an element, in other words it is not controlled by a case control command. It is always active if entered.

8. The MESUPER entries should normally be coded in small fixed field format. If coded in small format free field, Fname is limited to 8 characters. If coded in large format free field, Fname is limited to 16 characters. The filename may be extended to a continuation line. All filenames should be entered in lower case. Nastran will convert to upper case, and the SOL 600 translator will reconvert to lower case. The creation runs should also use lower case for all external superelement punch filenames for case-sensitive computer systems.
9. If there are no elements in the residual (that is, all elements are in the external superelements, PARAM,MARCND99,-1 is required to output the displacements in the Marc .out file regardless of the specified Case Control request.
10. The ASMBULK option in the creation runs is required for SOL 600 when outr options are specified. It is recommended whether or not outr options are specified. All .asm files (created by the ASMBULK option) for all external superelements should be included in the Bulk Data before any punch files (produced using the DMIGPCH option). See the following input file examples.
11. At present, an OP2 with results datablocks only can be produced by a SOL 600 External Superelement residual execution. OP2 files which combine geometry and results datablocks cannot be produced. Other outr options such as xdb, f06 and punch are also not available for SOL 600 External Superelement residual runs.
12. For the External Superelement Initial run(s) to generate punch and asm files, no Case Control output requests should be made and the following DMAP should be inserted in the Executive Control in order to prevent extra information (which will cause an error) for being inserted into the punch file:

```
compile extout
alter 'sdr2' $
delete /iug1,,,,, $
alter 'sdr2' (2) $
delete /igulo,,,,, $
```

13. If data exists after ENDDATA, including any characters on the ENDDATA line, after the word ENDDATA in the creation run input, this data must be removed.

Typical File Setup for External Superelement Creation Run for SOL 600 (same as for other solution sequences)

```
SOL 101
compile extout
alter 'sdr2' $
delete /iug1,,,,, $
alter 'sdr2' (2) $
delete /iuglo,,,,, $
CEND
TITLE = 2 SUPERELEMENTS AND THE RESIDUAL -- TEST PROBLEM NO.
EXTSE2A
SUBTITLE = 8 X 8 MESH OF QUAD4 ELEMENTS; GM-CMS PROJECT
EXTSEOUT (ASMBULK,DMIGPCH,EXTID=100)
SPC = 100
BEGIN BULK
aset1,123456,840,thru,848
CORD2R,1001,1002,,,,,,1.0
(rest of file same as any other Nastran run)
```

Typical File Setup for External Superelement Residual Run for SOL 600

```
SOL 600,101 outr=op2
CEND
TITLE = 2 SUPERELEMENTS AND THE RESIDUAL -- TEST PROBLEM NO. EXTSE2R
SUBTITLE = 8 X 8 MESH OF QUAD4 ELEMENTS; GM-CMS PROJECT
param,mextsee,1
SPC = 100
LOAD = 1000
DISP = ALL
K2GG=KAAX
M2GG=MAAX
BEGIN BULK
param,marcnd99,-1
force, 1000, 844, , 0.1, 0., 0., 1.
SPC1    100      12346     840      848
$2345678 2345678 2345678
mesuper      100 extse2a.pch
mesuper      200 extse2b.pch
include 'OUTDIR:extse2a.asm'
include 'OUTDIR:extse2b.asm'
include 'OUTDIR:extse2a.pch'
include 'OUTDIR:extse2b.pch'
ENDDATA
```

METADATA

Metadata for the input section

Defines the entry metadata.

Format:

1	2	3	4	5	6	7	8	9	10
METADATA		META							
	METAINFOR								

Example:

METADATA	Rear_Fender								
		This is line 1 of metadata							
		This is line 2 of metadata							
		This is line 3 of metadata...							

Descriptor	Meaning
META	Character handle of up to 24 characters identifying the metadata.
METAINFO	A open-ended list of strings of 64 characters that represents the user information.

Remarks:

1. META can include any character from defined bulk character set
2. METAINFO can contain basic Nastran characters and special symbols.

MFLUID**Fluid Volume Properties**

Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix.

Format:

1	2	3	4	5	6	7	8	9	10
MFLUID	SID	CID	ZFS	RHO	ELIST1	ELIST2	PLANE1	PLANE2	
	RMAX	FMEXACT							

Example:

MFLUID	3	2	15.73	1006.	3	4	S	N	
		100.							

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
CID	Identification number of rectangular coordinate system used to specify the orientation of the free surface (normal to X_3) and of planes of symmetry, if any. (Integer ≥ 0 or blank)
ZFS	Intercept of the free surface on the X_3 axis of the coordinate system referenced by CID. If X_3 of a grid > ZFS then there is no fluid. See Remark 3. (Real; Default means that the free surface is located at an infinitely large positive value of XFS.)
RHO	Density of the fluid (Real).
ELIST1	Identification number of an ELIST entry that lists the IDs of two-dimensional elements that can be wetted on one side by the fluid. Only those elements connected to at least one grid point below ZFS are wetted by the fluid. See Remarks 3. and 5. (Integer ≥ 0)
ELIST2	Identification number of an ELIST entry that lists the IDs of two-dimensional elements that can be wetted on both sides by the fluid. Only those elements connected to at least one grid point below ZFS are wetted by the fluid. (Integer ≥ 0 ; ELIST1 + ELIST2 > 0)
PLANE1, PLANE2	Planes of symmetry, antisymmetry, or no symmetry. "S" means that plane 1, which is the plane containing the X_1 and X_3 axes of CID, is a plane of symmetry. "A" means that plane 1 is a plane of antisymmetry. "N" means that it is neither. See Remark 5. Plane 2 uses "S", "A", or "N" for the X_2 and X_3 plane. (Character: "S", "A", or "N")
RMAX	Characteristic length. Interactions between elements with separation that is greater than RMAX will be neglected. (Real > 0.0 ; Default = 1.0E 10)
FMEXACT	Exact integration is used if the distance between two elements is less than FMEXACT times the square root of the area of the larger element. Otherwise, center point integration is used by default. (Real; Default = 1.0E 15)

Remarks:

1. The MFLUID entry must be selected with the Case Control command MFLUID = SID.
2. Several MFLUID entries corresponding to different fluid volumes can be used simultaneously.
3. The wetted side of an element in ELIST1 is determined by the presence or absence of a minus sign preceding the element's ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element's positive normal, as determined by applying the right-hand rule to the sequence of its corner points. The same element can appear on two ELIST entries, indicating that it forms a barrier between the unconnected fluids.
4. The fluid volume may be finite (interior) or infinite (exterior). The volume may be bounded by a free surface and one or two planes of structural symmetry. If structural symmetry is used, the structure must have the symmetric or antisymmetric boundary corresponding to the selection in fields 8 and 9. Interior fluids must have ELIST1 data and a free surface or plane of antisymmetry.
5. The planes of symmetry and/or antisymmetry defined in fields 8 and 9 must be planes of symmetry for the entire analysis. The user may apply appropriate structural boundary conditions at all grid points lying in these planes.
6. The current list of elements that may be placed in ELIST1 and ELIST2 include CTRIA3, CQUAD4, CTRIAR, and CQUADR.
7. The continuation entry is optional.
8. If there is ELIST1 data and no free surface nor plane of antisymmetry, the program assumes a special form of external fluid. These special external fluids must have a CID (field 3) such that the origin of the fluid coordinate system is near the center of the enclosed volume, since the singularity for volume change will be placed at the origin. Special external fluids are supported only in SOLs 103 and 107 through 112. If used in conventional solution sequences, System Fatal Message 3001 results for file 205.
9. See PARAM,VMOPT in “[Parameters](#)” on page 1409. VMOPT controls when the virtual mass is included in the mass matrix.
10. If any MFLUID entry is changed or added on restart then a complete re-analysis may be performed. Therefore, MFLUID entry changes or additions are not recommended on restart.
11. A tolerance is computed for each wetted element, with the value of $0.01\sqrt{2A}$, where A is the area of the element. If any grid point connected to the element lies within TOL below the free surface it is moved to the free surface.
12. Any element that has all grids on or above the free surface, after the grid points are moved by the procedures given in Remark 11. is removed from the ELIST. It is not included in the VM effects, and will produce no pressure output.

MGRSPR

Defines Grids to Add Soft Spring to Ground - SOL 600

This entry is used to add soft springs to ground at selected grids to stabilize the structure in a nonlinear analysis. It is most often used with contact to stabilize free-free bodies before they come into contact. The spring rates selected should be stiff enough to allow matrix decomposition but weak enough so they are not significant once full contact is achieved. Values on the order of 1.0E-6 to 1.0E-4 times the average main diagonal terms in the stiffness matrix for the grids selected are recommended.

Format:

1	2	3	4	5	6	7	8	9	10
MGRSPR	ID1	THRU	ID2	IDIR	K				

Example:

MGRSPR	100	THRU	200	123	10.0				
	500	THRU	520	123456	1.0				

Descriptor	Meaning
ID1	Starting grid ID. (Integer; Required; no Default)
THRU	Enter the character string THRU if more than one grid is desired.
ID2	Ending grid ID of the range. (Integer; or blank. If blank; ID2=ID1)
IDIR	Directions that the spring(s) will be added in, any unique combination of the integers 1 through 6 with no embedded blanks. (Integer; Required)
K	Spring rate. (Real; Required)

Remarks:

1. If springs are to be added to all grids, PARAM,MRSpring may be used instead of this entry.
2. Do not use PARAM,MRSpring and this entry in the same model.
3. Grids that do not exist in the range ID1 to ID2 will automatically not have springs in the Marc input.

MINSTAT

Option to Define Initial State Variables for SOL 600

This option is used to enter initial (stress free) temperatures calculated from a previous heat transfer analysis and saved on a t16 or t19 file. (MCHSTAT is used to define the temperatures that cause thermal strains). This entry may also be used to initialize other state variables if required.

Format:

1	2	3	4	5	6	7	8	9	10
MINSTAT	IDV	IOPT		INCR		IFORM	IPRT	NPST	
	“FILE”	Name							
	“ELEM”	ELE1	ELE2	INT1	INT2	LAY1	LAY2	VAL	
	“STATE”	NS	IS1	IS2	IS3	IS4	IS5	IS6	
		IS7	IS8	IS9	etc.				

Example:

MINSTAT	1	3		1		0		9	
	FILE		initial_state_example						

Descriptor	Meaning
IDV	State variable identifier (1=temperature). (Integer; Default = 1) (2,1) If more than one state variable is required, enter -1.
IOPT	Option of how to enter the data. (Integer; Default = 3) (2,2) <ul style="list-style-type: none"> 1 Use the “ELEM” continuation line for as many elements as necessary 2 Enter the data using user subroutine INITSV 3 Read the data from a t16 or t19 file (see IFORM)
INCR	Increment number on t16 or t19 file defining the new state values if IOPT=3. (Integer; no Default) (2,5)
IFORM	Designates whether a binary (t16) or formatted (t19) post file is used if IOPT=3. (Integer; Default = 0) (2,7) <ul style="list-style-type: none"> 0 Use binary (t16) file 1 Use formatted (t19) file
IPRT	Enter a value of 1 to suppress printing of state variable values defined in user subroutine INITSV (only applicable if IOPT=2). (Integer; Default = 0) (2,8)
NPST	Post Code ID to be read into this state variable. (Integer; Default = 9 [temperature]) (2,9) See MARCOUT for a list of the post codes.
FILE	Enter the character string FILE if IOPT=3. (Character; no Default; Required if IOPT=3)

Descriptor	Meaning
NAME	Enter the file name without the extension (.marc.t16 or .marc.t19). (Character; no Default) The file name must be entirely in lower case for case-sensitive computer systems and is limited to 56 characters. This file must be in the same directory as the Nastran input file.
ELEM	Enter the character string ELEM to start a list of elements and associated values if IOPT=1. (Character)
ELE1	First element with value VAL. (Integer; no Default; Required) (3,1)
ELE2	Last element with value VAL. (Integer; Default = ELE1) (3,2)
INT1	First integration point with value VAL. (Integer; no Default; Required) (3,3)
INT2	Last integration point with value VAL. (Integer; Default = INT1) (3,4)
LAY1	First cross-section layer with value VAL. (Integer; no Default; Required) (3,5)
LAY2	Last cross-section layer with value VAL. (Integer; Default = LAY1) (3,6)
VAL	New state value for these elements. (Real; no Default; Required) (4,1)
STATE	Enter the character string STATE to start a list of state variables. (Character)
NS	Number of state variables to be defined. (Integer; no Default; limited to 16 maximum)
ISi	State variable post codes. (Integer; no Default) (9,i) See MARCOUT entry for applicable post codes.

Remarks:

1. Only one MINSTAT entry may be entered in the input. If additional entries are found, the first will be used.
2. This entry maps to Marc's INITIAL STATE entry
3. (i,j) refer to Marc's INITIAL STATE (data block, field)
4. MINSTAT (and/or MCHSTAT) cannot be the only applied "loads". At least one standard load such as FORCE, PLOAD4 or a standard TEMP entry must be entered with a LOAD Case Control command that references the standard load(s). If there are no standard loads, please enter a dummy load with a very small magnitude and a LOAD Case Control command to reference it.

MISLAND**SOL 600**

Defines an island of connected elements that will be completely removed if the number of elements within the island becomes smaller than a specified value in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MISLAND	NELEM								

Example:

MISLAND	20								
---------	----	--	--	--	--	--	--	--	--

Descriptor	Meaning
NELEM	Number of elements to use as a limit for element deactivation. If the number of elements in an island is less than or equal to this number, all elements in this island are deactivated. (Integer; no Default; Required field)

Remarks:

1. If this entry is made in the model, it subdivides the entire mesh into islands of connected regions. If the number of elements in a particular island is less than or equal to the specified value, all elements of this island are deactivated. This check is performed after element deactivation has taken place where the deactivation can be due to model input options, deactivation user subroutines, or through material damage or failure. Two elements are considered connected if they share a node for line elements, an edge for 2-D elements, or a face for 3-D solid elements. This option is useful for cases where unconnected elements or regions of elements might exist after the neighboring elements have been deactivated. There is no check performed to see if the island to be deactivated has enough boundary conditions. Only the number of elements in the island is used for determining if the elements should be deactivated.
2. Only one MISLAND entry should be placed in the input. If more than one is entered, the first will be used.

MIXTURE**Constituents of "Composite" Material on Original and Potentially Damaged State - SOL 600**

Defines constituents of "composite" material on original and potentially damaged state. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MIXTURE	MID	ITYPE	IALPH	ISET	M1	FRACT1	M2	FRACT2	
	M3	FRACT3	M4	FRACT4	M5	FRACT5	etc.		

Example:

MIXTURE	120	3	1	200	10	.2	20	.2	
	30	.2	40	.2	50	.2			

Descriptor	Meaning
MID	Material ID (Integer; no Default, required field)
ITYPE	Mixture rule (Integer; Default = 1) <ul style="list-style-type: none"> 1 Weighted average of material properties based upon volume fraction. 2 Weighted average of Hooke's law based upon volume fraction 3 Weighted average of nonlinear stress strain curve based on volume fraction
IALPH	Flag controlling averaging procedure for thermal expansion (Integer; Default = 1) <ul style="list-style-type: none"> 1 Weighted average based upon volume fraction 2 Not currently available
ISET	ID of a SET3 containing the all the elements making up this material (Integer; no Default; Required)
Mi	Identification of a previously defined material, defined on a MATxxx entry (for example the MID on a MAT1 entry) (Integer; no Default)
FRACTi	Fraction for the ith component (see ITYPE). The sum of all FRACTi must add to 1.0 (Real; no Default)

Remarks:

1. Enter as many Mi, FRACTi components as desired. Each may have different properties, and failure criteria.
2. Fields 2-4 of each continuation entry are required, fields 5-9 may be left blank.

3. This option allows one to create a new material comprised of a number of other materials. The material behavior is based upon a "mixture" of the individual components using a mixture rule. Several of these mixture rules are only appropriate for linear elastic materials but also allow for temperature dependent material properties. The most sophisticated model (ITYPE=3) allows for the mixture of materials which undergo elastic-plastic behavior.
4. If void ratio or porosity is defined, it applied to all components in a uniform manner.
5. ITYPE 1 and 2 only support linear elastic material.
6. ITYPE 3 is not available using updated Lagrange - use PARAM,MARCPLAS,1 or PARAM,MARCPLAS,2 if any of the component materials have plasticity. The mixture will be limited to small strains, but large displacement, large rotation and follower forces and follower pressures can be included.
7. ITYPE 3 may not include the following material laws in any of the components.
 - Thermo-pore
 - Gurson damage
 - Simplified damage models 9 and 10
 - Gasket material
 - Shape memory material
 - Soils
 - User defined generalized stress strain
 - ORNL
 - Rigid-Plastic
 - Grain size effects
 - Rubber material (mooney, ogden, foam, gent, arruda-boyce)
 - Cohesive
8. Rebar elements and shell elements with offsets or non-composites with different properties for bending vs membrane (which use the Marc PSHELL option) may not be used for mixtures.
9. Within a layer, if the components are orthotropic or anisotropic, the preferred directions must be aligned.
10. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD* entries.

MKAERO1**Mach Number - Frequency Table**

Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

Format:

1	2	3	4	5	6	7	8	9	10
MKAERO1	m1	m2	m3	m4	m5	m6	m7	m8	
	k1	k2	k3	k4	k5	k6	k7	k8	

Example:

MKAERO1	.1	.7							
	.3	.6	1.0						

Descriptor	Meaning
mi	List of from 1 to 8 Mach numbers. (Real ≥ 0.0)
kj	List of from 1 to 8 reduced frequencies. (Real > 0.0)

Remarks:

1. Blank fields end the list, and thus cannot be used for 0.0.
2. All combinations of (mi, kj) will be used.
3. The continuation entry is required.
4. Multiple MKAERO1 entries are permitted.
5. For the lifting surface theories (Doublet-Lattice and Mach Box), the maximum value of kj should be less than one quarter of the number of boxes on a representative chord (i.e., $\text{MAX}(kj) < \bar{C}/4\Delta x$ where \bar{C} is the reference chord and Δx is a typical box chord length).
6. In SOL 146, the program selects only one value of mi to use in computing the dynamic response solution and, by default, the lowest value is selected. The PARAM,MACH,m entry may be specified to select a different value. If PARAM,MACH,m is specified, then the value of mi closest to m will be selected.
7. The very low nonzero value of kj required for aeroelastic divergence analysis of restrained vehicles with the K- and KE-methods of flutter analysis must be included on this entry.

MKAERO2**Mach Number - Frequency Table**

Provides a list of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

Format:

1	2	3	4	5	6	7	8	9	10
MKAERO2	m1	k1	m2	k2	m3	k3	m4	k4	

Example:

MKAERO2	.10	.30	.10	.60	.70	.30	.70	1.0	
---------	-----	-----	-----	-----	-----	-----	-----	-----	--

Descriptor	Meaning
mi	Mach numbers. (Real ≥ 0.0)
ki	Reduced frequencies. (Real > 0.0)

Remarks:

1. MKAERO2 will cause the aerodynamic matrices to be computed for the given sets of parameter pairs. Embedded blank pairs are skipped.
2. No continuation entries are allowed, but multiple MKAERO2 entries are permitted.
3. In SOL 146, the program selects only one value of mi to use in computing the dynamic response solution and, by default, the lowest value is selected. The PARAM,MACH,m entry may be specified to select a different value. If PARAM,MACH,m is specified, then the value of mi closest to m will be selected.
4. The very low nonzero value of ki required for aeroelastic divergence analysis of restrained vehicles with the K- and KE-methods of flutter analysis must be included on this entry.

MLAYOUT**SOL 600**

Selects layered composite shell output to be placed in Marc's t16 and/or t19 files and (if requested) to be transferred from Marc to the Nastran Database. The MARCOUT entry with LAYCODE of 3 or 103 must be used in conjunction with this entry. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MLAYOUT	L1	THRU	L2	BY	L3				
	L4	THRU	L5	BY	L6				

Example:

MLAYOUT	2	THRU	5						
	11	THRU	51	BY	10				
	52	THRU	54						
	21	THRU	20	BY	10				

Descriptor	Meaning
L1, L4	Starting layer of a sequence. (Integer; Required; no Default)
L2, L5	Ending layer of a sequence. (Integer; Default = L1 or L4)
L3, L6	Increment value for the sequence. (Integer; Default = 1)

Remarks:

1. This entry must be used in conjunction with MARCOUT with LAYCODE=3 or 103.
2. L2 must be larger than L1.
3. L1, L4, etc. must be positive integers of 1 or larger.
4. L2, L5 should not exceed the largest number of layers in the model.
5. Selection of all layers can lead to extremely large output files.
6. In the example, L1 could also be 1 instead of 2 and the largest layer could be 55 instead of 54 with causing an error.
7. If the layers designated on this entry are to apply to selected elements, enter the elements on the T16SEL entry. It is not currently possible to process different layered output for multiple groups of elements (i.e., it is not currently possible to request output of layers 1-20 for some elements and 1-10 for others.)

MNF600

Defines Auxiliary Data for MSC Adams MNF Files - SOL 600

Generated by the Marc portion of a SOL 600 execution. Used in SOL 600 only. (See also the MDMIOUT Bulk Data entry.)

Format:

1	2	3	4	5	6	7	8	9	10
MNF600	ID	ISTRESS	ISTRAIN	ISHELL	MASSU	LENGU	TIMEU	FORCU	

Example:

MNF600	100	1	1	3	2	7	3	2	
--------	-----	---	---	---	---	---	---	---	--

Descriptor	Meaning
ID	Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example, if the case control contains SUBCASE 20, ID would be 20. Must match the MDMIOUT entry. At present, ID is ignored and the first entry will apply to all subcases. (Integer; Default = 1)
ISTESS (2)	Flag to compute stress and place in the MNF file (0=no stress, 1=stress) (Integer; Default = 0)
ISTRAIN (3)	Flag to compute strain and place in the MNF file (0=no strain, 1=strain) (Integer; Default = 0)
ISHELL (4)	For shell elements, this entry describes which location the stresses or strains will be output to the MNF file. 1=top, 2=center, 3=bottom (Integer; Default = 1)
MASSU [2,1]	Mass units for MNF file. (Integer; Default = 1: kilogram) The following possible values may be entered: 1:kilogram (Default) 2:pound mass 3:slug 4:gram 5:ounce mass 6:kpound mass 7:megagram 8:dozen slug

Descriptor	Meaning
LENGU	Length units for MNF file (Integer; Default = 2: meter) The following possible values may be entered: 1:kilometer 2:meter (Default) 3:centimeter 4:millimeter 5:mile 6:foot 7:inch
TIMEU [2,3]	Time units for MNF file (Integer; Default = 3: second). The following possible values may be entered: 1:hour 2:minute 3:second (Default) 4:millisecond
FORCEU [2,4]	Force units for MNF file (Integer; Default = 1: newton). The following possible values may be entered: 1:Newton (Default) 2:poind force 3:kilogram force 4:ounce force 5:dyne 6:kNewton 7:kpound force

Remarks:

1. The MDMIOUT entry is the primary entry which generates an MNF file. This entry, MNF600 is only necessary if one or more of the fields is required to define non-default values, for example to generate stresses or strains or to specify the units.
2. The ID must be the same as the MDMIOUT ID.
3. Only one MNFDAT entry is allowed in an input file. If more than one is entered, the first will be used.
4. All remarks concerning MNF files for the MDMIOUT entry are also applicable to this entry.
5. (i) Indicates the corresponding field of Marc's MNF Parameter.
6. [i,j] Indicates the corresponding datablock and field of Marc's MNF units entry.

MODTRAK**Mode Tracking Parameters**

Specifies parameters for mode tracking in design optimization (SOL 200).

Format:

1	2	3	4	5	6	7	8	9	10
MODTRAK	SID	LOWRNG	HIGHRNG	MTFILTER					

Example:

MODTRAK	100	1	26	0.80					
---------	-----	---	----	------	--	--	--	--	--

Descriptor	Meaning
SID	Sets identification number that is selected in the Case Control Section with the MODTRAK command. See Remark 1. (Integer; no Default)
LOWRNG	Lowest mode number in range to search. See Remark 2. (Integer ≥ 0 ; Default = 0. If nonzero, LOWRNG < HIGHRNG.)
HIGHRNG	Highest mode number in range to search. See Remark 2. (Integer > 0 ; Default = number of eigenvalues extracted. If nonzero, LOWRNG < HIGHRNG.)
MTFILTER	Filtering parameter used in mode cross-orthogonality check. See Remark 3. (Real; Default = 0.9)

Remarks:

1. Only the designed modes for the subcase will be tracked. A designed mode is one that is used in the design model (in connection with either objective or constraints) and, therefore, identified on a DRESP1 entry.
2. The range of modes LOWRNG through HIGHRNG, inclusive, will be used to track the designed modes. If LOWRNG and HIGHRNG are both blank, then all computed modes will be used to search for the designed modes. Since large numbers of computed modes will result in higher computational costs, limiting the search range with LOWRNG and HIGHRNG is recommended.
3. Modes are considered to correlate if their mass normalized cross orthogonalities are greater than MTFILTER.

MOMAX

Conical Shell Static Moment

Defines a static concentrated moment load on a ring of a conical shell.

Format:

1	2	3	4	5	6	7	8	9	10
MOMAX	SID	RID	HID	S	MR	MP	MZ		

Example:

MOMAX	1	2	3	1.0	0.1	0.2	0.3		
-------	---	---	---	-----	-----	-----	-----	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
RID	Ring identification number. See the RINGAX entry. (Integer > 0)
HID	Harmonic identification number or a sequence of harmonics. See Remark 5. (Integer ≥ 0 or Character)
S	Scale factor. (Real)
MR, MP, MZ	Moment components in the r, ϕ , z directions. (Real)

Remarks:

1. MOMAX is allowed only if an AXIC entry is also present.
2. Load sets must be selected with the Case Control command LOAD=SID.
3. A separate entry is needed for the definition of the moment associated with each harmonic.
4. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.
5. If a sequence of harmonics is to be placed in HID, the form is as follows: "Sn1Tn2", where n1 is the start of the sequence and n2 is the end of the sequence; i.e., for harmonics 0 through 10, the field would contain "S0T10".

MOMENT**Static Moment**

Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	M	N1	N2	N3		

Example:

MOMENT	2	5	6	2.9	0.0	1.0	0.0		
--------	---	---	---	-----	-----	-----	-----	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0 or blank)
M	Scale factor. (Real)
Ni	Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni $\neq 0.0$ unless M is zero)

Remarks:

1. The static moment applied to grid point G is given by

$$\vec{m} = M \vec{N}$$

where \vec{N} is the vector defined by (N1, N2, N3). The magnitude of \vec{m} is equal to M times the magnitude of \vec{N} .

2. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. A CID of zero or blank references the basic coordinate system.
4. For scalar points see SLOAD.
5. For TYPE=12 or TYPE=13 on the TLOAD1, G is the ID of a rigid body: the MID of a rigid material (MATRIG) or the EID of a RBE2D. The MID of a rigid material and the EID of RBE2 must be different when both of a RBE2D and a rigid material are used with these TYPES. SOL 700 only.

MOMENT1**Follower Moment, Alternate Form 1**

Defines a concentrated moment at a grid point by specifying a magnitude and two grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT1	SID	G	M	G1	G2				

Example:

MOMENT1	6	13	-2.93	16	13				
---------	---	----	-------	----	----	--	--	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
M	Magnitude of moment. (Real)
G1, G2	Grid point identification numbers used to define the unit vector \vec{n} . (Integer > 0; G1 and G2 cannot be coincident.)

Remarks:

1. The concentrated moment applied to grid point G is given by

$$\vec{m} = M \vec{n}$$

where \vec{n} is a unit vector parallel to a vector from G1 to G2.

2. In the static solution sequences, SID must be selected by the LOAD Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, and 400, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

MOMENT2**Follower Moment, Alternate Form 2**

Defines a concentrated moment at a grid point by specification of a magnitude and four grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT2	SID	G	M	G1	G2	G3	G4		

Example:

MOMENT2	6	13	-2.93	16	13	17	13		
---------	---	----	-------	----	----	----	----	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
M	Magnitude of moment. (Real)
Gi	Grid point identification numbers used to determine the unit vector \vec{n} . (Integer > 0; G1 and G2 cannot be coincident; G3 and G4 cannot be coincident.)

Remarks:

1. The concentrated moment applied to grid point G is given by

$$\vec{m} = M \vec{n}$$

where \vec{n} is the unit vector parallel to the cross product of the vectors from G1 to G2, and G3 to G4.

2. In the static solution sequences, SID must be selected by the LOAD Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, and 400 if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

MONCARL**Parameters for Monte-Carlo simulation**

Parameters for conducting Monte-Carlo simulation using the non-parametric variability method (NPVM).

Format:

1	2	3	4	5	6	7	8	9	10
MONCARL	SID	VARTYP							
	STRVAR	MSVAR	KSVAR	BSVAR					
	FLUVAR	MFVAR	KFVAR	BFVAR					

Example:

MONCARL	100	DELTAVAR							
	STRVAR	0.001	0.05	0.002					
	FLUVAR	0.0	0.001	0.0					

Descriptor	Meaning
SID	Set identification number referenced by MONCARL case control. (Integer > 0; Required)
VARTYP	Select the type of variabilities that follow (Character; Default = ALPHAVAR) ALPHAVER Variabilities in this entry are Alpha variabilities. DELTAVAR Variabilities in this entry are Delta variabilities.
“STRVAR”	Indicates Structural variability parameters follow. (String)
MSVAR	Structural mass variability. (Real >= 0.0; Default = 0.0)
KSVAR	Structural stiffness variability. (Real >= 0.0; Default = 0.0)
BSVAR	Structural damping variability. (Real >= 0.0; Default = 0.0)
“FLUVAR”	Indicates Fluid variability parameters follow (String)
MFVAR	Fluid mass variability. (Real >= 0.0; Default = 0.0)
KFVAR	Fluid stiffness variability. (Real >= 0.0; Default = 0.0)
BFVAR	Fluid damping variability. (Real >= 0.0; Default = 0.0)

Remarks

1. Please use below as the guideline for selecting above mentioned structural and fluid alpha variabilities (which should be > 0.0):
 - a. Low variability level = 0.001
 - b. Medium variability level = 0.01

- c. High variability level = 0.05
- 2. Alpha variabilities are related to delta variabilities through the order of the matrix to be randomized, n (or equivalently the number of modes),

$$a = \delta \sqrt{\frac{2}{n+1}}$$

- 3. There is a limitation in the admissible values for the variability level. The maximum value for delta variability equals,

$$\delta_{\max} = \sqrt{\frac{n+1}{n+5}}$$

On a model with a large amount of modes ($n > 1$), the maximum value of delta tends to 1 ($\delta_{\max} \approx 1$). The maximum value of alpha variability using the above equations equals,

$$a_{\max} = \sqrt{\frac{2}{n+5}}$$

- 4. An unphysical (larger than permissible or negative value) selection of a variability would result in a FATAL.

MONCNCM**Nondimensional Normal Force and Pitching Moment**

Provides a stripwise aerodynamic normal force and pitching moment coefficients for CAERO1 based aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10
MONCNCM	NAME	LABEL							
	MREF	CAERID1	CAERID2	...	CAERIDn				

Example:

MONCNCM	LEFT	Normal force and pitching moment values for strips on the left wing.							
		2001	3001						

Descriptor	Meaning
NAME	Unique character string of up to 8 characters identifying the family of chord-wise strips. (Character)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the component. (Optional)
MREF	Fractional chord location of the aerodynamic strip about which the pitching moment is to be calculated. (Real; ≤ 0.0 MREF ≤ 1.0 ; Default = 0.25)
CAERID1	ID of a CAERO1 entry that contains aero boxes for which strip results are to be produced. (Integer > 0 or “ALL”). See Remarks 2. and 3.

Remarks:

1. The LABEL is optional.
2. Output is produced for all chordwise strips on the referenced CAERO1 entries. If a strip spans CAERO1 panels, results are output for the total strip.
3. If CAERID1 is the character string “ALL”, then output will be produced for all strips.
4. Strips are identified based on the y and z coordinates in the aerodynamic coordinate system.
5. If strips from separate CAERO1’s have the same y and z coordinate and the same strip width and share an xlocation (i.e., the leading edge of one strip equals the trailing edge from another) then they are processed as a single strip.
6. The normal force component is normalized by the dynamic pressure times the surface area of the strip. The moment component is normalized by the dynamic pressure times the surface area of the strip times the chord length at the center of the strip. The moment is calculated about the MREF location of the strip with the quarter-chord of the strip as the default.
7. Only CAERO1 id’s are supported.

MONDSP1**Displacement Monitor Point**

Defines a virtual point displacement response at a user-defined reference location (coordinates and coordinates system) as a weighted average of the motions at a set of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
MONDSP1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD	INDDOF	

Example:

MONDSP1	WING195	Wing twist at station 150.							
	5	WING150	1001	120	150.0	17.0	1002		

Descriptor	Meaning
NAME	Character string of up to 8 characters identifying the monitor point (Character)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point.
AXES	Component axes to monitor. (Any unique combination of the integers 1 through 6 with no embedded blanks.)
COMP	The name of an AECOMP or AECOMPL entry that defines the set of grid points over which the monitor point is defined.
CP	The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer ≥ 0 ; Default = 0)
X,Y,Z	The coordinates in the CP coordinate system at which the displacement is to be monitored.
CD	The identification number of a coordinate system in which the resulting displacement components are output. (Integer ≥ 0 ; Default = the coordinate system specified by the CP field)
INDDOF	Component numbers of all the independent grids from which the derived, dependent, monitor DOF's are to be computed. (Any unique combination of the integers 1 through 6 with no embedded blanks.) See Remark 3. (Default = 123)

Remarks:

1. The MONDSP1 is available for SOLs 101, 103, 105, 108, 109, 111, 112, 144, 146 and 200.
2. The entry can create either an aerodynamic or a structural MONDSP1, depending on the data provided in the COMP field. A structural MONDSP1 can have the same name as an aerodynamic MONDSP1, but two structural or two aerodynamic MONDSP1s cannot have the same name.

3. The INDDOF field defines the Ci field on the virtual RBE3; that is, it defines the components of the grids on the AECOMP that will be sampled to define the dependent (monitor point) displacement. Typically, the default is the correct choice. However, if there is only a single grid point, all six DOF's can be used.

MONGRP**Specification of a Monitor Group**

Defines a collection of monitor points into a group available for postprocessing.

Format:

1	2	3	4	5	6	7	8	9	10
MONGRP	GNAME	LABEL							
	NAME1	CLASS1	REAL1	INT1	STRING1				
	NAME2	CLASS2	REAL2	INT2	STRING2				
	NAME(m)	CLASS(m)	REAL(m)	INT(m)	STRING(m)				

Example:

MONGRP	WING	A COLLECTION OF MONITOR POINTS ON THE WING						
	WING195	SMONPT1		1	WL195			
	WING205	SMONPT1		2	WL205			
	WINGDSP	SMONDPI	20.0		GRID20			

Descriptor	Meaning
GNAME	A character string of up to 8 characters identifying the monitor group. (Character)
LABEL	A String comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point.
NAME(i)	Name of an existing monitor point. (Character)
CLASS(i)	Type of monitor point (either AMONPT1, AMONDPI, SMONPT1, SMONDPI, MONPT2, MONPT3)
REAL(i)	Optional real value to designate a real property of the monitor point.
INT(i)	Optional integer value to designate a integer property of the monitor point.
STRING	A label of the current monitor type. (Character string of no more than 32 character)

Remarks:

1. The MONGRP entry provides the user a means of specifying groups of monitor point data but does not produce any output on its own.
2. Aerodynamic (AMONPT1 and AMONDPI) and structural (SMONPT1, SMONDPI, MONPT2, MONPT3) cannot be mixed in a given group.
3. The REAL(i), INT(i) and STRING(i) data are provided to facilitate user display of the monitor data and can be used, for example, to label xy plots.
4. The LABEL should be unique across MONGRP entries.

MONPNT1**Integrated Load Monitor Point**

Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system. The integrated loads about this point over the associated nodes will be computed and printed for statics, dynamics and static aeroelastic trim analyses and form integrated loads on the nonlinear static aeroelastic database.

Format:

1	2	3	4	5	6	7	8	9	10
MONPNT1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD		

Example:

MONPNT1	WING155	Wing Integrated Load to Butline 155							
	34	WING		0.0	155.0	15.0			

Descriptor	Meaning
NAME	Character string of up to 8 characters identifying the monitor point (Character; no Default)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point. (Character; optional)
AXES	Component axes to monitor. (Any unique combination of the integers 1 through 6 with no embedded blanks.)
COMP	The name of an AECOMP or AECOMPL entry that defines the set of grid points over which the monitor point is defined. (Character; no Default)
CP	The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer ≥ 0 ; Default = 0)
X,Y,Z	The coordinates in the CP coordinate system about which the loads are to be monitored. (Real; Default = 0.0).
CD	The identification number of a coordinate system in which the resulting load components are output. (Integer ≥ 0 ; Default = the coordinate system specified by the CP field)

Remarks:

1. The MONPNT1 is available for SOLs 101, 108, 109, 111, 112, 144, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
2. The entry can create either an aerodynamic or a structural MONPNT1, depending on the data provided in the COMP field. A structural MONPNT1 can have the same name as an aerodynamic MONPNT1, but two structural or two aerodynamic MONPNT1s cannot have the same name.

3. In SOL 144, static aeroelasticity, the structural MONPNT1 integration is modified to account for the redistribution of loads and inertia caused by MPCs and rigid elements. In all other supported solution sequences, the integration is done using the g-set loads on all grid points specified on the referenced SET1 entry.

MONPNT2**Internal Load Monitor Point**

Element Monitor Output Results Item.

Format:

1	2	3	4	5	6	7	8	9	10
MONPNT2	NAME			LABEL					
	TABLE	TYPE	NDDLitem	EID					

Example:

MONPNT2	SB100	Leading edge stringer at root						
	STRESS	CBAR	SX2A	100				

Descriptor	Meaning
NAME	Unique character string of up to 8 characters identifying the monitor point (Character; no Default)
LABEL	An optional string comprising of no more than 56 characters (fields 3 through 9) that identifies the monitor point. (Character; Default = Blank)
TABLE	Type of output to be monitored. Options are STRESS, FORCE or STRAIN. (Character; no Default)
TYPE	Element type (Character; no Default)
NDDLitem	Component for this type to be monitored. This is the NDDL label for the particular Table and element type. (Character; no Default)
EID	Element ID. (Integer > 0)

Remarks:

1. The MONPNT2 is available for SOLs 101, 103, 108, 109, 111, 112, 144, 146 and 200.
2. Most element types have some items that can be monitored.
3. An assumption is made that the desired component is linear with respect to the displacement vector. If this assumption is not valid, the results will be incorrect.
4. Fictitious Table/Type/NDDLItems/EID generate a warning message and are ignored.
5. NDDL descriptions for Table=FORCE can be found in the [MSC Nastran DMAP Programmer's Guide](#) within the OEF datablock description. Table=STRESS and STRAIN are contained in the OES datablock description.

Once within the datablock description you can search for the element name (or better yet, element number, see the following table) you are interested in. There can be several different descriptions for an element type. For example, real vs. complex, thermal, stress vs. strain (within the OES description), linear vs nonlinear. In addition, the shell class of elements (quad4, quad8, quadr, tria3, tria6, triar) will have a composite form (quad4lc, quud8lc, quadrlc, tria3lc, .., i.e., basename + "LC"), corner or bilin stresses (basename + "C").

By looking at the comments contained in the text make sure you are reading from the appropriate section. The NDDLItem is labeled as the 'NAME' field within the [MSC Nastran DMAP Programmer's Guide](#).

You can also print the NDDL description for the entire database by running the following 4 statement bulk data file.

```
sol loadnndl  
compile nndl=nndl,list  
cend  
enddata
```

MONPNT3**Integrated Load Monitor Point**

Sums select Grid Point Forces to a user chosen monitor point.

Format:

1	2	3	4	5	6	7	8	9	10
MONPNT3	NAME	LABEL							
	AXES	GRIDSET	ELEMSET	CP	X	Y	Z	XFLAG	
	CD								

Example:

MONPNT3	t0	Fuselage station 1105						
	123456	1	2	0	30.0			ASM

Descriptor	Meaning
NAME	Unique character string of up to 8 characters identifying the monitor point (Character; Required)
LABEL	A optional string comprising of no more than 56 characters (fields 3 through 9) that identifies the monitor point.
AXES	Component axes about which to sum. Any unique combination of the integers 1 through 6 with no embedded blanks. (Integer; Required)
GRIDSET	Refers to a SET1 entry that has a list of grids to be included in the monitored point. (Integer; Required)
ELEMSET	Refers to a SET1 entry that has a list of elements to include at the monitored point. (Integer; optional)
CP	The identification number of a coordinate system in which the (x,y,z) coordinates are defined. (Integer ≥ 0 ; Default=0)
X,Y,Z	The coordinates in the CP coordinate system about which the forces are to be summed. (Real; Default = 0.0).
XFLAG	Exclusion flag. Exclude the indicated Grid Point Force types from summation at the monitor point. Default = blank (no type excluded). See Remark 4.
S	SPCforces
M	MPC forces
A, L, or P	applied loads
D	dmig's (and any other type not described above) at the monitored point.
C	contact forces (SOL 400 only).
CD	The identification number of a coordinate system in which the results are output. (Integer $>= 0$, Default = the coordinate system specified by the CP field.)

Remarks:

1. The MONPNT3 is available for SOLs 101, 103, 108, 109, 111, 112, 144, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN). Note that MONPNT3 output to csv file is available only to SOL 144.
 2. If ELEMSET is blank, no contributions are made from the set of elements attached to the grid.
 3. Fictitious grids or elements do not produce error or warning messages.
 4. For linear solution sequences, if the exclusion flags omit some grid point force types but not all of them (i.e; if the field is not blank and is not SMAD) then the following limitations exist:
 - The results for that MONPNT3 will not be exported to a CSV file, for SOL 144 only.
 - That MONPNT3 may not be used on a MONSUM, MONSUM1 or MONSUMT entry.
 - It is only available in SOLs 101, 103, 144 and SOL 200 (ANALYSIS = STATICS, MODES, or SAERO).
 5. MONPNT3 can be useful in calculating shear, moment and torque from the internal loads in a structure. For example, if a split is made in a fuselage component and all the grids that reside on this split are placed in the GRIDSET, differing internal loads resultants can be obtained based on the ELEMSET and XFLAG values. If a cut is made in a structure, there are three types of loads:
 - A - loads that come from the elements upstream of the cut.
 - B - loads that are applied to the grids on the cut from any other source.
 - C - loads that come from the elements downstream of the cut.Where A + B + C = 0
- Useful options for these two values are:
- a. If the ELEMSET includes all the elements that connect to the GRIDSET that are on the upstream/outboard part of the split and the XFLAG value is blank then the internal load is calculated using the elements that connect to the GRIDSET that are NOT included in ELEMSET. The direction of this load is reversed. So actually -C will be calculated which matches A+B. This in effect gives the resultants on the downstream/inboard side of the split pointing into upstream/ outboard direction which will include any loads applied to the GRIDSET from any source.
 - b. If the ELEMSET includes all the elements that connect to the GRIDSET that are on the upstream/outboard part of the split and the XFLAG value is SMAD then the internal load is calculated using the elements listed in ELEMSET. In that case, A will be calculated. This in effect gives the resultants on the upstream/outboard side of the split and does not include loads applied to the GRIDSET from any other source.
6. For SOL 400 (ANALYSIS = NLSTAT or NLTRAN), the following limitations exist:
 - The results for that MONPNT3 will not be exported to a CSV file, for SOL 144 only.
 - That MONPNT3 may not be used on a MONSUM, MONSUM1 or MONSUMT entry.
 - Thermal loads are not supported.

7. For linear solution sequences with XFLAG= blank or SMAD, the program applies a mini-EMA (element matrix assembly) algorithm that bypasses GPFDR (grid point force data recovery) and another modules to improve performance. This algorithm forms a stiffness matrix that just includes those elements specified in MONPNT3 entries. The monitor point results are computed directly from multiplying this matrix by an integration matrix that transfers forces to the monitor points. These results may be slightly different from the GPFDR results, such as clean zero forces versus small residual forces.

MONSUM

Linear Combinations of Monitor Point Components

Defines a new monitor result that is the weighted sum of existing monitor results. The existing monitor points do not need to be of the same type but they must be of similar type (see Remark 5.).

Format:

1	2	3	4	5	6	7	8	9	10
MONSUM	NAME	LABEL							
	NEWAXISA	MTYPE1A	NAME1A	AXES1A	COEF1A	NAME2A	AXES2A	COEF2A	
			NAME3A	AXES3A	COEF3A	etc.			
		MTYPE2A	NAME2A	etc.					
	NEWAXISB	MTYPE1 B	NAME1B	AXES1B	COEF1B	NAME2B	etc.		
		MTYPE1C	etc.						

Example: Create a new monitor point result by adding an aerodynamic monpnt1 and a monpnt3

MONSUM	AM1PSM3	Adding the wing lift to a free body load							
	3	AMONPNT1	WING	3	1.0				
		MONPNT3	FREEW	3	1400.	FREEW	5	-1000	

Alternate Format:

1	2	3	4	5	6	7	8	9	10
MONSUM	NAME				LABEL				
	MTYPE	NEWAXISA	NAME1A	AXES1A	COEF1A	NAME2A	AXES2A	COEF2A	
			NAME3A	AXES3A	COEF3A	etc.			
		NEWAXISB	NAME2A	AXES1B	COEF1B	NAME2B	AXIS2B	COEF2B	
			NAME3B	AXES3B	COEF3B	etc.			
		NEWAXISC	etc.						

Alternate Example: Scale an existing monitor point

MONSUM	ROOT	Scale from in-lbs to newton-cm and from lbs to newtons							
	AMONPNT1	123456	ROOT	123	4.482	ROOT	456	11.385	

Descriptor	Meaning
NAME	Character string of up to 8 characters identifying the monitor result. (Character)
LABEL	A string comprising of no more than 56 characters (fields 3 through 9) that identifies and labels the merged monitor result.
NEWAXISj	The component axis of the newly-created monitor point into which the summed quantity will be stored (integer, any unique combination of the integers 1 to 6 with no embedded blanks, see Remarks 7. and 8.)
MTYPij	Monitor type to be merged. (Character, one of “AMONPNT1”, “AMONDSP1”, “SMONPNT1”, “SMONDSP1”, or “MONPNT3”; no Default). See Remarks 5. and 6.

Descriptor	Meaning
MTYPE	Monitor type to be merged in the alternate format. (Character; one of “AMONPNT1”, “AMONDSP1”, “SMONPNT1”, “SMONDSP1”, or “MONPNT3”; no Default). See Remarks 5. and 6.
NAMEij	Name of the monitored quantity that is to be merged
AXESij	Component axes that are to be summed. (any unique combination of the integers 1 to 6 with no embedded blanks)
COEFij	Coefficient to be applied to the component(s) called out on AXESij field. (Real; Default = 1.0)

Remarks:

1. The MONSUM is available in SOLs 101, 103, 108, 109, 111, 112, 144, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
2. The LABEL is a 56 character string that should be unique.
3. The MONSUM can be used to update an existing monitor result by setting all the NAMEij terms equal to NAME. In this case, the alternate format is available and the NEWAXISj component is a scalar multiple of the original component:
$$\text{MONSUM}_j = M_{Rj} \text{ COEF}_j$$
4. When the NAME differs from the NAMEij values, it must be unique with respect to all monitor quantities. The result of the MONSUM entry is to create new monitor point(s) that are equal to:

$$\text{MONSUM}_j = \sum_i^n \text{COEF}_{ij} \text{MR}_{ij}$$

where MR_{ij} is the result from the individual component.

5. The merged monitor points must be of a similar type. “Similar” types are defined as:
Force and moment summation monitor points: AMONPNT1, SMONPNT1, MONPNT3
Average displacement monitor points: AMONDSP1 and SMONDSP1
6. For MONPNT1’s and MONDSP1’s, the MTYPij can be aerodynamic or structural.
MTYPij=AMxxx1 designates aerodynamic while SMxxx1 designates a structural monitor point.
7. If multiple components are to be summed, the NEWAXISj field must be the union of subsequent AXISij fields. If the subsequent AXISij field is blank, the components are determined from NEWAXISj.
8. If the NEWAXISj field indicates a single output, the AXISij fields must reference a single input, but it can be any value from 1 to 6.
9. The same component cannot be referenced multiple times on the NEWAXISj fields for a single MONSUM entry.
10. Structural monitor points may span superelements.

11. If multiple types are specified on a MONSUM, the resulting entry is of a type that appears on the MONSUM with the following order of precedence: smonpnt1, smondsp1, monpnt3, amonpnt1, amondsp1. E.g., a amonpnt1 and a monpnt3 appearing on the same entry will result in a monpnt3 regardless of which appears first.
12. If all the MTYPij values are of the same type, the alternate format provides a simplified interface.
13. A MONSUM can reference another MONSUM (including itself) as long as there is not a circular reference. A MONSUM cannot reference another MONSUM1.
14. MONPNT3's that have one, two or three excluded items (e.g., XFLAG=SMA) cannot be referenced on a MONSUM.
15. Since CP,X,Y,Z and CD are not known with certainty for a MONSUM, they are not printed as part of the monitor point output. Further, in SOL's 101 and 144, COMP=**SUM**, is used to indicate that the associated monitor point has been derived from a MONSUM.

MONSUM1

Linear combination of two or more monitor points.

Defines a new monitor result that is the weighted sum of existing monitor results. The location of the computed MONSUM1 is specified. The existing monitor points do not need to be of the same type but they must be of similar type (See Remark 3.)

Format:

1	2	3	4	5	6	7	8	9	10
MONSUM1	NAME	LABEL							
	CP	X	Y	Z	CD				
	NEWAXES	MTYP1	NAME1a	AXES1A	COEF1A	NAME2A	AXES2A	COEF2A	
			NAME3A	AXES3A	COEF3A	Etc			
		MTYP2	NAME2A	AXES1B	COEF1B	NAME2B	AXIS2B	COEF2B	
			NAME3B	AXES3B	COEF3B	Etc.	.		
		MTYP3	Etc						

Example: Create a new monitor point result by adding an aerodynamic monpnt1 and a monpnt3

MONSUM1	ROOT	SCALE FROM IN-LBS TO NEWTON-CM AND FROM LBS TO NEWTONS							
		500.	0.0	25.0	20	200			
	123456	MONPNT3	FS_1000	123	-4.482	FS_1000	356	-11.385	
		AMONPNT1	ROOT	123	4.482	R00T	456	11.385	

Alternate Format:

1	2	3	4	5	6	7	8	9	10
MONSUM1	NAME	LABEL							
	CP	X	Y	Z	CD				
	MTYPE	NEWAXISA	NAME1a	AXES1A	COEF1A	NAME2A	AXES2A	COEF2A	
			NAME3A	AXES3A	COEF3A	Etc			
		NEWASIXB	NAME2A	AXES1B	COEF1B	NAME2B	AXIS2B	COEF2B	
			NAME3B	AXES3B	COEF3B	Etc.	.		
		NEWAXISC	Etc						

Alternate Example: Create a new monitor point result by adding an aerodynamic monpnt1 and a monpnt3

MONSUM1	ROOT	SCALE FROM IN-LBS TO NEWTON-CM AND FROM LBS TO NEWTONS							
		500.	0.0	25.0	20				
	MONPNT3	123456	FS_1000	123	-4.482	FS_1000	456	-11.385	
	AMONPNT1	123456	ROOT	123	4.482	R00T	456	11.385	

Descriptor	Meaning
NAME	Character string of up to 8 characters identifying the monitor result (Character)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the merged monitor result.
CP	The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer > 0; Default = 0)
X,Y,Z	The coordinates in the CP coordinate system about which the loads are to be monitored. (Real; Default = 0.0).
CD	The identification number of a coordinate system in which the resulting load components are output. (Integer > 0; Default = the coordinate system specified by the CP field)
NEWAXESj	The component axis of the newly-created monitor point into which the summed quantity will be stored (integer, any unique combination of the integers 1 to 6 with no embedded blanks, see Remarks 5. and 6.)
MTYPi	Monitor type to be merged. (Character, one of “AMONPNT1”, “SMONPNT1”, or “MONPNT3”, no default). See Remarks 3. and 4.
NAMEij	Name of the monitored quantity that is to be summed.
AXESij	Component axes that are to be scaled. (any unique combination of the integers 1 to 6 with no embedded blanks. Must be a subset of NEWAXESj).Default =same as NEWAXESj
COEFij	Scaling coefficient to be applied to the component(s) called out on AXESij field. (Real, Default=1.0)

Remarks:

1. The LABEL is a 56 character string that should be unique.
2. The result of the MONSUM1 entry is to create new monitor point(s) as follows:

$$\text{MONSUM1}_j = \sum_i^n \text{COEF}_{ji} \text{MR}_i$$

Where MR_i is the result from the individual component.

3. The merged monitor points must be of a similar type. “Similar” types are defined as:
 - Force and moment summation monitor points: AMONPNT1, SMONPNT1, MONPNT3
4. For MONPNT1’s the MTYPij can be aerodynamic or structural. MTYPij=AMONPT1 designates aerodynamic while SMONPNT1 designates a structural monitor point.
5. If multiple components are to be summed, the NEWAXISj field must be the union of subsequent AXISij fields. If the subsequent AXISij field is blank, the components are the same as NEWAXISj.

6. If the NEWAXISj field indicates a single output, the AXISij fields must reference a single input, but it can be any value from 1 to 6. There may be as many as 6 AXISij and COEFij for each referenced NAMEij.
7. The same component cannot be referenced multiple times on the NEWAXISj fields for a single MONSUM1 entry.
8. If multiple types are specified on a MONSUM1, the resulting entry is of a type that appears on the MONSUM1 with the following order of precedence: SMONPNT1 (structural MONPNT1), MONPNT3, AMONPNT1 (aerodynamic MONPNT1). E.g., a AMONPNT1 (aerodynamic MONPNT1) and a MONPNT3 appearing on the same entry will result in a MONPNT3 regardless of which appears first.
9. A MONSUM1 can reference the results of another MONSUM1 or a MONSUMT as long as there is not a circular reference.
10. The MONSUM1 entry is available in SOLs 101,103,108,109,111,112,144,146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
11. The CP, CD, and X Y Z location coordinates are for reference only and do not affect the calculations.

MONSUMT

Linear combination of two or more monitor points with moment transfer

Defines a new monitor result that is the sum of existing monitor results. The existing monitor points do not need to be of the same type but they must be of similar type (See Remark 3) 3.)

Format:

1	2	3	4	5	6	7	8	9	10
MONSUMT	NAME	LABEL							
	AXES	CP	X	Y	Z	CD			
	MTYP1	NAME1a	NAME1b	NAME1c	NAME1d	NAME1e	NAME1f	NAME1g	
	MTYP2	NAME2a	NAME2b	NAME2c	NAME2d	NAME2e	NAME2f	NAME2g	

Example: Create a new monitor point result by adding an aerodynamic and a monpnt3

MONSUMT	FS_900	FUSELAGE STATION 900						
	123456		500.	0.0	25.0	20		
	MONPNT3	FS_1000	FS_1100	FS_1200	MONPNT3			
	AMONPNT1	HTP_ROOT	AMONPNT1					

Descriptor	Meaning
NAME	Character string of up to 8 characters identifying the monitor result (Character)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the merged monitor result.
CP	The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer > 0; Default = 0)
X,Y,Z	The coordinates in the CP coordinate system about which the loads are to be monitored. (Real; Default = 0.0).
CD	The identification number of a coordinate system in which the resulting load components are output. (Integer > 0; Default = the coordinate system specified by the CP field)
AXES	The component axis of the newly-created monitor point into which the summed quantity will be stored (integer, any unique combination of the integers 1 to 6 with no embedded blanks).
MTYPi	Monitor type to be merged. (Character, one of “AMONPNT1”, “SMONPNT1”, or “MONPNT3”, no default). See Remarks 3. and 4.
NAMEij	Name of the monitored quantity that is to be summed.

Remarks:

1. The LABEL is a 56 character string that should be unique.
2. The NAME must be unique with respect to all monitor quantities. The result of the MONSUMT entry is to create new monitor point(s) as follows:

$$\text{MONSUMT}_j = \sum_i^n T_{ji} \text{MR}_i$$

Where MR_i is the result from the monitor points being summed and T_{ji} is set of partial rigid body vectors for the locations of the monitor points being summed, with the origin at the X, Y, Z location.

3. The summed monitor points must be of a similar type. "Similar" types are defined as:
 - Force and moment summation monitor points: AMONPNT1, SMONPNT1, MONPNT3
4. For MONPNT1's the MTYPij can be aerodynamic or structural. MTYPij=AMONPNT1 designates aerodynamic while SMONPNT1 designates a structural monitor point.
5. If multiple types are specified on a MONSUMT, the resulting entry is of a type that appears on the MONSUMT with the following order of precedence: If multiple types are specified on a MONSUM1, the resulting entry is of a type that appears on the MONSUM1 with the following order of precedence: SMONPNT1 (structural MONPNT1), MONPNT3, AMONPNT1 (aerodynamic MONPNT1). E.g., a AMONPNT1 (aerodynamic MONPNT1) and a MONPNT3 appearing on the same entry will result in a MONPNT3 regardless of which appears first.
6. A MONSUMT can reference the results of another MONSUMT or a MONSUM1 as long as there is not a circular reference.
7. The MONSUMT entry is available in SOLs 101,103,108,109,111,112,144,146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).

MPC**Multipoint Constraint**

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

where u_j represents degree-of-freedom Cj at grid or scalar point Gj.

Format:

1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	A1	G2	C2	A2		
		G3	C3	A3	-etc.-				

Example:

MPC	3	28	3	6.2	2		4.29		
		1	4	-2.91					

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
Gj	Identification number of grid or scalar point. (Integer > 0)
Cj	Component number. (Any one of the Integers 1 through 6 for grid points; blank, zero or 1 for scalar points.)
Aj	Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)

Remarks:

1. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
2. The first degree-of-freedom (G1, C1) in the sequence is defined to be the dependent degree-of-freedom. A dependent degree-of-freedom assigned by one MPC entry cannot be assigned dependent by another MPC entry or by a rigid element.
3. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
4. The m-set degrees-of-freedom specified on this entry may not be specified on other entries that define mutually exclusive sets. See the “Degree-of-Freedom Sets” on page 1557 for a list of these entries.
5. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in “Parameters” on page 1409). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,3.

MPCADD**Multipoint Constraint Set Combination**

Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.

Format:

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

Example:

MPCADD	101	2	3	1	6	4			
--------	-----	---	---	---	---	---	--	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
Sj	Set identification numbers of multipoint constraint sets defined via MPC entries. (Integer > 0)

Remarks:

1. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
2. The Sj must be unique and may not be the identification number of a multipoint constraint set defined by another MPCADD entry.
3. MPCADD entries take precedence over MPC entries. If both have the same SID, only the MPCADD entry will be used.
4. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in “[Parameters](#)” on page 1409). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,101.
5. If Modules are present then this entry may only be specified in the main Bulk Data section.

MPCAX**Conical Shell Multipoint Constraint**

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

for conical shell coordinates, where u_j represents the degree-of-freedom C_j at ring RID j and harmonic HID j .

Format:

1	2	3	4	5	6	7	8	9	10
MPCAX	SID				RID1	HID1	C1	A1	
	RID2	HID2	C2	A2	-etc.-				

Example:

MPCAX	32				17	6	1	1.0	
	23	4	2	-6.8					

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
RID j	Ring identification number. (Integer > 0)
HID j	Harmonic identification number. (Integer > 0)
C j	Component number. (1 ≤ Integer ≤ 6)
A j	Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)

Remarks:

1. MPCAX is allowed only if an AXIC entry is also present.
2. The first degree-of-freedom in the sequence is assumed to be the dependent degree-of-freedom. A dependent degree-of-freedom assigned by one MPCAX entry cannot be assigned dependent by another MPCAX entry.
3. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
4. Dependent degrees-of-freedom appearing on MPCAX entries may not appear on OMITAX, SPCAX, or SUPAX entries.
5. See [Conical Shell Element \(RINGAX\)](#) (Ch. 3) in the *MSC Nastran Reference Guide* for further discussion of the problem.
6. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in “[Parameters](#)” on page 1409). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,32.

MPCD**Load Selectable Value for Y_m of Non Homogenous Multipoint Constraint**

Defines a load selectable value for Y_m of a MPCY entry.

Format:

1	2	3	4	5	6	7	8	9	10
MPCD	SID	GM1	CM1	YM1	GM2	CM2	YM2		

Example:

MPCD	700	101	2	.06					
------	-----	-----	---	-----	--	--	--	--	--

Descriptor	Meaning
SID	Set identification number of the MPCD entry. (Integer > 0)
GMi	Grid or scalar point identification number. Along with Ci it identifies the Equation of a MPCY. (Integer > 0)
Ci	Component number. (Any one of the Integers 1 through 6 for grid points, blank, zero or 1 for scalar points.)
YMi	Right hand side value of MPC equation. (Real)

Remarks:

1. In static solution sequences, the SID is selected by the LOAD Case Control command.
2. The Gi,Ci referenced on this entry must also be referenced on a MPCY Bulk Data entry and selected by an MPC Case Control command.
3. Values YMi will override the value specified on an MPCY Bulk Data entry.
4. The LOAD Bulk Data entry will not combine an MPCD load entry.
5. Two separate MPC equations may be pointed to per entry.

MPCREEP**SOL 600 Advanced Creep Options**

Specifies input values for Marc's creep parameter when creep analysis is performed using SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MPCREEP	ITYPE	IMPLK	IIMPLM	IHOW					

Example:

MPCREEP	1	0	0	0					
---------	---	---	---	---	--	--	--	--	--

Descriptor	Meaning
ITYPE	Type of explicit creep analysis enter one of the following: (Integer; Default = 0)
0	Normal creep (Maxwell Model)
1	Viscoplastic creep
2	Viscoplastic creep with nonassociative flow rule.
IMPLK	Flag determining if the explicit Kelvin model is to be used. (Integer; Default = 0)
0	Explicit Kelvin model is not used
1	Explicit Kelvin model is used.
IIMPLM	Flag determining if the implicit Maxwell creep or implicit viscoplastic model is to be used. (Integer; Default = 0)
0	These models are not used.
1	The implicit Maxwell creep or implicit viscoplastic model is used.
IHOW	Flag specifying how the Maxwell creep model or implicit viscoplastic model is used. (Integer; Default = 0)
0	Use elastic tangent
1	Use secant tangent
2	Use radial return

Remarks:

1. Set IHOW=0 or leave field blank if IIMPLM=0.
2. When using the implicit Maxwell creep model, the stress dependence must be in exponential form and the CRPLAW user subroutine cannot be used.
3. Only one MPCREEP entry may be entered in the input. If additional MPCREEP entries are found, the first will be used.
4. If ITYPE > 0, Bulk Data entry, MACREEP or MTCREEP is required in addition to this entry.

5. This entry maps to Marc's CREEP parameter.
6. This entry is not necessary if fields 2-5 are all zero or blank.
7. Kelvin creep requires user subroutines.

MPCY**Non Homogenous Multipoint Constraint**

Defines a multipoint constraint equation of the form

$$A_m u_m + \sum_i A_i u_i = Y_m$$

where u_m represents degree-of-freedom C_m at grid or scalar point G_m defined to be the dependent degree-of-freedom. u_i represents degree-of-freedom C_i at grid or scalar point G_i defined to be the independent degree-of-freedom. Y_m is a value for the equation.

Format:

1	2	3	4	5	6	7	8	9	10
MPCY	SID	GM	CM	AM	YM				
		G1	C1	A1	G2	C2	A2		
		G3	C3	A3	...				

Example:

MPCY	70	205	1	1.0	.02-3				
		1608	1	1.2					

Descriptor	Meaning
SID	Set identification number of a multi-point constraint set. (Integer > 0)
GM	Identification number of grid or scalar point. (Integer > 0)
Cj	Component number. (Any one of the Integers 1 through 6 for grid points; blank, zero or 1 for scalar points.)
AM	Coefficient. (Real nonzero value)
YM	Right hand side value. (Real; Default = 0.0)
Gi	Identification number of grid or scalar point. (Integer > 0)
Ci	Component number. (Any one of the Integers 1 through 6 for grid point; blank or zero for scalar points.)
Ai	Coefficient. (Real; Default = 0.0)

Remarks:

1. Multipoint constraint sets must be selected with Case Control command, MPC=SID.
2. The m-set degree-of-freedom specified on this entry may not be specified on other entries that define mutually exclusive sets. The GM term is an equation marker. If PARAM,AUTOMSET,YES is on Nastran may choose internally another Gi as the actual dependent degree-of-freedom.

MPHEAT

Maps to Marc's HEAT parameter for SOL 600 heat transfer analysis

Maps to Marc's HEAT parameter for SOL 600 heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
MPHEAT	IDIST	LHMAX	INTPT	ICONVT	ILAYC	LINR	NONH		

Example:

MPHEAT	1	1	1	0	3	1	0		
--------	---	---	---	---	---	---	---	--	--

Descriptor	Meaning
IDIST	Temperature distribution in thickness direction of heat transfer shell elements. (Integer; Default = 0)
0	Linear Variation (membrane)
1	Quadratic variation
LHMAX	Maximum number of latent heats associated with any material. (Integer; Default = 0)
INTPT	Controls integration point output. (Integer; Default = 0)
0	No integration point output
1	Gradients and fluxes at integration points
2	Save as 1 plus flux values at grid points
ICONVT	Flag to include convective terms. (Integer; Default = 0)
0	Do not include convective terms
2	Include convective terms
LAYC	Number of through thickness layers for heat transfer shells. (Integer; Default = 1)
LINR	Flag controlling linearization of surface energy and receding surface calculations. (Integer; Default = 1)
1	Linearize the calculations
2	Do not linearize the calculations
NONH	Flag to store nonhomogeneous density for postprocessing. (Integer; Default = 0)
0	Do not store the values
1	Store the values

MPROCS**Defines a Processor Entry to be Used in the SOL 600 Marc Parameter Section**

Defines a processor entry to be used in the SOL 600 Marc Parameter Section. It controls the use of vectorization and parallelization in the element assembly phase in Marc. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MPROCS	I2	I3	I4	I5	I6	I7			
	IRF	IFGF	IAXIS	WGHT					
	Dx	Dy	Dz	X	Y	Z			

Example:

MPROCS	1	1	1	1	11				
--------	---	---	---	---	----	--	--	--	--

Descriptor	Meaning
I2	Value of Marc's 2nd processor field. Number of CPUs to use. (Integer; Default = 0)
I3	Value of Marc's 3rd processor field. Optimal vector length (Defaults to 32 or 64 depending on the computer system). (Integer; Default = 0)
I4	Value of Marc's 4th processor field (enter 1 if beta matrices are to be formed in parallel). (Integer; Default = 0)
I5	Value of Marc's 6th processor field (enter 1 to use DDM single file input). (Integer; Default = 0)
I6	Value of Marc's 7th processor field -- Domain Decomposition Method. (Integer; Default = 0) Enter 11 to use Metis Best (best method of 12, 13 or 14) decomposition; Default. Enter 12 to use Metis Element-Based decomposition. Enter 13 to use Metis Node-Based decomposition. Enter 14 to use Vector decomposition. Enter 15 to use Radial decomposition. Enter 16 to use Angular decomposition.
I7	Value of Marc's 8th processor field. Enter 0 to use in-core storage for DDM single file creation. Enter 1 to use out-of-core storage. (Integer; Default = 0)
IRF	Island removal flag for domain decomposition. (Default = 0) 0 Do not remove islands 1 Attempt to remove islands
IFGF	Fine graph flag for domain decomposition. (Default = 0) 0 Coarse graph 1 Fine graph

Descriptor	Meaning
IAXIS	Control of point on axis of rotation for radial/angular domain decomposition
0	Use centroid of the boundary box of the model
1	User supplied point (supply X, Y, Z below)
WGHT	Element coefficient weight. Controls balance between computational costs of domains, range is 0.0 to 1.0. (Default = 1.0; Use full element weight)
0.0 -	Do not use element weight
Dx	First direction cosine of vector used for decomposition method 14, 15, or 16
Dy	Second direction cosine of vector used for decomposition method 14, 15, or 16
Dz	Third direction cosine of vector used for decomposition method 14, 15, or 16
X	X coordinate of point on axis
Y	Y coordinate of point on axis
Z	Z coordinate of point on axis

Remarks:

1. This entry should only be made for special cases when using DDM with Marc's single file parallel capability. Do not enter except for SOL 600 parallel executions.
2. Consult the Marc documentation volumes A, B, C for more detailed descriptions of this entry.
3. Enter only one of MPROCS entry in any given file. If more than one is entered, the first encountered will be used.
4. The continuation lines may be omitted if not required.
5. I2, I3, and I4 are no longer used and should be zero or blank.

MREVERS

Defines Which Elements Require Node Numbering to be Reversed - SOL 600

This entry is only used if the checks built into SOL 600 somehow fail to predict some elements which need node numbering reversed. Nastran allows clockwise as well as counter clockwise node numbering. Marc only allows counter clockwise numbering for most elements. SOL 600 has builtin logic to check for node reversal, however there may be some instances where Marc gives an “inside out” message during increment (before any nonlinear loads have been applied). This rarely happens unless field 7 of the GRID entry is set, in which case it sometimes happens. When this happens, the user can apply this entry to instruct SOL 600 how to renumber elements which have “inside out” messages during increment zero. (If “inside out” messages occur after increment zero, this is normally due to large loads and the element has either failed or the model needs to be re-meshed).

Format:

1	2	3	4	5	6	7	8	9	10
MREVERS	N1	M1	N2	M2	N3	M3	N4	M4	
	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	
	ID9	ID10	etc.						

Example:

MREVERS		2	3						
	500	510	520	888	889				
	950								

Descriptor Meaning

- | | |
|-----|--|
| Ni | Node number location to be reversed. (Integer; no Default; see Remark 1.) |
| Mi | Paired node number location to be reversed. (Integer; no Default; see Remark 1.) |
| IDi | Element identification number. (Integer; no Default) |

Remarks:

1. Example of node number locations are as follows:
For 4-node quads the node number locations are 1, 2, 3, 4
For 8-node hexas the node number locations are 1, 2, 3, 4, 5, 6, 7, 8
2. The 2nd and succeeding lines may be used to make it easy to enter elements with “inside out” messages in a preliminary marc.out file. The user can grep for “inside out”, save the message in a file, easily edit the file to retain only the element numbers and then add these to lines 2 and subsequent of the alternate format.
3. Up to four pairs of node locations may be reversed per entry. If additional pairs are necessary, repeat the entry and its continuation lines.
4. The example will reverse nodes locations 2 and 3 for elements 500, 510, 520, 888, 889 and 950.

5. This entry may only be used in MD Nastran R2.1 and later versions.
6. See PARAM,MAXIREVV for a similar option.

MRSSCON

Auxiliary data for RSSCON for use in SOL 600

Defines auxiliary data for RSSCON for use in SOL 600. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MRSSCON	ICONV	IDB	IDP	IDM	A	J	I	E	

Example:

MRSSCON	0	5000	5000	900	20.0	200.	100.	30.0E	
---------	---	------	------	-----	------	------	------	-------	--

Descriptor	Meaning
ICONV	Flag indicating how RSSCON rigid elements will be converted in SOL 600. (Integer; Default = 0) <ul style="list-style-type: none"> 0 Convert all RSSCON to CBEAM using values in fields 2-8 1 Convert all RSSCON to RBE2 using 3 degrees of freedom 2 Convert all RSSCON to RBE2 using 6 degrees of freedom
IDB	Starting CBEAM ID for all RSSCON's that are mapped to CBEAM's or RBE2 ID if RSSCON's are mapped to RBE2's (Integer; Default = 99000; see Remark 1.)
IDP	PBEAM ID for use by all RSSCON's in the model mapped to CBEAM's. See Remark 5. (Integer; Default = 99000)
IDM	MAT1 ID for use by all RSSCON's in the model mapped to CBEAM's. See Remark 6. (Integer; Default = 99000)
A	Area to be placed in the PBEAM (Real; Default = 10.0)
J	Torsional moment of inertia to be placed in the PBEAM (Real; Default = 20.0)
I	Moment of inertia (in both directions) to be placed in the PBEAM (Real; Default = 10.0)
E	Elastic modulus to be placed in the IDM MAT1 entry (Real; Default = 1.0E6)

Remarks:

1. ICONV=1 or 2 may cause MPC conflicts and/or may fail if AUTOMSET,YES is used.
2. RSSCON type=elem is not presently supported by SOL 600.
3. This entry applies to RSSCON type=grid where ES1, EA1, EB1 all have different coordinates, and if used ES2, EA2 and EB2 do not have the same coordinates as each other (see RSSCON entry for meaning of these values).
4. Only one MRSSCON entry should be included in the input data. If more than one is included, the first as determined by XSORT, will be used.

5. If IDP is entered, a PBEAM with and id of IDP and properties A, J, and I will automatically be created.
6. If IDM is entered, a MAT1 with a Young's modulus of E and Poisson ratio of 0.3 will automatically be created.

2550

MSTACK

Defines the Direction in Which 3D Solid Composites are Stacked -SOL 600

MSTACK

Defines the Direction in Which 3D Solid Composites are Stacked -SOL 600

Defines the direction in which 3D Solid Composites are stacked. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MSTACK	ID1	THRU	ID2	IDIR					

Example:

MSTACK	100	THRU	200	1					
--------	-----	------	-----	---	--	--	--	--	--

Descriptor Meaning

- | | |
|------|---|
| ID1 | Starting solid element ID (Integer; Required; no Default) |
| THRU | Enter the character string THRU if more than one element is desired |
| ID2 | Ending solid element in a range of ID1 to ID2 (Integer; optional; no Default) |
| IDIR | Stacking direction for this range of solid elements (Integer; Required; Default = 1) <ul style="list-style-type: none"> 1 Thickness direction is 1-2-3-4 face to 5-6-7-8 face (for CHEXA) 2 Thickness direction is 1-5-8-4 face to 2-6-7-3 face (for CHEXA) 3 Thickness direction is 2-6-5-1 face to 3-7-8-4 face (for CHEXA) |

Remarks:

1. Enter as many MSTACK lines as necessary to define any solid composites where the defaults do not apply. If IDIR is 1 for all elements in the range, this entry is not required.
2. This entry equates to Marc's EGEOM3 value for solid composite elements (for example element 149).

MT16SEL

Limits results to selected elements or grids for t16 and t19 for SOL 600

Limits elements and/or grid results to selected elements or grids for t16 and t19 file results. Used in SOL 600.

Format

1	2	3	4	5	6	7	8	9	10
MT16SEL	TYPE	ID1	THRU	ID2	BY	ID3			
	TYPE	ID1	THRU	ID2	BY	ID3			

Example:

MT16SEL	GRID	1	THRU	100					
	GRID	2525	THRU	3000	BY	10			
	ELEM	100	THRU	500	BY	2			
	ELEM	1000	THRU	2000					

Descriptor	Meaning
TYPE	Type of output - Enter one of the values "GRID", or "ELEM", for "GRID", nodal results such as displacement, spc force, etc are selected. For "ELEM", element results such as stress, strain, etc. are selected. This entry should normally be used in combination with MARCOUT unless the MARCOUT defaults are satisfactory for the model.
ID1	Starting ID of above selection. (Integer; no Default required value)
ID2	Ending ID of above selection. (Integer; Default = ID1)
ID3	Increment by value. (Integer; Default = 1)

Remarks:

1. It is highly recommended that all grid and element output be placed on the t16 file since it is not usually known where the maximum values will occur and the max/min values could easily be missed. Also, if all grid/element values are not selected, contour plots could be misleading. To output all grid/element results in the t16 file, do not make any MT16SEL entries.
2. This entry may not be used when OUTR options (which requires a t16 to op2 conversion) on the SOL 600 entry are requested. If this entry is made together with any OUTR options, this entry will be ignored and a warning message issued.
3. ID2 must be blank, zero or greater than ID1 (if ID2 is blank or zero, only ID1 will be used)
4. ID3 must not be negative. If can be blank or zero in which case it is reset to one.
5. This entry (with the exception of Remark 3) activates Marc's POST version 13 and overrides any other POST version specifications such as PARAM,MARCPOST.
6. This entry can be used to also obtain a t19 file with selected element and/or grid results by placing PARAM,MARCT19,1 in the bulk data.

2552

MT16SEL

Limits results to selected elements or grids for t16 and t19 for SOL 600

7. It is not presently possible to only create a t19 file with selected element/grid results using SOL 600.
If a t19 file is desired, a t16 file must also be created.

MT16SPL

Split a Marc t16 file into one or more smaller t16 files in SOL 600

Determines how to split a Marc t16 file into one or more smaller t16 files in SOL 600. Splitting of a large t16 file is sometimes necessary if the postprocessor aborts due to the large amount of data or if the results need to be transferred to another computer for postprocessing.

Format: This entry is available in small field format only:

1	2	3	4	5	6	7	8	9	10
MT16SPL	IOPT	NVECT	FNAME						
	ID1	START	LAST	INCR	(remark 4)				
	ID2	START	LAST	INCR	LastFew				

Examples:

MT16SPL	0								
	1	5	100	5	3				

MT16SPL	0		old1.mar	c.t16					
	1	10	80	10	3				
	2	90	100	5	4				

MT16SPL	1	1							
	1	100							

MT16SPL	3	2	old2.t16						
	1	10	100	10	3				

Create several new t16 files with 2 increments each using original increments 10, 20, ... 98, 99, 100. This is a restart job and uses old2.t16 (from a standalone Marc run). New files will be created as follows:

jid.0001.t16 with old increments 10, 20
 jid.0002.t16 with old increments 30, 40
 jid.0003.t16 with old increments 50, 60
 jid.0004.t16 with old increments 70, 80
 jid.0005.t16 with old increments 90, 98
 jid.0006.t16 with old increments 99, 100

Descriptor	Meaning
IOPT	Option of how to split up the “old” t16 file. (Integer; Default = 0)
0	Split into as many new t16 files as there are continuation lines of this entry, use the current jid.marc.t16 file, FNAME should not be entered.
1	Split into as many new t16 files with NVECT original increments each, use the current jid.marc.t16 file, FNAME should not be entered.

Descriptor	Meaning
2	Split into as many new t16 files as there are continuation lines of this entry, use a previously generated t16 file whose name is specified by FNAME.
3	Split into as many new t16 files with NVECT original increments each, use a previously generated t16 file whose name is specified by FNAME.
NVECT	Number of increments to be placed on each new t16 file (only used if IOPT is 1 or 3) (Integer; no Default)
FNAME	Original t16 filename - Only used if IOPT is 2 or 3. (Character; no Default) The filename may extend from fields 4-9. If it is more than 8 characters long, this entry must made in fixed format. the entire t16 filename including the t16 extension should be entered. No upper case letters may be used. (Integer > 0)
IDi	New t16 plot ID (must start with 1 and increase by 1 on each continuation line.) (Integer; no Default)
START	Starting Marc increment to be placed on new t16 file. (Integer; Default = 1)
LAST	Last Marc increment in range of start-last-incr to be n new t16 file. (Integer; Default is last increment on original t16 file if FNAME is blank. If FNAME is not blank, LAST must be an accurate value which can be obtained from the .sts file of the original run that produced FNAME.)
INCR	Increments to be used for start-last-incr tgo b e on new t16 file. (Integer; Default = 1)
Last Few	In addition to START,LAST,INCR the last several increments may be placed on the t16 file. In the first example increments 5, 10, 15, ..., 100, 99, 98 are placed on the new t16 file. LastFew may only be entered on the last line. (Integer; Default = 2)

Remarks:

1. Sometimes large SOL 600 models do not converge on a user wishes to examine output at unknown time intervals. To determine what is happening, it is frequently necessary to plot the results at several output intervals. In fact, sometimes the last increment may have bogus results due to divergence. At present, MSC GUI programs sometimes are not capable of postprocessing the large amount of data one would like to include in a single f16 file. This option allows you to break up the t16 file into one or more smaller files.
2. GUI's might only be able to handle one increment per t16 file for extremely large models. To specify this, only enter Id and START as in the 3rd example.
3. All t16 files will have the geometry as well as the specified output increments.
4. LastFew may only be entered on the last line.
5. Nastran may be restarted to perform this step. To do so, enter
SOL 600, ID t16split=fname
Where fname is the jid of the original job.
6. The new files will be named jid.ID.t16. Examples are as follows:

Case 1 - Split up t16 as part of current run starting with jid1.dat as the Nastran input SOL 600 will create jid1.marc.t16 with a full set of output increments (unless reduced by you).

jid1.0001.t16
jid1.0002.t16
etc.

Case 2 - Split up a t16 file formed by a previous SOL 600 run named jid1.dat (the t16 file is named jid1.marc.t16). The current Nastran input file to split up the original t16 is named jid2.dat. The new t16 files will be designated:

jid2.00001.t16
jid2.00002.t16
etc.

7. Nastran will spawn Marc's pldump2000 program to split up the original t16 file.
8. If FNAME is entered (IOPT=2 or 3), OUTR options (on SOL600,ID) will be ignored. If FNAME is blank and IOPT=0 or 1 and OUTR options are requested, the t16op2 translator process the full (unsplit) jjid.mar.t15 file. Future implementations may allow processing of the split t16 files.
9. Only one MT16SPL (plus many continuation lines as necessary) is allowed per job.

MTABRV

Defines a List of Tables to Modify and/or Positions of Values - SOL 600

In some cases, a model is built with tables defined backwards. For example, for a stress-strain curve the compression position of the curve might really need to be the tension portion and visa versa. This entry allows them to be reversed. In addition, this entry allows a user to add a point to the lower and/or upper end of the tables in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MTABRV	ID1	THRU	ID2	IREV	XLOW	YLOW	XHIGH	YHIGH	

Example:

MTABRV	1	THRU	10	1	-100.	-1.0E7	0.01	1.0E7	
	21	THRU	25	1					
	45	THRU	48				25.5	66523.	

Descriptor	Meaning				
ID1	Starting table ID to be reversed. (Integer; Required; no Default)				
THRU	Enter the character string THRU if more than one grid is desired.				
ID2	Ending table ID of the range. (Integer; or blank. If blank; ID2=ID1)				
IREV	Option to reverse (flip) positive and negative values and change the sign for tables in the range ID1 to ID2. (Integer; Default = 0) <table border="0" style="margin-left: 20px;"> <tr> <td>0</td> <td>Do not reverse</td> </tr> <tr> <td>1</td> <td>Reverse</td> </tr> </table>	0	Do not reverse	1	Reverse
0	Do not reverse				
1	Reverse				
XLOW	Option to add a point to the lower end of these tables. If so, XNEG is the lower end X value to be added (after reversal, if IREV=1). (Real; Default = 0.0)				
YLOW	Option to add a point to the lower end of these tables. If so, YNEG is the lower end Y value to be added (after reversal, if IREV=1). (Real; Default = 0.0)				
XHIGH	Option to add a point to the upper end of these tables. If so, XPOS is the upper end X value to be added (after reversal, if IREV=1). (Real; Default = 0.0)				
YHIGH	Option to add a point to the upper end of these tables. If so, YPOS is the upper end Y value to be added (after reversal, if IREV=1). (Real; Default = 0.0)				

Remarks:

- Missing tables in the range ID1 to ID2 will be ignored and no error will be produced.
- This capability is limited to tables defined using TABLES1, TABLED1 and/or TABLEM1 and the behavior in the x and y directions must be linear.

3. Values XLOW, YLOS, XHIGH, YHIGH are not reversed and the signs are not changed even if IREV=1.
4. The first example reverses tables 1-10 and also adds a point at the lower and upper end. The second example reverses tables 21-25, no additional points are added. The third example adds a point to the upper end of tables 45-48, does not reverse the tables or add a point to the lower end.
5. This entry will reverse both X and Y of the specified tables and is not capable of reversing only X or Y.

MTCREEP**AUTO THERM CREEP Iteration Control - SOL 600**

Controls a transient thermal creep analysis. This entry or the MACREEP entry is required if ITYPE is not zero on the MPCREEP entry in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MTCREEP	ID	Tchg	Nmax	Iasmb	Ttot	Tincr	Tcrep	Nsub	
	Maxit	Nupd	VV1	VV2	VV3	IABS			

Example:

MTCREEP	11	1.0	99999	1	2000.	10.	1000.	50	
	5	1	.5	.1	.05				

Descriptor	Meaning	
ID	Identification number of a matching NLPARM Case Control command (for statics) or entry (for dynamics). (Integer; no Default; Required field)	
Tchg	Maximum temperature change to be used per step of the stress analysis. (Real; no Default; Required value) (2,1)	
Nmax	Maximum number of increments allowed. (Integer; Default = 999999) (2,2)	
Iasmb	Reassembly interval for element matrices. (Integer; Default = 1) (2,3)	
Ttot	Total transient time from corresponding heat transfer analysis. (Real; no Default) (2,4)	
Tincr	Suggested time increment for creep analysis (Real; no Default) (3,1)	
Tcrep	Total creep time to be covered in this creep analysis (Real; no Default) (3,2)	
Nsub	Maximum number of subincrements to be allowed during this creep analysis (Integer; Default = 50). (3,3)	
Maxit	Maximum number of iterations allowed to modify the time step during any increment. (Integer; Default = 5) (3,4)	
Nupd	Number of increments between stiffness matrix updates. (Integer; Default = 1) (3,5)	
VV1	Tolerance value #1. (Real; see below for defaults) (4,1)	
	IABS=0	Enter the tolerance on the creep strain increment relative to the elastic strain. (Default = 0.5)
	IABS=1	Enter the maximum creep strain increment. (Default = 0.01)
VV2	Tolerance value #2 (Real; see below for defaults) (4,2)	
	IABS=0	Enter the tolerance on the stress change per increment divided by the total stress. (Default = 0.1)
	IABS=1	Enter the maximum stress increment. (Default = 100.0)

Descriptor	Meaning
VV3	Tolerance on low stress point cutoff. Points with a stress lower than this ratio relative to the maximum stress in the structure are not used in the creep tolerance checking. (Real; Default = 0.05) (4,3)
IABS	Flag controlling relative or absolute convergence testing. (Integer; Default = 0) (4,5)
0	Relative checking is used
1	Absolute checking is used

Remarks:

1. This entry maps to Marc's AUTO THERM CREEP entry.
2. This entry will be used instead of AUTO STEP or AUTO INCREMENT entries in the Marc file. It is suggested that if this entry is used, NLAUTO and NLSTRAT should not be specified (and will be ignored if entered).
3. Bulk Data entries, MPCREEP and MCHSTAT, must also be entered in addition to this entry.
4. (i,j) refer to Marc's AUTO THERM CREEP (data block, field).
5. Bulk Data entries MACREEP and MTCREEP should not be entered in the same input file.

MTHERM**Iteration Control for Automatic Thermal Loading for Structural Analysis Following a SOL 600 Heat Transfer Analysis**

Iteration control for automatic thermal loading for structural analysis following a SOL 600 heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
MTHERM	ID	Tchg	Nmax	Iasmb	Ttot	Tincr			

Example:

MTHERM	21	1.0	99999	1	2000.	10.			
--------	----	-----	-------	---	-------	-----	--	--	--

Descriptor	Meaning
ID	Identification number of a matching Case Control NLPARM command (for statics) or entry (for dynamics). (Integer; no Default; Required field)
Tchg	Maximum temperature change to be used per step of the stress analysis. (Real; no Default; Required value) (2,1)
Nmax	Maximum number of increments allowed. (Integer; Default = 999999) (2,2)
Iasmb	Reassembly interval for element matrices. (Integer; Default = 1) (2,3)
Ttot	Total transient time from corresponding heat transfer analysis. (Real; no Default) (2,4)
Tincr	Suggested time increment for creep analysis. (Real; Default = 0.0; leave blank if analysis is not a creep analysis) (2,5)

Remarks:

1. This entry maps to Marc's AUTO THERM entry.
2. This entry will be used instead of AUTO STEP or AUTO INCREMENT entries in the Marc file. It is suggested that if this entry is used, NLAUTO and NLSTRAT should not be specified (and will be ignored if entered).
3. (i,j) refer to Marc's AUTO THERM (data block, field).
4. This entry may be omitted in which case an AUTO THERM entry as follows will be used
 AUTO THERM
 1., 9999, 0, 1.0

NHRMPRM

Parameters for Nonlinear frequency response analysis (SOL 128)

Format:

1	2	3	4	5	6	7	8	9	10
NHRMPRM	PARAM1	VAL1	PARAM2	VAL2					

Example:

NHRMPRM	MNCSF	3	NLHDIAG	1					
---------	-------	---	---------	---	--	--	--	--	--

Var Name	Type/ Default	Description
NLHRED	I = -1	Use reduction for nonlinear analysis.
NHPLUS	I = 20	Number of extra points to avoid aliasing.
NLHTOL	R = 1.0E-5	Tolerance for convergence.
NLHTWK	R = 1.1	Push-off factor for line search in case convergence fails.
MXICODE0	I = 5	If solution fails to converge for mxicode0 steps, new trial displacement is used.
TICPNCH	I = 0	If TICPNCH = -1 then write a TIC Bulk Data entry for each degree-of-freedom in the d-set to be included in a subsequent transient analysis run.
MNCSF	I = 1000	Max number of non-converged sequential frequencies for RESET.
MFRINT	R = 10.	Maximum frequency interval for RESET.
NLHDIAG	I = 0	Output additional diagnostic messages.
CONTP	I = 0	Continuation procedure.

Remarks:

- Following parameters can also be specified using PARAM option: NLHRED, NHPLUS, NLHTOL, NLHTWK, TICPNCH and MXICODE0.
See description under PARAM for more details about these variables.
In case a parameter is specified in NHRMPRM and also using PARAM, the value specified using PARAM entry is used.
- Parameters MNCSF or MFRINT can be used to reset initial condition in case of non-convergence.
If both these parameters are specified, the condition which is satisfied first is used.
- CONTP =1 is used to initiate continuation procedure method. In this case, following parameters are not used in the analysis: NLHRED, MNCSF, MFRINT. Currently, the CONTP option is available for frequency-independent analysis only.
Here, NLHRED is set to 0 to not use the reduction method. And initial condition for the continuation procedure is determined using the results from two previously converged solutions.

NLADAPT

Additional Parameters for Automatic Load or Time Stepping

Defines additional parameters for automatic load or time stepping used with enhanced nonlinear in SOL 400. *NLADAPT is an obsolete option from MD Nastran 2010 onwards. It is recommended that NLADAPT be replaced by NLSTEP.*

Format:

1	2	3	4	5	6	7	8	9	10
NLADAPT	ID								
	“STEP”	RSMALL	RBIG	TSMIN	TSMAX	NSMAX	NRECYC	SFACT	
		IDAMP	DAMP						
	“CREEP”	RAC	TCSTRN	TCSTRC	TCOFF				

Example:

NLADAPT	700								
	STEP								
		4							
	CREEP	1							

NLADAPT	800								
	CREEP	1							
	STEP								
		4							

Descriptor **Meaning**

ID	Identification number of a NLPARM or entry. (Integer > 0)
“STEP”	Keyword indicating the following entries are for enhanced general load step or time step convergence. (Character)
RSMALL	Smallest scale factor for time step changes. See Remark 5. (Real; Default= 0.1)
RBIG	Largest scale factor for time step changes. See Remark 5. (Real; Default = 1.5)
TSMIN	Smallest ratio of a time step to the total time. (Real; Default = total time divided by number of time steps)
TSMAX	Largest ratio of a time step to the total time. (Real; Default = 0.5)
NSMAX	Maximum number of steps allowed. (Integer; Default = 99999)
NRECYC	Number of recycles per increment. (Integer; Default = 10)
SFACT	Scale factor for time step changes. See Remark 6. (Real; Default = 1.2)
IDAMP	Enter 4 to activate artificial damping. (Integer 0 or 4; Default = 0)
DAMP	Damping factor for activated artificial damping. (Real; Default = 2.E-4)

“CREEP”	Keyword indicating the following entries are for enhanced creep analysis. (Character)	
RAC	Flag controlling relative or absolute convergence. (Integer 0 or 1; Default = 0)	
0	Relative checking used.	
1	Absolute checking used.	
TCSTRN	Creep strain tolerance. (Real)	
RAC = 0	Enter the tolerance on the creep strain increment divided by the elastic strain. (Real; Default = 0.5)	
RAC = 1	The maximum creep strain increment allowed. (Real; Default = 0.1)	
TCSTRS	Creep stress tolerance (Real)	
RAC = 0	Enter the tolerance on the stress increment divided by the total stress. (Real; Default = 0.1)	
RAC = 1	enter the maximum stress increment. (Real; Default = 100.0)	
TCOFF	Tolerance on low stress point cutoff. Points with a stress lower than this ratio relative to the maximum stress in the structure are not used in the creep tolerance checking. (Real; Default = 0.05)	

Remarks:

1. This entry must point to an existing NLPARM or entry and is only used in SOL 400.
2. The keywords and their associated entries may occur in any order.
3. This entry computes an initial time step TINIT=1.0/NINC if pointing to a NLPARM entry and TINT=1.0/DT if pointing to a entry.
4. If this entry is used, it is preferred KMETHOD on the NLPARM or METHOD on the be set to PFNT, FNT, ITER, or AUTO.
5. The scale factor is defined as the new step size divided by the time step size.
6. SFACT is only used when artificial damping is activated.

NLAUTO

Parameters for Automatic or Fixed Load/Time Stepping - SOL 600

Defines parameters for automatic or fixed load/time stepping used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
NLAUTO	ID	TINIT	TFINAL	RSMALL	RBIG	TSMIN	TSMAX	NSMAX	
	NRECYC	IENHAN	IDAMP	NSTATE	NCUT	LIMTAR	IFINISH	FTEMP	
	SFACT	IFLAG	IDTAB	DAMP	IDMPFLG		IPHYS	I313	
	CRITERIA	SETID	Y1	X1	Y2	X2	Y3	X3	
		Y4	X4						

Example:

NLAUTO	1	.01	1	1	10	-1.5	.5		
	5		1			-1			
	1.2								

Descriptor	Meaning
ID	Identification number referenced by the NLPARM, , or TSTEP Case Control command for the applicable subcase. See Remarks 1. and 2. Include a NLPARM or entry for the subcase in addition to the NLAUTO entry. See Remarks 1. and 2. (Integer > 0; no Default)
TINIT (2,1)	Initial time step. (Real; Default is determined by NLPARM NINC or DT, see Remark 8.)
TFINAL (2,2)	Total time period. If OUTR options are used, for a static analysis, TFINAL must be 1.0. (Real; Default is 1.0 or Remark 8.)
RSMALL (2,3)	Smallest ratio between steps. (Real; Default = 0.1 or Remark 7.)
RBIG (2,4)	Largest ratio between steps. (Real; Default = 10.0 or Remark 7.)
TSMIN (2,5)	Minimum time step. (Real; Default is total time divided by number of time steps or Remark 7.)
TSMAX (2,6)	Maximum time step. (Real; Default is total time or Remark 7.)
NSMAX (2,7)	Maximum number of steps allowed. If IENHAN=2, NSMAX is the number of fixed time steps. (Integer; Default = 999999 or Remark 7.)

Descriptor	Meaning
NRECYC (2,8)	Desired number of recycles per increment. For beam or shell bending NRECYC should be set to 10. If large deformation is anticipated, NRECYC = NO is necessary for plate and shell buckling. PARAM,MARCD,0 can be used to set NRECYC=10 without entering NLAUTO. (Integer; Default = 5 or Remark 7.)
IENHAN (2,9)	Enter 1 to activate the enhanced scheme, 0 otherwise. Enter 2 to use fixed time stepping (enter the number of time steps using NSMAX) (Integer)
IDAMP (2,10)	Artificial damping control. (Integer; Default = 1, or Remark 7.) Enter 0 to not use artificial damping Enter 1 to use artificial damping if time step is less than minima time step. Reduce the minimum time step by 1000. IDAMP=1 is usually necessary (in combination with NRECYC=10) for plate and shell buckling models. Enter 2 to always use artificial damping. The value to use is given by DAMP which scales the damping matrix. Enter 4 to always use artificial damping which is determined by the strain energy in the first increment of the present load case. DAMP is used to scale the strain energy. Enter 5 to set the time step as if artificial damping was being used, but do not actually add artificial damping. Enter 6 to add artificial damping when the minimum time step has been reached.
NSTATE (3,1)	Number of states for post file. (Integer; enter only if ienhan = 1)
NCUT (3,2)	Maximum number of times to cut down time step in an increment. (Integer) (Default = 10 or Remark 7.; enter only if ienhan = 1)
LIMTAR (3,3)	Enter 0 to create criteria as limits, 1 to treat criteria as targets. (Integer, Default = 0 or Remark 7.; enter only if ienhan = 1)
IFINISH (3,4)	Enter 1 to finish time period when all nodal temperatures fall below FTEMP. Enter -1 if all nodal temperatures should exceed FTEMP. Enter 0 to omit temperature check. (Integer; enter only if ienhan = 1, Default = 0 or Remark 7.)
FTEMP (3,5)	Finish temperature, use with IFINISH. (Real; enter only if ienhan = 1, Default = 0.0 or Remark 7.)
SFACT (3,6)	Scale factor for time step changes other than changes due to user criteria. (Real) (Default = 1.2 or Remark 7.; enter only if ienhan = 1)
IFLAG (3,7)	Enter flag to override CREEP and DYNAMIC parameters as specified in the Marc input parameter section for this load case. (Integer) 0 Do not override parameters. 1 Turn off CREEP and DYNAMICS. 2 Turn off CREEP. 3 Turn off DYNAMICS.
IDTAB (3,8)	Table ID scaling damping factor (see next item) (Integer)

Descriptor	Meaning
DAMP (3,9)	Damping factor for artificial damping. The number entered here depends on the IDAMP option. If IDAMP is 1, the damping matrix is scaled by setting this factor to be the ratio of the initial damping energy to the initial strain energy (Defaults to 1e-5). If IDAMP is 2, the damping matrix is directly scaled by this factor. If IDAMP is 4, the estimated total damping energy in the subcase will be this factor times the estimated total strain energy. (Default value of 2.e-4 or Remark 7.)
IDMPFLG (3,10)	Enter 1 to put states reached by the above IDMPFLG flag on the post file. (Integer)
IPHYS (3,12)	Flag to determine if automatic physical criteria should be added and how analysis should proceed if they are not satisfied. (Integer) 2 Do not add automatic physical criteria. Stop when any user criteria are not satisfied. 1 Add automatic physical criteria. Stop when any user criteria are not satisfied. -1 Add automatic physical criteria. Continue when any user criteria are not satisfied. -2 Do not add automatic physical criteria. Stop when any user criteria are not satisfied.
I313 (3,13)	Flag to check if dynamic integration error checks should be made while determining the timestep (single step Humbolt and Newmark-Beta only) (Integer, Default = 0 or Remark 7.). 0 Skip error check. 1 Include error check.
CRITERIA (4,1)	Enter an integer corresponding to the criteria desired. See Remark 1. (Integer; no Default)
SETID (4,2)	Case Control Set ID of nodes or elements for which this criteria will apply. Restriction: Must be one of first 25 sets entered in the case control. Leave blank if "ALL" is desired. (Integer; Default is all or Remark 7.)
Y1 (4,3)	Time step adjustment value. See Remark 10. (Real)
X1 (4,4)	Time step adjustment value. See Remark 10. (Real)
Y2 (4,5)	Time step adjustment value. See Remark 10. (Real)
X2 (4,6)	Time step adjustment value. See Remark 10. (Real)
Y3 (4,7)	Time step adjustment value. See Remark 10. (Real)
X3 (4,8)	Time step adjustment value. See Remark 10. (Real)

Descriptor	Meaning
Y4 (4,9)	Time step adjustment value. See Remark 10. (Real)
X4 (4,10)	Time step adjustment value. See Remark 10. (Real)

Remarks:

- Enter the following index in the CRITERIA field (a limit of 9 criteria may be specified and the usual option is to specify none).

1	Strain Increment	2	Plastic Strain Increment
3	Creep Strain Increment	4	Normalized Creep Strain Increment
5	Stress Increment	7	Strain Energy Increment
8	Temperature Increment	9	Displacement Increment
10	Rotation Increment	12	Stress
13*100+n	State Variable n		

- Values entered on NLAUTO override values with the same meaning if entered elsewhere (for example, on the NLPARM or entry).
- If the NLAUTO entry is used, there should also be a corresponding NLPARM or . The matching NLPARM entry must have KMETHOD=AUTO or ITER. If is the matching entry, then field 6 must be blank or have the value ADAPT.
- Values such as (3,7) indicated corresponding item on Marc's AUTOSTEP data block 3 field 7.
- Items (3,7) to (3,13) were implemented starting with MSC Nastran 2004.0.4 and are not in previous versions.
- The continuation lines may be omitted if not needed. If one of the continuation lines is needed, all proceeding continuation lines must be entered and at least one value per line is specified (no blank lines are allowed).
- If an NLAUTO field is blank for the second and following subcases, the value will be assumed to be the same as that of the proceeding subcase for the same field. If this is not the behavior that is desired, be sure not to leave fields blank that should vary between the current and previous subcases.
- Dynamics TINIT and TFINAL are determined from and TABLED1 entries. They are ignored if entered using NLAUTO. For statics, TINIT and TFINAL, if specified on NLAUTO override the values from NLPARAM.
- Fixed time stepping is triggered by setting NSMAX equal to the number of steps (N) and IENHAN=2. It may also be triggered by using bulk data PARAM,MARCITER,N.
- The time step is adjusted based upon:

```

if   X<X1      Y(calculated)/Y1
if   X1<X<X2    Y(calculated)/Y2
if   X2<X<X3    Y(calculated)/Y3
if   X3<X<X4    Y(calculated)/Y4

```

where

Criterion	X	Y
1	Strain	Strain increment
2	Plastic strain	Plastic strain increment
3	Creep strain	Creep strain increment
4	Creep strain	Creep strain increment/Elastic strain
5	Stress	Stress increment
7	Strain energy	Strain energy increment
8	Temperature	Temperature increment
9	Displacement	Displacement increment
10	Rotation	Rotation increment
12	Stress	Stress increment/Stress
13*100+n	State variable n	Increment in state variable n

NLBSH3D

Nonlinear 3-dimensional rotor loading

Defines connectivity of a three-dimensional nonlinear load as a function of relative deflection and velocity of a grid pair which in turn is a function of rotor speed.

Format:

1	2	3	4	5	6	7	8	9	10
NLBSH3D	SID	GA	GB	Blank	LID-1	LID-2			

Descriptor	Meaning
SID	Nonlinear load set identification number. (Integer > 0; Required)
GA	Inner grid. (Integer > 0; Required)
GB	Outer grid. (Integer > 0; Required)
LID-1	Load vs deflection group ID of a LBSH3DG (Integer > 0; Required)
LID-2	Load vs velocity group ID of a LBSH3DG (Integer > 0; Required)

Remarks:

1. NLBSH3D are selected with the Case Control command NONLINEAR = SID.
2. In order to ensure correct results, GRID points GA and GB must satisfy the following three conditions:
 - a. They must both be grid points
 - b. They must be coincident
 - c. They must have parallel displacement coordinate systems
 The program checks for the above conditions. If they are not satisfied, the program will terminate the run with an appropriate fatal message.
3. Multiple NLBSH3D may reference the same SID but require different GA, GB grid pairs.
4. Multiple NLBSH3D entries may refer to the same LID independent of SID.
5. Non-linear load is calculated as a function of relative deflection and velocity between GRID points GA and GB and as function of rotor speed. Current non-linear load = load (deflection (rotor speed)). Used with LBSH3DG entries to define a nonlinear load.
6. GA and GB must have all 6 DOF in D-set. Both grids should not be dependent on any RBE or MPC and must not be constrained by an SPC. The program checks for active DOFs of both the grids. If they are not satisfied, the program will terminate the run with an appropriate fatal message.

NLCYSYM

Defines Information to Perform Nonlinear Cyclic Symmetry Analysis using SOL 600

A limited capability is available to analyze structures with a geometry and a loading varying periodically about a symmetry axis. This capability is termed Nonlinear Cyclic Symmetry. This is done by developing a special set of tying constraints on the boundaries of the “pie-sliced” segment being analyzed. For more details on the type of structures that may be analyzed using this method and its restrictions, please see the Remarks.

Format:

1	2	3	4	5	6	7	8	9	10
NLCYSYM	ID	DC1	DC2	DC3	X	Y			
	ANG	TOL	IRB	LinFlag					

Examples:

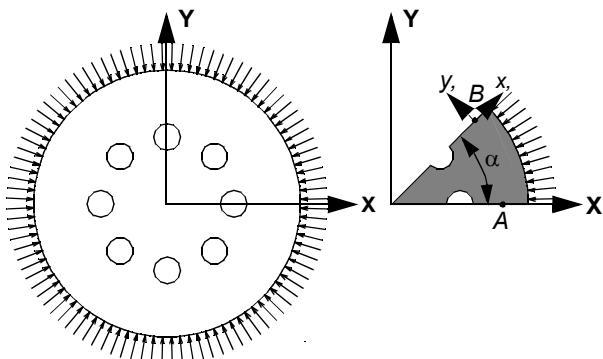
NLCYSYM	0.0	0.0	1.0	0.0	0.0	0.0			
	30.0		-1	-1					

Descriptor	Meaning
ID	Identification number not presently used. (Integer or blank, Default = 0)
DC1	First component of the direction cosine of the symmetry axis. (Real; Default = 0.0)
DC2	Second component of the direction cosine of the symmetry axis. (Real; Default = 0.0)
DC3	Third component of the direction cosine of the symmetry axis. (Real; Default = 0.0)
X	X coordinate of a point on the symmetry axis. (Real; Default = 0.0)
Y	Y coordinate of a point on the symmetry axis. (Real; Default = 0.0)
Z	Z coordinate of a point on the symmetry axis. (Real; Default = 0.0)
ANG	Cyclic angle in degrees. (Real; no Default)
TOL	Cyclic symmetry tolerance. (Real; Default = 0.5 times the minimum element size)
IRB	Rigid body mode suppression flag. (Integer; Default = -1) <ul style="list-style-type: none"> -1 Automatically suppress rigid body mode. 0 Do not suppress the rigid body mode. N Suppress rigid body mode at grid point N. (N > 0)
LinFlag	Linearization flag to be used if a cyclic symmetric structure consists of quadratic elements. (Integer; Default = 1)

Descriptor	Meaning
1	The outer boundary of the structure is described based on the corner nodes only. Multipoint constraints due to cyclic symmetry are not assigned to midside nodes. Instead, they are linearly tied to the corresponding corner nodes.
-1	The outer boundary of the structure is described using a quadratic field. Due to cyclic symmetry, full quadratic multipoint constraints are set up; they are assigned both to corner and midside nodes.

Remarks:

1. A full and “pie-shaped” structure to be analyzed using nonlinear cyclic symmetry shown below:



Looking at points and on this segment, the displacement vectors should fulfill:

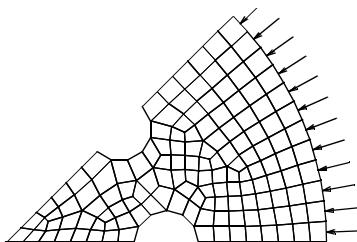
$$u'_B = u_A$$

which can also be written as:

$$u_B = R u_A$$

where the transformation matrix R depends on the symmetry axis (which, in the example above, coincides with the global Z-axis) and the sector angle. The input for the nonlinear cyclic symmetry option consists of the direction vector of the symmetry axis, a point on the symmetry axis and the sector angle. The following items should be noted:

- a. The meshes do not need to line up on both sides of a sector (for example, see the following figure)



- b. Any shape of the sector sides is allowed provided that upon rotating the sector $360/\alpha$ times about the symmetry axis over the sector angle will result in the complete model.
- c. The nonlinear cyclic symmetry option can be combined with standard contact. In this case, both sides of the cyclic symmetry sectors need to belong to the same contact body.
- d. The nonlinear cyclic symmetry option can be combined with global remeshing (not currently available).
- e. In a coupled thermo-mechanical analysis, the temperature is forced to be cyclic symmetric (TA=TB) (not currently available).
- f. A nodal point on the symmetry axis is automatically constrained in the plane perpendicular to the symmetry axis.
- g. Possible rigid body motion about the symmetry axis can be automatically suppressed.
- h. Cyclic Symmetry is valid for:
 - Primarily for continuum elements. However, the presence of beams and shells is allowed, but there is not connection of shells to shells across the symmetry plane, so the shell part can, for example, be a turbine blade and the volume part is a turbine rotor. The blade is connected to the rotor and if there are 20 blades, 1/20 of the rotor is modeled plus one complete blade.
 - Nonlinear static analysis including remeshing as well as coupled analysis
 - Valid for all analysis involving contact
 - Valid also for: eigenvalue analysis such as buckling or modal extraction, and linear transient dynamic analysis.
 - Cyclic Symmetry is invalid for pure heat transfer.
 - The contact status can be viewed. Grids tied across the symmetry plane will have a value of 2.

To prevent confusion, it must be emphasized that the cyclic symmetry feature described above is different than linear cyclic symmetry commonly used in modal analysis where physical quantities such as x_n , displacements, forces, stresses and temperature in the n-th segment are expanded in a Fourier series with terms of the cyclic components, u^k , in the fundamental region, like:

$$x_n = \frac{1}{N} u^0 + \sqrt{\frac{2}{N}} \sum_{k=1}^{K} [u^{k,c} \cos(n-1)k\alpha + u^{k,s} \sin(n-1)k\alpha] + \frac{(-1)^{n-1}}{\sqrt{N}} u^{N/2}$$

where k is the harmonic order; N is the total number of sectors; α is the fundamental inter-sector phase shift defined as $2\pi/N$; and K is defined as:

$$K = \begin{cases} \frac{N-1}{2} & \text{if } N \text{ is odd} \\ \frac{N-2}{2} & \text{if } N \text{ is even} \end{cases}$$

There are considerable savings in both computing time and data storage associated with the use of the linear cyclic symmetry concept. Assuming a finite element model with a sector size of m degrees of freedom, a real-valued cyclic symmetry approach leads, in the worst case, to one eigenvalue problem of size m and $(N-1)/2$ eigenvalue problems of size $2m$. A complex approach leads to N eigenvalue problems of size m ; while the full analysis leads to a single, but very costly, eigenvalue problem of size Nm .

Although linear cyclic symmetry can reduce the problem size greatly, it is restricted to linear analysis, and the sector must have its surface mesh on the symmetry planes to be identical on each side of the sector. The nonlinear cyclic symmetry implemented in SOL 600 can be used for nonlinear problems, such as contact, and the nodes do not need to line up on both symmetry planes of the sector.

NLDAMP**Damping Constants - SOL 600**

Defines damping constants for nonlinear analysis when Marc is executed from SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
NLDAMP	EID1	EID2	ALPHA	BETA	GAMMA				

Examples:

NLDAMP	1	2000	.025	4.5	1.0				
--------	---	------	------	-----	-----	--	--	--	--

Descriptor	Meaning
EID1	First element for which the damping values will be used. (Integer ≥ 0 ; Required)
EID2	Last element for which the damping values will be used. (Integer ≥ 0 or blank)
ALPHA	Mass Matrix Multiplier. (Real; Default = 0.0)
BETA	Stiffness Matrix Multiplier. (Real; Default = 0.0)
GAMMA	Numerical Damping Multiplier. (Real; Default = 0.0)

Remarks:

1. This entry matches Marc's Damping definition.
2. NLDAMP is recognized only when Marc is executed from SOL 600.

NLFREQ

Frequency List for Nonlinear Harmonic Response

Defines explicit forcing frequencies for nonlinear harmonic response.

Format:

1	2	3	4	5	6	7	8	9	10
NLFREQ	ID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	-etc.-						

Examples:

NLFREQ	17	3.	7.	21.	14.				
--------	----	----	----	-----	-----	--	--	--	--

Descriptor **Meaning**

ID Identification number referenced by the NLFREQ field (field 5) of an NLHARM Bulk Data entry. (Integer > 0)

F1...Fn Forcing frequency values in cycles per unit time. (Real ≥ 0.0)

Remarks:

1. The frequencies may be specified in any order; they do not have to be in ascending or descending order.
2. Only one NLFREQ or NLFREQ1 entry with the same ID is allowed.

NLFREQ1

Frequency List for Nonlinear Harmonic Response, Alternate Form 1

Defines a set of forcing frequencies for nonlinear harmonic response by specification of a starting frequency, frequency increment, and the number of increments/decrements desired.

Format:

1	2	3	4	5	6	7	8	9	10
NLFREQ1	ID	F1	DF	NDF					

Examples:

NLFREQ1	17	3.	2.	6					
---------	----	----	----	---	--	--	--	--	--

Descriptor	Meaning
ID	Identification number referenced by the NLREQ field (field 5) of an NLHARM Bulk Data entry. (Integer > 0).
F1	First forcing frequency in the set. (Real ≥ 0.0)
DF	Frequency increment. See Remark 1. (Real $<> 0.0$; Required)
NDF	Number of frequency increments/decrements. (Integer > 0; Default = 1)

Remarks:

1. If DF is positive, it defines a frequency increment. In this case, the first excitation frequency will be F1 with all subsequent excitations frequencies being of larger value than F1. If DF is negative, it defines a frequency decrement. In this case, the first excitation frequency will still be F1, but now all subsequent excitations frequencies will be of smaller value than F1. In both cases, the initial conditions for a frequency are the response configuration at the previous excitation frequency. The initial conditions for the first frequency are zero.
2. Only one NLREQ or NLFREQ1 entry with the same ID is allowed.

NLHARM

Parameters for Nonlinear Harmonic Response Control

Defines parameters for nonlinear harmonic response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
NLHARM	ID	SUBFAC	NHARM	NLFREQ					

Examples:

NLHARM	41		3	15					
--------	----	--	---	----	--	--	--	--	--

Descriptor	Meaning
ID	Identification number referenced by the NLHARM Case Control command. (Integer > 0)
SUBFAC	Factor for capturing sub-harmonic response. See Remark 3. (Integer ≥ 1 ; Default = 1)
NHARM	The number of harmonics to include in the solution. See Remark 2. (Integer > 0)
NLFREQ	Identification number of the NLFREQ or NLFREQ1 entry specifying the forcing frequency list. (Integer > 0)

Remarks:

1. The NLHARM ID must be unique among all NLHARM entries.
2. The greater the degree of nonlinearity, the larger the number of harmonics required to find a solution (if one exists). The number of harmonics (NHARM) used may influence the ability of the algorithm to find solutions to the nonlinear problem posed. However, the larger the number of harmonics, the more computational effort is required to find a solution.
3. The response frequencies in linear harmonic analysis are the same as the forcing frequency. Permanent oscillations whose frequencies are a fraction of the forcing frequency ($\frac{1}{2}, \frac{1}{4}, \dots$) may occur in a nonlinear system; these oscillations are known as sub-harmonic response.

NLHEATC

Defines Numerical Analysis Parameters for SOL 600 Heat Transfer Analysis

Format:

1	2	3	4	5	6	7	8	9	10
NLHEATC	ID	MLS	MRC	MMIN	NPOS	IASMB			
	TCHG	TEVAL	TERR						

Examples:

NLHEATC	9999	10	1	1	1				
	10.	999999.	20.						

Descriptor	Meaning
ID	Not presently used, leave blank.
MLS	Maximum number of load steps in the run. (Integer; Default = 4). [2,1]
MRC	Maximum number of recycles during an increment due to temperature dependent material properties. (Integer; Default = 3) [2,2]
MMIN	Minimum number of recycles during an increment. This value is forced even if convergence appears to occur in a fewer number of cycles. (Integer; Default = 1) [2,3]
NPOS	Nonpositive definite flag. (Integer; Default = 0) [2,7] <ul style="list-style-type: none"> 0 Solution of nonpositive definite matrices will fail. 1 Solution of nonpositive definite matrices is forced (computer time is greater if nonpositive matrices do not exist)
IASMB	Flag to assemble conductivity matrix. (Integer; Default = 0) [2,12] <ul style="list-style-type: none"> 0 Conductivity matrix is not assembled each iteration 1 Conductivity matrix is assembled each iteration
TCHG	Maximum nodal temperature change allowed. (Real; Default = 20.0) [3,1]
TEVAL	Maximum temperature change before properties are re-evaluated and matrices reassembled. (Real; Default = 100.0) [3,2]
TERR	Maximum error in temperature estimate used for property evaluation. It provides a recycling capability to improve accuracy for highly nonlinear heat transfer such as latent heat and radiation. (Real; Default = 0.0) [3,3]

Remarks:

- Only one NLHEATC entry should be entered.
- This entry maps to Marc's CONTROL history definition entry for heat transfer. [i,j] indicates the datablock and field of this Marc entry.

NLMOPTS**Nonlinear Multiple Options**

Specifies nonlinear material options and composite options. The nonlinear material options are for advanced materials used in SOL400. The composite options (TSHEAR and INLAM, CPROJ) are applicable to all solution sequences in which Layered Solid Elements (PCOMPLS) are available, namely, SOL600, SOL400, SOL200 (analysis only), and all linear solution sequences between SOL101 and SOL112.

Format:

1	2	3	4	5	6	7	8	9	10
NLMOPTS	"CREEP"	valc1	valc2	valc3	valc4				
	"ASSM"	vala							
	"TSHEAR"	vals							
	"LRGSTRN"	valle							
	"HEMICUBE"	Value	NPIXEL		CUTOFF	FRACTION	FACCN	FACTOL	
	"TEMPP"	valtd							
	"TEMGO"	vmaptg							
	"SPROPMAP"	PROPMAP	PROPBH	DIRECT	THICKOP	IPRINT			
	"SPCRMPT"	vramp							
	"DEACTEL"	vald1	vald2						
	"ENTHALP"	valclu	valen1						
	"MAPTOL"	vmptol							
	"INLAM"	vcoord	cproj						

Examples:

NLMOPTS	CREEP	0							
	HEMICUBE	1	500		0	0.01			

Descriptor	Meaning
“CREEP”	Keyword indicating that the formulation for creep analysis. (Character Default CREEP). Creep data should be entered using MATVP, not the CREEP bulk data entry. There are 2 kinds of creep models (a) Explicit Creep - ValC1 and ValC2 are used for this.
valc1	0 : Maxwell Model Creep (default) 1 : Explicit Viscoplastic Creep (only supported via user subroutine CRPLAW)
valc2	(Blank or 0) : (Default) 1: (Reserved)
	(b) Implicit Creep - ValC3 and ValC4 are used for this.
valc3	0 : Implicit Creep off 1 : Implicit Creep on
valc4	(Blank or 0) : elastic tangent (Default) 1: secant tangent 2: radial return
	Note that: (1) ValC3 = 0 means Explicit Creep (see ValC1). ValC3 = 1 means Implicit Creep. (2) ValC4 is only used in conjunction with Valc3=1. The recommended value for ValC4 is 1 (secant tangent).
“ASSM”	Keyword indicating that the item following applies to assumed strain. See Remark 3. (Character; Default = See Remark 3.)
vala	ASSUMED for assumed strain formulation OFF for no assumed strain formulation.
“TSHEAR”	Keyword indicating that the item following applies to a parabolic shear distribution through the shell thickness (or ply thickness). See Remark 4.
vals	TSHEAR for parabolic distribution. (Character; Default = blank)
“LRGSTRN”	Keyword indicating that the item following applies to a formulation for large strain. (Default; Integer = 0) -1 No large strain formulation. 0 Mean normal return. 1 Hypoelasticity and additive plasticity with mean normal return. 2 Hyperelasticity and multiplicative plasticity with radial return.
“HEMICUBE”	Keyword to select the view factor calculation method. See Remarks 2. and 7.

Descriptor	Meaning	
Field 3	VALUE=0 - Use Nastran finite difference, contour integration, or Gaussian integration method (Default).	=1 - (HEMI) Use pixel based modified hemicube method. This method is not available for axisymmetric view factor calculations.
Field 4	NPIXEL - Enter the number of pixels (Default = 500).	
Field 6	CUTOFF - Enter the fraction of the maximum view factor that is to be used as a cutoff. View factors calculated below this cutoff are ignored (Default = 0).	
Field 7	FRACTION - Enter the fraction of the maximum view factors that is to be treated implicitly. View factors values smaller than this cutoff are treated explicitly (Default = 1.0E-2).	
Field 8	FACCNT - Set 1 to activate explicit treatment of reflection matrix. Default = 0)	
Field 9	FACTOL - Tolerance to be used on the above iteration on the Poljak equations.	
“TEMPP”	Keyword indicating temperature distribution across the thickness of advanced heat shell elements. Internal invisible scalar points will be created to store these extra variables. LINE and QUAD options require shell elements to have full 3-D thermal material data. When the temperature results of these internal points are applied to calculating thermal loading effect in a separate mechanical analysis, this keyword must also be specified in the structural model (See Remarks 11., 12. and 13.).	
valtd	CONS	Constant distribution. (Default)
	LINE	Linear distribution. (Character)
	QUAD	Quadratic distribution.
“TEMGO”	Keyword indicating user request identified in .f06 file, the mapping of user supplied and internally generated grids for the linear and quadratic thermal shell elements. See Remark 8.	
vmaptg	NO	Do not show grid mapping. (Character; Default)
	YES	Show mapping.
“SPROPMAP”	Keyword controlling automated inclusion of nonlinear property options for SOL 400. See Remark 9.	
PROPMAP	(Integer ≥ 0 ; Default 0)	
	-1	No mapping considered
	0	Map to full integration elements associated with nonlinear property extension entries when necessary
	1	Map to reduced integration elements associated with nonlinear property extension entries when necessary

Descriptor	Meaning
	<p>2 Map to full integration elements associated with nonlinear property extension entries when possible</p> <p>3 Map to reduced integration elements associated with nonlinear property extension entries when possible</p> <p>Note: The nonlinear property extension entries refer to: PBARN1, PBEMN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 entries.</p> <p>To ensure that the automatic mapping is unique for an element ID, whenever advanced elements are detected / specified by the user, ELTPRT will be automatically called.</p>
PROPBEH	Property behavior flag. Valid values are 1D, 3D, PLST, PSTR, AXI. (Default = None for 1D, 3D. PLST for 2D; Character)
DIRECT	Layer direction flag. Valid values; 1, 2, 3. (Integer > 0; Default 1)
THICKOP	For 2D solids: out of plane thickness. (Real ≥ 0.0 ; Default 1.0)
IPRINT	Print flag for generated property flags.
	<p>0 Blank (Default)</p> <p>1 Print</p> <p>2 Punch</p> <p>3 Print and Punch</p>
“SPCRMPT”	Keyword indicating heat transfer SPC is to be applied instantaneously or by linear step. (Character Default SPCMPT)
vramp	<p>0 SPC applied instantaneously. (Integer or blank; Default = blank)</p> <p>1 SPC applied by linear step.</p>
“DEACTEL”	Keyword controlling default behavior and output diagnostics for element deactivation
vald1	Select default deactivation rules for each physics pass in the analysis. See Remark 9.
	<p>-1 Default deactivation rules are always ignored. (Default in single physics analysis)</p> <p>0 Default deactivation rules are always applied. (Default in coupled analysis)</p> <p>1 Default deactivation rules are applied in a mechanical pass only.</p> <p>2 Default deactivation rules are applied in a thermal pass only.</p>
vald2	Select diagnostic output behavior for active and deactivated elements.
	<p>0 A summary of all deactivated elements is printed. If no elements get deactivated output is skipped. (Default)</p> <p>1 A detailed list of all deactivated elements is printed. If no elements get deactivated output is skipped.</p> <p>2 A summary of all deactivated and all remaining active elements is printed.</p>

Descriptor	Meaning
3	A detailed list of all deactivated and all remaining active elements is printed.
"ENTHALP"	Keyword controlling the scheme used to form the capacitance matrix and the enthalpy vector for elements with nonlinear property extensions in transient heat transfer. See Remark 10.
valclu	Select option to form capacitance matrix for elements with nonlinear property extensions.
0	Use same scheme as that for linear elements without property extensions
1	Use consistent capacitance matrix
2	Use coupled capacitance matrix
3	Use lumped capacitance matrix
valen1	Select option to form enthalpy vector for elements with nonlinear property extensions
0	Form enthalpy using automatic scheme. (Default)
1	Form enthalpy explicitly through time integration for material behavior
2	Form enthalpy via multiplication of temperature with capacitance
"MAPTOL"	Keyword controlling the tolerance used to check if a structural node lies inside a thermal element.
vmptol	Value of MAPTOL. (Real > 0.0; Default = 0.2)
INLAM	(Character; Default Blank)
vcoord	Selects coordinate system for interlaminar stress output.
Blank	Using the original coordinate system for interlaminar stress output
BOTT	Using the ply coordinate system of bottom layer of the interlaminates
TOP	Using the ply coordinate system of above layer of the interlaminates
cproj	Activates projection system for projecting X-axis of CORDM onto layer plane. See remark 5.
OFF	Does not project (Default).
ON	Projection scheme is activated.

Remarks:

1. The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.

2. The material and property related keywords associated with the entry (CREEP, ASSM, TSHEAR, LRGSTRN, TEMPP and ENTHALP) are only applicable to elements associated with PAXISYM, PBARN1, PBEMN1, PRODN1, PSHLN1, PSHLN2, PSLDN1, PLCOMP, PCOMPLS, and PCOHE entries. Other keywords SPCRMPT, HEMICUBE, DEACTEL, MAPTOL) are more generally applicable. Keyword SPCRMPT applies to all SPC temperature boundary conditions, HEMICUBE applies to radiation boundary conditions and DEACTEL applies to all elements on the job.
3. The setting of ASSUMED STRAIN in SOL 400 is automatic for the following three elements with some conditions.
 - a. 4 nodes full integration Plane Stress (PLPLANE + PSHLN2 with PSTRS -- BEH4=PSTRS, INT4=L);
 - b. 4 nodes full integration Plane Strain (PLPLANE + PSHLN2 with PSTRN -- BEH4=PLSTRN, INT4=L); and
 - c. 8 nodes full integration HEXA (PSOLID + PSLDN1 -- BEH8=SOLID, INT8=L,)

The application conditions are summarized here.

OFF Assumed strain always turned off.

ASSUMED Assumed strain will be turned on manually for 3 element types (PSTRS, PSTRN, PSLDN1) under the following cases:

- a. Elastic / Elasto-Plastic materials using small strain formulation (i.e., PARAM,LGDISP,-1 / NLMOPTS,LRGSTRN,-1). This includes isotropic and non-isotropic materials.
- b. Elastic / Elasto-Plastic materials using large strain additive formulation (i.e., PARAM,LGDISP,1 / NLMOPTS,LRGSTRN,0 or 1). This includes isotropic and non-isotropic materials.

Assumed strain flag will be turned off for all materials using the Multiplicative Formulation (NLMOPTS,LRGSTRN,2).

Default Assumed Strain will be turned on automatically for 3 element types (PSTRS, PSTRN, PSLDN1) under the following cases:

- a. Isotropic Elastic materials using small strain formulation (i.e., PARAM,LGDISP,-1 / NLMOPTS,LRGSTRN,-1)
- b. Isotropic Elastic materials using large strain additive formulation (i.e., PARAM,LGDISP,1 / NLMOPTS,LRGSTRN,0 or 1)

Note that this turns on Total Lagrange Formulation currently by default for the isotropic elastic materials.

Assumed Strain will be turned off automatically for the following cases:

- a. Any material that uses Updated Lagrange in either Additive or Multiplicative Formulation
- b. Any non-isotropic material

4. TSHEAR allows a parabolic shear distribution for the BEH4=DCT, INT4=L or LRIH (shell) elements, and BEH8 or BEH20=SLCOMP, INTi=L, Q (Layered Solid) or ASTN (Layered Solid Shell) elements.
5. In Figure 9-114, plane ABCD is the layer plane of interest. Z_m is normal to the layer plane (or is the thickness direction). X_m is the projection of the X axis of the MCID coordinate system on the layer plane. $Z_m \times X_m$ gives the Y_m . Angle θ is measured counter-clockwise from X_m axis about the Z_m axis.

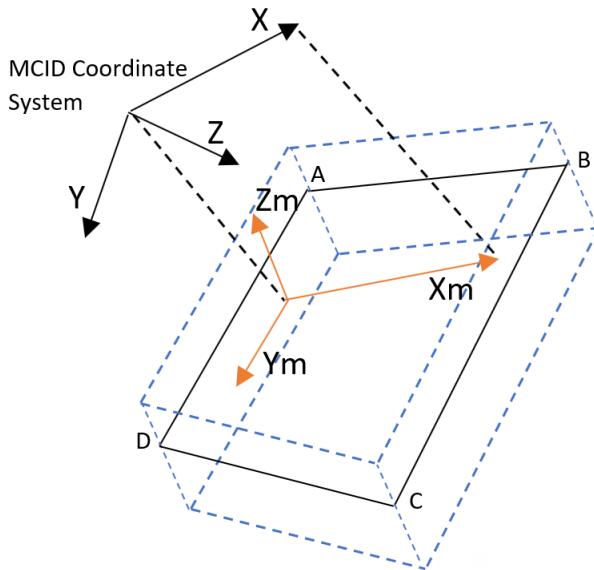


Figure 9-114 Projection Scheme

6. NLMOPTS,LRGs,valle can be used separately or in conjunction with the PARAM,LGDISP option. Two separate cases can be identified:
 - a. Only NLMOPTS,LRGs specified. In this case, valle = -1 and valle = 0 mean the same, i.e., no large strain flags are turned on for the applicable elements identified in remark 2. Valle = 1 flags the additive framework and valle = 2 flags the multiplicative framework.
 - b. Both NLMOPTS,LRGs and PARAM,LGDISP,N ($N > -1$) specified. In this case, valle = -1 supercedes the PARAM,LGDISP option and turns off all large displacement / large strain flags for the applicable elements identified in remark 2. Also, valle = 0 behaves the same way as valle = 1 (i.e., for the applicable elements, both options flag the additive framework).
7. Definition of the radiation exchange matrix as in the *MSC Nastran Thermal User's Guide*, Eq. 0-15.

$$R = \sigma [A\epsilon - A\alpha(A - F(I - \alpha))^{-1}F\epsilon]$$

in which the reflection matrix is:

$$[A - F(I - \alpha)]^{-1}$$

Note that the reflection term is costly, since it involves the factorization of a dense matrix.

For the SOL 400 Newton's method, the previous exchange matrix multiplied by a function of the temperature is added to the stiffness. If the exchange matrix is dense, which is generally the case, the sparse stiffness matrix consequently also becomes dense, and the factorization of the stiffness matrix becomes much more expensive.

The input options are as follows. All options with the exception of the "faccnt" option, are available in SOLs 153, 159, and 400. The "faccnt" option is only available in SOL 400:

hemi, view, npixel, ndiv, cutoff, fraction, faccnt, factol

where

view	1 to flag hemicube method. Default is 0. The following options are available only if view = 1.
npixel	Number of pixels per quarter section, where total number of pixels = $(2 \cdot \text{npixel})^2$. Default is 500.
ndiv	Not used, but originally intended for axisymmetry. Reserve this one for the future to do adaptive mesh refinement, with ndiv defined as the maximum allowable number of subdivisions.
cutoff	Factor below which viewfactors will be set to zero. This one is applied during the last viewfactor calculation. It results in a less dense exchange matrix and therefore a faster calculation of the reflection term and of the stiffness matrix factorization. Default is 0.0.
fraction	Factor below which radiation exchange matrix terms are not added to the stiffness matrix. For each equation in the radiation exchange matrix, the fraction is multiplied by the diagonal term, and all terms in the equation less than this value is not added to the stiffness matrix. This procedure does not affect the density of the radiation exchange matrix or the cost of calculating it, including the reflection term, but reduces the density of the stiffness matrix and the cost of factorizing the stiffness matrix. Note that the full radiation exchange matrix still is used to calculate fluxes. Default is 0.01.
faccnt	Set to 1 to activate explicit treatment of reflection matrix. Default is 0. In this procedure, the radiation exchange matrix is constructed to be: $R = \sigma[A\varepsilon - A\alpha F\varepsilon]$

Note that the expensive reflection term is absent. This reduced form of the radiation exchange matrix is added to the stiffness matrix. Since the reflection matrix never is calculated and factorized, the calculation of the radiation exchange matrix is significantly cheaper. Since this reduced radiation exchange matrix is less dense, the factorization of the stiffness matrix also is significantly faster.

For the flux calculation, an iterative procedure is used based on iterating towards a solution simultaneously satisfying both Poljak equations in the *MSC Nastran Thermal User's Guide*, Eqs. 6-11 and 6-12, respectively.

$$A\{q\}_e^{IN} = [F]\{q\}_e^{OUT}$$

$$\{q\}_e^{OUT} = \sigma[\varepsilon]\{u_e\}^4 + [I - \varepsilon]\{q\}_e^{IN}$$

The value of “faccnt” also can be set to the maximum allowable number of iterations to be used in this procedure. If the value is set to 1, the maximum allowable number is internally set to 100. This procedure currently is available only in SOL 400 and is not available for wavelength-dependent emissivities.

The “cutoff” and “fraction” parameters can be specified concurrently with this option.

factol tolerance to be used on the previous iterative procedure on the Poljak equations.

8. TEMGO,YES results in the following type of print.

USER GRID POINT(TOP) MAPPING TO BOT/MID INTERNAL GRID										
TOP	1	2	3	4	5	6	7	8	9	10
BOT	101000001	101000002	101000003	101000004	101000005	101000006	101000007	101000008	101000009	101000010
MID	0	0	0	0	0	0	0	0	0	0
...										

9. The “SPROPMAP” keyword provides a convenient option to automatically flag secondary properties like PBARN1, PBEMN1, PRODN1, PSHEARN, PSHLN1, PSHLN2, and PSLDN1. Note that these secondary property entries expose the user to a set of sophisticated 2-D continuum and 3-D beam, shell and continuum elements in SOL 400. The rules governing the generation of the additional properties are many and are a function of the problem dimension, material type, and procedure. All these rules have been incorporated into the automatic generation option. These are briefly summarized in the table below for the default SPROPMAP = 0 case:

Secondary Property	Primary Property	Dimension	Material	Notes	Unsupported Features of Primary Entry
PBARN1	PBARL	1-D	MAT4	Note 4	
			MATS1	Note 1	
			MATEP		
			MATF		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	

Secondary Property	Primary Property	Dimension	Material	Notes	Unsupported Features of Primary Entry
PBEMN1	PBEAML	1-D	MAT4	Note 4	Tapered Sections
			MATS1	Note 1	
			MATEP		
			MATF		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	
PRODN1	PROD	1-D	MAT4	Note 4	J,C
			MATS1	Note 1	
			MATEP		
			MATF		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	
PSHEARN	PSHEAR	3-D	MAT4	Note 4	F1,F2
			MAT8		
			MATS1	Note 1	
			MATS8	Note 1	
			MATEP		
			MATF		
			MATORT		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	
PSHLN1	PSHELL	3-D	MAT4	Note 4	TS/T, nondefault Z1 and Z2, $121/T^3$
			MAT5	Note 4	
			MATS1	Note 1	
			MATS8	Note 1	
			MATEP		
			MATF		
			MATORT		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	

Secondary Property	Primary Property	Dimension	Material	Notes	Unsupported Features of Primary Entry
PSHLN1	PCOMP/ PCOMPG	3-D	MAT4	Note 4	FT, GE, LAM options other than BLANK and SYM, SOUTi
			MAT5	Note 4	
			MAT8		
			MATS1	Note 1	
			MATS2	Note 1	
			MATS8	Note 1	
			MATEP		
			MATF		
			MATORT		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	
PSHLN2	PLPLANE	2-D	MAT4	Note 4	
			MAT5	Note 4	
			MATG	Note 5	
			MATS1	Note 1	
			MATS3	Note 1	
			MATS8	Note 1	
			MATEP		
			MATF		
			MATORT		
			MATSMA	Note 6	
			MATVE		
			MATVP	Note 7	
PSLDN1	PSOLID	3-D	MAT4	Note 4	IN, ISOP, FCTN
			MAT5	Note 4	
			MATS1	Note 1	
			MATEP		
			MATF		
			MATORT		
			MATSMA		
			MATVE		
			MATVP		

Note:	1. Only when NLMOPTS, LRGSTRN, 1 is flagged or HGENPLAS is provided NLSTEP entry for coupled analysis.
	2. For shape memory materials, define through MATSMA only the thermo-mechanical model is available.
	3. For creep, define through MATVP, VALC=0 must be set on NLMOPTS for explicit formulation.
	4. Only when phase changes are flagged.
	5. Gasket materials for BEH4=COMPS or AXCOMP, INT4=L.
	6. For shape memory materials, define through MATSMA and BEH=PLSTRS only the thermo-mechanical model is available.
	7. For creep, define through MATVP and BEH=PLSTRS VALC=0 must be set on NLMOPTS for explicit formulation.
	8. The “SPROPMAP” does not support MATDIGI.

In the previous table, the Secondary Property (Column 1) is automatically flagged depending on the Primary Property (Column 2) and the Primary/Secondary Materials (Column 4). The Comments column (Column 5) indicates any special conditions that are followed by the program while doing this mapping. The Unsupported Features column (Column 6) indicates the list of options on the Primary Property entry that are not supported by the Secondary Property.

It should be noted that the Secondary Property additions offer a wide range of element formulations that primarily cater to nonlinear extension but can also serve for linear applications. By default, the PROPMAP field is set to 0 and maps to the full integration elements of the secondary property only if necessary. For a less memory intensive element, the PROPMAP field can be manually set to 1 and this then maps to the equivalent reduced integration element of the secondary property only if available and necessary. PROPMAP field can be manually set to 2 or 3 when secondary properties need to be tagged to all elements in the model. In this case, the special conditions given in the 5th column above are ignored and the mapping is carried out for both linear and nonlinear materials (if allowed). PROPMAP = 2 maps to full integration elements whenever possible and PROPMAP = 3 maps to reduced integration elements whenever possible.

- a. Property mapping is considered for all possible elements (i.e., PROPMAP = 0/1 is automatically converted to PROPMAP = 2/3) for the following conditions:
 - When IDAMP (damping scheme specified through the NLSTEP entry) > 0 is used for static analysis. Note that the damping scheme is only available for elements with nonlinear property extensions.
 - When CRITTID (user criteria specified through the NLSTEP entry) and the associated TABSCTL entry refer to displacement / stress / strain criteria for all elements in the model. Note that the user criteria are only supported for elements with nonlinear property extensions.

- When a small displacement analysis is flagged for thermo-mechanical coupled analysis specified through SUBSTEP or for real-time thermo-mechanical chained analysis specified through TEMP(LOAD,HSUB,HSTEP,HTIME). Note that for real-time temperatures to be transferred from the heat pass to the structural pass, all elements need to be processed in the nonlinear program flow and this is accomplished by flagging all possible elements with nonlinear property extensions.
- b. Property mapping is provided for elements that are associated with special procedures / flags:
 - Elements whose grids are specified on the VCCT Bulk Data entry. VCCT is only supported for grids of elements with nonlinear property extensions.
 - Elements that are specified on the IPSTRN / ISTRESS commands. These are only supported for elements with nonlinear property extensions.

It should be noted also that while the NLMOPTS,SPROPMAP option is powerful and allows a user-friendly way to map to advanced elements, it does not provide a substitute for all cases. Defaults are provided for the most common cases on the NLMOPTS,SPROPMAP entry itself. This includes PROPBEN - especially useful to distinguish between plane strain and plane stress, DIRECT - the layer thickness direction for gaskets, THICKOP - the out-of-plane thickness for the planar case. It should be noted that these defaults are global defaults and apply to all the elements for which the secondary properties are added. If such defaults are not applicable (for e.g., different out-of-plane thicknesses), it is the user's responsibility to add individual secondary property entries.

IPRINT can be used to get additional information on the secondary properties that have been added. Default is 0 in which case, there is no print-out. IPRINT = 1 allows the print-out in the .f06 file, IPRINT = 2 allows the print-out in the .pch file, IPRINT = 3 allows print-out in both the .f06 and .pch files.

Example 1: 8 noded CHEXA, PSOLID, MAT1	
SPROPMAP	Added Secondary Property
-1	No mapping considered
0	No mapping done - not necessary
1	No mapping done - not necessary
2	PSLDN1: C8, BEH8 = SOLID, INT8 = L, BEH8H = SOLID, INT8H = L
3	PSLDN1: C8, BEH8 = SOLID, INT8 = LRIH, BEH8H = SOLID, INT8H = LRIH

Example 2: 4 noded CQUAD4, PSHELL, MATF

SPROPMAP	Added Secondary Property
-1	No mapping (note that MATF will not work in this case)
0	PSHLN1: C4, BEH4 = DCT, INT4 = L, BEH4H = DCT, INT4H = L
1	PSHLN1: C4, BEH4 = DCT, INT4 = LRIH, BEH4H = DCT, INT4H = L
2	PSHLN1: C8, BEH8 = DCT, INT4 = L, BEH4H = DCT, INT4H = L
3	PSHLN1: C4, BEH4 = DCT, INT4 = LRIH, BEH4H = DCT, INT4H = L

9. The **vald1** value in “DEACTEL” determines in which physics pass the default deactivation rules are applied. If **vald1** is positive the deactivation rules are applied in the specified pass only and in all other passes the defaults are switched off, meaning that in those passes all elements remain active unless a DEACTEL Bulk Data input deactivates some of them.

The default deactivation rules are summarized in the following table:

Element Types	By Default Deactivated in Pass
Connector elements CWELD, CFAST, CSEAM	Thermal
Rigid body elements Rxxx	Thermal
CONMi and CMASSi elements	Thermal
Interface elements CIFxxx	Thermal
Surface elements CHBDYx	Mechanical
Heat convection elements CONVx	Mechanical
Radiation boundary elements RADBC	Mechanical
CDAMP4 elements originating from CMB or CMS fields in BCBODY input	Mechanical

The default for elements of all other types is that they are active in every pass.

10. The “ENTHALP” keyword is applicable to elements associated with the nonlinear property extensions PRODN1, PSHEARN, PSHLN1, PSLDN1 and for axisymmetric heat transfer shells specified through PAXISYM. It provides different options to form the capacitance matrix and enthalpy vector for these elements through **valclu** and **valenl** respectively.
- The **valclu** value allows the choice of 3 different capacitance formulations: consistent, coupled (average of consistent and lumped) and lumped. The default value of **valclu** = 0 allows the nonlinear elements to follow the same scheme as that for linear elements; lumped for rods and 3D shells, coupled for axisymmetric shells and linear hexes, pentas, tetras and consistent for quadratic hexes, pentas and tetras. Values of **valclu** > 0 allow the user to pick specific capacitance formulations for all elements.

The **valenl** value allows the user the choice of 2 different enthalpy vector formulations: time integration of the enthalpy at each integration point or product of the nodal temperature with the capacitance. The default value of **valenl = 0** allows the program to pick automatically: use the time integration option for nonlinear specific heat / latent heat, use the nodal product option for linear specific heat. Values of **valenl > 0** allow the user to pick the approach. Note that for nonlinear problems, the time integration approach is more accurate.

11. To compute the thermal loads of 3D thermal shells in a separate mechanical run, a consistent temperature distribution type must be specified in mechanical models, i.e. LINE must be defined if the thermal model has a linear temperature distribution using LINE, or QUAD must be specified for the thermal model with a quadratic temperature distribution defined by QUAD. If LINE or QUAD is specified, the program will switch the structural shell elements to advanced elements automatically by generating PSHLN1 entries internally if no PSHLN1 or PSHLN2 entries are specified in the original model.
12. MSC NASTRAN thermal analysis does not differentiate between top and bottom faces while applying loads or boundary conditions on shell, even though the top and bottom temperatures may be different through specifying TEMPP = LINE or QUAD.
13. TEMPP keyword is supported in coupled analysis or in separate runs of thermal and mechanical analyses. This keyword is not supported in chained thermal and mechanical analyses.

NLOUT

Selects Additional Nonlinear Output Quantities as Referenced By NLSTRESS Case Control Command

Selects additional nonlinear output quantities as referenced by NLSTRESS Case Control Command in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
NLOUT	ID	AIO1	AIO2	AIO3	AIO4	AIO5	AIO6	AIO7	
	AIO8	AIO9	etc.						
	ESV	SV2	SV3	etc.					

Example:

NLOUT	1	TOTTEMP							
-------	---	---------	--	--	--	--	--	--	--

NLOUT	1	TOTTEMP							
	ESV	SV2	SV3						

Descriptor **Meaning**

- | | |
|------|---|
| ID | Identification number of a NLOUT entry. Selected by the NLOUT keyword on the NLSTRESS Case Control command. (Integer > 0) |
| AIOi | Additional Output request. See Remark 3. (Character) |
| ESV | Keyword indicating the following are user state variables, it is optional. |
| SVi | State variable names selected for output. |

Remarks:

1. The keywords can appear in any order.
2. Any item selected that is inappropriate for the element will be ignored.
3. The request codes, meaningful only to elements who refer to PRODN1, PBARN1, PBEMN1, PSHLN1, PSHLN2, PSLDN1, PLCOMP, PCOMLS, or PCOHE entries, are as follows:

Additional Output Code Keywords	
Keyword	Description
CCASTRSS	Components of Cauchy Stress
CTOTSTRN	Components of Total Strain
CELASTRN	Components of Elastic Strain
CPLASTRN	Components of Plastic Strain

Additional Output Code Keywords	
Keyword	Description
CCRPSTRN	Components of Creep Strain
CTHMSTRN	Components of Thermal Strain
TSTRNPS	Thickness Strain for Plane Stress
MAJESTRN	Major Engineering Strain
MINESTRN	Minor Engineering Strain
CURVOL	Current Volume
ORGVOL	Original Volume
TOTTEMP	Total Temperature
INCTEMP	Incremental Temperature
EQVMSTRS	Equivalent von Mises Stress
EQSTRSA	Equivalent Stress/Yield Stress Ratio
EQELSTRN	Equivalent Elastic Strain
EQPLSTRN	Equivalent Plastic Strain
EQCRSTRN	Equivalent Creep Strain
TTSTRNED	Total Strain Energy Density
ELSTRNED	Elastic Strain Energy Density
PLSTRNED	Plastic Strain Energy Density
PLSTRNRT	Plastic Strain Rate
ILNMSTRS	Interlaminar Normal Stress. See Remark 5.
ILSHSTRS	Interlaminar Shear Stress. See Remark 5.
ILSHTKCX	X-component of the interlaminar shear stress for thick composite shells.
ILSHTKCY	Y-component of the interlaminar shear stress for thick composite shells
CSTRSCRD	Components of Stress Preferred System
GSKTCLST	Gasket Pressure
GSKTCLSR	Gasket Closure
PGSKTCLS	Plastic Gasket Closure
FAILIDX	Failure Index (%)
TOTVSV1	Total Value of First State Variable
TOTVSV2	Total Value of Second State Variable
TOTVSV3	Total Value of Third State Variable

Additional Output Code Keywords	
Keyword	Description
EQPHSTRN	Equivalent phase transformation strain
EQTWSTRN	Equivalent TWIN strain
EQTPSTRN	Equivalent TRIP strain 75
CPHSTRN	Phase transformation strain tensor
VOLFMART	Volume fraction of Martensite

4. User state variable name following the ESV keyword can be the default nominal name or user defined name in UDSESV.
5. ILNMSTRS and ILSHSTRS are stress tensors formed in the basic coordinate system, their principal directions are the directions of stress vector of interlaminar normal and shear stresses respectively. More precisely, the tensor components σ_{ij} satisfy below equation:

$$(\sigma_{ij} - \delta_{ij}\sigma)n_j = 0$$

where;

n_j is the stress vector direction,

σ is the normal of stress vector, and

$\delta_{ij} = 0$ when $i \neq j$ and $\delta_{ij} = 1$ when $i = j$.

NLOUTUD**User Defined Output Requests For Nonlinear Explicit Analysis.**

User defined output requests for elements or Lagrangian grid points. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
NLOUTUD	NID	GROUP	UNAME	DTOUT	STEPS	OUTNAME			
+	EL1/G1	EL2/G2	Eli/Gli						

Example:

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE myuds 'SCA.MDSolver.Obj.Uds.Dytran.InitOut'
```

In Bulk Data:

NLOUTUD	4	MYUDS	EEXOUT	0.01		OUTPUTR1			
+	1	2	3	4					

Field	Contents
NID	Unique output number. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	Name User subroutine name associated with the entry. (Character; default=EEXOUT) EEXOUT: Output for elements GEXOUT: Output for Lagrangian grid points
DTOUT	Output time interval. (Real > 0.0; default=blank)
STEPS	Output step interval. (Integer > 0; default=blank)
OUTNAME	Name for output request. (Character; required.)
Eli	Element id for user output. (Integer > 0; required)
Gi	Grid point id for user output. (Integer > 0; required)

Remarks:

1. Only can be used for SOL 700.
2. UNAME can be EEXOUT or GEXOUT.
3. Only one of DTOUT and STEPS is available.

4. When UNAME is set to EEXOUT, element id's are required for user output and when UNAME is set to GEXOUT, grid point id's are required for user output.

5. The file names that will be generated are:

{jobname}_{OUTNAME}_{cycle number}.[THS,ARC]

NLTERM

Parameters for Nonlinear Static Analysis Control

Defines a set of parameters for nonlinear static analysis iteration strategy. NLSTEP is the preferred option for SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
NLTERM	ID	NINC	DT	KMETHOD	KSTEP	MAXITER	CONV	INTOUT	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS	LSTOL	
	MAXBIS				MAXR		RTOLB	MINITER	

Example:

NLTERM	15	5		ITER					
--------	----	---	--	------	--	--	--	--	--

Descriptor	Meaning
ID	Identification number. (Integer > 0)
NINC	Number of increments. See Remark 16. (Integer > 0)
DT	Incremental time interval for creep analysis. See Remark 3. (Real ≥ 0.0 ; Default = 0.0 for no creep.)
KMETHOD	Method for controlling stiffness updates. See Remark 4. (Character = "AUTO", "ITER", "SEMI", "FNT", or "PFNT"; Default = "AUTO" for SOL 106, "AUTO" for SOL 400 with non-contact analysis, and "FNT" for SOL 400 with contact analysis.)
KSTEP	Number of iterations before the stiffness update for ITER method. See Remarks 5. For the FNT and PFNT usage of KSTEP, please see Remark 19. (Integer ≥ -1 ; Default = 5 for SOL 106 and 10 for SOL 400)
MAXITER	Limit on number of iterations for each load increment. See Remark 6. (Integer $\neq 0$; Default = 25)
CONV	Flags to select convergence criteria. See Remarks 7., 21., and 22. (Character = "U", "P", "W", "V", "N", "A" or any combination; Default = "PW". (See Remark 4 for additional default comment.)
INTOUT	Intermediate output flag. See Remark 8. (Character = "YES", "NO", "ALL" or Integer > 0 for SOL 400 only; Default = NO)
EPSU	Error tolerance for displacement (U) criterion. See Remarks 4., 16., 17. and 20. (Real > 0.0 ; Default = 1.0E-2)
EPSP	Error tolerance for load (P) criterion. See Remarks 4., 16. and 17. (Real > 0.0 ; Default = 1.0E-2)
EPSW	Error tolerance for work (W) criterion. See Remarks 4., 16., 17. and 20. (Real > 0.0 ; Default = 1.0E-2)

Descriptor	Meaning
MAXDIV	Limit on probable divergence conditions per iteration before the solution is assumed to diverge. See Remark 9. (Integer $\neq 0$; Default = 3)
MAXQN	Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark 10. (Integer ≥ 0 ; Default = MAXITER for all methods except PFNT. For PFNT, Default = 0)
MAXLS	Maximum number of line searches allowed for each iteration. See Remark 11. (Integer ≥ 0 ; Default = 4 for all methods except PFNT. For PFNT, Default = 0)
FSTRESS	Fraction of effective stress ($\bar{\sigma}$) used to limit the subincrement size in the material routines. See Remark 12. (0.0 < Real < 1.0; Default = 0.2)
LSTOL	Line search tolerance. See Remark 12. (0.01 < Real < 0.9; Default = 0.5)
MAXBIS	Maximum number of bisections allowed for each load increment. See Remark 13. (-10 \leq MAXBIS \leq 10; Default = 5 except for MAXITER < 0; Default = 0 if MAXITER < 0)
MAXR	Maximum ratio for the adjusted arc-length increment relative to the initial value. See Remark 14. (1.0 \leq MAXR \leq 40.0; Default = 20.0)
RTOLB	Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 15. (Real > 2.0; Default = 20.0)
MINITER	Minimum number of iterations for each increment, SOL 101 with contact and SOL 400 only. (Integer > 0 ; Default = 1; In contact analysis, Default = 2) When high accuracy is required, it is also recommended to set MINITER = 2.

Remarks:

1. The NLPARM entry is selected by the Case Control command NLPARM = ID. Each solution subcase requires an NLPARM command.
2. In cases of static analysis (DT = 0.0) using Newton methods, NINC is the number of equal subdivisions of the load change defined for the subcase. Applied loads, gravity loads, temperature sets, enforced displacements, etc., define the new loading conditions. The differences from the previous case are divided by NINC to define the incremental values. In cases of static analysis (DT = 0.0) using arc-length methods, NINC is used to determine the initial arc-length for the subcase, and the number of load subdivisions will not be equal to NINC. In cases of creep analysis (DT > 0.0), NINC is the number of time step increments.
3. For creep analysis, the unit of DT must be consistent with the unit used on the CREEP entry that defines the creep characteristics. Total creep time for the subcase or step is DT multiplied by the value in the field NINC; i.e., DT*NINC. For SOL 400 with advanced nonlinear elements (elements with PSLDN1, PSHLN1, etc.) for creep analysis, if the total time does not equal 1.0, then the NLSTEP Bulk Data entry must be used. For SOL 600 DT is ignored except for creep analyses.

4. The stiffness update strategy is selected in the KMETHOD field.
 - If the AUTO option is selected, the program automatically selects the most efficient strategy based on convergence rates. At each step the number of iterations required to converge is estimated. Stiffness is updated, if (i) estimated number of iterations to converge exceeds MAXITER, (ii) estimated time required for convergence with current stiffness exceeds the estimated time required for convergence with updated stiffness, and (iii) solution diverges. See Remarks 9. and 13. for diverging solutions.
 - If the SEMI option is selected, the program for each load increment (i) performs a single iteration based upon the new load, (ii) updates the stiffness matrix, and (iii) resumes the normal AUTO option.
 - If the ITER option is selected, the program updates the stiffness matrix at every KSTEP iterations and on convergence if $KSTEP \leq MAXITER$. However, if $KSTEP > MAXITER$, stiffness matrix is never updated. Note that the modified Newton-Raphson iteration method is obtained by selecting the ITER option and $KSTEP = MAXITER$. The "U" convergence test is not used if ITER-1 is selected.
 - If the FNT option is selected, the program will use the full Newton iteration method for which the stiffness matrix will be updated at every iteration. FNT option is available for SOL 400 only. For SOL 106, please use "KMETHOD=ITER and KSTEP=1" instead. In comparison with the PFNT method, the defaults for FNT are EPSU=0.01, EPSW=0.01 and MAXLS=4. See Remark 19.
 - If the PFNT option is selected, the program will use the Pure Full Newton iteration method. The PFNT method is the same as the FNT method except that the defaults for PFNT method are internally set as EPSU=-0.01, EPSW=-0.01, and MAXLS=0. The PFNT method is available for SOL 400 only. See Remark 19.
5. For AUTO and SEMI options, the stiffness matrix is updated on convergence if KSTEP is less than the number of iterations that were required for convergence with the current stiffness.
6. The number of iterations for a load increment is limited to MAXITER. If the solution does not converge in MAXITER iterations, the load increment is bisected and the analysis is repeated. If the load increment cannot be bisected (i.e., MAXBIS is attained or $MAXBIS = 0$) and MAXDIV is positive, the best attainable solution is computed and the analysis is continued to the next load increment. One best solution is computed for SOL 106 and 4 best solutions are computed for SOL 400. The analysis is terminated if the solution still diverges. If MAXDIV is negative, the analysis is terminated immediately.

For SOL 400 only, MAXITER can be negative. If MAXITER is negative, the solution is continued to the end of the current step, even if the solution is divergent. In this case, the best attainable solution is computed for each load increment. The default for MAXBIS = 0, if $MAXITER < 0$. Also, for SOL 400, the value of MAXITER for the AUTO method is an approximation. The program will try to obtain a converged solution if it senses the solution can converge.
7. The test flags (U = displacement error, P = load equilibrium error, W = work error, V = vector component method, N = length method, and A = auto switch) and the tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All the requested criteria (combination of U, P, W, V and/or N) are satisfied upon convergence. For SOL 400, if the U criterion is selected together with P or W,

then for the first iteration of a load increment, the U criterion will not be checked. For SOL 400 if CONV = ‘blank’ the code will use a default of “UPW” if no contact analysis and “PV” if a contact analysis is performed. See the *MSC Nastran Handbook for Nonlinear Analysis* for more details on convergence criteria. For V and N, see Remark 21. For A, see Remark 22.

8. INTOUT controls the output requests for displacements, element forces and stresses, etc. YES or ALL must be specified in order to be able to perform a subsequent restart from the middle of a subcase.

INTOUT	Output Processed
YES	For every computed load increment.
NO	For the last load of the subcase or step.
ALL	For every computed and user-specified load increment.

- For the Newton family of iteration methods (i.e., when no NLPCI command is specified), the option ALL is equivalent to option YES since the computed load increment is always equal to the user-specified load increment.
 - For SOL 400 only, if the adaptive time stepping scheme is used (i.e., when a NLAUTO Bulk Data entry with the same ID is specified), INTOUT is allowed to be Integer > 0. In this case, the load step is divided into INTOUT increments for output. For example, if INTOUT=5, the output will be at load increments 0.2, 0.4, 0.6, 0.8, and 1.0. Please note that INTOUT defines only the output load increments, which are different from the analysis increments. The analysis load increment size is smaller than or equal to the output load increment size.
 - For arc-length methods (i.e., when the NLPCI command is specified) the computed load increment in general is not going to be equal to the user-specified load increment, and is not known in advance. The option ALL allows the user to obtain solutions at the desired intermediate load increments.
 - For SOL 600 only, the default is YES (see Remark 18.)
9. The ratio of energy errors before and after the iteration is defined as divergence rate (E^i), i.e.,

$$E^i = \frac{\{\Delta u^i\}^T \{R^i\}}{\{\Delta u^i\}^T \{R^{i-1}\}}$$

Depending on the divergence rate, the number of diverging iteration (NDIV) is incremented as follows:

If $E^i \geq 1$ or $E^i < -10^{12}$, then $NDIV = NDIV + 2$

If $-10^{12} < E^i < -1$, then $NDIV = NDIV + 1$

The solution is assumed to diverge when $NDIV \geq |\text{MAXDIV}|$. If the solution diverges and the load increment cannot be further bisected (i.e., MAXBIS is attained or MAXBIS is zero), the stiffness is updated based on the previous iteration and the analysis is continued. If the solution diverges again in the same load increment while MAXDIV is positive, the best solution is computed and the analysis is continued to the next load increment. If MAXDIV is negative, the analysis is terminated on the second divergence.

10. The BFGS update is performed if $\text{MAXQN} > 0$. As many as MAXQN quasi-Newton vectors can be accumulated. The BFGS update with these QN vectors provides a secant modulus in the search direction. If MAXQN is reached, no additional ON vectors will be accumulated. Accumulated QN vectors are purged when the stiffness is updated and the accumulation is resumed.
11. The line search is performed as required, if $\text{MAXLS} > 0$. In the line search, the displacement increment is scaled to minimize the energy error. The line search is not performed if the absolute value of the relative energy error is less than the value specified in LSTOL.
12. The number of subincrements in the material routines (elastoplastic and creep) is determined so that the subincrement size is approximately $\text{FSTRESS} \cdot \bar{\sigma}$ (equivalent stress). FSTRESS is also used to establish a tolerance for error correction in the elastoplastic material; i.e.,

$$\text{error in yield function} < \text{FSTRESS} \cdot \bar{\sigma}$$

If the limit is exceeded at the converging state, the program will exit with a fatal message. Otherwise, the stress state is adjusted to the current yield surface.
13. The number of bisections for a load increment/arc-length is limited to the absolute value of MAXBIS. Different actions are taken when the solution diverges depending on the sign of MAXBIS. If MAXBIS is positive, the stiffness is updated on the first divergence, and the load is bisected on the second divergence. If MAXBIS is negative, the load is bisected every time the solution diverges until the limit on bisection is reached. If the solution does not converge after $|\text{MAXBIS}|$ bisections, the analysis is continued or terminated depending on the sign of MAXDIV. See Remark 9.
14. MAXR is used in the adaptive load increment/arc-length method to define the overall upper and lower bounds on the load increment/arc-length in the subcase; i.e.,

$$\frac{1}{\text{MAXR}} \leq \frac{\Delta l_n}{\Delta l_o} \leq \text{MAXR}$$

where Δl_n is the arc-length at step n and Δl_o is the original arc-length. The arc-length method for load increments is selected by an NLPCI Bulk Data entry. This entry must have the same ID as the NLPARM Bulk Data entry.

15. The bisection is activated if the incremental rotation for any degree-of-freedom ($\Delta\theta_x$, $\Delta\theta_y$, or $\Delta\theta_z$) exceeds the value specified by RTOLB. This bisection strategy is based on the incremental rotation and controlled by MAXBIS.
16. The default for NINC is 10, except if there is a GAP, Line Contact, Heat Transfer or PARAM,NLTOL,0, in which case the default is 1. Default tolerance sets are determined based on model type and desired accuracy. Accuracy is under user control and can be specified on the PARAM, NLTOL entry. For SOL 106, NLTOL's value is used only if the CONV, EPSU, EPSP and EPSW

fields are blank, and if NINC is set to a value of 10 or larger. Otherwise, the NLTERM selection will be overridden. The overridden values are CONV=PW, EPSP=1.0E-3, and EPSW=1.0E-7. For SOL 400, NLTERM's value is used if the value of CONV, EPSU, EPSP, or EPSW fields are blank. The tables below list tolerances according to NLTERM selections:

Table 25 Default Tolerances for Static Nonlinear SOL 106 Models Without Gaps, Contact or Heat Transfer

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
1	High	PW	_____	1.0E-2	1.0E-3
2	Engineering	PW	_____	1.0E-2	1.0E-2
3	Prelim Design	PW	_____	1.0E-1	1.0E-1
None	Engineering	PW	_____	1.0E-2	1.0E-2

Table 26 Default Tolerances for Static Nonlinear SOL 106 Models With Gaps or Contact (Enter NLTOL Values of 0 or 2 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
2	Engineering	PW	_____	1.0E-3	1.0E-5
None	Engineering	PW	_____	1.0E-3	1.0E-5

Table 27 Default Tolerances for Static Nonlinear SOL 106 or 153 Models With Heat Transfer (Enter NLTOL Value of 0 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
None	Very high	PW	_____	1.0E-3	1.0E-7

Table 28 Default Tolerances for Static Nonlinear SOL 400 Models Without Gaps, Contact or Heat Transfer

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	-	1.0E-3	1.0E-3
1	High	PW	-	1.0E-2	1.0E-3
2	Engineering	PW	-	1.0E-2	1.0E-2
3	Prelim Design	PW	-	1.0E-1	1.0E-1
None	Engineering	PW	-	1.0E-2	1.0E-2

Table 29 Default Tolerances for Static Nonlinear SOL 400 Models With Gaps or Contact (Enter NLTOL Values of 0 or 2 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	-	1.0E-3	1.0E-3
2	Engineering	PW	-	1.0E-3	1.0E-3
None	Engineering	PW	-	1.0E-3	1.0E-3

17. The method to compute the energy (work) error is different for SOL 106 and SOL 400. For SOL 106, the energy error is computed based on the residue forces. While, for SOL 400, the energy error computed is the total energy error, which is based on the nonlinear forces acting on the structure. At the start of the iteration, these two methods give approximately the same value. However, near convergence, the SOL 106 method will field a much smaller value than that provided by the SOL 400 method. The difference in these two methods is reflected in the default values shown in Remark 16. The reason for a new method used in SOL 400 is that it gives the true error of the physical energy. On the other hand, the error computed in SOL 106 has no counter part in the physical world.
18. For SOL 600, the only fields used are ID, NINC, DT (creep only), KMETHOD and INTOUT, however, PARAM,MARCOTIM is recommended instead of INTOUT. For other fields, advanced convergence controls are available using NLAUTO, NLSTRAT and PARAM,MARCDATA Bulk Data entries. For SOL 600, if INTOUT is specified all NLPARM's in the file must use the same values. The first INTOUT encountered is what is actually used. The default for INTOUT is YES. For SOL 600, the initial time step for each subcase is 1/NINC of the NLPARM applicable to that subcase. If TINIT or the NLAUTO entry is entered it overrides 1/NINC as the initial time step. For arc length methods NLPCI with the same ID as NLPARM must be entered and if AIFRACT or the NLSTRAT entry is entered it will override 1/NINC as the initial increment size. Beware that NLSTRAT entries, if used, must be entered for each subcase as well as for "subcase zero". The ID of NLSTRAT do not correspond to the NLPARM Id or to the subcase ID but are numbered sequentially starting with zero for Marc increment zero, one for the first subcase (regardless of its ID) etc. For KMETHOD only, strings AUTO and ITER are supported. If any other string is entered it will be assumed to be the same as AUTO in SOL 600.
19. For FNT and PFNT methods, whether the stiffness matrix will be updated between the convergence of a load increment and the start of the next load increment depends on the value of KSTEP. In this case, KSTEP = -1, 'BLANK', or 1. A user fatal error will be issued if other value is input. If KSTEP = 1, the stiffness matrix will not be updated. If KSTEP = 'BLANK', the program will decide whether to update depending element type. If KSTEP = -1, the stiffness matrix will be forced to be updated.
20. If EPSU > 0.0, the displacement error is computed with respect to the total displacements. If EPSU < 0.0, the displacement error is computed with respect to the delta displacements of a load increment. If EPSW > 0.0, the energy error is computed with respect to the total energy. If EPSW < 0.0, the energy error is computed with respect to the delta energy of a load increment. The options EPSU < 0.0 and EPSW < 0.0 are available for SOL 400 only.

21. V and N are additional methods for convergence checking using the displacement (U) and/or load (P) criteria. V stands for vector component checking. In this method, convergence checking is performed on the maximum vector component of all components in the model. N stands for length checking. In this method, the length of a vector at a grid point is first computed by the SRSS (square root of the sum of the squares) method. Then convergence checking is performed on the maximum length of all grid points in the model. For example, if CONV=UV, then V checking method will be performed with the U criteria, i.e., the maximum displacement component of all displacement components in the model is used for convergence checking. For V and N, the EPSU is always negative, i.e., the error is computed with respect to the delta displacements of a load increment, even if positive value is requested by users. CONV=V is the same as CONV=UPV. If both V and N are specified; V will take precedence over N. For example, CONV=VN is the same as CONV=V.
- By default, for UPV or UPN, separate checks are made over force and moment vectors, and translation and rotation vectors. While the force/translation check is valid always, the moment or rotation check is only valid for 6 DOF elements (beams, shells, etc.). In certain cases (i.e., simply supported or hinged structures where moments are numerically small, small rotation problems), it may be beneficial to turn off the additional convergence testing done for moments and/or rotations.
22. For SOL 400, the convergence checking flag "A" is implemented. "A" means automatically switching to an appropriate convergence checking flag if an unappropriated one is selected for a particular problem. For example, for the problem of stress-free contact analysis, the convergence checking flag PV is inappropriate because this may result of zero divided by zero in convergence checking computation. In this case, PV is switched to UV automatically if A is selected and the residual force is small, i.e., PVA → UVA . The legal combinations for A and PA, UA, WA, PVA, UVA, PNA, and UNA. The rules for auto-switching are that P is switched to U, U is switched to P, and W is switched to UP. For example, PVA → UVA , PVA → UNA , etc. For all other combinations, the A selection is ignored, for example, UPA is the same as UP.
23. For SOL 101 contact, NLPARM can be used to control nonlinear solution process, such as the number of load increments. Since the only source of nonlinearity comes from contact, some fields, however, are neither relevant to the iterative solution nor considered as user-controllable. These fields include DT(=0.0), KMETHOD(=AUTO), KSTEP(=500), INTOUT(=NO) and MAXLS(=0). NLPARM is not required for running SOL 101 contact jobs. If it is not provided by the user, Nastran will create one. All the values of its fields are printed in F06 file under N O N - L I N E A R I T E R A T I O N S O L U T I O N C O N T R O L P A R A M E T E R S. Listed below are the remaining defaults for SOL 101 contact analysis.

NINC	MAXITER	CONV	MAXDIV	MAXQN	FSTRESS	LSTOL	MAXBIS	MAXR	RTOLB	MINITER
10	25	"PV"	3	0	0.2	0.5	5	20.0	20.0	2

Note that the defaults for EPSU, EPSP and EPSW are different for NINC < 10 and NINC ≥ 10.

	EPSU	EPSP	EPSW
NINC < 10	1.e-3	1.e-3	1.e-7
NINC ≥ 10	1.e-2	1.e-2	1.e-2

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

NLPCI**Parameters for Arc-Length Methods in Nonlinear Static Analysis**

Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis (SOL 106 and SOL 400). This entry will be used if a subcase contains an NLPARM command (NLPARM = ID).

Format:

1	2	3	4	5	6	7	8	9	10
NLPCI	ID	TYPE	MINALR	MAXALR	SCALE		DESITER	MXINC	

Example:

NLPCI	10	CRIS	1.0	1.0			12	10	
-------	----	------	-----	-----	--	--	----	----	--

Descriptor	Meaning
ID	Identification number of an associated NLPARM entry. (Integer > 0)
TYPE	Constraint type. See Remark 2. (Character: “CRIS”, “RIKS”, or “MRIKS”; Default = “CRIS”)
MINALR	Minimum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 3. and 4. (0.0 < Real \leq 1.0; Default = 0.25)
MAXALR	Maximum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 3. and 4. (Real \geq 1.0; Default = 4.0)
SCALE	Scale factor (w) for controlling the loading contribution in the arc-length constraint. SOL 106 only. (Real \geq 0.0; Default = 0.0)
DESITER	Desired number of iterations for convergence to be used for the adaptive arc-length adjustment. See Remarks 3. and 4. (Integer > 0; Default = 12)
MXINC	Maximum number of controlled increment steps allowed within a subcase. See Remark 5. (Integer > 0; Default = 20)

Remarks:

1. The NLPCI entry is selected by the Case Control command NLPARM = ID. There must also be an NLPARM entry with the same ID. However, for creep analysis (DT \neq 0.0 in NLPARM entry), the arc-length methods cannot be activated, and the NLPCI entry is ignored if specified. The NLPCI entry is not recommended for heat transfer analysis in SOL 153. Arc-length method is not supported in contact analysis.
2. The available constraint types are as follows:

TYPE = “CRIS”:

$$\{u_n^i - u_n^0\}^T \{u_n^i - u_n^0\} + w^2 (\mu^i - \mu^0)^2 = \Delta l_n^2$$

TYPE = "RIKS":

$$\{u_n^i - u_n^{i-1}\}^T \{u_n^i - u_n^0\} + w^2 \Delta \mu^i = 0$$

TYPE = "MRIKS":

$$\{u_n^i - u_n^{i-1}\}^T \{u_n^{i-1} - u_n^0\} + w^2 \Delta \mu^i (\mu^{i-1} - \mu^0) = 0$$

where:

w = the user-specified scaling factor (SCALE)

μ = the load factor

Δl = the arc-length

The constraint equation has a disparity in the dimension by mixing the displacements with the load factor. The scaling factor (w) is introduced as user input so that the user can make constraint equation unit-dependent by a proper scaling of the load factor μ . As the value of w is increased, the constraint equation is gradually dominated by the load term. In the limiting case of infinite w, the arc-length method is degenerated to the conventional Newton's method.

3. The MINALR and MAXALR fields are used to limit the adjustment of the arc-length from one load increment to the next by:

$$\text{MINALR} \leq \frac{\Delta l_{\text{new}}}{\Delta l_{\text{old}}} \leq \text{MAXALR}$$

The arc-length adjustment is based on the convergence rate (i.e., number of iterations required for convergence) and the change in stiffness. For constant arc-length during analysis, use MINALR = MAXALR = 1.

4. The arc-length Δl for the variable arc-length strategy is adjusted based on the number of iterations that were required for convergence in the previous load increment (I_{\max}) and the number of iterations desired for convergence in the current load increment (DESITER) as follows:

$$\Delta l_{\text{new}} = \Delta l_{\text{old}} \sqrt{\frac{\text{DESITER}}{I_{\max}}}$$

5. The MXINC field is used to limit the number of controlled increment steps in case the solution never reaches the specified load. This field is useful in limiting the number of increments computed for a collapse analysis.
6. NLPCI does not support general contact in SOL 400.

NLRGAP**Nonlinear Load Proportional to Gap**

Defines a nonlinear radial (circular) gap for transient response or nonlinear harmonic response.

Format:

1	2	3	4	5	6	7	8	9	10
NLRGAP	SID	GA	GB	PLANE	TABK	TABG	TABU	RADIUS	

Example:

NLRGAP	21	3	4	XY	3	10	6	1.6	
--------	----	---	---	----	---	----	---	-----	--

Descriptor	Meaning
SID	Nonlinear load set identification number. (Integer > 0)
GA	Inner (e.g., shaft) grid for radial gap. (Integer > 0)
GB	Outer (e.g., housing) grid for radial gap. (Integer > 0)
PLANE	Radial gap orientation plane: XY, YZ, or ZX. (Character; Default = XY.)
TABK	TABLED1 ID defining gap stiffness vs. time or frequency (Integer > 0) or the TABLED1 ID defining gap force vs. penetration (Integer < 0), or TABLED5 ID defining frequency and (gap force vs. penetration) for nonlinear harmonic response only. See Remark 11.
TABG	Table ID for radial gap clearance as function of time for transient response or frequency for nonlinear harmonic response. (Integer > 0)
TABU	Table ID for radial coefficient of friction as function of time for transient response or frequency for nonlinear harmonic response. (Integer > 0)
RADIUS	Shaft radius. (Real \geq 0.0; Default = 0.0)

Remarks:

1. NLRGAP must be selected with the Case Control command NONLINEAR = SID.
2. Multiple NLRGAP entries with the same SID are allowed.
3. The NLRGAP is not an element, but a nonlinear load similar to the NOLINi Bulk Data entries. It computes the relative displacements of GA and GB in the selected plane and applies appropriate nonlinear loads to simulate the radial contact.
4. The degrees-of-freedom in the XY, YZ, or ZX planes (depending on the PLANE) of GA and GB must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation. If RADIUS is > 0.0, then the in-plane rotation degree-of-freedom must also be in the solution set.

5. The NLRGAP is limited to use in direct transient or nonlinear harmonic response solution sequences. When enforced motion is used, the NLRGAP requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
6. The XY, YZ and ZX planes are relative to the displacement coordinate systems of GA and GB. In order to ensure correct results, points GA and GB must satisfy the following three conditions:
 - a. They must both be grid points
 - b. They must be coincident
 - c. They must have parallel displacement coordinate systemsThe program checks for the above conditions. If they are not satisfied, the program terminates the execution with an appropriate fatal message. The coincident check can be skipped by specifying system(648)=1. The default is 0.
7. The shaft radius is used only for the computation of friction induced torque.
8. In the underlying equations, a positive coefficient of friction is consistent with counter-clockwise shaft rotation from axis 1 towards axis 2 (anti-clockwise). A negative coefficient of friction is consistent with clockwise shaft rotation from axis 2 towards axis 1 (clockwise). See [Figure 9-115](#).
9. Nonlinear forces for the grids referenced on the NLRGAP can be output with the NLLOAD Case Control command. See [Figure 9-115](#) for the sign conventions.

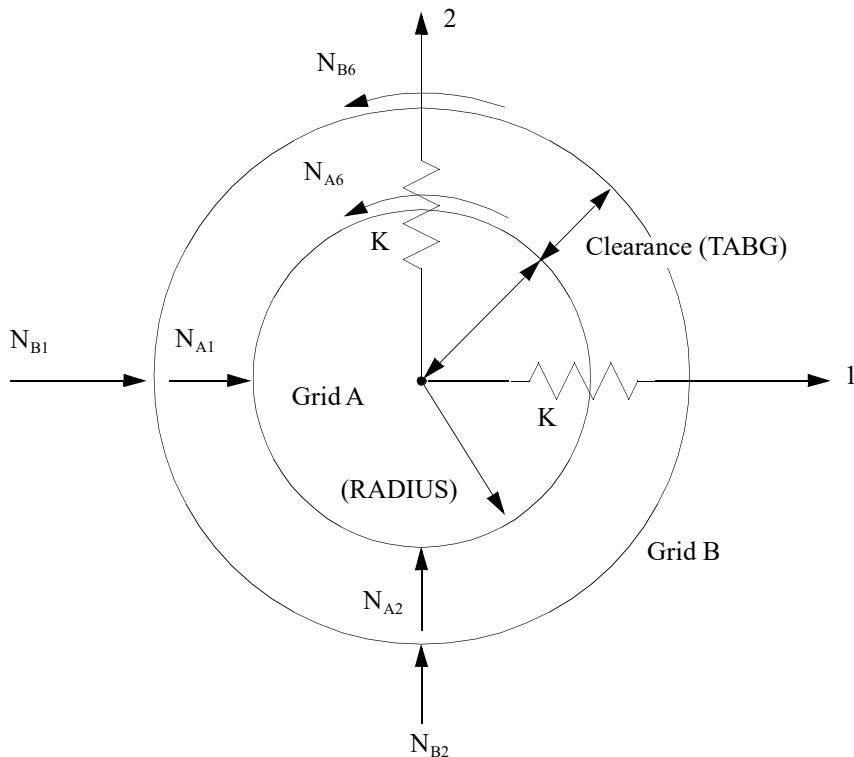


Figure 9-115 Radial Gap Orientation and Nonlinear Load Sign Conventions

10. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

11. TABK may reference either a TABLED1 ID or a TABLED5 ID.

For transient response, only a TABLED1 entry may be referenced. There are two possible scenarios:

- a. The number entered in the TABK field is a positive integer. In this case the integer references a TABLED1 entry defining pairs of time vs. gap stiffness.
- b. The number entered in the TABK field is a negative integer. In this case the absolute value of the integer references a TABLED1 entry defining pairs of gap penetration vs. gap force.

For nonlinear harmonic response either a TABLED1 entry or a TABLED5 entry may be referenced leading to three possible scenarios:

- a. The number entered in the TABK field is a positive integer referencing a TABLED1 entry. In this case, the pairs of values defined on the TABLED1 entry are frequency vs. gap stiffness.
- b. The number entered in the TABK field is a negative integer, the absolute value of which references a TABLED1 entry. In this case, the pairs of values defined on the TABLED entry are gap penetration vs. gap force.

- c. The number entered in the TABK field is a positive integer referencing a TABLED5 entry. In this case, the pairs of values defined on the TABLED5 entry are frequency vs. a TABLED1 ID that defines pairs of gap penetration vs. gap force.
- 12. Forces due to TABK and TABU at GA and GB are only present when the gap is closed. A moment is applied only when the gap is closed and RADIUS > 0.0.

NLRSFD**Nonlinear Transient Load Proportional to a Squeeze Film Damper**

Defines a nonlinear transient radial squeeze film damper.

Format:

1	2	3	4	5	6	7	8	9	10
NLRSFD	SID	GA	GB	PLANE	BDIA	BLEN	BCLR	SOLN	
	VISCO	PVAPCO	NPORT	PRES1	THETA1	PRES2	THETA2	NPNT	
	OFFSET1	OFFSET2	GROUP NAME	NAME2					
	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	RDATA8	

Example:

NLRSFD	100	1001	1002	XY	1.0	2.0	0.05	LONG	
	2.1	300.0	1	100.0	30.0	120.0	90.0	51	
	0.01	0.0	GRPNAMEx						
	0.0				0.0	0.	0.0	0.0	

Alternate Example with UDS:

NLRSFD	100	1001	1002	XY	1.0	1.0	1.0		
	1.0	1	1	1.0	1.0	1.0	1.0	1	
	1.0	1.0	MYSUB	EXT					
	1.0E-4								

Descriptor	Meaning
SID	Nonlinear load set identification number. (Integer > 0; Required)
GA	Inner (e.g., damper journal) grid for squeeze film damper. (Integer > 0; Required)
GB	Outer (e.g., housing) grid for squeeze film damper. (Integer > 0; Required)
PLANE	Radial gap orientation plane: XY, XZ, or ZX. See Remark 1. (Character; Default = XY)
BDIA	Inner journal diameter. (Real > 0.0; Required)
BLEN	Damper length. (Real > 0.0; Required)
BCLR	Damper radial clearance. (Real > 0.0; Required)
SOLN	Solution option: LONG or SHORT bearing. (Character; Default = LONG)
VISCO	Lubricant dynamic viscosity. (Real > 0.0; Required)
PVAPCO	Lubricant vapor pressure. (Real; Required)
NPORT	Number of lubrication ports: 1 or 2 (Integer; no Default)
PRES1	Boundary pressure for port 1. (Real \geq 0.0; Required if NPORT = 1 or 2)

Descriptor	Meaning
THETA1	Angular position for port 1. ($0.0 \leq \text{Real} > 360.0$; Required if NPURT = 1 or 2). See Remark 2.
PRES2	Boundary pressure for port 2. ($\text{Real} \geq 0.0$; Required if NPURT = 2).
THETA2	Angular position for port 2. See Remark 2. ($0.0 \leq \text{Real} < 360.0$; Required if NPURT = 2)
NPNT	Number of finite difference points for damper arc. (Odd Integer ≤ 201 ; Default = 101)
OFFSET1	Offset in the SFD direction 1. (Real; Default = 0.0)
OFFSET2	Offset in the SFD direction 2. (Real; Default = 0.0)
GRPNAM	The GRPNAM is the name used for the group on the FMS CONNECT SERVICE statement; this is used with an external User Defined Service (UDS). See Remark 6. (Character or Blank)
EVALNAME	Value passed to the external User Defined Service as argument "evalname". (Character or Blank)
RDATAi	Parameters passed to the external User Defined Service as arguments "parmi". (Real; Default = 0.0). See Remark 7.

Remarks:

1. The XY, YZ, and ZX planes are relative to the displacement coordinates of GA and GB. The plane coordinates correspond to the NLRSFD directions 1 and 2.

In order to ensure correct results, points GA and GB must satisfy the following three conditions:

- a. They must both be grid points
- b. They must be coincident
- c. They must have parallel displacement coordinate systems

The program checks for the above conditions. If they are not satisfied, the program terminates the execution with an appropriate fatal message.

2. The angular measurement is counterclockwise from the displacement x-axis for the XY plane, the y-axis for the YZ plane, and the z-axis for the ZX plane.
3. OFFSET1 = Damper housing ID center offset displacement relative to OD center in the horizontal direction. Entered as a positive value for horizontally to the left (negative x-direction) displacement (inches).
4. OFFSET2 = Damper housing ID center offset displacement relative to OD center in the vertical direction. Entered as a positive value for downward (negative y-direction) displacement (inches). Positive entry typically used for -1 g compensation.
5. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

6. If GRPNAME is blank, the NLRSFD entry used the values defined in the fields prior to GRPNAME. If GRPNAME contains a valid group (as defined on the FMS CONNECT SERVICE entry), the values in fields BDIA, BLEN, BCLR, SOLN, VISCO, PVAPCO, NPORT, PRES1, THETA1, PRES2, THETA2, NPNT, OFFSET1 and OFFSET2 are ignored and the NLRSFD will obtain its characteristics from an external source via the CONNECT SERVICE architecture.
7. Parameters may be changed by the external service; the changed values will be stored and returned on a subsequent call.
8. The squeeze films damper defined via the CONNECT SERVICE architecture in NLRSFD does not support rotors defined using ROTORSE.

NLSTEP

Describes the Control Parameters for Mechanical, Thermal and Coupled Analysis in SOL 400 and for Contact Analysis in SOL 101.

Specifies the convergence criteria, step size control and numerical procedure for time/load stepping in SOL 400. For multi-physics, it controls both structural and thermal analysis. Defines analysis preference and control parameters for contact analysis in SOL 101.

There are three groups of data that can be entered through this option:

1. General data which defined parameters that may be used for a variety of simulations. This data is provided by the GENERAL keyword.
2. Selecting the type of procedure used to control the time/load stepping procedure. These procedures are activated by the keywords: LCNT, FIXED, ADAPT, or ARCLN (arc length or continuation method). Only one of the keywords may be chosen in a loadcase.
3. Data associated with the physics type that are activated by the keywords: MECH, HEAT, COUP, and RCHEAT. One can enter as many as necessary.

The NLSTEP is selected by the Case Control Command NLSTEP=ID.

Format: (For SOL 400)

1	2	3	4	5	6	7	8	9	10
NLSTEP	ID	TOTTIME	CTRLDEF						
"GENERAL"	MAXITER	MINITER	MAXBIS	CREEP					
"FIXED"	NINC	NO							
"ADAPT"	DTINITF	DTMINF	DTMAXF	NDESIR	SFACT	INTOUT	NSMAX		
	IDAMP	DAMP	CRITTID	IPHYS	LIMTAR	RSMALL	RBIG		
	ADJUST	MSTEP	RB	UTOL					
"ARCLN"	TYPE	DTINITFA	MINALR	MAXALR		NDESIRA	NSMAXA		
"HEAT"	CONVH	EPSUH	EPSPH	EPSWH	KMETHODH	KSTEPH			
	MAXQNH	MAXLSH	LSTOLH						
"MECH"	CONV	EPSU	EPSP	EPSW	KMETHOD	KSTEP	MRCOVN		
	MAXQN	MAXLS	LSTOL	FSTRESS					
"COUP"	HGENPLAS	HGENFRIC							
"RCHEAT"	SOLVER	DRLXCA	ARLXCA	BALENG	DAMPC	GRVCON	CSGFAC		
	NRLOOP	OUTINV	DTIME1						

Format: (For SOL 101)

1	2	3	4	5	6	7	8	9	10
NLSTEP	ID		CTRLDEF						
	"LCNT"	NINCC	CONVC	EPSUC	EPSPC	EPSWC	MAXDIVC	MAXBIS	
		MAXITERC	MINITERC						

Example: (Fixed stepping, 30 increments, total time 4.3, max 5 bisections)

NLSTEP	10	4.3							
	GENERAL			5					
	FIXED	30							
	MECH	PV		0.01					

Example: (Automatic stepping, total time 4.3. Start out with 2%)

NLSTEP	20	4.3							
	GENERAL			5					
	ADAPT	0.02	1.-5		5		20		

Example: (Using SEVERELY default setting with automatic time stepping)

NLSTEP	10	4.3	SEVERELY						
	ADAPT								

Example: (Select the default control parameters for accuracy preference in SOL 101)

NLSTEP	10		LCACCU						
--------	----	--	--------	--	--	--	--	--	--

Example: (Four increments, P convergence criterion with error tolerance 1.e-4. The other parameters are defaults for performance preference in SOL 101)

NLSTEP	10		LCPERF						
	LCNT	4	P		1.e-4				

Descriptor	Meaning
ID	Identification number. (Integer > 0)
TOTTIM	Total time for the load case. (Real; Default = 1.0)

Descriptor	Meaning
CTRLDEF	<p>This Keyword is applicable in SOL101 linear static contact analysis or in SOL400 nonlinear static and transient analysis.</p> <p>For SOL400, the values "QLINEAR", "MILDLY", and "SEVERELY" are available along with the option of leaving blank. See Remark 14. for detail description of these three keyword options and the blank option.</p> <p>For SOL101, the values "LCPERF" or "LCACCU" are available along with the option of leaving blank. See Remark 15. for detail description of these two keyword options. If left blank the values as described under the "LCNT" keyword entry will be used.</p> <p>If SOL400 keyword values are present in a SOL101 linear static contact analysis run or SOL101 keyword values are present in a SOL400 nonlinear static run, the keywords are ignored.</p> <p>In SOL400 nonlinear transient analysis, the use of keywords "QLINEAR", "MILDLY", and "SEVERELY" or blank <i>cannot</i> properly set a "FIXED" NINC value or an "ADAPT" DTINITF value so the user <i>still must</i> set whichever one of these keyword entries value is relevant for a transient run.</p> <p>The keywords "QLINEAR", "MILDLY", and "SEVERELY" are ignored for a RC Network heat transfer simulation.</p>
"GENERAL"	Keyword for parameters used for overall analysis.
MAXITER	Maximum number of iterations allowed for each increment. See Remark 6. (Integer; Default = 10)
MINITER	Minimum number of iterations needed for each increment. (Integer >0; Default = 1) In contact analysis or CTRLDEF = SEVERELY, Default = 2). When high accuracy is required, it is also recommended to set MINITER = 2.
MAXBIS	Maximum number of bisections allowed in the current step. (Default = 10). See remark 17. for more information.
CREEP	Creep will be used in the current load case. If set to 1 creep will be used; if set to 0 creep will not be used (even if creep is defined for a material). (Integer 0 or 1; Default = 0). Adaptive load stepping based upon creep can be flagged through CRITTID below. For enhanced nonlinear elements only.
"FIXED"	Keyword to indicate that fixed time stepping is to be used. See Remark 1.
NINC	Number of increments for fixed time stepping. (Integer > 0; Default = 50). The time step of each increment will be TOTTIME/NINC.
NO	Interval for output. Every NO-th increment will be saved for output. (Integer ≥ 0 ; Default = 1).
"ADAPT"	Keyword to indicate that the adaptive load stepping procedure should be used. See Remark 1.

Descriptor	Meaning
DTINITF	Initial time step defined as fraction of total load step time (TOTTIM). (Real; Default 0.01). If DTINITF>=DTMAXF then, DTINITF is reset to DTMAXF. If CTRLDEF is set to QLNEAR, the user should set DTINITF equal to TOTTIM.
DTMINF	Minimum time step defined as fraction of total load step time (TOTTIM). (Real; Default 1e-5).
DTMAXF	Maximum time step defined as fraction of total load step time (TOTTIM). (Real; Default 0.5). For most nonlinear problems, this should be between 0.05 and 0.2 for dynamic simulations.
NDESIR	Desired number of iterations per increment. See Remark 2. (Integer; Default = 4)
SFACT	Factor for increasing time steps due to number of iterations. See Remark 4. (Real; Default = 1.2).
INTOUT	Output flag. Integer ≥ -1 . (Default = 0) <ul style="list-style-type: none"> -1 Only the last increment of the step will be output. 0 Every computed load increment will be output. > 0 The output will be obtained at INTOUT equally spaced intervals. The time step will be temporarily adjusted if necessary in order to reach these points in time.
NSMAX	Maximum number of increments in the current load case. (Integer; Default = 99999). The job will stop if this limit is reached.
IDAMP	Flag for activating artificial damping for static analysis. (Integer, Default = 0). See Remarks 27. <ul style="list-style-type: none"> 0 No damping considered 4 Artificial damping is always turned on 5 Artificial damping is not turned on. But time step is adjusted based on damping energy as 4 6 When the time step reaches the minimum value and artificial damping is turned on
DAMP	Damping ratio. (Real; Default = 2.e-4).
CRITTID	ID of Bulk Data TABSCTL entry which defines the user criteria to use. See Remark 5. (Integer; Default 0)
IPHYS	Flag to determine if automatic physical criteria should be added and how analysis should proceed if a user criterion is not satisfied. (Integer; Default = 2) It is recommended to use 1. <ul style="list-style-type: none"> 2 Do not add automatic physical criteria; stop when any user criterion is not satisfied

Descriptor	Meaning
-2	Do not add automatic physical criteria; continue when user criteria are not satisfied
1	Add automatic physical criteria; stop when any user criterion is not satisfied
-1	Add automatic physical criteria; continue when any user criterion is not satisfied
LIMTAR	Enter 0 to treat user criteria as limits, 1 to treat user criteria as targets. (Integer; Default = 0). Only used if a user criterion is given through CRITTID. See Remark 5.
RSMALL	Smallest ratio between time step changes due to user criteria. (Real; Default = 0.1)
RBIG	Largest ratio between time step changes due to user criteria. (Real; Default = 10.0)
ADJUST	Time step skip factor for automatic time step adjustment. Only for dynamics. See Remark 16. (Integer; Default = 0).
MSTEP	Number of steps to obtain the dominant period response. See Remark 16. ($10 \leq \text{Integer} \leq 200$ or = -1; Default = 10).
RB	Define bounds for maintaining the same time step for the stepping function during the adaptive process. See Remark 16. ($0.1 \leq \text{Real} \leq 1.0$; Default = 0.6).
UTOL	Defines tolerance on displacement. ($.0001 \leq \text{Real} \leq 1.0$; Default = 1.0)
"LCNT"	Keyword to indicate that the contact analysis in SOL101 should be used. See Remark 15.
NINCC	Number of increments. (Integer > 0; Default = 10 for CTRLDEF= ""; Default=1 for CTRLDEF="LCPERF" or "LCACCU")
CONVC	Flags to select convergence criteria. (Character="U", "P", "W", "V", or any combination; Default = "PV" for CTRLDEF = "" or "LCPERF"; Default = "UPV" for CTRLDEF= "LCACCU"). See Remark 19.
EPSUC	Error tolerance for displacement (U) criterion. (Real > 0.0; Default = 1.0E-3 for CTRLDEF = "LCPERF"; Default = 1.0E-2 for CTRLDEF= "" or "LCACCU")
EPSPC	Error tolerance for load (P) criterion. (Real > 0.0; Default = 1.0E-3 for CTRLDEF = "LCPERF"; Default = 1.0E-2 for CTRLDEF= "" or "LCACCU")
EPSWC	Error tolerance for work (W) criterion. (Real > 0.0; Default = 1.0E-7 for CTRLDEF = "LCPERF"; Default = 1.0E-2 for CTRLDEF= "" or "LCACCU")
MAXDIVC	Limit on probable divergence conditions per iteration before the solution is assumed to diverge. (Integer $\neq 0$; Default = 3 for CTRLDEF = "" or "LCPERF"; Default = 5 for CTRLDEF= "LCACCU").
MAXBISC	Maximum number of bisections allowed for each load increment. (-10 \leq Integer \leq 10; Default = 5)
MAXITERC	Limit on number of iterations for each load increment. (Integer ≥ 0 , Default = 25)
MINITERC	Minimum number of iterations of a load increment. (Integer > 0; Default = 1. In contact analysis, Default = 2).

Descriptor	Meaning
"ARCLN"	Keyword to indicate that an arc length load stepping procedure is to be used. It does not support general contact in SOL 400. See Remark 1.
TYPE	Constraint type. (Character: "CRIS", "RIKS", or "MRIKS"; Default = "CRIS") See Remark 7.
DTINITFA	Initial time step defined as a fraction of the load step time (TOTTIME) for the arc-length procedure. (Real; Default .01)
MINALR	Minimum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 8. and 9. (0.0 < Real < 1.0; Default = 0.25)
MAXALR	Maximum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 8. and 9. (Real > 1.0; Default = 4.0)
NDESIRA	Desired number of iterations for convergence to be used for the adaptive arc-length adjustment. (Integer > 0; Default = 4) See Remarks 8. and 9.
NSMAXA	Maximum number of increments in the current load case. (Integer; Default = 1000). The job will stop if this limit is reached
"HEAT"	Keyword for parameters for heat transfer analysis.
CONVH	Flags to select convergence criteria. (Character = "U", "P", "W", "V", "N", "A" or any combination. (Default UPW)
EPSUH	Error tolerance for temperature (U) criterion. (Real; Default =1.0E-2)
EPSPH	Error tolerance for heat flux (P) criterion. (Real; Default =1.0E-2)
EPSWH	Error tolerance for work (W) criterion. (Real; Default =1.0E-2)
KMETHODH	Method for controlling stiffness updates. (Character = "PFNT", "AUTO" or "ITER"). Default = "AUTO". See Remark 22.
KSTEPH	Number of iterations before the stiffness update for the ITER method. (Integer; Default = 1).
MAXQNH	Maximum number of quasi-Newton correction vectors to be saved on database. (Integer; Default = MAXITER). Not used for PFNT.
MAXLSH	Maximum number of line searches allowed for each iteration. (Integer; Default = 4). Not used for PFNT.
LSTOLH	Line Search tolerance. (0.01 < Real < 0.9; Default = 0.5)
"MECH"	Keyword for parameters for mechanical analysis
CONV	Flags to select convergence criteria. (Character = "U", "P", "W", "V", "N", "A" or any combination; Default = PV). See Remark 19.
EPSU	Error tolerance for displacement (U) criterion. (Real; Default = -0.1)
EPSP	Error tolerance for residual load (P) criterion. (Real; Default = 0.1)
EPSW	Error tolerance for work (W) criterion. (Real; Default = 0.1)

Descriptor	Meaning
KMETHOD	Method for controlling stiffness updates. (Character = "PFNT" or "ITER"; Default = "PFNT"). See Remark 22 .
KSTEP	Number of iterations before the stiffness update for the ITER method. (Integer; Default = 10).
MRCOVN	Flag to specify if rotations and moments should be included in the convergence testing when CONV is set to UV, UN, PV, PN, UPV or UPN: (Integer; Default = 3) <ul style="list-style-type: none"> 0 check on forces, moments, displacements and rotations 1 check on forces, moments and displacements 2 check on forces, displacements and rotations 3 check on forces and displacements If there is no V and N in the CONV, MRCOVN is ignored and all translational and rotational quantities are used to compute the convergence criteria.
MAXQN	Maximum number of quasi-Newton correction vectors to be saved on the database. (Integer; Default = MAXITER). Not used for PFNT.
MAXLS	Maximum number of line searches allowed for each iteration. (Integer; Default = 4). Not used for PFNT.
LSTOL	Line Search tolerance. ($0.01 < \text{Real} < 0.9$; Default = 0.5). Not used for PFNT.
FSTRESS	Fraction of effective stress used to limit the sub increment size in material routines ($0.0 < \text{Real} < 1.0$; Default = 0.2) For non material enhanced elements only.
"COUP"	Keyword for parameters for coupled analysis.
HGENPLAS	Conversion factor for heat generated due to plasticity. (Real; Default = 0.0)
HGENFRIC	Conversion factor for heat generated due to friction. (Real; Default = 0.0)
"RCHEAT"	Keyword to indicate that RC Heat Transfer Analysis is to be performed. See Remark 10 .
SOLVER	The Relaxation scheme to be used. See Remark 12 . (Character; Default = "SNSOR")
DRLXCA	Diffusion node convergence criterion. See Remark 11 . ($\text{Real} \geq 0$; Default = 1.e-3)
ARLXCA	Arithmetic node convergence criterion. See Remark 11 . ($\text{Real} \geq 0.0$; Default = 1.0e-3 degrees)
BALENG	Allowable system energy imbalance. See Remark 11 . ($\text{Real} \geq 0.0$; Default 0.0 energy/time)
DAMPC	Damping constant. ($\text{Real} \geq 0.0$; Default 0.0 non dimensional)
GRVCON	Gravitation constant. ($\text{Real} \geq 0.0$; Default $9.81 \text{ length/time}^2$)
CSGFAC	Time step control factor. See Remark 13 . ($\text{Real} \geq 0.0$; Default 1.0 nondimensional)
NRLOOP	Number of relaxation loops allowed. (Integer ≥ 0 ; Default 5 loop)
OUTINV	Output interval. See Remark 13 . ($\text{Real} \geq 0.0$; Default 60.0 time)
DTIME1	Time step. See Remark 13 . ($\text{Real} \geq 0.0$; Default 0.0 time)

Remarks:

1. Note that the following applies:
 - a. Only one of LCNT, FIXED, ADAPT, ARCLN, or RCHEAT time/load stepping scheme can be used on a specific NLSTEP entry.
Note that only LCNT may be used in SOL 101. If other selection of load stepping scheme is used, it will be ignored, and the default value of LCNT will be applied
 - b. FIXED or ADAPT may be used for a single physics STEP or for a coupled physics STEP/SUBSTEP.
 - c. If no LCNT, FIXED, ADAPT, or ARCLN appear on a NLSTEP entry, then the default is FIXED with 50 increments.
 - d. The ARCLN method is not supported in contact analysis.
 - e. The ARCLN method should not be used in transient dynamic analysis.
 - f. The ARCLN method should not be used in conjunction with HEAT or COUP.
 - g. The ARCLN method cannot be used with the Intel MKL PARDISO solver.
 - h. In one simulation, one cannot use both HEAT and RCHEAT.
 - i. One cannot use both COUP and RCHEAT.
 - j. The LCNT keyword can only be used in a SOL 101 analysis.
2. The desired number of recycles (NDESIR) can be used in static mechanical, dynamic mechanical (see Remark 16) and heat transfer.
3. In coupled analysis, the time step change is calculated separately for heat and mechanical and the smallest of the two is used.
4. When the time step is increased due to desired number of recycles, the previous time step is multiplied with SFACT. When the time step is decreased the factor is calculated internally based upon the minimum time step.
5. User criteria can be given in the TABSCTL entry via CRITTID. These criteria include rotation, displacements, stresses, strains, creep strains. The time step is decreased if the current value of the value is larger than the user specified limit. If LIMITAR is equal to 1 ("target") it also increases the time step for the next increment if the current value is smaller than the target value given.
6. If MAXITER is given a negative value and the MAXITER number of iterations are obtained, convergence is assumed and the analysis will continue with the next increment.
7. The "ARCLN" entry is applicable to "MECH" analysis only and is ignored for creep analysis. The available constraint types are as follows.

TYPE = "CRIS":

$$\{U_n^i - U_n^O\}^T \{U_n^i - U_n^O\} + w^2 (\mu^i - \mu^O)^2 = \Delta l_n^2$$

TYPE = "RIKS":

$$\{U_n^i - U_n^{i-1}\}^T \{U_n^i - U_n^O\} + w^2 \Delta \mu^i = 0$$

TYPE = "MRIKS":

$$\{U_n^i - U_n^{i-1}\}^T \{U_n^{i-1} - U_n^O\} + w^2 \Delta \mu^i (\mu^{i-1} - \mu^O) = 0$$

where:

w = user specified scaling factor (SCALEA)

μ = load factor

l = the arc-length

The constraint equation has a disparity in the dimension by mixing the displacements with the load factor. The scaling factor (w) is introduced as user input so that the user can make constraint equation unit-dependent by a proper scaling of the load factor (μ). As the value of w is increased, the constraint equation is gradually dominated by the load term. In the limiting case of infinite, the arc-length method is degenerated to the conventional Newton's method

8. The MINALR and MAXALR fields are used to limit the adjustment of the arc-length from one increment to the next by:

$$\text{MINALR} \leq \Delta l_{\text{new}} / \Delta l_{\text{old}} \leq \text{MAXALR}$$

The arc-length adjustment is based on the convergence rate (i.e., number of iterations required for convergence) and the change in stiffness. For constant arc-length during analysis, use:

$$\text{MINALR} = \text{MAXALR} = 1$$

9. The arc-length l for the variable arc-length strategy is adjusted based on the number of iterations that were required for convergence in the previous increment (I_{\max}) and the number of iterations desired for convergence in the current increment (NDESIRA) as follows:

$$\Delta l_{\text{new}} = \Delta l_{\text{old}} (NDESIRA / I_{\max})^{1/2}$$

10. This entry is required for a non finite element, Resistance-Capacitor network method of analysis for heat transfer.
11. Convergence is determined by the combination of DRLXCA, ARLXCA, and BALENG. DRLXCA and ARLXCA determine if relaxation is met on a node by node basis, rather than a residual vector length.
12. If in Case Control the ANALYSIS=RCNS, then valid values of SOLVER are:

SNSOR (Default)	Successive over-relaxation method
STDSTL	An iterative solver aimed at the fourth root of a quartic for the network equations (good for strong radiation dependence).

If in Case Control the ANALYSIS=RCNT, then valid values are

SNDUFR (Recommended)	An unconditionally stable, explicit method based on a modified Dufort-Frankel scheme
SNFRDL	Fast, accurate explicit forward differencing transient method
FWDBKL	Implicit forward/backward differencing Crank Nicolson method
SNADE	Alternating direction explicit method
ATSDUF	SNFUFR with automatic time step based on ERRMIN/ERRMAX
ATSFBK	FWDBKL with automatic time step based on ERRMIN/ERRMAX
SNTSM	Weighted implicit forward/backward differencing method
SNTSM3	Weighted implicit forward/backward differencing method
SNTSM1	Weighted implicit forward/backward differencing method
SNTSM4	Weighted implicit forward/backward differencing method

If SOLVER is left blank or set to SNSOR and ANALYSIS=RCNT then internally the RC code will select SNDUFR.

13. About the time step:

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

14. In SOL 400, CTRLDEF entry is only valid for "MECH", "HEAT" and "COUP". The description and default of each option is listed below.

CTRLDEF	<p>Keyword to select the default setting of the control parameters. (Character = "QLINEAR", "MILDLY", "SEVERELY". No default.)</p> <p>Stress analysis (NLSTAT, NLTRAN):</p> <pre> QLINEAR : FIXED = 1 inc, ADAPT : initial time ratio = 1.0 (maximum time ratio = 1.0) MILDLY : FIXED = 10 inc, ADAPT : initial time ratio = 0.1 SEVERELY : FIXED = 50 inc, ADAPT : initial time ratio = 0.01 Blank : FIXED = 50 inc, ADAPT : initial time ratio = 0.01 </pre> <table border="0"> <thead> <tr> <th></th><th>CONV</th><th>EPSU*</th><th>EPSP</th><th>EPSW</th><th>KMETHOD</th></tr> </thead> <tbody> <tr> <td>QLINEAR</td><td>PV</td><td>0.001</td><td>0.001</td><td>---</td><td>PFNT</td></tr> <tr> <td>MILDLY</td><td>PV</td><td>0.01</td><td>0.01</td><td>---</td><td>PFNT</td></tr> <tr> <td>SEVERELY</td><td>PV</td><td>0.1</td><td>0.01</td><td>---</td><td>PFNT</td></tr> <tr> <td>Blank</td><td>PV</td><td>0.1</td><td>0.1</td><td>---</td><td>PFNT</td></tr> </tbody> </table> <p>[Note] EPSU* is a suggestion when user wants to set CONV=UPV</p> <p>Thermal analysis (HSTAT, HTRAN) :</p> <pre> OLINEAR : FIXED = 1 MILDLY : FIXED = 10 SEVERELY : FIXED = 50 Blank: : FIXED = 50 </pre> <table border="0"> <thead> <tr> <th></th><th>CONVH</th><th>EPSUH</th><th>EPSPH</th><th>EPSWH</th><th>KMETHOD</th></tr> </thead> <tbody> <tr> <td>OLINEAR</td><td>UPW</td><td>0.001</td><td>0.001</td><td>0.001</td><td>AUTO</td></tr> <tr> <td>MILDLY</td><td>UPW</td><td>0.01</td><td>0.01</td><td>0.01</td><td>AUTO</td></tr> <tr> <td>SEVERELY</td><td>UP</td><td>0.01</td><td>0.01</td><td>---</td><td>AUTO</td></tr> <tr> <td>Blank</td><td>UPW</td><td>0.01</td><td>0.01</td><td>0.01</td><td>AUTO</td></tr> </tbody> </table> <p>For stress analysis, both (1) CTRLDEF and (2) "FIXED" or "ADAPT" have to be set. For heat analysis, both CTRLDEF and "FIXED" have to be set. Otherwise the standard default setting for "FIXED" or "ADAPT" will be set.</p>		CONV	EPSU*	EPSP	EPSW	KMETHOD	QLINEAR	PV	0.001	0.001	---	PFNT	MILDLY	PV	0.01	0.01	---	PFNT	SEVERELY	PV	0.1	0.01	---	PFNT	Blank	PV	0.1	0.1	---	PFNT		CONVH	EPSUH	EPSPH	EPSWH	KMETHOD	OLINEAR	UPW	0.001	0.001	0.001	AUTO	MILDLY	UPW	0.01	0.01	0.01	AUTO	SEVERELY	UP	0.01	0.01	---	AUTO	Blank	UPW	0.01	0.01	0.01	AUTO
	CONV	EPSU*	EPSP	EPSW	KMETHOD																																																								
QLINEAR	PV	0.001	0.001	---	PFNT																																																								
MILDLY	PV	0.01	0.01	---	PFNT																																																								
SEVERELY	PV	0.1	0.01	---	PFNT																																																								
Blank	PV	0.1	0.1	---	PFNT																																																								
	CONVH	EPSUH	EPSPH	EPSWH	KMETHOD																																																								
OLINEAR	UPW	0.001	0.001	0.001	AUTO																																																								
MILDLY	UPW	0.01	0.01	0.01	AUTO																																																								
SEVERELY	UP	0.01	0.01	---	AUTO																																																								
Blank	UPW	0.01	0.01	0.01	AUTO																																																								

15. "LCPERF" specifies the performance preference during analysis, while "LCACCU" prefers accuracy for analysis. These keywords must be defined if the smart contact in SOL 101 default is required. Specification of LCNT keyword is optional. Listed below are default control parameters of contact in SOL 101 if LCNT keyword or some of its fields are not defined. If the CTRLDEF field is blank, default control parameters will be same as those of NLPARM Bulk Data entry.

CTRLDEF	NINCC	CONVC	EPSUC	EPSPC	EPSWC	MAXDIVC	MAXBISC	MAXITERC	MINITERC
LCPERF	1	PV	1.e-3	1.e-3	1.e-7	3	5	25	2
LCACCU	1	UPV	1.e-2	1.e-2	1.e-2	5	5	25	2
	10	PV	1.e-2	1.e-2	1.e-2	3	5	25	2

The default error tolerances EPSPC and EPSWC are relaxed for the accuracy preference LCACCU. These relaxations are imperative to avoid convergence problems with an additional displacement convergence criterion. The default control parameters are selected based on the test results of typical contact in SOL 101 models. Therefore, these empirical data are only served for the purpose of general convenience. The users can customize any control parameters while performing their analysis.

For some contact models, specifying "LCPERF" or "LCACCU" preference may result in poor convergence. In these cases, the user may increase the number of increments (NINCC) to achieve optimal results.

16. If $10 \leq \text{MSTEP} \leq 200$: activating frequency based adaptive time step adjustment with bounds (RB) when specified increment (controlled by ADJUST) converges. It cannot be combined with iteration based adaptive time step adjustment, controlled by NDESIR, and NDESIR is always ignored in this case. If MSTEP = -1: activating frequency based adaptive time step adjustment without bounds (RB). It can be combined with iteration based adaptive time step adjustment controlled by NDESIR. If NDESIR=0, time step is only adjusted after the increment converges.
17. If MAXBIS is positive and the solution does not converge after MAXBIS bisections, the best solution is computed and the analysis is continued to the next time step. If MAXBIS is negative and the solution does not converge in $|\text{MAXBIS}|$ bisection, the analysis is terminated. It is recommended to enter 0.
18. SOL400 only support the displacement error computation with respect to the delta displacement of a load increment for EPSU. Although a positive EPSU is assigned, internally it will be changed into a negative value. If EPSW > 0.0, the energy error is computed with respect to the total energy. If EPSW < 0.0, the energy error is computed with respect to the delta energy of a load increment.
19. The test flags (U = displacement error, P = load equilibrium error, W = work error, V = vector component method, N = length method, and A = auto switch) and the tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All the requested criteria (combination of U, P, W, V and/or N) are satisfied upon convergence. For SOL 400, if the U criterion is selected together with P or W, then for the first iteration of a load increment, the U criterion will not be checked. For SOL 400 if CONV = 'blank' the code will use a default of "UPW" if heat analysis and "PV" if a structural stress analysis is performed. See the *MSC Nastran Handbook for Nonlinear Analysis* for more details on convergence criteria. For V and N, see Remark 20. For A, see Remark 21.
20. V and N are additional methods for convergence checking using the displacement (U) and/or load (P) criteria. V stands for vector component checking. In this method, convergence checking is performed on the maximum vector component of all components in the model. N stands for length checking. In this method, the length of a vector at a grid point is first computed by the SRSS (square root of the sum of the squares) method. Then convergence checking is performed on the maximum length of all grid points in the model. For example, if CONV=UV, then V checking method will be performed with the U criteria, i.e., the maximum displacement component of all displacement components in the model is used for convergence checking. For V and N, the EPSU is always negative, i.e., the error is computed with respect to the delta displacements of a load increment, even if positive value is requested by users. CONV=V is the same as CONV=UPV. If both V and N are specified; V will take precedence over N. For example, CONV=VN is the same as CONV=V.

By default, for UPV or UPN, separate checks are made over force and moment vectors, and translation and rotation vectors. While the force/translation check is valid always, the moment or rotation check is only valid for 6 DOF elements (beams, shells, etc.). In certain cases (i.e., simply supported or hinged structures where moments are numerically small, small rotation problems), it may be beneficial to turn off the additional convergence testing done for moments and/or rotations.

21. For SOL 400, the convergence checking flag "A" is implemented. "A" means automatically switching to an appropriate convergence checking flag if an unappropriated one is selected for a particular problem. For example, for the problem of stress-free contact analysis, the convergence checking flag PV is inappropriate because this may result of zero divided by zero in convergence checking computation. In this case, PV is switched to UV automatically if A is selected and the residual force

is small, i.e., PVA → UVA . The legal combinations for A are PA, UA, WA, PVA, UVA, PNA, and UNA. The rules for auto-switching are that P is switched to U, U is switched to P, and W is switched to UP. For example, PVA → UVA , PVA → UNA , etc. For all other combinations, the A selection is ignored, for example, UPA is the same as UP.

22. The stiffness update strategy is selected in the KMETHOD and KMETHODH field for mechanical and thermal analysis, respectively.
 - AUTO may be used for thermal analysis only. PFNT and ITER may be used in both mechanical and thermal analysis.
 - In the thermal analysis, if the AUTO option is selected, the program automatically selects the most efficient strategy based on convergence rates. At each step the number of iterations required to converge is estimated. Stiffness is updated, if (i) estimated number of iterations to converge exceeds MAXITER, (ii) estimated time required for convergence with current stiffness exceeds the estimated time required for convergence with updated stiffness, and (iii) solution diverges. See Remarks 17. and 23. for diverging solutions. Available for the heat transfer pass only.
 - If the ITER option is selected, the program updates the stiffness matrix at every KSTEP iterations and on convergence if KSTEP < MAXITER. However, if KSTEP > MAXITER, stiffness matrix is never updated. Note that the modified Newton-Raphson iteration method is obtained by selecting the ITER option and KSTEP = MAXITER.
 - If the PFNT option is selected, the program will use the Pure Full Newton iteration method.
23. The ratio of energy errors before and after the iteration is defined as divergence rate (E^i) , i.e.,

$$E^i = \frac{\{\Delta u^i\}^T \{R^i\}}{\{\Delta u^i\}^T \{R^{i-1}\}}$$

Depending on the divergence rate, the number of diverging iteration (NDIV) is incremented as follows:

If $E^i \geq 1$ or $E^i < -10^{12}$, then $NDIV = NDIV + 2$

If $-10^{12} < E^i < -1$, then $NDIV = NDIV + 1$

The solution is assumed to diverge when $NDIV \geq |MAXDIVC|$. If the solution diverges and the load increment cannot be further bisected (i.e., MAXBIS is attained or MAXBIS is zero), the stiffness is updated based on the previous iteration and the analysis is continued. If the solution diverges again in the same load increment while MAXDIVC is positive, the best solution is computed and the analysis is continued to the next load increment. If MAXDIVC is negative, the analysis is terminated on the second divergence.

24. Please note that in NLTRAN analysis, the output is also influenced by NLPACK. For example, one NLTRAN analysis has NLSTEP as:

```
NLSTEP,900,0.2
,fixed,2000,20
,mech,u
```

It has total 2000 increments, and asks output every 20 increments. Therefore, total output is 100 time step. In NASTRAN, default value of NLPACK for NLTRAN is 100, in this model, therefore, NASTRAN will write the results to OP2 after collecting 100 output, i.e., this model will write results to OP2 only one time. With intermediate output request, only one OP2 file will be created.

If NLPACK=1, NASTRAN will write results to OP2 for every output request. In this model, NASTRAN will write results to OP2 for every output request, i.e., writing 100 times in this model. With "intermediate output request, 100 OP2 files will be created.

If NLPACK=2, NASTRAN will write results to OP2 for every two output requests. In this model, NASTRAN will write results to OP2 for every two output request, i.e., writing 50 times in this model. With "intermediate output request, 50 OP2 files will be created.

For NO of FIXED in NLSTEP, it is similar to INTOUT of ADAPT.

25. If Modules are present then this entry may only be specified in the main Bulk Data section.
26. NLSTEP entry does not work for creep analysis when using the traditional elements. In SOL 400, NLPARM entry must be used to perform creep analysis with the traditional elements model.
27. Artificial damping takes effect in the following way with different options.

IDAMP=4: Artificial damping is always turned on. Both time stepping and added damping are controlled through the step.

- A comparison of the incremental damping energy to the predicted incremental total strain energy is used as a criterion for time step control.
- The damping factor to be used is computed based on the estimated damping energy and the estimated total energy for the step.

IDAMP=5: Artificial damping is not turned on. Similar algorithm as IDAMP = 4 except that only the time stepping is controlled based on the damping.

- A comparison of the incremental damping energy to the predicted incremental total strain energy is used as a criterion for time step control.

IDMAP=6: When the time step reaches the minimum value and the analysis is going to stop prematurely, two attempts are made to avoid a premature exit:

- The increment is repeated with a new time step = 10 tmin and quasi-static damping is added in a manner similar to the IDAMP = 4 scheme.
- If this does not work, the increment is repeated with a new time step = 100 tmin and the process is repeated. Note that once damping is turned on to avoid the premature stop, it remains on for the rest of the step.

NLSTRAT

Strategy Parameters for SOL 600 Nonlinear Structural Analysis

Defines strategy parameters for nonlinear structural analysis used in SOL 600 only. For SOL 600 Heat Transfer, see NLHEATC.

Format:

1	2	3	4	5	6	7	8	9	10
NLSTRAT	ID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	etc				

Example:

NLSTRAT	501	CONVTYP	4	RESPF	.015	ALPHA	.05		
	KNONPOS	1							

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

ID	Identification number referenced by a Case Control command with regard to time steps or load steps (such as SUBCASE). If ID = 0, the values entered will be used for Marc increment zero. For the first subcase ID = 1; for the second subcase ID = 2, etc. If there are no subcases in the model, enter ID = 1. If NLSTRAT with ID > 0 is entered and if the ISOLVER option is entered, another NLSTRAT entry with ID = 0 and the same ISOLVER option must be entered. If NLSTRAT is used, there must be an NLSTRAT entry for each subcase. Multiple NLSTRAT entries with the same ID are not allowed. (Integer ≥ 0)
----	---

PARAMi	Name of the NLSTRAT parameter. Allowable names are given in Table 9-27 . (Character).
--------	---

VALi	Value of the parameter. See Table 9-27 . (Real or Integer)
------	--

Table 9-27 Parameters

Name	Description, Type and Value Convergence Criteria
CONVTYP	Convergence Criteria -- (Integer) - If not set, value will be determined by NLPARM or entry - CONTROL(2,4) The possibilities are: 0 -- Convergence based on residuals 1 -- Convergence based on displacements 2 -- Convergence based on energy 4 -- Convergence based on residuals or displacements 5 -- Convergence based on residuals and displacements
IRELABS	Flag for relative or absolute convergence criteria (Integer) CONTROL(2,5) 0 -- Testing is done on relative error 1 -- Testing is done on absolute value 2 -- Testing is done on relative error testing unless reactions or incremental displacements are below minimum value, in which case absolute tolerance testing is used.
RCK1	Used for Relative Checking - Maximum residual force ratio (maximum allowable value of maximum residual force divided by maximum reaction force) or displacement ratio (maximum allowable value of the change in displacement increment divided by displacement increment) depending on CONVTYP. (Real ≥ 0 ; CONTROL(3,1); Default = 0.1)
RCK2	Used for Relative Checking - Maximum residual moment ratio or if autoswitch is on, the rotation ratio depending on CONVTYP (Real ≥ 0 ; CONTROL(3,2); Default = no checking)
RCK3	Used for Relative Checking - Minimum reaction force ratio or minimum displacement ratio depending on CONVTYP. (Real ≥ 0 ; CONTROL(3,3); no Default; if 0.0, checking is bypassed or absolute testing is performed)
RCK4	Used for Relative Checking - Minimum moment ratio or rotation ratio depending on CONVTYP. (Real ≥ 0 ; CONTROL(3,4))
ABCK1	Used for Absolute Checking - Maximum residual force ratio or displacement ratio depending on CONVTYP. (Real ≥ 0 ; CONTROL(3,5); Default = no checking)
ABCK2	Used for Absolute Checking - Maximum residual moment ratio or rotation ratio depending on CONVTYP. (Real ≥ 0 ; CONTROL(3,6) - Default = no checking)
MAXDI	Maximum change in displacement increment divided by displacement increment (Real) Default is no checking (real) - CONTROL (3a-1) - Enter only if CONVTYP is 4 or 5.
MAXRI	Maximum change in rotational increment divided by rotational increment (Real) Default is no checking (Real) - CONTROL (3a-2) -Enter only if CONVTYP is 4 or 5.
MINDI	Minimum change in displacement increment divided by displacement increment (real) Default is no checking (Real) - CONTROL (3a-3) - Enter only if CONVTYP is 4 or 5.
MINRI	Minimum change in rotational increment divided by rotational increment (real) Default is no checking (Real) - CONTROL (3a-4) - Enter only if CONVTYP is 4 or 5.

Table 9-27 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
MAXD	Maximum value of displacement increment (Real) Default is no checking (real) - CONTROL (3a-5) - Enter only if CONVTYP is 4 or 5.
MAXR	Minimum value of rotational increment (real) Default is no checking (Real) - CONTROL (3a-6) - Enter only if CONVTYP is 4 or 5.
IPRCOV	Flag controls printing of convergence. CONTROL(2,9) (Integer; Default = 0 = no print)
AUTOSW	Flag to turn on or off Marc's Auto Switch (Integer) Controls switching between convergence testing of residuals and displacements when residuals are small - CONTROL (2,11). 0 -- Off (Default unless NLAUTO entry is entered) 1 -- On (Default only if NLAUTO entry is entered)
Newton Iterations	
MAXSTEP	Maximum number of load steps. CONTROL(2,1) (Integer; Default = 9999)
MAXREC	Maximum number of recycle steps per load step (Integer; Default = 3) If set to a negative value, if convergence is not obtained after maxrec recycles, a warning is issued and the analysis proceeds to the next step (not recommended) CONTROL(2,2)
MINREC	Minimum number of recycle steps per load step. CONTROL(2,3) (Integer; Default = 0)
IKMETH	Newton method (Integer; Default = 1) 1= Full Newton, 2 = Modified Newton, 3 = Newton-Raphson with strain correction, 8 = Secant stiffness CONTROL(2,6)
IKUPD	Reassembly interval of stiffness and mass. AUTO LOAD (2,2) DYNAMIC CHANGE (2,5) (Integer)
IKNONPOS	Solve a non positive definite stiffness (1) or not (0). CONTROL(2,7) For jobs with multiple subcases, use IKNONPOS instead of NONPOS if some subcases should handle nonpositive-definite systems and other should not. (Integer)
IKINIT	Initial stiffness control (Integer) CONTROL (2,10) 0 - Normal full contribution 1 - For Mooney material, reduced contribution of hydrostatic pressure based on initial stress 2 - No initial stiffness 3 - Use stress at beginning of increment but not in the last iteration 4 - Use only positive stresses in initial stress stiffness (faster than option 0 and is always stable for thin shell structures).
General Parameter	
STRAINS	Scale factor for strain increments. PARAMETERS(2,1) (Real ≥ 0 ; Default = 1.0)
PENBOUN	Penalty value to enforce certain boundary conditions. PARAMETERS(2,2) (Real)

Table 9-27 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
FSTRESS	Fraction of the hydrostatic pressure subtracted from the stress tensor in initial stress calculation. PARAMETERS(3,5) (Real)
Load Step or Time Step Control	
MAXTSC	Maximum number of allowable time step cuts. AUTO LOAD (2,3) (Integer ≥ 0) 0 - No automatic restart from the previously converged step >1 - Maximum number of time step cutbacks allowed.
Transient Analysis Damping Parameters	
BETA	Beta parameter used by Newmark-beta procedure. PARAMETERS (2,5) (Real; Default = 0.25)
GAMMA	Gamma parameter used by Newmark-beta procedure. PARAMETERS(2,6) (Real; Default = 0.5)
GAMMA1	Gamma1 parameter used by Single Step Houbolt procedure. PARAMETERS (2,7) (Real; Default = 1.5)
GAMMA2	Gamma parameter used by Single Step Houbolt procedure. PARAMETERS (2,8) (Real; Default = -0.5)
Solver-Related Parameters	
ISOLVER	Type of solver (Integer ≥ 0) SOLVER(2,1) 0 - Profile Direct Solver 2 - Sparse Iterative 4 - Sparse Direct 6 - Hardware provided direct sparse 8 - Multifrontal direct sparse (Default) 9 - The CASI iterative solver will be used 10 - The mixed direct/iterative solver will be used. The other NLSTRAT entries that apply to solvers 2 and 9 may also be used for solver 10 but are not required. 11 - Pardiso direct solver 12 - MUMPS parallel direct solver (Note: Solvers that do not require other NLSTRAT entries can be specified by PARAM,MARCSOLV. For solvers 11 or 12 please also see parameters MUMPSOLV and MRTHREAD)
ISYMM	Nonsymmetric solver option (Integer ≥ 0 ; Default = 0) SOLVER (2,2) 0 - Symmetric solver 1 - Non symmetric solver
NONPOS	Nonpositive Definite solver option (Integer ≥ 0 ; Default = 0) SOLVER (2,3) 0 - Error if system is nonpositive-definite 1 - Solve nonpositive definite systems if possible For jobs with multiple subcases, use IKNONPOS instead of NONPOS if some subcases should handle nonpositive-definite systems and other should not.

Table 9-27 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
MBYTE	Solver type 6 or 8 memory option (Integer ≥ 0) SOLVER (2,8). Enter the number of 4-byte words in millions to be used if solver type 6 (SGI only) or solver type 8 (all other systems) is to be used. For example, if 96MB is needed, enter 96.
MAXITER	Maximum number of iterations (Iterative solver only). (Integer; Default = 1000) Enter a negative value if program is to continue even though iterations have not fully converged. SOLVER (3,1)
PREVITER	Enter 1 if the previous solution is to be used as the initial trial value (Iterative solver only) (Integer > 0 ; Default = 0) SOLVER (3,2)
PRECOND	Preconditioner Option (Iterative solver only) (Integer > 0) SOLVER (3,3) For the CASI solver (solver 9): 1- CASI solver with standard preconditioner 0- CASI solver with primal preconditioner For the standard iterative solver (solver 2): 3 - Use diagonal preconditioner 4 - Use scaled-diagonal preconditioner 5 - Use incomplete Cholesky preconditioner
CJTOL	Enter Conjugate Gradient Convergence Tolerance (Iterative solver only) (Real; Default = 0.001) SOLVER (4,1)
<p>Note: Arc Length and Other Parameters for Marc's AUTO INCREMENT Option</p> <p>An NLPCI entry is needed in addition to the options below to trigger the AUTO INCREMENT option. It is usually used for post-buckling problems.)</p>	
AITYPE	Arc Length Method. AUTO INCREMENT (2,8) (Integer ≥ 0 ; Default = 3) 0 standard load control 1 - Crisfield quadratic constraint method 2 - Riks/Ramm linear constraint method 3 - Modified Riks/Ramm (linear constrain method) 4 - Crisfield, switch to modified Riks/Ramm if no real root found
AIMAXCUT	Maximum number or time step cutbacks. AUTO INCREMENT (2,9) (Integer ≥ 0 ; Default if not entered = 5) 0 - No automatic restart (cutbacks) from previous converged step are allowed >0 - Maximum number of load step cutbacks Note for shell buckling problems it is best to set this value to 5 or 10.
AIFRACT	Fraction of total load increment that is applied in the first cycle of the first increment. AUTO INCREMENT (2,1) (Real) Note this value is not set from NLPARM. It needs to be entered if AUTO INCREMENT is to be used for buckling problems.
AIMAXINC	Maximum number of increments. For most problems, this value should be entered and set to a large value such as 99999. AUTO INCREMENT (2,2) (Integer > 0)

Table 9-27 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
AINRECYC	Desired number of recycles per increment used to increase or decrease load steps. AUTO INCREMENT (2,3) (Integer > 0; Default = 3)
AIMAXF	Maximum fraction of the total load that can be applied in any increment. It is recommended that for most nonlinear problems, this value be 0.1 or smaller. AUTO INCREMENT (2,4) (Real; Default=0.05 if the model does not have contact and 0.01 if the model has contact). Note for shell buckling problems it is best not to set this value (i.e., leave it as default). If AIMAXF is set to a small value, the problem will probably diverge and/or get bad results.
AIARCM	Maximum arc length multiplier (norm of displacement vector to initial arc length). AUTO INCREMENT (2,5) (Real; Default is fraction of load divided by initial fraction of load)
AITOTT	Total Time period to be covered, used in conjunction with contact analysis. AUTO INCREMENT (2,6) (Real; Default = 1.0)
AIARC0	Fraction of the initial arc length to define a minimal arc length. AUTO INCREMENT (2,7) (Real; Default = 0.01)
Fully Coupled Heat-Structural Analysis Controls	
TCHANGE	Maximum nodal temperature change allowed. CONTROL(4,1) (Real; Default = 20.0)
TEVAL	Maximum nodal temperature change allowed before properties are re-evaluated and matrices reassembled. CONTROL(4,2) (Real; Default = 100.0)
TERROR	Maximum error in temperature estimates used for property evaluation. CONTROL(4,3) (Real; Default = 0.0; which bypasses the test)
Contact-Related Parameters	
ANG2D	Angle at which a node separates from a convex corner or becomes stuck in a concave corner in 2D contact (Real; Default = 8.625 degrees) PARAMETERS (3,1)
ANG3D	Angle at which a node separates from a convex corner or becomes stuck in a concave corner in 3D contact (Real; Default = 20.0 degrees) PARAMETERS (3,2)
Other Parameters	
DRILLF	The factor used to calculate the drilling mode for shell elements (types 22, 75, 138, 139, 140) DRILLF=1.0E-6*K6ROT. The default for DRILLF (0.0001) and K6ROT (100.0) produce the same results. If K6ROT is entered, it will be used for DRILLF unless DRILLF is also entered. DRILLF has precedence over K6ROT. (Real; Default = 0.0001) PARAMETERS (3,6)
REZONEF	Incremental displacement scale factor after a rezoning increment (Real; Default=1.0). Note that a value of 1.0 improves friction convergence but may result in an inside-out element. PARAMETERS (3,7)
UGAS	Universal Gas Constant (Real; Default = 8.314 J mol ⁻¹ K ⁻¹) PARAMETERS (4,1)

Table 9-27 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
TOFSET	Offset temperature between user units and absolute zero temperature (Real; Default=273.15 which is correct for Centigrade). If temperature units are Kelvin (K) or Rankine (R), enter a negative value and the temperature offset is set to zero PARAMETERS (4,2)
TWEIGHT	Thermal properties evaluation weight (Real; Default = 0.5) PARAMETERS (4,3)
SPFACT	Surface projection factor for single step Houbolt method (Real; Default = 0.0) PARAMETERS (4,4)
STEFAN	Stefan Boltzman Constant (Real; Default = 5.67051E-8 W / m ² K ⁴) PARAMETERS (4,5)
PLANKS	Planks second constant. (Real; Default = 14387.69 micro MK PARAMETERS (4,6)
CLIGHT	Speed of light in a vacuum. (Real; Default = 2.9979E14 micro M/s PARAMETERS (4,7)
RAPMAX	Maximum change in the incremental displacement in a Newton-Raphson iteration. (Real; Default = 1.0E30) PARAMETERS (4,8)
FISTIF	Initial friction stiffness for model 6 used in first cycle of an increment to define the friction stiffness matrix in cases where a touching node has a zero normal force and the amount of sliding does not exceed the elastic sticking limit. (Real; Default = 0.0 in which case the program calculates it.) PARAMETERS (5,1)
SNGMIN	Minimum value that indicates a singularity if a direct solver is used. (Real; Default = 0.0 in which case the value is set internally by the program.) PARAMETERS (5, 2)
RTMAX	Maximum change in temperature per iteration in radiation simulations. (Real; Default = 10 times the maximum error in temperature estimate or 100.0) PARAMETERS (5,3)
New Items for Version 2005 r3 and Subsequent	
IASMBL	Assembly flag. If set to 1, the stiffness matrix is assembled each iteration. Note that this switches off the modified Newton-Raphson procedure if chosen. (Integer)

Table 9-27 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
INNER	For some material models, such as damage, cracking, and Chaboche, there is an inner iteration loop to insure accuracy. The maximum number of iterations allowed can be set here. (Integer; Default = 50)
RIGLNK	Rigid Link Rotation Tolerance: Maximum allowable value of the change in rotation increment at the retained nodes of RBE2, rigid link 80 or beam-shell offset nodes. Default is 0.0, in which case, no checking on rigid link rotations takes place. (Real)
RLROTT	Rigid Link Rotation Tolerance: Maximum allowable value of the change in rotation increment at the retained nodes of RBE2, rigid link 80 or beam-shell offset nodes. Default is 0.0, in which case, no checking on rigid link rotations takes place. (Real) Note: If CONVTYP is 4 or 5, the rigid link rotation tolerance entered here circumvents the corresponding value RIGLNK above. For all versions with PARAM,MARCVR less than 12, the rigid link rotation tolerance if left at 0, is reset to 0.001 radians to ensure backward compatibility for RBE2. In this case, the rigid link rotation tolerance should be set to a negative number to by-pass the check.
ENRGCH	Maximum allowable value of the change in energy increment. Enter only if CONVTYP=2. (Real; Default = 0.1)

Remarks:

1. This entry matches Marc's CONTROL, AUTO LOAD, DYNAMIC CHANGE, PARAMETERS, and SOLVER definitions.
2. NLSTRAT is recognized only when Marc is executed from SOL 600.
3. Correlation between NLSTRAT names and Marc CONTROL entry fields

2-1 MAXSTEP	3-1 RCK1	4-1 MAXDI	5-1 MAXENRG	6-1 TCHANGE
2-2 MAXREC	3-2 RCK2	4-2 MAXRI		6-2 TEVAL
2-3 MINREC	3-3 RCK3	4-3 MINDI		6-3 TERROR
2-4 CONVTYP	3-4 RCK4	4-4 MINRI		6-4 VOLTMAX
2-5 IRELABS	3-5 ABCK1	4-5 MAXD		
2-6 IKMETH	3-6 ABCK2	4-6 MAXR		7-1 ESRELER
2-7 IKNONPOS	4-7 RIGLNK	4-7 RLROTT		7-2 ESABSER
2-8 Not Used				
2-9 IPRCONVs			5-1 ENRGCH	
2-10 IKINT				
2-11 AUTOSW				

2-12 IASMBL

2-13 INERR

- Items 6-4,7-1,7-2 are not presently available using SOL 600).
- 3-1 to 3-6 is entered only if CONVTYP=0, 4 or 5.
- 4-1 to 4-6 is entered only if CONVTYP=1, 4 or 5.
- 5-1 is entered only if CONVTYP=2.

4. Correlation between NLSTRAT names and Marc PARAMETERS entry fields

2-1 STRAINS	3-1 ANG2D	4-1 UGAS	5-1 FISTIF
2-2 PENBOUN	3-2 ANG3D	4-2 TOFSET	5-2 SNGMIN
2-3 PFPLAS	3-3 RATE0	4-3 TWEIGHT	5-3 RTMAX
2-4 PFFLUID	3-4 RATEC	4-4 SPFACT	
2-5 BETA	3-5 FSTRESS	4-5 STEFAN	
2-6 GAMMA	3-6 DRILLF	4-6 PLANKS	
2-7 GAMMA1	3-7 REZONEF	4-7 CLIGHT	
2-8 GAMMA2		4-8 RAPMAX	

5. Correlation between NLSTRAT names and Marc SOLVER entry fields

2-1 ISOLVER	3-1 MAXITER	4-1 CJTOL
2-2 ISYMM	3-2 PREVITER	
2-3 NONPOS	3-3 PRECOND	
2-8 MBYTE		

6. Correlation between NLSTRAT names and Marc AUTO INCREMENT entry fields

2-1 AIFRACT	2-6 AITOTT
2-2 AIMAXINC	2-7 AIARC0
2-3 AINRECYC	2-8 AITYPE
2-4 AIMAXF	2-9 AIMAXCUT
2-5 AIARCM	

7. Correlation between NLSTRAT names and Marc AUTO LOAD entry fields

2-2 IKUPD
2-3 MAXTSC

8. Correlation between NLSTRAT names and Marc DYNAMIC CHANGE entry fields
2-5 IKUPD
9. The ISOLVER must be the same for all load cases (and Phase 0) or Marc will abort. It is recommended that all other SOLVER items also be the same for the entire run. If ISOLVER is a value other than 8, NLSTART with ID=0 must be entered with the desired ISOLVER type. Multiply NLSTRAT entries with ID=1, 2, etc. may be used to change other values if desired.
10. ISYMM must be the same for all load cases. All discussions from note 10 apply to ISYMM.

NOLIN1**Nonlinear Load as a Tabular Function**

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$\text{Function of displacement: } P_i(t) = S \cdot T(u_j(t)) \text{ or } P_i(f) = S \cdot T(u_j(f)) \quad (9-30)$$

$$\text{Function of velocity: } P_i(t) = S \cdot T(\dot{u}_j(t)) \text{ or } P_i(f) = S \cdot T(\dot{u}_j(f)) \quad (9-31)$$

where t is time, f is frequency and $u_j(t/f)$ and $\dot{u}_j(t/f)$ are the displacement and velocity at point GJ in the direction of CJ.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN1	SID	GI	CI	S	GJ	CJ	TID		

Example:

NOLIN1	21	3	4	2.1	3	10	6		
--------	----	---	---	-----	---	----	---	--	--

Descriptor	Meaning
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. ($0 < \text{Integer} \leq 6$; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	$1 \leq \text{Integer} \leq 6$	$11 \leq \text{Integer} \leq 16$
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

TID Identification number of a TABLEDi entry. (Integer > 0)

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.
2. Nonlinear loads may not be referenced on DLOAD entry.

3. All degrees-of-freedom referenced on NOLIN1 entries must be members of the solution set. This means the e-set (EPOINT entry) for modal formulation and the d-set for direct formulation.
4. Nonlinear loads as a function of velocity (9-31) are denoted by components ten greater than the actual component number; i.e., a component of 11 is component 1 (velocity). The velocity is determined by

$$\dot{u}_{j,t} = \frac{u_{j,t} - u_{j,t-1}}{\Delta t}$$

where Δt is the time step interval and $u_{j,t-1}$ is the displacement of GJ-CJ for the previous time step.

5. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
6. RC network solver does not support NOLIN1 for thermal analysis.
7. When enforced motion is used, the NOLIN1 requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
8. An alternate option for averaging nonlinear loads is available with DIAG 10, although it is *not* recommended.

NOLIN2

Nonlinear Load as the Product of Two Variables

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$P_i(t) = S \cdot X_j(t) \cdot X_k(t) \text{ or } P_i(f) = S \cdot X_j(f) \cdot X_k(f)$$

where t is time, f is frequency and $X_j(t/f)$ and $X_k(t/f)$ can be either displacement or velocity at points GJ and GK in the directions of CJ and CK.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN2	SID	GI	CI	S	GJ	CJ	GK	CK	

Example:

NOLIN2	14	2	1	2.9	2	1	2		
--------	----	---	---	-----	---	---	---	--	--

Descriptor	Meaning
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. ($0 \leq \text{Integer} \leq 6$; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ, GK	Grid, scalar, or extra point identification number. (Integer > 0)
CJ, CK	Component number for GJ, GK according to the following table:

Type of Point	Displacement	Velocity
Grid	$1 \leq \text{Integer} \leq 6$	$11 \leq \text{Integer} \leq 16$
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR=SID.
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN2 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. GI-CI, GJ-CJ, and G-K-CK may be the same point.

5. Nonlinear loads may be a function of displacement ($X = u$) or velocity ($X = \dot{u}$). Velocities are denoted by a component number ten greater than the actual component number; i.e., a component of 10 is component 0 (velocity). The velocity is determined by

$$\dot{u}_t = \frac{u_t - u_{t-1}}{\Delta t}$$

where Δt is the time step interval and u_{t-1} is the displacement of GJ-CJ or GK-CK for the previous time step.

6. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
7. RC network solver does not support NOLIN2 for thermal analysis.
8. When enforced motion is used, the NOLIN2 requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
9. An alternate option for averaging nonlinear loads is available with DIAG 10, although it is *not* recommended.

NOLIN3

Nonlinear Load as a Positive Variable Raised to a Power

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$P_i(t) = \begin{cases} S \cdot [X_j(t)]^A, X_j(t) > 0 \\ 0, X_j(t) \leq 0 \end{cases} \quad \text{or} \quad P_i(f) = \begin{cases} S \cdot [X_j(f)]^A, X_j(f) > 0 \\ 0, X_j(f) \leq 0 \end{cases}$$

where t is time, f is frequency and $X_j(t/f)$ may be a displacement or a velocity at point GJ in the direction of CJ.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN3	SID	GI	CI	S	GJ	CJ	A		

Example:

NOLIN3	4	102		-6.1	2	15	-3.5		
--------	---	-----	--	------	---	----	------	--	--

Descriptor	Meaning
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which the nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. ($0 \leq \text{Integer} \leq 6$; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ	Grid, scalar, extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	$1 \leq \text{Integer} \leq 6$	$11 \leq \text{Integer} \leq 16$
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

A Exponent of the forcing function. (Real)

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.

2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN3 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. Nonlinear loads may be a function of displacement ($X_j = u_j$) or velocity ($X_j = \dot{u}_j$). Velocities are denoted by a component number ten greater than the actual component number; e.g., a component of 16 is component 6 (velocity). The velocity is determined by

$$\dot{u}_{j,t} = \frac{u_{j,t} - u_{j,t-1}}{\Delta t}$$

where Δt is the time step interval and $u_{j,t-1}$ is the displacement of GJ-CJ for the previous time step.

5. Use a NOLIN4 entry for the negative range of $X_j(t)$.
6. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
7. RC network solver does not support NOLIN3 for thermal analysis.
8. When enforced motion is used, the NOLIN3 requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
9. An alternate option for averaging nonlinear loads is available with DIAG 10, although it is *not* recommended.

NOLIN4

Nonlinear Load as a Negative Variable Raised to a Power

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$P_i(t) = \begin{cases} -S \cdot [-X_j(t)]^A, X_j(t) < 0 \\ 0, X_j(t) \geq 0 \end{cases} \quad \text{or} \quad P_i(f) = \begin{cases} -S \cdot [-X_j(f)]^A, X_j(f) < 0 \\ 0, X_j(f) \geq 0 \end{cases}$$

where t is time, f is frequency and $X_j(t/f)$ may be a displacement or a velocity at point GJ in the direction of CJ.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN4	SID	GI	CI	S	GJ	CJ	A		

Example:

NOLIN4	2	4	6	2.0	101		16.3		
--------	---	---	---	-----	-----	--	------	--	--

Descriptor	Meaning
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 < Integer \leq 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	$1 \leq$ Integer ≤ 6	$11 \leq$ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

A Exponent of forcing function. (Real)

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.

2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN4 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. Nonlinear loads may be a function of displacement ($X_j = u_j$) or velocity ($X_j = \dot{u}_j$). Velocities are denoted by a component number ten greater than the actual component number; i.e., a component of 10 is component 0 (velocity). The velocity is determined by

$$\dot{u}_{j,t} = \frac{u_{j,t} - u_{j,t-1}}{\Delta t}$$

where Δt is the time step interval and $u_{j,t-1}$ is the displacement of GJ-CJ for the previous time step.

5. Use a NOLIN3 entry for the positive range of $X_j(t)$.
6. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
7. RC network solver does not support NOLIN4 for thermal analysis.
8. When enforced motion is used, the NOLIN4 requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
9. An alternate option for averaging nonlinear loads is available with DIAG 10, although it is *not* recommended.

NSM**Non Structural Mass Entry by ID**

Defines a set of non structural mass.

Format:

1	2	3	4	5	6	7	8	9	10
NSM	SID	TYPE	ID	VALUE	ID	VALUE	ID	VALUE	
	ID	VALUE	-etc.-						

Example:

NSM	3	PSHELL	15	.022					
-----	---	--------	----	------	--	--	--	--	--

Descriptor	Meaning
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
ID	Property or Element ID. (Integer > 0)
VALUE	NSM value (Real)

Remarks:

1. Non structural mass sets must be selected with Case Control command NSM = SID.
2. For CCONEAX the element ID is $1000 \cdot ID + i$, where $i = 1$ to number of harmonics.
3. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

NSM1**Alternate Form for NSM Entry**

Defines non structural mass entries by VALUE, ID list.

Format:

1	2	3	4	5	6	7	8	9	10
NSM1	SID	TYPE	VALUE	ID	ID	ID	ID	ID	
	ID	ID	ID	etc.	-				

Example:

NSM1	3	ELEMENT	.044	1240	1500	THRU	1600	BY	
	2	2440	THRU	2560					

Alternate Form and Example(s), applicable when NASTRAN SYSTEM(444)=0 (IFPSTAR=NO):

(All must be in FIELD 5 and non continuation is allowed)

NSM1	SID	TYPE	VALUE	ID	THRU	ID			
NSM1	SID	TYPE	VALUE	ALL					
NSM1	SID	TYPE	VALUE	ID	THRU	ID	BY	N	

Descriptor	Meaning
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
VALUE	NSM value (Real)
ID	Property or Element ID. (Integer > 0 or “ALL” or “THRU” or “BY” or N (the BY increment))

Remarks:

1. Non structural mass sets must be selected with Case Control command NSM = SID.
2. For CCONEAX the element ID is $1000 \cdot ID + i$, where $i = 1$ to number of harmonics.
3. PBEAML and PBCOMP are treated as PBEAM, PBARL is treated as PBAR, and PCOMP or PCOMPG is treated as PSHELL; therefore a command such as:

NSM1,12,PCOMP,0.045,ALL

would for example get all PSHELLs in the file. The converted PCOMPs or PCOMPGs plus any existing PSHELLS would have .045 added to their nonstructural mass.

4. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

NSMADD

Non Structural Mass Set Combination

Defines non structural mass as the sum of the sets listed.

Format:

1	2	3	4	5	6	7	8	9	10
NSMADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	S10	etc.	-				

Example(s):

NSMADD	3	17	18	19	20	22	23	24	
	25	26	27	28					
NSMADD	3	29	40	50	55				

Descriptor	Meaning
SID	Identification number of non structural mass set. (Integer > 0)
Si	Identification numbers of non structural mass sets defined via NSM, NSML, NSM1, and NSML1 entries. (Integer > 0; SID ≠ Si)

Remarks:

1. The non structural mass sets must be selected with the Case Control command NSM = SID.
2. No Si may be the identification number of a non structural mass set defined by another NSMADD entry.
3. NSMADD entries take precedence over NSM, NSML, NSM1 or NSML1 entries. If both have the same set ID, only the NSMADD entry will be used.
4. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

NSML**Lumped Non Structural Mass Entry by ID**

Defines a set of lumped non structural mass.

Format:

1	2	3	4	5	6	7	8	9	10
NSML	SID	TYPE	ID	VALUE	ID	VALUE	ID	VALUE	
	ID	VALUE	-etc.-						

Example:

NSML	3	PSHELL	15	.66					
------	---	--------	----	-----	--	--	--	--	--

Descriptor	Meaning
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
ID	Property or Element ID. (Integer > 0)
VALUE	A lumped mass value to be distributed. (Real)

Remarks:

1. If TYPE = ELEMENT is used, line element (CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD) IDs cannot be mixed with Area element (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D) IDs.
2. For Area elements the calculation is $NSM = \text{VALUE}/\sum_{\text{elements}} \text{Area}$ and for Line elements the calculation is $NSM = \text{VALUE}/\sum_{\text{elements}} \text{Length}$.
3. Non structural mass sets must be selected with Case Control command NSM = SID.
4. This entry is not allowed for the CCONEAX element.
5. This entry will cause an equivalent NSM entry to be generated using the computed value for NSM.
6. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.
7. Undefined property/element IDs are ignored.

NSML1

Alternate Form for NSML Entry

Defines lumped non structural mass entries by VALUE, ID list.

Format:

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ID	ID	ID	ID	ID	
	ID	ID	ID	etc.	-				

Example:

NSML1	3	ELEMENT	.044	1240	1500				
-------	---	---------	------	------	------	--	--	--	--

Alternate Form and Example(s):

NSML1	SID	TYPE	VALUE	ID	THRU	ID	ID	THRU	
	ID	ID	THRU	ID	ID	THRU	ID	ID	
	THRU	ID	...						

NSML1	15	PSHELL	.067	1240	THRU	1760			
	2567	THRU	2568	35689	THRU	40998			
	76	THRU	300						

NSML1	SID	TYPE	VALUE	ID	THRU	ID	BY	N	
	ID	THRU	ID	BY	N	...			

NSML1	3	PSHELL	.067	1240	THRU	1760	1763	1764	
	2567	THRU	2568	35689	TO	40999	BY	2	
	76666	76668	79834						

(ALL must be in FIELD 5 and no continuation is allowed)

NSML1	SID	TYPE	VALUE	ALL					
NSML1	59	PTUBE	.0123	ALL					

Descriptor	Meaning
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)

Descriptor	Meaning
VALUE	A lumped mass value to be distributed (Real)
ID	Property or Element ID. (Integer > 0 or "ALL" or "THRU" or "TO" or "BY" or N (the BY increment))

Remarks:

1. If TYPE = ELEMENT is used, line element (CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD) IDs cannot be mixed with Area element (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D) IDs.
2. For Area elements the calculation is $NSM = VALUE / \sum_{elements} \text{Area}$ and for Line elements the calculation is $NSM = VALUE / \sum_{elements} \text{Length}$.
3. For NSML1 entries with multiple "THRU" and "THRU,BY" and "ID" lists or any such combination of entries, the $NSM = VALUE / \sum_{elements} \text{Area}$ and for Line elements the calculation is $NSM = VALUE / \sum_{elements} \text{Length}$ is based on the individual parent card plus all continuation entries. If an element appears more than once in these multiple combinations, its area or length will be used multiple times in the sum.
4. Nonstructural mass sets must be selected with Case Control command NSM=SID.
5. This entry is not allowed for the CCONEAX element.
6. PBEAML and PCOMP are treated as PBEAM, PBARL is treated as PBAR, and PCOMP or PCOMPG is treated as PSHELL; therefore a command such as:
`NSML1,12,PCOMP,1.35,ALL`
would, for example, get all PSHELLs in the file. The converted PCOMPs or PCOMPGs plus any existing PSHELLS would have a mass of 1.35 added to their nonstructural mass.
7. The ELSUM Case Control command will give a summary of both structural and non structural mass by element or property type.
8. With the "THRU" and "THRU", "BY" forms, blanks fields are allowed for readability. Any combination of a list of IDs and "THRU" and "THRU", "BY" is allowed. The "THRU" and "BY" lists may have missing IDs. That is the list of IDs in a THRU range need not be continuous.
9. Undefined property/element IDs are ignored.

NTHICK**Nodal Thickness Values - SOL 600**

Defines nodal thickness values for beams, plates and/or shells. This is the Marc's nodal thickness option used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
NTHICK	ID1	ID2	THICK						

Example:

NTHICK	151	180	0.255						
--------	-----	-----	-------	--	--	--	--	--	--

Descriptor	Meaning
ID1	First Nodal ID to which the thickness applies. (Integer > 0)
ID2	Last Nodal ID to which the thickness applies. (Integer; Default = EID1)
THICK	Thickness for all beam, plate or shell elements connecting the nodes specified. (Real > 0.0)

Remarks:

1. The option allows specification of beam, plate and/or shell thickness on a nodal basis. Thickness values specified on property entries overrides values specified by this entry.
2. For all elements including composite elements, nodal thickness is the total thickness.
3. Discontinuities must be modeled using property entries.

OMIT**Omitted Degrees-of-Freedom**

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

Format:

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

Example:

OMIT	16	2	23	3516			1	4	
------	----	---	----	------	--	--	---	---	--

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

Remarks:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive o-set. They may not be specified on other entries that define mutually exclusive sets. See “[Degree-of-Freedom Sets](#)” on page 1557 for a list of these entries.
2. Up to 24 degrees-of-freedom may be specified on a single entry.
3. In many cases it may be more convenient to use OMIT1, ASET, or ASET1 entries.
4. In nonlinear analysis, degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, nonlinear degrees-of-freedom must not be specified on OMIT or OMIT1 entries.

OMIT1**Omitted Degrees-of-Freedom, Alternate Form 1**

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
OMIT1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	-etc.-					

Example:

OMIT1	3	2	1	3	10	9	6	5	
	7	8							

Alternate Format and Example:

OMIT1	C	G1	"THRU"	G2					
OMIT1	0	17	THRU	109					

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

Remarks:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive o-set. They may not be specified on other entries that define mutually exclusive sets. See "[Degree-of-Freedom Sets](#)" on page 1557 for a list of these entries.
2. If the alternate format is used, not all points in the range G1 through G2 have to be defined. Undefined points will collectively produce a warning message but will otherwise be ignored.
3. In nonlinear analysis, degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, nonlinear degrees-of-freedom must not be specified on OMIT or OMIT1 entries.

OMITAX

Omitted Conical Shell Degrees-of-Freedom

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

Format:

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

Example:

OMITAX	2	6	3	4	7	1			
--------	---	---	---	---	---	---	--	--	--

Descriptor	Meaning
RID _i	Ring identification number. (Integer > 0)
HID _i	Harmonic identification number. (Integer ≥ 0)
C _i	Component number(s). (Any unique combination of the Integers 1 through 6 with no embedded blanks.)

Remarks:

1. OMITAX is allowed only if an AXIC entry is also present.
2. Up to 12 degrees-of-freedom may be specified on this entry.
3. Degrees-of-freedom appearing on OMITAX entries may not appear on MPCAX, SUPAX, or SPCAX entries.

OUTPUT

Output Control for Adaptive Analysis

Output control for p-adaptive analysis.

Format:

1	2	3	4	5	6	7	8	9	10
OUTPUT	SID								
ELSET=n, cmd1=(option1, option2, etc.), cmd2=(option1, etc.), etc.									
ELSET=m, -etc.-									

Example:

OUTPUT	127								
ELSET=12, DISP=PRINT, STRESS=(PRINT,PUNCH), STRAIN=PUNCH									
ELSET=42, STRESS=PRINT,BY=1									

Descriptor	Meaning	Type	Default
SID	DATAREC ID selected in Case Control. See Remark 1.	Integer > 0	Required
ELSET	ID of SET entry containing sets of elements with results that will be processed. See Remark 1.	Integer > 0	999999
cmdi	Output commands.	See below	
optioni	Specifies one or more of the following output options. The following options may be specified in any order without regard to field boundaries.	See below	Required
DISP	Request for calculating displacements. See Remark 1a.	Character	DISP = PRINT
VELO	Request for calculating velocities. See Remarks 1a., 7., and 8.	Character	VELO = NONE
ACCE	Request for calculating accelerations. See Remarks 1a., 7., and 8.	Character	ACCE = NONE
STRESS	Request for calculating stresses. See Remark 1a.	Character	STRESS = PRINT
STRAIN	Request for calculating strains. See Remark 1a.	Character	STRAIN = NONE
FORCE	Request to output forces/length in shell elements or forces in beam elements.	Character	FORCE = NONE
ERROR	Request for error estimate table. See Remark 1a.	Character	ERROR = PRINT

Descriptor	Meaning	Type	Default
PVAL	Request for new pval values. See Remark 1a.	Character	PVAL = PRINT
LAST	Request to print results of last analysis in an adaptive analysis. See Remark 1b.	Character	LAST = YES
BY	Request to print intermediate results in an adaptive analysis. See Remark 1c.	Integer ≥ 0	BY = 0
FIRST	Request to print results of first analysis in an adaptive analysis. See Remark 1b.	Character	FIRST = YES

Remarks:

1. ELSET = n indicates the start of a new set of commands. Commands appearing after ELSET apply only to elements in SET n.
 - a. For cmdi: DISP, VELO, ACCE, STRESS, STRAIN, FORCE, ERROR, and PVAL the allowable optioni are PRINT, PLOT, PUNCH, REAL, IMAG, PHASE, or NONE. If more than one option is desired, enclose in parentheses; e.g., DISP = (PRINT, PUNCH).
 - b. For cmdi: STRAIN, the allowable optioni are STRCUR, FIBER, PRINT, PLOT, PUNCH, or NONE. If more than one option is desired, enclose in parentheses; e.g., STRAIN = (FIBER, PRINT, PUNCH). The options STRCUR and FIBER are for shell elements, they are ignored for other elements. For STRCUR membrane strain and curvature are output, for FIBER, strains in the fibers Z1 and Z2 are output. Z1 and Z2 are specified on the PSHELL Bulk Data entry. The Default is STRCUR. Either STRCUR or FIBER should be specified, but not both.
 - c. For cmdi: FIRST and LAST, the allowable optioni are YES and NO. For example, "FIRST = YES".
 - d. For cmdi: BY, the allowable optioni is an integer greater than or equal to 0. optioni specifies that cmdi will be processed at every adaptive cycle that is a multiple of optioni.
2. Only the output (displacements, stresses, etc.) requested will be either printed or stored for postprocessing. optioni = PRINT (for print in F06 file), PUNCH (for print in punch file), and PLOT (for calculation to be used by postprocessing but not printing) can be used in any combination. For example, DISP = (PRINT), STRESS = (PRINT,PUNCH) will result in printing of displacement data in the F06 file and printing of the stress data in both the F06 file and the punch file.
3. If an element is specified in more than one ELSET = n, then the union of all commands will be performed on that element.
4. SET = 999999 is a reserved set that includes all elements.
5. A command and its options must be specified entirely on the same entry.
6. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
7. VELO and ACCE output commands are only available for transient and frequency response problems.

8. For modal transient and modal frequency analyses with the default matrix data recovery method, requests of velocity or acceleration output must be accompanied by the displacement request for the same set of elements (ELSET). The complex output formats of displacements, velocities, and accelerations are specified by the REAL, IMAG, or PHASE option of the DISP command.
9. The REAL or IMAG option (the default) is used to request rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
10. The PHASE option is used to request polar format (magnitude and phase) of complex output. Phase output is in degrees.

OUTRCV**Output Options for p-elements**

Defines options for the output of displacements, stresses, and strains of p-elements.

Format:

1	2	3	4	5	6	7	8	9	10
OUTRCV	SID	SETID							
	OPTION1					OPTION2			

Examples:

OUTRCV	150	160							
	CID=2					VIEW=3*3*9			

OUTRCV	3	5							
--------	---	---	--	--	--	--	--	--	--

Descriptor	Meaning	Type	Default
SID	Identification number. SID is selected by the OUTRCV Case Control command.	Integer > 0	Required
SETID	Set identification number of a SET Case Control command that appears after the SETS DEFINITION or OUTPUT(POST) command.	Integer > 0	999999
OPTIONi	Specifies one or more of the following options. The following options may be specified in any order without regard to field boundaries.	See CID and VIEW below	
CID	Specifies the output coordinate system for all stresses, strains, and displacements, except displacements at points defined by GRID entries. CID = 0 specifies the basic coordinate system; and CID = id specifies a CORDij entry. See Remark 4.	Integer ≥ 0	CID = 0
VIEW	Specifies the intervals for displacement, stress, and strain $\xi \cdot \eta \cdot \zeta$ is the number of subdivisions in $\xi \cdot \eta \cdot \zeta$ of the element's output recovery parametric system. See Remark 5.	Three Integers separated by “*”	VIEW = 3*3*3

Descriptor	Meaning	Type	Default
PROJ	Specifies the orientation of a convective coordinate system for shells. PROJ = X specifies the coordinate axis in the CID system which is projected to define the x-axis of the convective coordinate system (tangent system) for shells and beams. Ignored for solids. A minus sign specifies the reverse direction. See Remarks 5. and 10. for more details.	Character; F, X, Y, Z, -X, -Y, -Z	PROJ = X
NORMAL	Specifies the positive direction of the outward normal for shell elements in the CID coordinate system. For NORMAL=R, the positive direction of the outward normal is the exiting arrow side of a radius vector from the origin of the CID system to the element center. For NORMAL = E, the positive direction of the outward normal is the z-axis of the element coordinate system. A minus sign specifies the reverse direction. See Remark 10. for more details.	Character; R, E, -R, -E, X, Y, Z, -X, -Y, -Z	NORMAL = R
THETA	Angle in degrees which rotates the convective system defined with CID and PROJ. THETA is measured in the tangent plane of the shell from the projected axis (selected in PROJ) to the x-axis of the final output coordinate system. For shell elements only.	Real	THETA = 0.

Remarks:

1. OUTRCV is intended for p-elements only and specifies the coordinate system and density used for displacement, stress, strain, and force output. OUTRCV is used only for output and has no effect on the solution.
2. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
3. Sets referenced by SETID are defined on the SET command after the SETS DEFINITION or OUTPUT(POST) command. Any p-element not referenced by the SET = SETID Case Control command will use the defaults listed above for CID and VIEW.
4. If an element is referenced by more than one OUTRCV entry then a warning message will be issued and the last OUTRCV will be applied to the element.

5. $\xi^*\eta^*j$ represents the ξ , η and ζ subdivisions in the solid element's output recovery parametric system. Both "*" delimiters are required. η is ignored for the CPENTA and CTETRA element and ξ is ignored for the CTETRA, CQUAD4, and CTRIA3 element.
6. The elements referenced by the SET = SETID command are labeled in the stress output as VUHEXA, VUPENTA, VUTETRA, VUQUAD, VUTRIA, and VUBEAM. They may be renamed via the PARAM,VUHEXA; PARAM,VUPENTA; PARAM,VUTETRA; PARAM,VUQUAD; PARAM,VUTRIA; and PARAM,VUBEAM entries.
7. Only one OUTRCV Case Control command is allowed. Multiple OUTRCV Bulk Data entries with the same SID are allowed to specify multiple element sets with different output coordinate systems.
8. The displacement output at locations defined by the GRID Bulk Data entry are determined by the CD value located on the GRID Bulk Data entry.
9. For p-version shell elements, the default output coordinates system is the convective coordinate system tangent to the shell mid surface. The x-axis of the convective system is the projected x-axis of the basic system. For p-version beam elements, the output system is the convective coordinate system tangent to the beam axis, oriented from grid A to grid B, as specified on the CBEAM entry.
10. The PROJ and NORMAL options for shells are described in the following list.

PROJ	Defines the orientation of the output coordinate system for stresses, strains and forces in shell elements. The reference system for PROJ is the CID coordinate system.
PROJ = F	Stresses, strains and forces of shells are output in the fixed CID. This option should be used if a postprocessor requires the results in terms of 3D vectors or tensors. For example, stress tensors with 6 components. The option does not produce output in the f06 file.
PROJ = X, Y, Z	The x- or y- or z-axis of the CID system is projected on to the shell tangent plane, the projected vector defines the x-axis of the convective coordinate system for output of stresses, strains and forces.
NORMAL	Specifies the positive normal direction of the output coordinate system for stresses, strains, and forces in shell elements. The reference system for NORMAL is the CID coordinate system.
NORMAL = R	The positive direction of the normal is the exiting arrow of the position vector from the origin of the CID system to the element center.
NORMAL = E	The positive direction of the normal is the z-axis of the element coordinate system.
NORMAL = X	The positive direction of the outward normal is the exiting arrow of the x-axis.
NORMAL = Y,Z	See above.

Entries P

PAABSF

Frequency-Dependent Absorbers Element Property

Format:

1	2	3	4	5	6	7	8	9	10
PAABSF	PID	TZREID	TZIMID	S	A	B	K	RHOC	

Example:

PAABSF	44	38	47						
--------	----	----	----	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number that matches the identification number of the corresponding CAABSF entry. (Integer > 0)
TZREID	Identification number of a TABLEDi entry that defines the resistance as a function of frequency. The real part of the impedance. See Remark 1. (Integer ≥ 0 or blank)
TZIMID	Identification number of a TABLEDi entry that defines the reactance as a function of frequency. The imaginary part of the impedance. See Remark 1. (Integer ≥ 0 or blank)
S	Impedance scale factor. (Real; Default = 1.0)
A	Area factor when 1 or 2 grid points are specified on the CAABSF entry. (Real > 0.0 ; Default = 1.0)
B	Equivalent structural damping coefficient. See Remark 1. (Real ≥ 0.0 ; Default = 0.0)
K	Equivalent structural stiffness coefficient. See Remark 1. (Real ≥ 0.0 ; Default = 0.0)
RHOC	Constant used in data recovery for calculating an absorption coefficient. RHO is the media density, and C is the speed of sound in the media. (Real; Default = 1.0)

Remarks:

- At least one of the four fields TZREID, TZIMID, B, or K must be specified.
- If only one grid point is specified on the CAABSF entry, then the impedance $Z(f) = Z_R + iZ_I$ is the total impedance at the point. If two grids are specified, then the impedance is the impedance per unit length. If three or four points are specified, then the impedance is the impedance per unit area.
 $Z_R(f) = TZREID(f) + B$ and $Z_I(f) = TZIMID(f) - K/(2\pi f)$.
- The resistance represents a damper quantity B. The reactance represents a quantity of the type $(\omega M - K/\omega)$. The impedance is defined as $Z = p/u$ where p is the pressure and u is the velocity.
- The impedance scale factor S is used in computing element stiffness and damping terms as:

$$k = \frac{A}{S} \cdot \frac{2\pi f Z_I(f)}{Z_R^2 + Z_I^2} \int \text{(of shape functions)}$$

$$b = \frac{A}{S} \cdot \frac{Z_R(f)}{Z_R^2 + Z_I^2} \int \text{(of shape functions)}$$

The value of $(Z_R^2 + Z_I^2)$ must be greater than machine epsilon--a machine dependent constant in the neighborhood of 1.E-15. The scale factor S can be used to ensure this constraint while retaining the same units.

5. The output for the element is specified by the STRESS Case Control command and consists of the resistance, reactance, and absorption coefficient. The absorption coefficient is defined as:

$$a = \frac{4(Z_R/\rho c)}{(Z_R/\rho c + 1)^2 + (Z_I/\rho c)^2}$$

PACABS**Acoustic Absorber Property**

Defines the properties of the acoustic absorber element.

Format:

1	2	3	4	5	6	7	8	9	10
PACABS	PID	SYNTH	TID1	TID2	TID3		CUTFR	B	
	K	M							

Example:

PACABS	12		1	2	3	3.5	500.0		
--------	----	--	---	---	---	-----	-------	--	--

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
SYNTH	Request the calculation of B, K, and M from the tables TIDI below. (Character = “YES” or “NO”; Default = “YES”)
TID1	Identification of the TABLEDi entry that defines the resistance. See Remark 2. (Integer > 0 or blank)
TID2	Identification of the TABLEDi entry that defines the reactance. See Remark 2. (Integer > 0 or blank)
TID3	Identification of the TABLEDi entry that defines the weighting function. See Remark 2. (Integer > 0 or blank)
CUTFR	Cutoff frequency for tables referenced above. (Real > 0.0)
B, K, M	Equivalent damping, stiffness and mass values per unit area. (Real ≥ 0.0)

Remarks:

1. PACABS is referenced by a CHACAB entry only.
2. If SYNTH = “YES”, then TID1 and TID2 must be supplied (TID3 is optional) and the equivalent structural model will be derived from tables TIDI. If TID3 is blank, then the weighting function defaults to 1.0.
3. If SYNTH = “NO”, then the equivalent structural model will be derived from one of B, K, or M.
4. The continuation entry is optional.
5. All data defined in tables TIDI must be a function of frequency in cycles/unit time.

PACBAR

Acoustic Barrier Property

Defines the properties of the acoustic barrier element.

Format:

1	2	3	4	5	6	7	8	9	10
PACBAR	PID	MBACK	MSEPTM	FRESON	KRESOM				

Example:

PACBAR	12	1.0	0.01	400.0					
--------	----	-----	------	-------	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MBACK	Mass per unit area of the backing material. (Real > 0.0)
MSEPTM	Mass per unit area of the septum material. (Real > 0.0)
FRESON	Resonant frequency of the sandwich construction in hertz. (Real > 0.0 or blank)
KRESOM	Resonant stiffness of the sandwich construction. (Real > 0.0 or blank)

Remarks:

1. PACBAR is referenced by a CHACBR entry only.
2. Either FRESON or KRESOM must be specified, but not both.

PACINF**Acoustic Conjugate Infinite Element Property**

Defines the properties of acoustic conjugate infinite elements.

Format:

1	2	3	4	5	6	7	8	9	10
PACINF	PID	MID	RIO	XP	YP	ZP			

Example:

PACINF	100	10	5	0.	1.	2.			
--------	-----	----	---	----	----	----	--	--	--

Descriptor	Meaning
PID	Property Identification Number of PACINF entry. (Integer > 0)
MID	Material Identification Number of a MAT10 entry. (Integer > 0)
RIO	Radial Interpolation Order. (Integer > 0; no Default)
XP, YP, ZP	Coordinates of the Pole of the Infinite Elements (in the Basic Coordinate System).

Remark:

1. The location of the pole together with the connecting grid points of the element, define the geometry of the element, see [Figure 9-116](#).

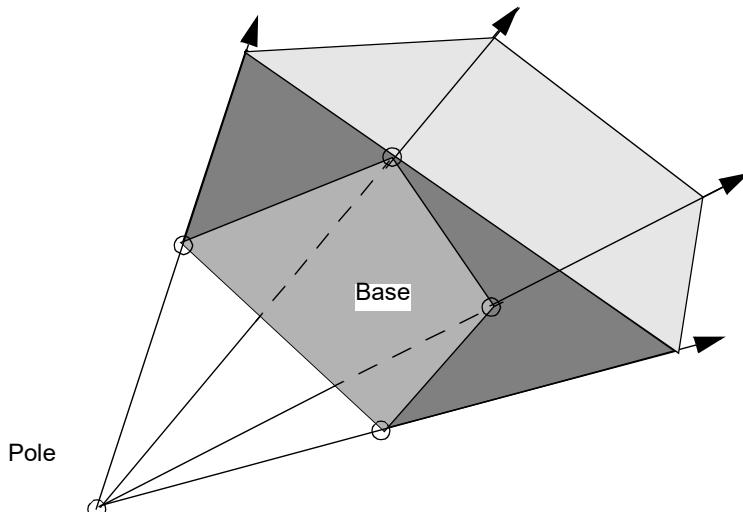


Figure 9-116 Geometry of Infinite Element

2. The radial interpolation order required depends on the directivity of the pressure field.

PAERO1**Aerodynamic Panel Property**

Defines associated bodies for the panels in the Doublet-Lattice method.

Format:

1	2	3	4	5	6	7	8	9	10
PAERO1	PID	B1	B2	B3	B4	B5	B6		

Example:

PAERO1	1	3							
--------	---	---	--	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number referenced by a CAERO1 entry. (Integer > 0)
Bi	Identification number of CAERO2 entries for associated bodies. Embedded blanks are not allowed. (Integer ≥ 0 or blank)

Remarks:

1. The associated bodies must be in the same aerodynamic group, as specified in the IGID field on CAERO2 entry.
2. If there are no bodies, the entry is still required (with Bi fields blank).
3. The Bi numbers above must appear on a CAERO2 entry to define these bodies completely.

PAERO2**Aerodynamic Body Properties**

Defines the cross-sectional properties of aerodynamic bodies.

Format:

1	2	3	4	5	6	7	8	9	10
PAERO2	PID	ORIENT	WIDTH	AR	LRSB	LRIB	LTH1	LTH2	
	THI1	THN1	THI2	THN2	THI3	THN3			

Example:

PAERO2	2	Z	6.0	1.0	22	91	100		
	1	3							

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
ORIENT	Orientation flag. Type of motion allowed for bodies. Refers to the aerodynamic coordinate system of ACSID. See AERO entry. (Character = "Z", "Y", or "ZY")
WIDTH	Reference half-width of body and the half-width of the constant width interference tube. (Real > 0.0)
AR	Aspect ratio of the interference tube (height/width). (Real > 0.0)
LRSB	Identification number of an AEFACT entry containing a list of slender body half-widths at the end points of the slender body elements. If blank or zero, the value of WIDTH will be used. (Integer ≥ 0 or blank)
LRIB	Identification number of an AEFACT entry containing a list of interference body half-widths at the end points of the interference elements. If blank or zero, the value of WIDTH will be used. (Integer ≥ 0 or blank)
LTH1, LTH2	Identification number of AEFACT entries for defining θ arrays for interference calculations. (Integer ≥ 0)
THIi, THNi	The first and last interference element of a body to use the θ_1 array; the others use the θ_2 array. (Integer ≥ 0)

Remarks:

1. The half-widths (given on AEFACT entries referenced in fields 6 and 7) are specified at division points. The number of entries on an AEFACT entry used to specify half-widths must be one greater than the number of elements.
2. The half-width at the first point (i.e., the nose) on a slender body is usually 0.0; thus, it is recommended (but not required) that the LRSB data is supplied with a zero first value.
3. THIi and THNi are interference element numbers on a body. The first element is one for each body.

4. A body is represented by a slender body surrounded by an interference tube. The slender body creates the downwash due to the motion of the body, while the interference tube represents the effects upon panels and other bodies.

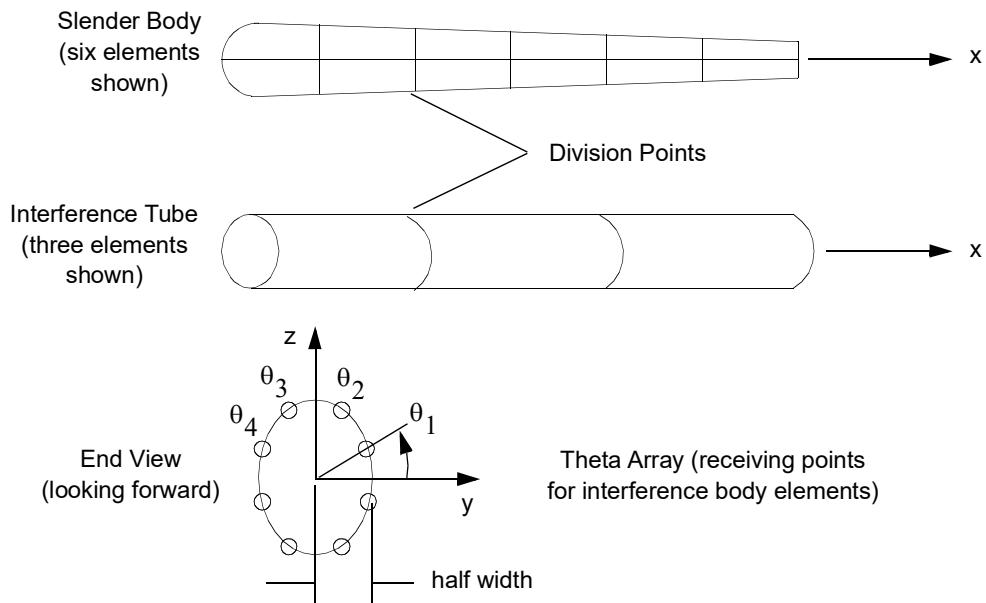


Figure 9-117 Idealization of Aerodynamic Body

5. The angles θ_1 and θ_2 are input in degrees using the aerodynamic element coordinate system as the reference coordinate system for defining the theta points.
6. Distribution of the theta points need not be uniform. A theta point must be placed a finite distance from any aerodynamic box edge; preferably the box edge would be equidistant from any two theta points. This aerodynamic coordinate system is defined on the AERO Bulk Data entry.
7. For half models, the theta arrays LTH1 and LTH2 should encompass a full 360 degree range.

PAERO3**Aerodynamic Panel Property**

Defines the number of Mach boxes in the flow direction and the location of cranks and control surfaces of a Mach box lifting surface.

Format:

1	2	3	4	5	6	7	8	9	10
PAERO3	PID	NBOX	NCTRL		X5	Y5	X6	Y6	
	X7	Y7	X8	Y8	X9	Y9	X10	Y10	
	X11	Y11	X12	Y12					

Example:

PAERO3	2001	15	1		0.	65.			
	78.	65.	108.	65.	82.	97.5	112.	97.5	
	86.	130.	116.	130.					

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
NBOX	Number of Mach boxes in the flow direction. (0 < Integer < 50)
NCTRL	Number of control surfaces. (Integer 0, 1, or 2)
X5 through Y12	Locations of points 5 through 12, which are in the aerodynamic coordinate system, to define the cranks and control surface geometry. (Real)

Remarks:

- For an illustration of the geometry, see the CAERO3 entry description.
- If $Y5 \leq 0.0$, there is no leading edge crank. Also, if $Y6 \leq 0.0$, there is no trailing edge crank.
- If $NCTRL = 0$, no continuations are required. If $NCTRL = 1$ or 2 , then NCTRL continuations are required.
- $Y7 \geq Y8, Y9 \geq Y10$, and $Y11 \geq Y12$.
- The number of Mach boxes in the spanwise direction (NSB) may be found from the following formula:

$$NSB = \text{INT} \left[\frac{\beta \cdot y_{max}}{\left(\frac{x_{max}}{NBOX + 0.5} \right)} + 0.5 \right]$$

where:

$$\beta = \sqrt{M^2 - 1}$$

x_{max} = maximum chordwise direction

y_{max} = maximum spanwise direction

NBOX = initial number of boxes specified in field 3

The number of Mach boxes in the streamwise direction may then be computed from:

$$\text{NBOX} = \text{INT} \left[\frac{x_{max}}{\left(\frac{\beta \cdot y_{max}}{NSB - 0.5} \right)} + 0.999 \right]$$

The number of chordwise boxes specified by the user ($\text{NBOX} \geq 50$) will be replaced by a floating point number (usually slightly higher than NBOX). The method contracts the mesh equally in both dimensions until a box edge lies on the surface tip. This mesh size is then used to compute the number of chordwise boxes.

Note:

A minimum of seven Mach boxes in the flow direction (NBOX) is recommended.

PAERO4**Aerodynamic Strip Properties**

Defines properties of each strip element for Strip theory.

Format:

1	2	3	4	5	6	7	8	9	10
PAERO4	PID	CLA	LCLA	CIRC	LCIRC	DOC1	CAOC1	GAPOC1	
	DOC2	CAOC2	GAPOC2	DOC3	CAOC3	GAPOC3	-etc.-		

Example:

PAERO4	6001	1	501	0	0	0.0	0.0	0.0	
	0.50	0.25	0.02	0.53	0.24	0.0			

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
CLA	Select Prandtl-Glauert correction. (Integer = -1, 0, 1; Default = 0) <ul style="list-style-type: none"> -1 Compressibility correction made to lift curve slope data for a reference Mach number. 0 No correction and no list needed. (Default) +1 No correction and lift curve slope provided by a list as a function of strip location and Mach number.
LCLA	Identification number of the AEFFACT entry that lists the lift curve slope on all strips for each Mach number on the MKAEROi entry. See Remark 7(b.) below. (Integer = 0 if CLA = 0, > 0 if CLA ≠ 0)
CIRC	Select Theodorsen's function $C(k)$ or the number of exponential coefficients used to approximate $C(k)$. (Integer = 0, 1, 2, 3; Default = 0; Must be zero if CLA ≠ 0.) <ul style="list-style-type: none"> 0 Theodorsen function. 1, 2, 3 Approximate function with $b_0, b_1, \beta_1, \dots, b_n, \beta_n$ $n = 1, 2, 3$.
LCIRC	Identification number of the AEFFACT entry that lists the b, β values for each Mach number. See Remark 7c., 7d., and 7e. below; variable b 's and β 's for each mi on the MKAEROi entry. (Integer = 0 if CIRC = 0, > 0 if CIRC ≠ 0)
DOCi	d/c distance of the control surface hinge aft of the quarter-chord divided by the strip chord. (Real ≥ 0.0)
CAOCi	c_a/c control surface chord divided by the strip chord. (Real ≥ 0.0)
GAPOCi	g/c control surface gap divided by the strip chord. (Real ≥ 0.0)

Remarks:

- PAERO4 is required for Strip theory with three fields (DOCi, CAOCi, GAPOCi) specified per strip.

2. If CLA = -1, lift curve slope data at one Mach number are needed on the AEFACT entry.
3. If CAOCi = 0.0, there is no control surface.
4. If GAPOCivb = 0.0, there is no slot flow.
5. If GAPOCi < 0.01, then 0.01 is used.
6. Embedded blank fields are not permitted.
7. [Table 9-28](#) lists the lift curve slope or lag function selection and the AEFACT entry formats used for Strip theory:

Table 9-28 Strip Theory Function Selections and AEFACT Entry Formats

Theodorsen Function	Data Type Input	Parameter Combinations				Number of Words	Entry Format Index
		CLA	LCLA	CIRC	LCIRC		
Exact	Lift Curve Slope $c_{l\alpha_i} = 2\pi$	0	0	0	0	No AEFACT entry required	
	$c_{l\alpha_i}$ Input, Uses Prandtl-Glauert Correction	-1	ID	0	0	(NSTRIP+1)	a.
	$c_{l\alpha_i}$ Input, for All m's on MKAERO Entry	1	ID	0	0	(NSTRIP+1)*NMACH	b.
Approximate Coefficients	$b_{0i}, b_{1i}, \dots, b_{li}$, etc.	0	0	1	ID	4*NMACH	c.
		0	0	2	ID	6*NMACH	d.
		0	0	3	ID	8*NMACH	e.

Entry Format

- a. AEFACT, ID, $m_1, c_{l\alpha_1}, c_{l\alpha_2}, \dots, c_{l\alpha_{NSTRIP}}$
- b. AEFACT, ID, $m_1, c_{l\alpha_{11}}, c_{l\alpha_{21}}, \dots, c_{l\alpha_{NSTRIP1}}, m_2, c_{l\alpha_{11}}, c_{l\alpha_{12}}, c_{l\alpha_{21}}, c_{l\alpha_{22}}, \dots, c_{l\alpha_{NSTRIP1}}, c_{l\alpha_{NSTRIP2}}$, for all m on MKAEROi data entry
- c. AEFACT, ID, $m_1, b_{01}, b_{11}, \beta_{11}, m_2, b_{02}, b_{12}, P_{12}, m_3$, etc.
- d. AEFACT, ID, $m_1, b_{01}, b_{11}, \beta_{11}, b_{21}, \beta_{21}, m_2$, etc.
- e. AEFACT, ID, $m_1, b_{01}, b_{11}, \beta_{11}, b_{21}, \beta_{21}, b_{31}, \beta_{31}, m_2$ etc.
8. A control surface rotation is positive when the trailing edge moves in the negative z-direction of the aerodynamic element coordinate system; see the *MSC Nastran: Aeroelastic Analysis User's Guide*.

PAERO5**Aerodynamic Panel Property**

Defines properties of each strip element for Piston theory.

Format:

1	2	3	4	5	6	7	8	9	10
PAERO5	PID	NALPHA	LALPHA	NXIS	LXIS	NTAUS	LTAUS		
	CAOC1	CAOC2	CAOC3	CAOC4	CAOC5				

Example:

PAERO5	7001	1	702	1	701	1	700		
	0.0	0.0	5.25	3.99375	0.0				

Describer	Meaning
PID	Property identification number. (Unique Integer > 0)
NALPHA	Number of angle of attack (α) values to be input for each Mach number (mi) on the MKAERO1 or MKAERO2 entry. (Integer > 0)

NALPHA	Meaning
1	α is the same value for all strips; enter one value, in units of degrees, on the AEFACT entry for each Mach number.
Number of Strips	α is different for each strip; enter α 's, in units of degrees, in the following order: $m_1, \alpha_1, \alpha_2, \dots, m_2, \alpha_1, \alpha_2, \dots$, etc.

LALPHA	ID number of the AEFACT entry that lists the α 's for the strips at each Mach number in the MKAERO1 or MKAERO2 entry. (Integer > 0)
NXIS	Number of dimensionless chord coordinates (ξ) to be input. (Integer ≥ 0 , Default = 0)

NXIS	Meaning
0	No ξ 's are required. (Default)
1	ξ 's are the same for all strips; enter values for one strip on the AEFACT entry (ξ_h if NTHICK > 0, or ξ_m and ξ_h if NTHICK = 0)
Number of Strips	ξ 's have to be input for each strip ($\xi_{h1}, \xi_{h2}, \dots, \xi_{hNSPAN}$, if NTHICK > 0, or $\xi_{m1}, \xi_{h1}, \xi_{m2}, \xi_{h2}, \dots, \xi_{mNSPAN}, \xi_{hNSPAN}$, β_{hNSPAN} if NTHICK = 0)

LXIS	Identification number of AEFACT entry that lists the ξ values for the strip in order indicated by values of NXIS and NTHICK. (Integer = 0 if $c_a = 0$ and NTHICK > 0 or LXIS>0 if $c_a = 0$ and/or NTHICK = 0)
NTAUS	Parameter used to select the number of thickness ratio (τ) values to be input. (Integer ≥ 0 , Default = 0)

NTAUS	Meaning
0	No τ 's are required. (Default)
1	τ 's are the same for all strips; enter $(\tau_1, \tau_{h1}, \tau_{t1})$ values for one strip on AEFACT entry.
Number of Strips	τ 's must be input for each strip on an AEFACT entry in the following order: $(\tau_1, \tau_{h1}, \tau_{t1}, \tau_2, \tau_{h2}, \tau_{t2}, \dots, \tau_{NSPAN}, \tau_{hNSPAN}, \tau_{tNSPAN})$

LTAUS	Identification number of AEFACT entry that lists the τ values for the strips. (Integer = 0 or blank if NTAUS = 0, LTAUS > 0 if NTAUS > 0)
CAOCi	c_a/c = control surface chord divided by the strip chord. (Real ≥ 0.0)

Remarks:

1. The continuation entry is required for Piston theory with one entry (CAOCi) per strip.
2. Embedded blank fields are not allowed on the continuation entry.
3. If CAOCi = 0.0, there is no control surface.
4. [Table 9-29](#) lists the thickness data input and AEFACT entry format used for Piston theory.

Table 9-29 Thickness Data Input and AEFACT Entry Format for Piston Theory

Type of Input	Parameter Combinations						Number of Words	Entry Format Index
	CAOG	NGHICK	NXIS	LXIS	NTAUS	LTAUS		
No control surfaces, Integrals input are same for all strips	0.	ID ^(a)	0	0	0	0	6	a.
With control surfaces, Integrals input, same hinge on all strips	$\neq 0$	ID ^(b)	1	ID ^(c)	0	0	12 1	b. c.
With control surfaces, Integrals input, variable hinge	$\neq 0$	ID ^(b)	NSTRIP	ID ^(d)	0	0	12 NSTRIP	b. d.
No control surfaces, thickness inputs are same for all strips	0.0	0	1	ID ^(f)	1	ID ^(e)	3 2	e. f.
With control surfaces, thickness inputs are same for all strips	$\neq 0.0$	0	1	ID ^(f)	1	ID ^(e)	3 2	e. f.
With control surfaces, thickness inputs vary for strips	$\neq 0.0$	0	NSTRIP	ID ^(h)	NSTRIP	ID ^(g)	3*NSTRIP 2*NSTRIP	g. h.

Entry Format

- a. AEFACT, ID, $I_1, I_2, I_3, I_4, I_5, I_6$
- b. AEFACT, ID, $I_1, \dots, I_6, J_1, \dots, J_6, I_1, I_2, I_3, I_4, I_5, I_6$
- c. AEFACT, ID, ξ_h
- d. AEFACT, ID, $\xi_{h1}, \xi_{h2}, \xi_{h3}, \dots, \xi_{hNSTRIP}$
- e. AEFACT, ID, τ_m, τ_h, τ_t
- f. AEFACT, ID, ξ_m, ξ_h
- g. AEFACT, ID, $\tau_{m1}, \tau_{h1}, \tau_{t1}, \tau_{m2}, \tau_{h2}, \tau_{t2}, \dots, \tau_{mNSTRIP}, \tau_{hNSTRIP}, \tau_{tNSTRIP}$
 $\tau_{m1}, \tau_{h1}, \tau_{t1}, \tau_{m2}, \tau_{h2}, \tau_{t2}, \dots, \tau_{mNSTRIP}, \tau_{hNSTRIP}, \tau_{tNSTRIP}$
- h. AEFACT, ID, $\xi_{m1}, \xi_{h1}, \xi_{m2}, \xi_{h2}, \dots, \xi_{mNSTRIP}, \xi_{hNSTRIP}$

5. The following table lists the angle-of-attack distribution and AEFACT entry formats used for Piston theory.

Type of Distribution	Parameter Combinations		Number of Words	Entry Format Index
	NALPHA	LALPHA		
Equal angle of attack on all strips	1	ID	2*NMACH	a.
Unequal angle of attack	NSTRIP	ID	(1 + NSTRIP) * NMACH	b.

Entry Format

- a. AEFACT, ID, $m_1, \alpha_1, m_2, \alpha_2, \dots,$
- b. AEFACT, ID, $m_1, \alpha_{11}, \alpha_{21}, \alpha_{31}, \dots, \alpha_{NSTRIP1}, m_2, \alpha_{12}, \alpha_{22}, \dots, \alpha_{NSTRIP2}, m_2,$ etc., for all m on MKAEROi entry.
- c. A control surface rotation is positive when the trailing edge moves in the negative z-direction of the aerodynamic element coordinate system; see the *MSC Nastran: Aeroelastic Analysis User's Guide*.

PANEL**Panel Definition for Coupled Fluid-Structural Analysis**

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

Format:

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

Example:

PANEL	BKDOOR	103							
-------	--------	-----	--	--	--	--	--	--	--

Descriptor	Meaning
NAMEi	Panel label. (Character)
SETIDi	Identification number of a SET1 or SET3 entry that lists the grid points, elements or properties of the panel. On this entry, the SET1 should only be used for grid definition. (Integer > 0)

Remarks:

1. If a set of grid points is referenced, the set must list only structural grid points.
2. If an element is assigned to a panel, it is recommended that all of its connection points belong to the same panel.
3. If a set of elements is referenced, the set must list only structural elements. The panel will consist of all grid points that are connection points of these elements.
4. If a set of property identifiers is referenced, the properties must be referenced by structural elements. The panel will consist of all grid points that are connection points of elements referencing one of the properties contained in the set.
5. NAMEi is used only for labeling the output of the panel participation factors (cf. the description of the PFMODE and PFPANEL Case Control commands).

PARAM

Parameter

Specifies values for parameters used in solution sequences or user-written DMAP programs.

Format:

1	2	3	4	5	6	7	8	9	10
PARAM	N	V1	V2						

Example:

PARAM	IRES	1							
-------	------	---	--	--	--	--	--	--	--

Descriptor	Meaning
N	Parameter name (one to eight alphanumeric characters, the first of which is alphabetic).
V1, V2	Parameter value based on parameter type, as follows:

Type	V1	V2
Integer	Integer	Blank
Real, single-precision	Real	Blank
Character	Character	Blank
Real, double-precision	Double-precision real	Blank
Complex, single-precision	Real or blank	Real or blank
Complex, double-precision	Double-precision real	Double-precision real

Remarks:

- See [Parameters, 783](#) for a list of parameters used in solution sequences that may be set by the user on PARAM entries.
- If the large field entry format is used, the second physical entry line must be present, even though fields 6 through 9 are blank except for SOL 700.
- The first 8 characters of N must be unique. N with more than 8 characters is normally only used by SOL 700.
- If the Bulk Data involves the use of part superelements or external superelements, the following points should be noted regarding the use of the PARAM Bulk Data entry:
 - PARAM entries specified in the Main Bulk Data portion of the input data apply *only to the residual and not to the part superelements or external superelements*.
 - PARAM entries specified in the BEGIN SUPER portion of the Bulk Data for a part superelement or an external superelement apply *only to that superelement*.

- c. The most convenient way of ensuring that PARAM entries apply not only to the residual, but also to all part superelements and external superelements is to specify such PARAM entries in Case Control, not in the Main Bulk Data. This is particularly relevant for such PARAMs as POST.
- 5. If Modules are present then this entry may only be specified in the main Bulk Data section.

PARAMARC

Parallel Domain Decomposition in Marc from SOL 600

Specifies parallel regions for domain decomposition in nonlinear analysis when Marc is executed from MSC Nastran. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
PARAMARC	ID	KIND	NPROC	IDSOLVE	ACCUR				
	IDP	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	IDP	ID8	ID9	etc.					

Example: To create 4 parallel processes using Marc's single file input

PARAMARC	51		4	0	1.0E-4				
----------	----	--	---	---	--------	--	--	--	--

Example: To create 2 parallel processes by specifying element numbers

PARAMARC	201	2	2						
	1	1	THRU	5000	BY	1			
	2	5001	5002	5005	5010	5012	5013	5015	
	2	5020	5030	5040	5050				

Descriptor	Meaning
ID	Identification number of the PARAMARC entry -- Not presently used. (Integer)
KIND	Designates how parallel domains are created. (Integer = 0 or Blank; Default = 0)
0	Parallel processing is accomplished using Marc's single file input. (Marc Version 2005 or subsequent must be used. The command line to execute Marc is changed from -np N (or -nproc N) to -nps N where N is the number of processors. The maximum number of processors for Marc is 256. Continuation lines may not be entered for KIND=0.)
2	Parallel domains will be specified by the user and the continuation lines are required.
NPROC	Number of parallel processes requested. See Remark 11. (Integer; Default = 0)
IDSOLV	Inter-domain solution type. See Remark 11. (Integer; Default = 0)
0	Iterative Solver
1	Direct Solver
ACCUR	Inter-domain solve accuracy. See Remark 11. (Real; Default = 1.0E-2) For an accurate solution of difficult nonlinear models, ACCUR should be set to 1.0E-4.

Describer	Meaning
IDP	Parallel process ID for following group of elements or grid ID's. See Remark 5. (Integer > 0; Required)
ID(i)	Element number (if KIND=2). (Integer > 0; Required if KIND=2)
THRU	Enter THRU in field 4 (may be omitted if I2 and INC are not required. The word TO instead of THRU may also be used.)
ID2	Ending element number. (Integer > 0)
BY	Enter BY in field 6 (may be omitted in INC=1)
INC	Increment ID's. (Integer; Default = 1, may be negative but not zero)

Remarks:

1. The PARAMARC entry is recognized only when Marc is executed from within SOL 600.
2. Use of KIND=2 provides more control than does the automatic (KIND=0) option.
3. If parallel jobs are run on different computers across a network, as opposed to using multiple processors in the same box, a host file is normally needed. Consult MSC technical support to determine how to setup a host file for your computer system. Use of the host file is triggered by Bulk Data PARAM,MARCHHOST,Name.
4. Continuation lines should not be entered unless KIND=2.
5. The continuation entries should be entered as many times as necessary to completely define each parallel region for KIND=2.
6. The string TO may be substituted for THRU if so desired.
7. For PC Windows systems, the default type of MPI for SOL 600 is Intel MPI. The first time a parallel job is run, the user may be prompted for domain\user_name and then for password. Since MSC Nastran is a batch process, the user will not normally see the prompts and the job may appear to hang. If the job appears to hang, carefully enter the following information in exactly the same way you enter it to login into your PC:


```
domain\user_name
      password
```
8. If multiple computers are used across a network, all computers must normally be the same type of computer, run the same operating system, be in the same domain, have the same user name and passwords. Also, a host file is required to describe the machines to be used. Further details are provided in the *SOL 600 Parallel Processing User's Guide*.
9. See PARAM,MRPARALL for additional notes concerning SOL 600 parallel processing.
10. If running on a PC, see PARAM,MARCMPII for options to keep the small MPI service (for example, ismpd.exe for Intel MPI) running or not after the Marc portion of the job has completed.
11. Parallel processing for SOL 600 is accomplished using domain decomposition. A solution of each domain is performed using its own processor. When each of the individual processor solutions converge, the overall solution at each increment is put together from the domains and iterated until convergence occurs. Variables IDSOLV and ACCUR control how this final iteration is performed

and its accuracy. IDSOLV=0 (iterative solver) will usually be faster than IDSOLV=1 (direct solver), however it is sometimes necessary that ACCUR be reduced to 1.0E-3 or 1.0E-4 to obtain sufficient accuracy for difficult nonlinear models, particularly those with contact, post-buckling, or large strain plasticity.

PAXISYM

Properties of Axisymmetric Line Element - SOL 400

Defines the properties of axisymmetric line elements.

Format:

1	2	3	4	5	6	7	8	9	10
PAXISYM	PID	MID		T1	T2	ANAL			

Example:

PAXISYM	98	17		0.1					
---------	----	----	--	-----	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number of a CAXISYM entry. (Integer > 0)
MID	Identification number of a MAT1, MAT2, MAT3, MAT8, MATORT, or MATHE entry. Identification number of a MAT4 or MAT5 for heat transfer. See Remark 3. (Integer > 0)
T1	Thickness at G1. (Real > 0)
T2	Thickness at G2. (Real \geq 0)
ANAL	Analysis type, IS=structural, IH=heat, ISH=structural-heat (CHAR Default=ISH)

Remarks:

1. PAXISYM identification entries should be unique with respect to all other property entries.
2. For elements with only two grids, only T1 is applicable. For elements with three grids, if T2 is blank or 0.0 then constant thickness is assumed.
3. The MID entry may point to MAT1, MAT2, MAT3, MAT8, MATORT, or MATHE entries for structures and MAT4 or MAT5 entries for heat transfer. The tables below show associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Implicit Structural Materials					
MAT1	MAT2	MAT3	MAT8	MATORT	MATHE
MATVE	<MATVE>	<MATVE>	<MATVE>	<MATVE>	MATVE
MATVP	MATVP	MATVP		MATVP	
MATEP	MATEP	MATEP	MATEP	MATEP	
MATF	MATF	MATF	MATF	MATF	
MATS1		MATS3	MATS8	MATSORT	

<MATVE> refers to the ALTERNATE format for type ORTHO

Heat Materials	
MAT4	
	MAT5

MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries. MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

4. The CAXISYM element uses PLOADX1 for loading.
5. The element does not support composite materials, via PCOMP or PCOMPG.

PAXSYMH**Linear Axisymmetric Harmonic Element Properties**

Defines the properties of a linear axisymmetric harmonic element.

Format:

1	2	3	4	5	6	7	8	9	10
PAXSYMH	PID	MID	CID	NHARM	INT				

Example:

PAXSYMH	100	10	5						
---------	-----	----	---	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number. (Integer > 0). See Remarks 1. and 5.
MID	Identification number of MAT1 or MAT9 entry. (Integer > 0).
CID	Identification number of element coordinate system. (Integer ≥ 0 ; Default = 0). See Remark 2..
NHARM	Harmonic index. (Integer ≥ 0 ; Default = 1). See Remark 3..
INT	Integration scheme. (Integer 0, 1, 2 or 3; Default = 0). See Remark 4.

Remarks:

1. This entry is referenced by the CQUADX and CTRIA9 entries.
2. The Y-axis of CID defines the axis of symmetry while its X-Y plane defines the two-dimensional plane of the axisymmetric harmonic element. The X-axis of CID thus defines the radial direction.
3. The gyroscopic matrix is generated for the element only for the case of NHARM = 1 (the default).
4. The integration scheme INT selects the number of Gauss points used for matrix generation as indicated below:

INT	Order	Gauss Points	Gauss Points
		CQUADX	CTRIAX
1	1x1	1	1
2	2x2	4	3
3	3x3	9	7

INT=0 uses INT=2 (no mid-side nodes) or INT=3 (mid-side nodes). The default INT=0 is recommended. The use of INT=1 will produce a singular stiffness and is meant for experimental use only.

5. PAXSYMH is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PAXSYMH property entries have unique identification numbers with respect to all other property entries, else unexpected grouping results may occur. There must be uniqueness among PAXSYMH, PLPLANE and PSHELL entries.

PBAR**Simple Beam Property**

Defines the properties of a simple beam element (CBAR entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2	I12						

Example:

PBAR	39	6	2.9		5.97				
				2.0	4.0				

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 2. and 3. (Integer > 0)
A	Area of bar cross section. (Real; Default = 0.0)
I1, I2, I12	Area moments of inertia. See Figure 9-118 . (Real; $I1 \geq 0.0$, $I2 \geq 0.0$, $I1*I2 \geq I12^2$; Default = 0.0)
J	Torsional constant. See Figure 9-118 . (Real; Default = $\frac{1}{2}(I_1 + I_2)$ for SOL 600 and 0.0 for all other solution sequences)
NSM	Nonstructural mass per unit length. (Real)
Ci, Di, Ei, Fi	Stress recovery coefficients. (Real; Default = 0.0)
K1, K2	Area factor for shear. See Remark 5. (Real or blank)

Remarks:

- Both continuation entries may be omitted.
- For structural problems, MID must reference a MAT1 material entry.
- For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
- See the CBAR entry description for a discussion of bar element geometry.
- The transverse shear stiffnesses times unit length in planes 1 and 2 are $K1*A*G$ and $K2*A*G$, respectively, where G is the shear modulus. The default values for K1 and K2 are infinite; in other words, the transverse shear flexibilities are set equal to zero. K1 and K2 are ignored if $I12 \neq 0$. K1 and K2 must be blank if A = 0.0.

6. The stress recovery coefficients C1 and C2, etc., are the y and z coordinates in the bar element coordinate system of a point at which stresses are computed. Stresses are computed at both ends of the bar. For conventional element, only bending components of strain and stress are outputted at points C1 and C2, etc. Membrane components of strain and stress are outputted as axial value. For advanced nonlinear element, the bending and membrane components of strain and stress are superposed and outputted together at the points C1 and C2, etc., there is no axial value individually.
7. For response spectra analysis on stress recovery coefficients, the CBEAM element entry should be used because bar element results will be inaccurate.
8. [Figure 9-118](#) describes the PBAR element coordinate system.

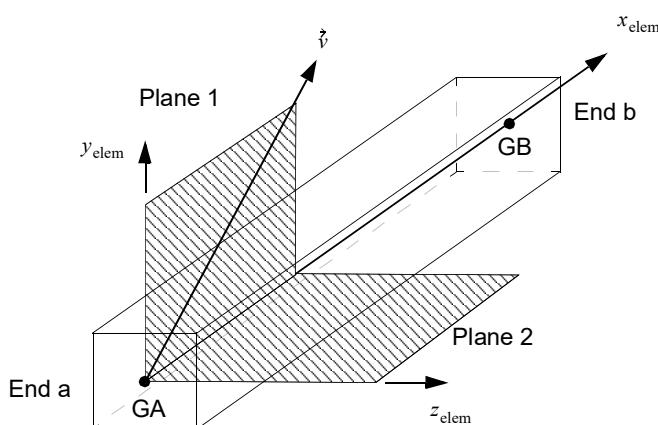
where:

$$I1 = I_{zz_{\text{elem}}}$$

$$I2 = I_{yy_{\text{elem}}}$$

$$I12 = I_{zy_{\text{elem}}}$$

$$J = I_{xx_{\text{elem}}}$$



[Figure 9-118](#) PBAR Element Coordinate System

9. For cross-sections that are not doubly symmetric, when a beam is loaded through the centroid, it may in addition to bending undergo rotation. The CBAR element, by default, does not represent this behavior because the shear center is not explicitly accounted for on a CBAR entry.

By definition, for the CBAR element, the load is applied at the centroid and not at the shear center. If load application at the shear center is desired, appropriately applied offsets can be used on the CBAR entry or the CBAR/PBAR element replaced by the CBEAM/PBEAM element which by default explicitly applies the load through the shear center.

If warping considerations and bi-moment calculations are important, the CBEAM/PBEAM element should be used.

10. Mass moment of inertial formulation has changed in Version 2003. System (398) may be used to select the formulation in pre-Version 2004 systems.
11. PBAR is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBAR property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBAR, PBARL, PBRSECT entries.

PBART

Simple Beam Cross-Section Property

Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.

Format:

1	2	3	4	5	6	7	8	9	10
PBART	PID	MID	GROUP	TYPE					
	DIM1	DIM2	DIM3	DIM4	DIM5	DIM6	DIM7	DIM8	
	DIM9	-etc.-	NSM						

Example:

PBART	39	6		I					
	14.	6.	.5	.5	.5	.5	.2		

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GROUP	Cross-section group. See Remarks 6. and 8. (Character; Default = "MSCBMLO")
TYPE	Cross-section type. See Remarks 6. and 8. and Figure 9-119. (Character: "ROD", "TUBE", "TUBE2", "I", "CHAN", "T", "BOX", "BAR", "CROSS", "H", "T1", "I1", "CHAN1", "Z", "CHAN2", "T2", "BOX1", "HEXA", "HAT", "HAT1", "DBOX" for GROUP="MSCBMLO")
DIMi	Cross-sectional dimensions. (Real > 0.0 for GROUP = "MSCBMLO")
NSM	Nonstructural mass per unit length. NSM is specified after the last DIMi. (Default = 0.0)

Remarks:

- For structural problems, PBART entries must reference a MAT1 material entry.
- PBART is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBART property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBAR, PBART, PBRSECT entries.
- See CBAR entry for a discussion of bar element geometry.
- For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.
- For response spectra analysis on stress recovery coefficients, the CBEAM element should be used because results for the CBAR element will not be accurate.

6. The GROUP is associated with an FMS CONNECT statement that specifies the evaluator. A reserved GROUP name is “MSCBML0”. Users may create their own cross-section types. Each of the types will require one or more subroutines to convert DIMi information to geometric property information contained on a PBAR entry and optimization information. See [Building and Using the Sample Programs](#) in the *MSC Nastran Utilities Guide* for a discussion on how to include a user-defined beam library.
7. A function of this entry is to derive an equivalent PBAR entry. Any sorted echo request will also cause printout and/or punch of the derived PBAR.
8. For GROUP = “MSCBML0”, the cross-sectional properties, shear flexibility factors, and stress recovery points (C, D, E, and F) are computed using the TYPE and DIMi as shown in [Figure 9-119](#) through [Figure 9-122](#). The figures show the origin of the cross section, but the PBARL does not account for differences between the shear center and the neutral axis and the properties are computed relative to the neutral axis. This results in approximations for section types ‘T’, ‘CHAN’, ‘T’, ‘CHAN1’, ‘T1’, ‘CHAN2’, ‘T2’, ‘L’ and ‘BOXI’. The PBEAML provides a better representation of section properties in these cases and is recommended.

The PBARL does not account for offsets between the neutral axis and the shear center. Therefore, the CHAN, CHAN1 and CHAN2 cross-sections may produce incorrect results. The PBEAML is recommended.

9. For DBOX section, the default value for DIM5 to DIM10 are based on the following rules:
 - a. DIM5, DIM6, DIM7 and DIM8 have a default value of DIM4 if not provided.
 - b. DIM9 and DIM10 have a default value of DIM6 if not provided.

Note:

The above default value rules for DIM5 to DIM10 are not applicable to design optimization property value update.

10. The finite element formulation (FEF) utilized for arbitrary beam cross section is selected as the default method for computing sectional properties for all supported cross section types of PBARL when GROUP=MSCBML0. The original beam equations which are based on thin-walled assumptions can be accessed via Bulk Data entry ‘MDLPRM,TWBRBML,1’. Thus the shear stiffness factors K1 and K2 are calculated as in Remark 11. of the PBMSECT entry and the element will behave according to the Timoshenko beam theory by default. Thus the resulting CBAR will have non-infinite K1/K2 shear flexibility factors, and therefore will not behave as a Bernoulli-Euler.
11. For optimization, individual DIMx of PBARL can be selected as designed properties even with the finite element formulation.

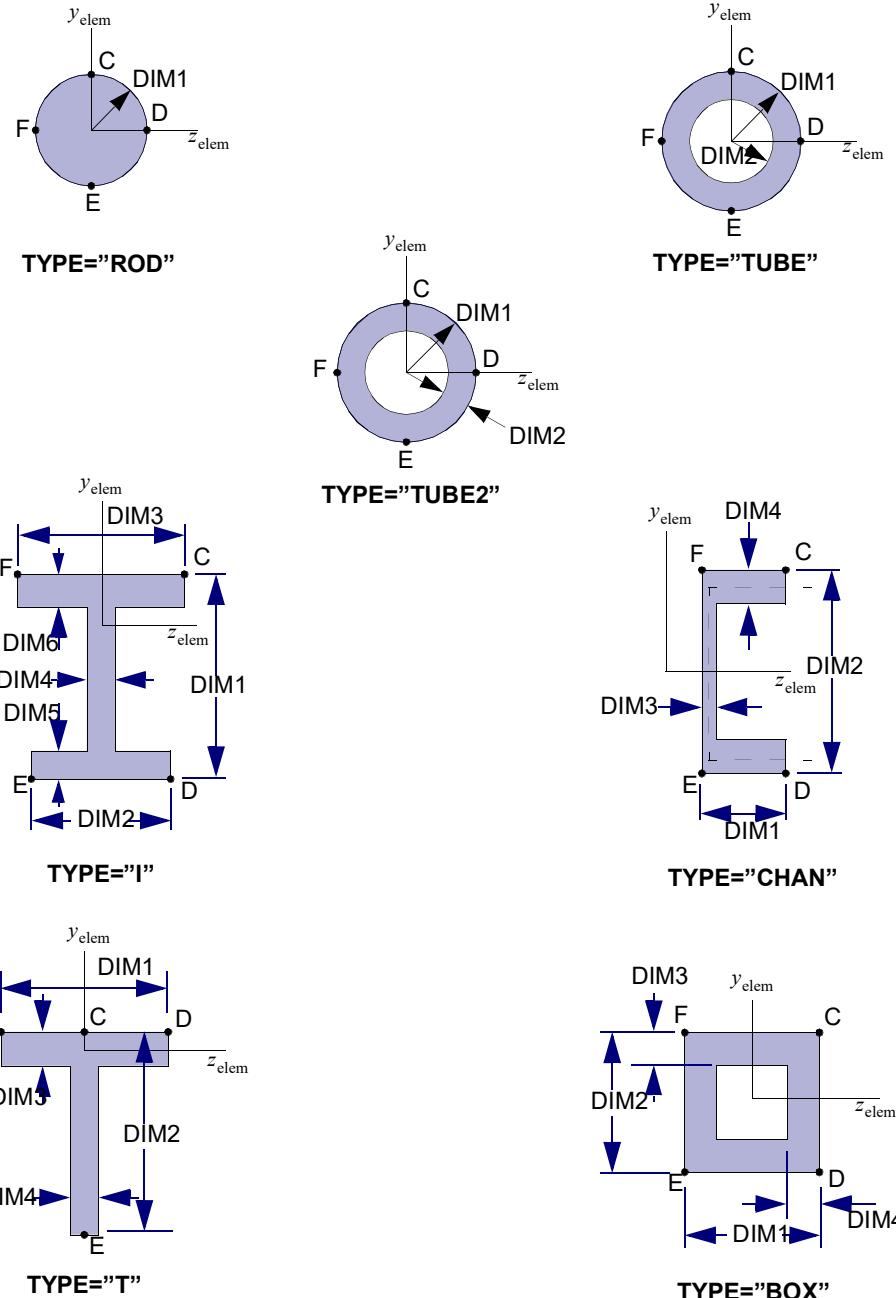


Figure 9-119 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"

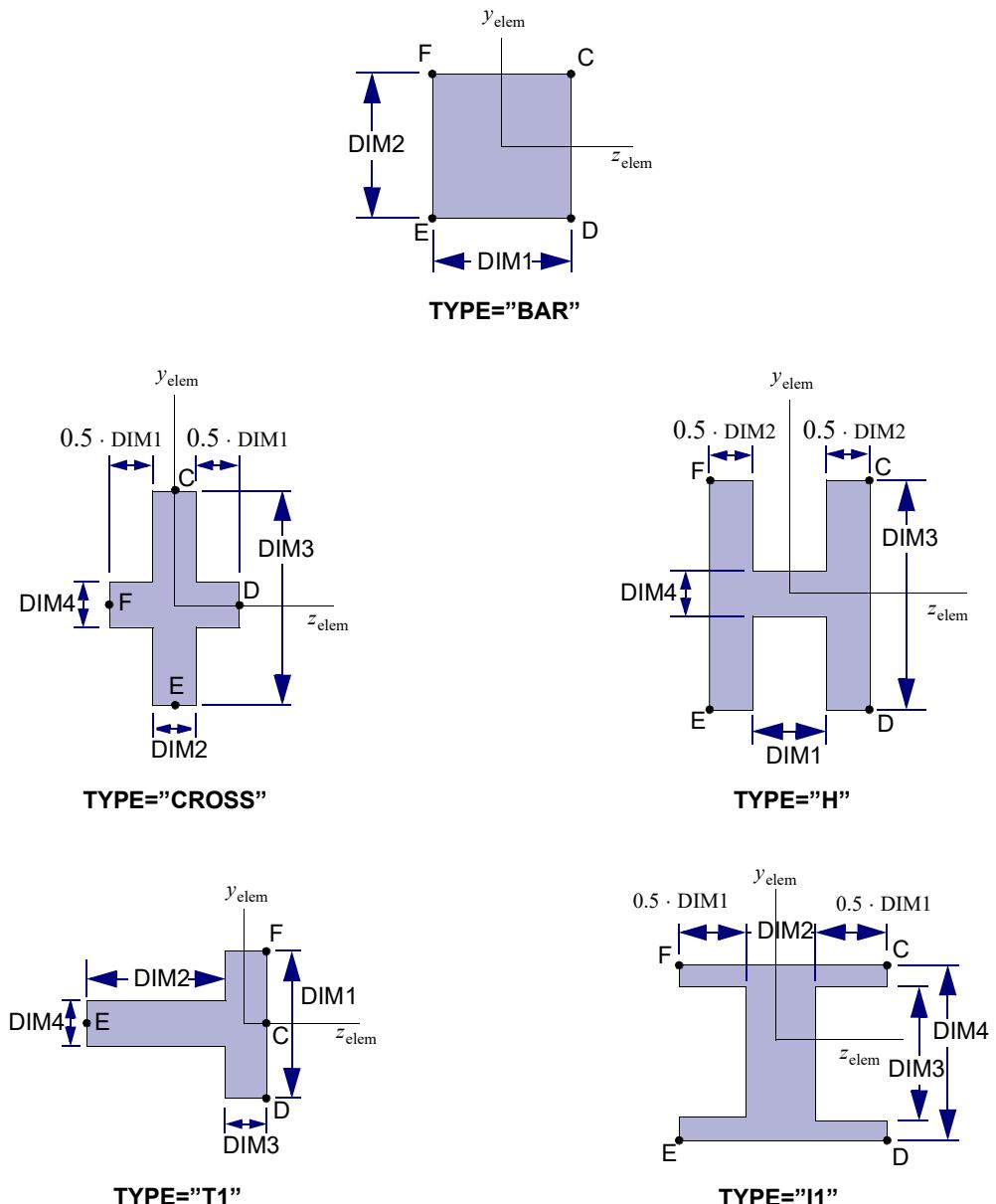


Figure 9-120 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

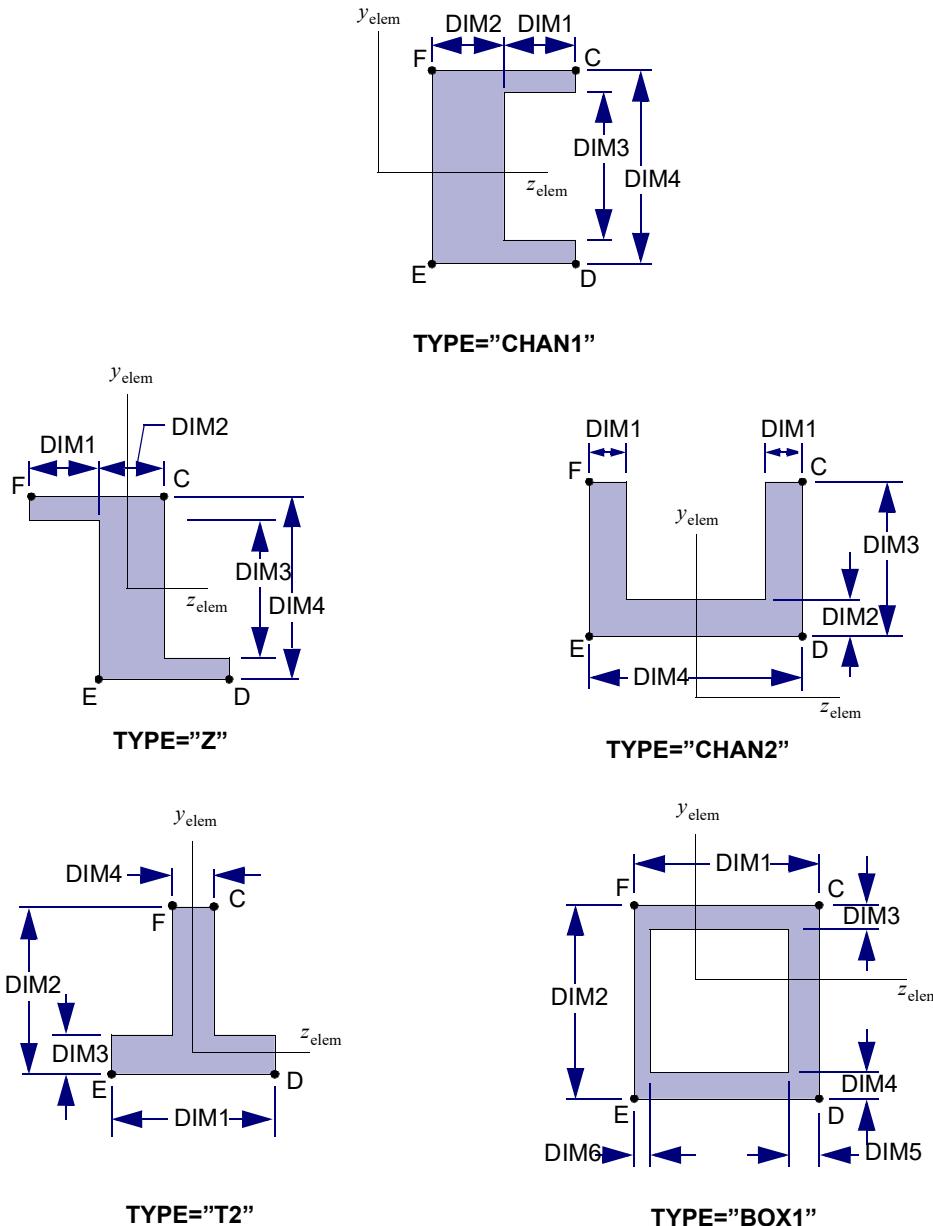


Figure 9-121 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

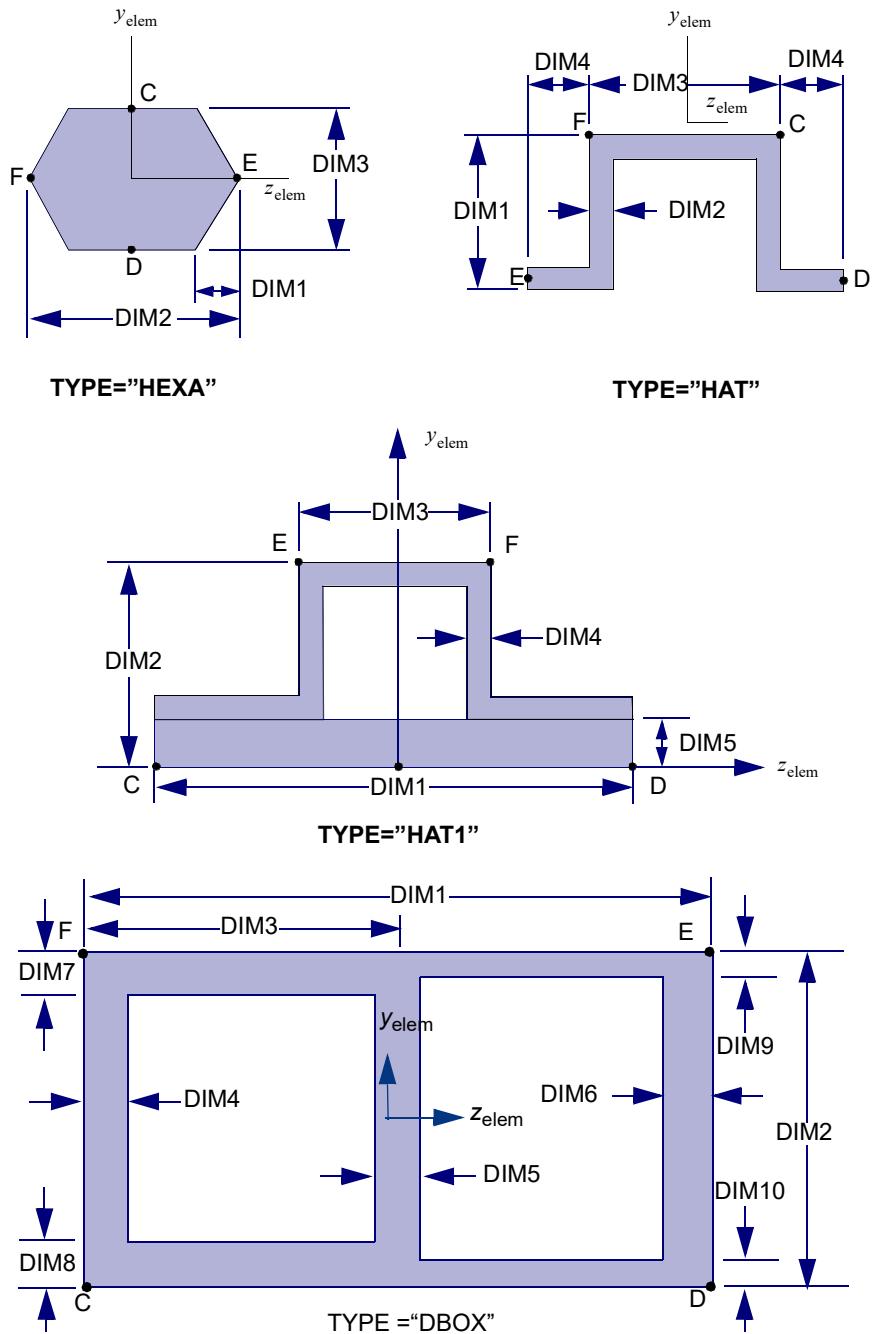


Figure 9-122 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

PBARN1

Nonlinear Property Extensions for a PBAR or PBARL Entry

Specifies additional nonlinear properties for elements that point to a PBAR or PBARL entry in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
PBARN1	PID	MID		SECT					
	“C2”	BEH2	INT2						

Example:

PBARN1	29	73							
--------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number of an existing PBAR entry. (Integer > 0,)
MID	Material ID. Remark 10. (Integer ≥ 0)
SECT	Section integration. SECT = “S” a smeared cross section is used for integration. SECT = “N” a numerically integrated cross section is used. See Remark 7. (Character Default S or blank)
C2	Keyword indicating that items following apply to elements with two end grids. (Character)
BEH2	Element structural behavior. See Remark 4. (Character Default BAR)
INT2	Integration scheme. See Remarks 4. and 5. (Character Default LC)

Remarks:

1. The PID above must point to an existing PBAR or PBARL Bulk Data entry and is honored only in SOL 400.
2. MID if blank (or 0) use the MID value on the PBAR or PBARL entry. If > 0 it will override the MID value on the PBAR entry.
3. The MID entry may point to the MAT1 entry. The table below shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Implicit Structural Materials
MAT1
MATVE
MATVP
MATEP

Implicit Structural Materials
MATF
MATS1
MATSMA

4. BEH2 refers to the nonlinear structural behavior of the BAR element. An underlined item delineates a default.

Structural Classification of Elements		
Element Structural Type	BEHAV CODE	Integration Code
Bar	BAR	<u>LC</u> LCC LS

5. Integration codes in Remark 4. are:

INT CODE	Integration Type
LC	Linear/Cubic
LCC	Linear/Cubic Closed section
LS	Linear-shear

6. Normal buckling modes for beams with nonlinear extensions can be computed through the ANALYSIS=BUCK step. The buckle modes can be evaluated through a linear perturbation step about a linear or nonlinear prestressed state. The current limitation is that lateral buckling mode computations (eg. Lateral-torsional buckling seen in compression flanges of open section beams) are not supported for these elements.

7. Smeared section support.

Primary BAR Property	INT CODE	SECT	Interpolation	Usage	COMMENTS
PBAR	LC	S	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Both Thin-Walled and Solid Sections with elastic materials. Euler-Bernoulli Beam.	Smeared section properties only are supported. Only linear elastic materials via MAT1 card can be provided.
	LS	S	Linear Interpolation for axial, transverse displacements and rotations	Both Thin-Walled and Solid Sections with elastic materials. Captures transverse shear effects - useful for deep beams	Smeared section properties only are supported. Only linear elastic materials via MAT1 card can be provided.
	LC or LS	N			Not Supported
	LCC	S			Not Supported
	LCC	N			Not Supported
PBART	LC or LS	S			Not Supported
	LC	N	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Solid Sections. Euler-Bernoulli Beam	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials are supported. MATHE/MATHP are not supported.

Primary BAR Property	INT CODE	SECT	Interpolation	Usage	COMMENTS
	LS	N	Linear Interpolation for axial displacements, transverse displacements and rotations	Solid Sections. Captures transverse shear effects - useful for deep beams	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported.
	LCC	S			Not Supported
	LCC	N	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Thin-Walled Sections. Euler-Bernoulli Beam.	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported.

8. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation.
9. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available.
10. The structural element damping coefficient, GE, is not supported on elements which reference PBARN1.

PBCOMP

Beam Property (Alternate Form of PBEAM)

Alternate form of the PBEAM entry to define properties of a uniform cross-sectional beam referenced by a CBEAM entry. This entry is also used to specify lumped areas of the beam cross section for nonlinear analysis and/or composite analysis.

Format:

1	2	3	4	5	6	7	8	9	10
PBCOMP	PID	MID	A	I1	I2	I12	J	NSM	
	K1	K2	M1	M2	N1	N2	SYMOPT		
	Y1	Z1	C1	MID1					
	Y2	Z2	C2	MID2					
	-etc.-								

Example:

PBCOMP	39	6	2.9						
							1		
	-0.5	1.2	0.1	18					
	0.2	0.9	0.15						

Descriptor	Meaning
PID	Property identification number. See Remark 1. (Integer > 0)
MID	Material identification number. See Remarks 3. and 6. (Integer > 0)
A	Area of beam cross section. (Real > 0.0)
I1	Area moment of inertia in plane 1 about the neutral axis. See Remark 7. (Real > 0.0)
I2	Area moment of inertia in plane 2 about the neutral axis. See Remark 7. (Real > 0.0)
I12	Area product of inertia. See Remark 7. (Real; Default = 0.0, but $I1 \cdot I2 - (I12)^2 > 0.0$)
J	Torsional stiffness parameter. See Remark 7. (Real > 0.0; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real > 0.0; Default = 0.0)
K1, K2	Shear stiffness factor K in K^*A^*G for plane 1 and plane 2. See Remark 5. (Real > 0.0; Default = 1.0)
M1, M2	The (y,z) coordinates of center of gravity of nonstructural mass. See the figure in the CBEAM entry description. (Real; Default = 0.0)
N1, N2	The (y,z) coordinates of neutral axis. See the figure in the CBEAM entry description. (Real; Default = 0.0)
SYMOPT	Symmetry option to input lumped areas for the beam cross section. See Figure 9-124 and Remark 8. ($0 \leq \text{Integer} \leq 5$; Default = 0)

Descriptor	Meaning
Yi, Zi	The (y,z) coordinates of the lumped areas in the element coordinate system. See Remark 1. (Real)
Ci	Fraction of the total area for the i-th lumped area. (Real > 0.0; Default = 0.0)
MIDi	Material identification number for the i-th integration point. See Remark 6. (Integer > 0)

Remarks:

1. PBCOMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBCOMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.
2. The second continuation entry may be repeated 18 more times. If SYMOPT = 5 a maximum of 21 continuation entries is allowed; i.e., a maximum of 20 lumped areas may be input. If SYMOPT = 1 through 4, the total number of areas input plus the total number generated by symmetry must not exceed 20. If these are not specified, the program defaults, as usual, to the elliptically distributed eight nonlinear rods. See [Figure 9-123](#).

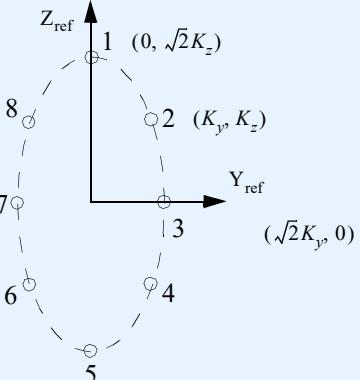
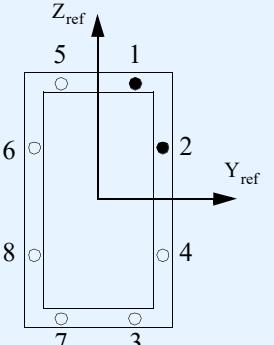
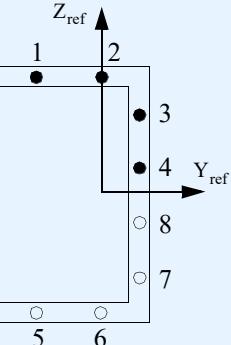
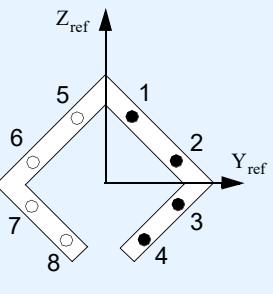
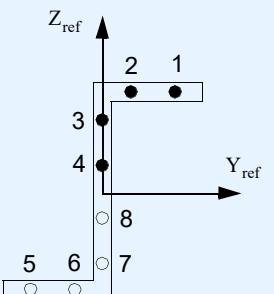
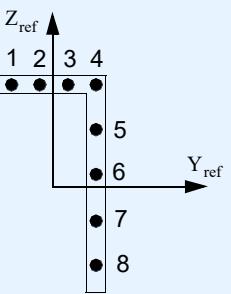
 <p><u>SYMOPT = 0</u> No continuation entries Symmetric about Y_{ref} and Z_{ref}</p> $K_y = \sqrt{\frac{I_{zz}}{A}}, K_z = \sqrt{\frac{I_{yy}}{A}}, C1 = \frac{1}{8}$ <p>I_{zz} - moment of inertia about z-axis I_{yy} - moment of inertia about y-axis</p>	 <p><u>SYMOPT = 1</u> (w/continuation entries) Symmetric about Y_{ref} and Z_{ref}</p> <p>$Y1 = Y3 = -Y5 = -Y7$ $Z1 = -Z3 = Z5 = -Z7$, etc.</p>	 <p><u>SYMOPT = 2</u> Symmetric about Y_{ref}</p> <p>$Y1 = Y5$ $Z1 = -Z5$, etc.</p>
 <p><u>SYMOPT = 3</u> Symmetric about Z_{ref}</p> <p>$Y1 = -Y5, Z1 = Z5$, etc.</p>	 <p><u>SYMOPT = 4</u> Mirror Symmetry about Y_{ref} and Z_{ref}</p> <p>$Y1 = -Y5, Z1 = -Z5$, etc.</p>	 <p><u>SYMOPT = 0 or 5</u> No symmetry</p>

Figure 9-123 PBCOMP Entry SYMOPT Type Examples with 8 Lumped Areas

Notes:

Integration points (lumped area) are numbered 1 through 8.

User-specified points are denoted by ● and the mirrored points are denoted by ○.

3. For structural problems, MID and MIDi must reference a MAT1 material entry. For material nonlinear analysis, the material should be perfectly plastic since the plastic hinge formulation is not valid for strain hardening. For heat transfer problems, MID and MIDi must reference a MAT4 or MAT5 material entry.
4. For the case where the user specifies I1, I2 and I12 on the parent entry, the stress-output location may also be specified on continuation entries. The (y,z) coordinates specified on these entries will serve as stress output locations with the corresponding Ci's set to 0. Stress output is provided at the first four lumped area locations only. If one of the symmetry options is used and fewer than four lumped areas are input explicitly, the sequence of output locations in the imaged quadrants is shown in [Figure 9-123](#). For one specific example in the model shown in Remark 8. ([Figure 9-124](#)), output can be obtained at points 1 and 2 and in the image points 3 and 4.
5. Blank fields for K1 and K2 are defaulted to 1.0. If a value of 0.0 is used for K1 and K2, the transverse shear stiffness becomes rigid and the transverse shear flexibilities are set to 0.0.
6. The values E_0 and G_0 are computed based on the value of MID on the parent entry. MID is will follow the same symmetry rules as Ci depending on the value of SYMOPT. If the MIDi field on a continuation entry is blank, the value will be that of MID on the parent entry. MIDi values may be input on continuations without the corresponding Yi, Zi, and Ci values to allow different stress-strain laws.
7. If the lumped cross-sectional areas are specified, fields I1, I2, and I12 will be ignored. These and other modified values will be calculated based on the input data (Yi, Zi, Ci, MIDi) as follows:

$$y_{NA} = \frac{\sum_{i=1}^n Y_i C_i E_i}{\sum_{i=1}^n C_i E_i}$$

$$z_{NA} = \frac{\sum_{i=1}^n Z_i C_i E_i}{\sum_{i=1}^n C_i E_i}$$

$$\bar{A} = A \sum_{i=1}^n \frac{C_i E_i}{E_0}$$

$$\bar{I}_1 = A \sum_{i=1}^n \frac{C_i E_i (Y_i - y_{NA})^2}{E_o}$$

$$\bar{I}_2 = A \sum_{i=1}^n \frac{C_i E_i (Z_i - z_{NA})^2}{E_o}$$

$$\bar{I}_{12} = A \sum_{i=1}^n \frac{C_i E_i (Y_i - y_{NA})(Z_i - z_{NA})}{E_o}$$

$$J = J \sum_{i=1}^n \frac{C_i G_i}{G_o}$$

where n is the number of lumped cross-sectional areas specified.

8. For a doubly symmetric section (SYMOPT = 1), if the lumped areas are specified on either axis, the symmetry option will double the areas. For example, for the section shown in [Figure 9-124](#), points 2 and 4 are coincident and so are points 6 and 8. In such cases, it is recommended that users input the value of area as half of the actual value at point 2 to obtain the desired effect.

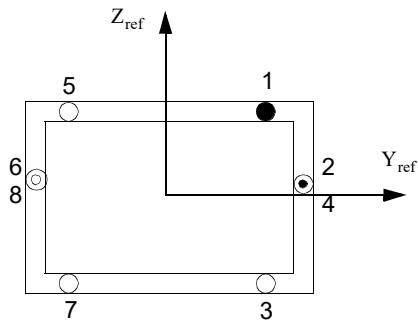


Figure 9-124 Doubly Symmetric PBCOMP Section

9. For SYMOPT =0, 5, or blank at least three Yi and one Zi coordinates must be nonzero. All of the points (input plus mirrored) should not lie on a straight line. All of the points (input plus mirrored) should not lie on a straight line or a FATAL message will be issued.

PBEAM**Beam Property**

Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams.

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM	PID	MID	A(A)	I1(A)	I2(A)	I12(A)	J(A)	NSM(A)	
	C1 (A)	C2 (A)	D1 (A)	D2 (A)	E1 (A)	E2 (A)	F1 (A)	F2 (A)	

The next two continuations are repeated for each intermediate station as described in Remark 6. and SO and X/XB must be specified.

	SO	X/XB	A	I1	I2	I12	J	NSM	
	C1	C2	D1	D2	E1	E2	F1	F2	

The last two continuations are:

	K1	K2	S1	S2	NSI(A)	NSI(B)	CW(A)	CW(B)	
	M1(A)	M2(A)	M1(B)	M2(B)	N1(A)	N2(A)	N1(B)	N2(B)	

Example:

Tapered beam with A=2.9 at end A and A=5.3 at end B.

PBEAM	39	6	2.9	3.5	5.97				
			2.0	-4.0					
	YES	1.0	5.3	56.2	78.6				
			2.5	-5.0					
			1.1		2.1		0.21		
					0.5		0.0		

Descriptor	Meaning	Default Values
PID	Property identification number. (Integer > 0)	Required
MID	Material identification number. See Remarks 1. and 3. (Integer > 0)	Required
A(A)	Area of the beam cross section at end A. (Real > 0.0)	Required
I1(A)	Area moment of inertia at end A for bending in plane 1 about the neutral axis. See Remark 10. (Real > 0.0)	Required
I2(A)	Area moment of inertia at end A for bending in plane 2 about the neutral axis. See Remark 10. (Real > 0.0)	Required

Descriptor	Meaning	Default Values
I12(A)	Area product of inertia at end A. See Remark 10. (Real, but $I_1 \cdot I_2 - (I_{12})^2 > 0.0$)	0.0
J(A)	Torsional stiffness parameter at end A. See Remark 10. (Real ≥ 0.0 but > 0.0 if warping is present)	Default= $\frac{1}{2}(I_1 + I_2)$ for SOL 600 and 0.0 for all other solution sequences
NSM(A)	Nonstructural mass per unit length at end A. (Real)	0.0
Ci(A), Di(A)	The y and z locations ($i = 1$ corresponds to y and $i = 2$ corresponds to z) in element coordinates relative to the shear center (see the diagram following the remarks) at end A for stress data recovery. (Real)	$y = z = 0.0$
Ei(A), Fi(A)		
SO	Stress output request option. See Remark 9. (Character)	Required*
	“YES” Stresses recovered at points Ci, Di, Ei, and Fi on the next continuation.	
	“YESA” Stresses recovered at points with the same y and z location as end A.	
	“NO” No stresses or forces are recovered.	
X/XB	Distance from end A in the element coordinate system divided by the length of the element See Figure 9-125 in Remark 10. (Real, $0.0 < x/xb \leq 1.0$)	Required* See Remark 6.
A, I1, I2, I12,	Area, moments of inertia, torsional stiffness parameter, and nonstructural mass for the cross section located at x. (Real; J > 0.0 if warping is present.)	See Remark 7.
J, NSM		
Ci, Di, Ei, Fi	The y and z locations ($i = 1$ corresponds to y and $i = 2$ corresponds to z) in element coordinates relative to the shear center (see Figure 9-125 in Remark 10.) for the cross section located at X/XB. The values are fiber locations for stress data recovery. Ignored for beam p-elements. (Real)	
K1, K2	Shear stiffness factor K in K^*A^*G for plane 1 and plane 2. See Remark 12. (Real)	1.0, 1.0
S1, S2	Shear relief coefficient due to taper for plane 1 and plane 2. Ignored for beam p-elements. (Real)	0.0, 0.0
NSI(A), NSI(B)	Nonstructural mass moment of inertia per unit length about nonstructural mass center of gravity at end A and end B. See Figure 9-125. (Real)	0.0, same as end A

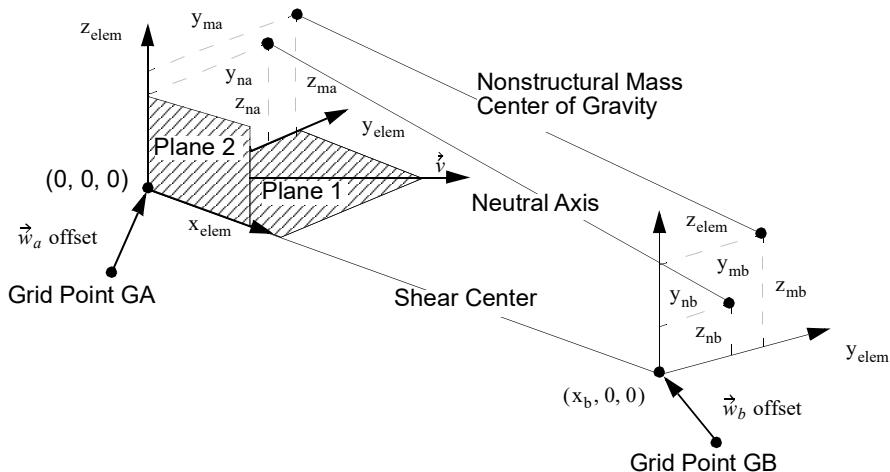
Descriptor	Meaning	Default Values
CW(A), CW(B)	Warping coefficient for end A and end B. Ignored for beam p-elements. See Remark 11. (Real)	0.0, same as end A
M1(A), M2(A), M1(B), M2(B)	(y,z) coordinates of center of gravity of nonstructural mass for end A and end B. See Figure 9-125 . (Real)	0.0 (no offset from shear center), same values as end A
N1(A), N2(A), N1(B), N2(B)	(y,z) coordinates of neutral axis for end A and end B. See Figure 9-125 . (Real)	0.0 (no offset from shear center), same values as end A

Remarks:

- For structural analysis, MID must reference a MAT1 material entry (SOL 600). The beam may be described by any valid stress-strain law. A plastic hinge is not used for SOL 600 and SOL 700; instead, a standard nonlinear analysis is performed.
- For material nonlinear analysis, MID may also reference a MATS1 entry, but the material properties must be defined as elastic-perfectly plastic; for example, H = 0.0 on the MATS1 entry. Also, only one-eighth of the length at each end of the element abides by material nonlinear law; i.e., the element is modeled as a plastic hinge. Any other type of material property specification may yield inaccurate results.
- For heat transfer analysis, MID must reference a MAT4 or MAT5 material entry.
- If no stress data at end A is to be recovered and a continuation with the SO field is specified, then the first continuation entry, which contains the fields C1(A) through F2(A), may be omitted.
- If SO is “YESA” or “NO”, the third continuation entry, which contains the fields C1 through F2, must be omitted. If SO is “YES”, the continuation for Ci, Di, Ei, and Fi must be the next entry. The blank fields are defaulted to 0.0 on these continuations.
- The rules for the continuations entries are:
 - The second and third continuation entries, which contain fields SO through F2, may be repeated nine more times for intermediate X/XB values for linear beam elements. The order of these continuation pairs is independent of the X/XB value; however, one value of X/XB must be 1.0, corresponding to end B. The intermediate stress output requests will be ignored in the nonlinear solution sequences (SOLs 106 and 129).
 - The value of X/XB must be unique among the stations of a PBEAM. Duplication of X/XB is not permitted.
 - The fourth and fifth continuation entries, which contain fields K1 through N2(B), are optional and may be omitted if the default values are appropriate.
- If any fields 4 through 9 are blank on the continuation with the value of X/XB = 1.0, then the values for A, I1, I2, I12, J and NSM are set to the values given for end A. For the continuations that have intermediate values of X/XB between 0.0 and 1.0 and use the default option (any of the fields 4 through 9 are blank), a linear interpolation between the values at ends A and B is performed to obtain the missing section properties.

8. Blank fields for K1, K2 are defaulted to 1.0. If a value of 0.0 is used for K1 and K2, the transverse shear flexibilities are set to 0.0 and field G on the MAT1 entry selected by MID must be nonzero.
9. If end B forces are desired and station data are input, use “YES” or “YESA” in the SO field of the record with X/XB=1. If station data are not input you will get end B forces if forces are requested.
10. [Figure 9-125](#) describes the PBEAM element coordinate system.

$$\begin{array}{lll}
 I_{11} = I_{(zz)_{na}} & N1(A) = y_{na} & N1(B) = y_{nb} \\
 I_{12} = I_{(yy)_{na}} & N2(A) = z_{na} & N2(B) = z_{nb} \\
 I_{12} = I_{(zy)_{na}} & M1(A) = y_{ma} & M1(B) = y_{mb} \\
 J = I_{(xx)_{na}} & M2(A) = z_{ma} & M2(B) = z_{mb}
 \end{array}$$



[Figure 9-125](#) PBEAM Element Coordinate System

11. The warping coefficient CW is represented in the following differential equation for the torsion of a beam about the axis of the shear centers:

$$G \frac{d}{dx} \left(J \frac{d\theta}{dx} \right) - E \frac{d^2}{dx^2} \left(CW \frac{d^2\theta}{dx^2} \right) = m$$

where:

G = shear modulus
 J = torsional stiffness
 E = Young's modulus
 θ = angle of rotation at any cross-section
 m = applied torsional moment per unit length

Note: CW has units of (length)⁶.

12. The shear stiffness factors K_1 and K_2 adjust the effective transverse shear cross-section area according to the Timoshenko beam theory. Their default values of 1.0 approximate the effects of shear deformation. To neglect shear deformation (i.e., to obtain the Bernoulli-Euler beam theory), the values of K_1 and K_2 should be set to 0.0.
13. In nonlinear analysis the location of the 8 plastic rods is the same on the PBEAM entry as it is on the SYMOPT field on the PBCOMP entry when SYMOPT = 0. Please see the [Figure 9-123](#) for more information on the SYMOPT field.
14. For SOL 600, for structural analysis, MID must reference a MAT1 material entry.
15. RC network solver only supports constant cross section beam for thermal analysis.
16. PBEAM is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEAM property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.

PBEAM3

Three-node Beam Property

Defines the properties of a three-node beam element (CBEAM3 entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM3	PID	MID	A(A)	IZ(A)	IY(A)	IYZ(A)	J(A)	NSM(A)	
	CY(A)	CZ(A)	DY(A)	DZ(A)	EY(A)	EZ(A)	FY(A)	FZ(A)	
	SO(B)		A(B)	IZ(B)	IY(B)	IYZ(B)	J(B)	NSM(B)	
	CY(B)	CZ(B)	DY(B)	DZ(B)	EY(B)	EZ(B)	FY(B)	FZ(B)	
	SO(C)		A(C)	IZ(C)	IY(C)	IYZ(C)	J(C)	NSM(C)	
	CY(C)	CZ(C)	DY(C)	DZ(C)	EY(C)	EZ(C)	FY(C)	FZ(C)	
	KY	KZ	NY(A)	NZ(A)	NY(B)	NZ(B)	NY(C)	NZ(C)	
	MY(A)	MZ(A)	MY(B)	MZ(B)	MY(C)	MZ(C)	NSIY(A)	NSIZ(A)	
	NSIYZ(A)	NSIY(B)	NSIZ(B)	NSIYZ(B)	NSIY(C)	NSIZ(C)	NSIYZ(C)	CW(A)	
	CW(B)	CW(C)	STRESS						
	WC(A)	WYC(A)	WZC(A)	WD(A)	WYD(A)	WZD(A)	WE(A)	WYE(A)	
	WZE(A)	WF(A))	WYF(A)	WZF(A)	WC(B)	WYC(B)	WZC(B)	WD(B)	
	WYD(B)	WZD(B)	WE(B)	WYE(B)	WZE(B)	WF(B)	WYF(B)	WZF(B)	
	WC(C)	WYC(C)	WZC(C)	WD(C)	WYD(C)	WZD(C)	WE(C)	WYE(C)	
	WZE(C)	WF(C)	WYF(C)	WZF(C)					

Example:

PBEAM3	1010	2	2.9	3.5	5.97			1.0	
	0.2	3.0	-1.2	2.6	2.0	0.5			
	YES		1.0	23.6	34.7				
	1.1	3.2							
	YESA		3.2	2.1	3.2			1.0	
	0.8		0.5						
	0.9	1.0		1.5					
		1.0							

Descriptor	Meaning
PID	Property identification number. (Integer > 0; Required)
MID	Material identification number. See Remark 1. (Integer > 0; Required)
A(A)	Area of the beam cross-section at end A. (Real > 0.0; Required)
IZ(A)	Area moment of inertia at end A about local z-axis and the neutral axis. (Real > 0.0; Required)

Descriptor	Meaning
IY(A)	Area moment of inertia at end A about local y-axis and the neutral axis. (Real > 0.0; Required)
IYZ(A)	Area product of inertia at end A about local y- and z-axes and the neutral axis. If y- and z- axes are principal axes, then IYZ(A)=0.0. (Real, but $I_y \cdot I_z - I_{yz}^2 > 0.0$; Default = 0.0)
J(A)	Torsional stiffness parameter at end A. (Real > 0.0; Default = IZ+IY)
NSM(A)	Nonstructural mass per unit length at end A. (Real; Default = 0.0)
$Ci(j)$, $Di(j)$ $Ei(j)$, $Fi(j)$	The local y and z coordinates ($i=Y, Z$) at point j ($j=A, B, C$), used for stress output. (Real; Default = 0.0)
A(j), IZ(j), IY(j) IYZ(j), J(j), NSM(j)	Area, moments of inertia, torsional stiffness parameter and nonstructural mass for the cross-section at j ($j=B, C$). (Real; See Remark 2.)
SO(j)	Stress output request option at j ($j=B, C$). (Character; Default = "YESA") <ul style="list-style-type: none"> "YES" Stresses are recovered at Ci, Di, Ei, and Fi on the next continuation. "YESA" Stresses are recovered at points with the same (y, z) location at end A
KY, KZ	Shear effectiveness factors for local y- and z-directions. (Real > 0.0, Default = 1.0)
NY(j), NZ(j)	Local (y, z) coordinates of neutral axis for j ($j=A, B, C$). (Real, Default = 0.0 at end A and same values as end A for $j=B, C$)
MY(j), MZ(j)	Local (y, z) coordinates of nonstructural mass center of gravity at j ($j=A, B, C$). (Real, Default = 0.0 at end A and same values as end A for $j=B,C$)
NSIY(j), NSIZ(j)	Nonstructural mass moments of inertia per unit length about local y and z-axes, respectively, with regard to the nonstructural mass center of gravity at j ($j=A, B, C$). (Real, Default = 0.0 at end A and same values as end A for $j=B, C$)
NSIYZ(j)	Nonstructural mass product of inertia per unit length about local y and z-axes, respectively, with regard to the nonstructural mass center of gravity at j ($j=A, B, C$). (Real, Default = 0.0 at end A and same values as end A for $j=B, C$)
CW(j)	Warping coefficient at j ($j=A, B, C$). (Real ≥ 0.0 ; Default = 0.0 at end A; same values as end A for $j = B, C$)
STRESS	Location selection for stress, strain and force output. (Character; Default = "GRID"; See Remark 3.)
Wi(j)	Values of warping function at stress recovery points $i = C, D, E$ and F , at location $j=A, B$, and C . (Real; Default = 0.0 at end A and same values as end A for $j=B, C$)
WYi(j), WZi(j)	Gradients of warping function in the local (y, z) coordinate system at stress recovery points $i=C, D, E$, and F , at location $j=A, B$, and C . (Real; Default = 0.0 at end A and same values as end A for $j=B, C$.)

Remarks:

1. For structural analysis, MID must reference a MAT1, MAT2 or MAT8 material entry.
2. If any fields 4 through 9, for values of A, IZ, IY, IYZ, J and NSM at end B or C, are blank, then those values for end B or C are set to the values given for end A.
3. If STRESS=“GRID”, then the stresses, strains and forces are recovered at A, B and C. If STRESS=“GAUSS”, then the stresses, strains and forces are recovered at Gauss integration points, $\xi = \{1/\sqrt{3}, 1/\sqrt{3}, 0\}$. The beam cross-section properties at these points are interpolated from those at A, B and C.
4. If all fields of $W_i(j)$, $WY_i(j)$ and $WZ_i(j)$ ($i=C, D, E, F$ and $j=A, B, C$), are left blank, both stresses and strains due to the warping effect will not be recovered at the stress recovery points.
5. PBEAM3 is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEAM3 property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.
6. When PBMSECT is used the resulting PBEAM3 image printed after the:

*** USER INFORMATION MESSAGE 4379 (IFP9B)

THE USER SUPPLIED PBMSECT BULK DATA ENTRIES ARE REPLACED BY THE FOLLOWING PBEAM3 ENTRIES.

contains special composite related data and will look different from the above entry description.

PBEAMD

PBEAMD usage is no longer recommended and will be removed in a future version. Use PBEAM, PBDISCR, PBSPOT, PBEAM71 instead.

PBEAML**Beam Cross-Section Property**

Defines the properties of a beam element by cross-sectional dimensions.

Format:

(Note: n = number of dimensions and m = number of intermediate stations)

1	2	3	4	5	6	7	8	9	10
PBEAML	PID	MID	GROUP	TYPE					
	DIM1(A)	DIM2(A)	-etc.-	DIMn(A)	NSM(A)	SO(1)	X(1)/XB	DIM1(1)	
	DIM2(1)	-etc.-	DIMn(1)	NSM(1)	SO(2)	X(2)/XB	DIM1(2)	DIM2(2)	
	-etc.-	DIMn(2)	NSM(m)	-etc.-	SO(m)	X(m)/XB	DIM1(m)	-etc.-	
	DIMn(m)	NSM(m)	SO(B)	1.0	DIM1(B)	DIM2(B)	-etc.-	DIMn(B)	
	NSM(B)								

Example:

PBEAML	99	21		T					
	12.	14.8	2.5	2.6		NO	0.4	6.	
	7.	1.2	2.6		YES	0.6	6.	7.8	
	5.6	2.3		YES					

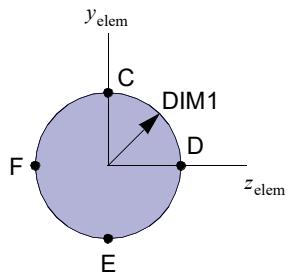
Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GROUP	Cross-section group. (Character; Default = "MSCBML0")
TYPE	Cross-section shape. See Remark 4. (Character: "ROD", "TUBE", "TUBE2", "L", "I", "CHAN", "T", "BOX", "BAR", "CROSS", "H", "T1", "I1", "CHAN1", "Z", "CHAN2", "T2", "BOX1", "HEXA", "HAT", "HAT1", "DBOX" for GROUP = "MSCBML0")
DIMi(j)	Cross-section dimensions at end A, intermediate station j and end B. (Real > 0.0 for GROUP = "MSCBML0")
NSM(j)	Nonstructural mass per unit length. (Default = 0.0)
SO(j),SO(B)	Stress output request option for intermediate station j and end B. (Character; Default = "YES")
YES	Stresses recovered at all points on next continuation and shown in as C, D, E, and F.

Descriptor	Meaning
NO	No stresses or forces are recovered.
X(j)/XB	Distance from end A to intermediate station j in the element coordinate system divided by the length of the element. (REAL, 0.0<x(j)/xb ≤1.0, default=1.0)

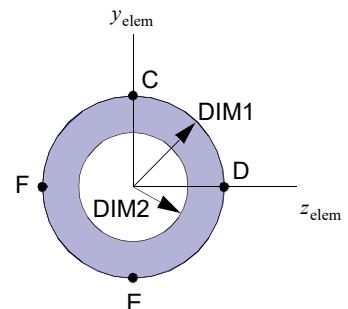
Remarks:

1. For structural problems, PBEAML entries must reference a MAT1 material entry.
2. PBEAML is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEAML property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.
3. For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.
4. See the PBEAM entry description for a discussion of beam-element geometry.
5. If any of the fields NSM(B), DIMi(B) are blank on the continuation entry for End B, the values are set to the values given for end A. For the continuation entries that have values of X(j)/XB between 0.0 and 1.0 and use the default option (blank field), a linear interpolation between the values at ends A and B is performed to obtain the missing field.
6. The GROUP is associated with a FMS CONNECT statement, which specifies the evaluator. A reserved GROUP name is “MSCBML0”. Users may create their own cross-section types. Each of the types will require a one or more subroutines to convert DIMi information to geometric property information contained on a PBEAM entry. See [Building and Using the Sample Programs](#) in the *MSC Nastran Utilities Guide* for a discussion of how to include a user-defined beam library.
7. For GROUP = “MSCBML0”, the cross-sectional properties, shear flexibility factors and stress recovery points are computed using the TYPE and DIMi as shown in . The element coordinate system is located at the shear center.
8. A function of this entry is to derive an equivalent PBEAM entry. Any sorted echo request will also cause printout and/or punch of the derived PBEAM.
9. Beams can have no more than 14 dimensions per station. The total number of dimensions at all stations must be less than 200. The transfer of data with the beam server is limited to 4000 words.
None of these limits are exceeded with the MSC beam library, but a user defined beam library could.
10. The finite element formulation (FEF) utilized for the arbitrary beam cross section is selected as the default method for computing sectional properties for all supported cross section types of PBEAML when GROUP=MSCBML0. The original beam equations, which are based on thin-walled assumption can be accessed via Bulk Data entry ‘MDLPRM,TWBRBML,1’.
11. For optimization, individual DIMx of PBEAML can be selected as designed properties even with finite element formulation.

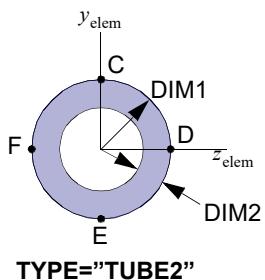
12. The origin of the y_{elem} and z_{elem} coordinate system for the section types is at the shear center for all element types when using the FEF. For section types 'L', 'T', 'T1', 'T2' and 'HAT1', no shear center calculation is calculated with the original beam equations and the origins of these types are at the locations specified in the following figures. For these sections types, this can result in a difference in the stress recovery points and the neutral axis locations between the beam equations and the FEF. The FEF results are considered more exact. For the remaining types with the beam equation, the shear center is either computed or is obvious due to symmetry considerations.



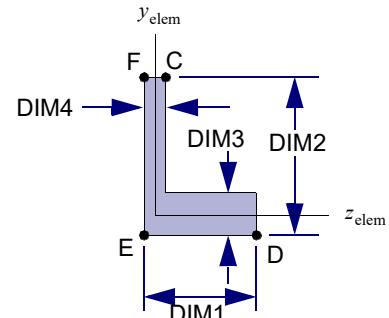
TYPE="ROD"



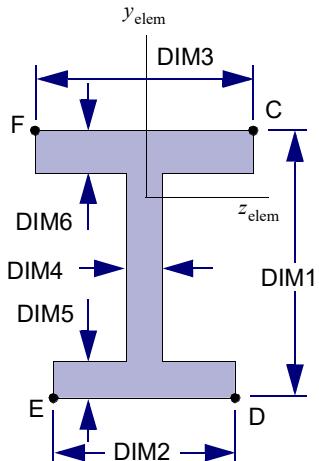
TYPE="TUBE"



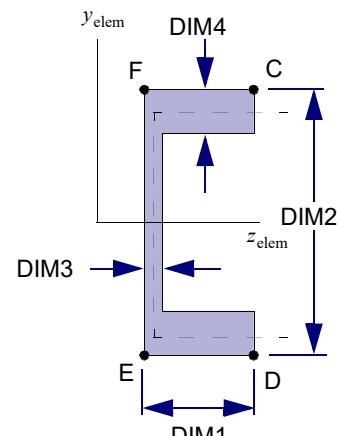
TYPE="TUBE2"



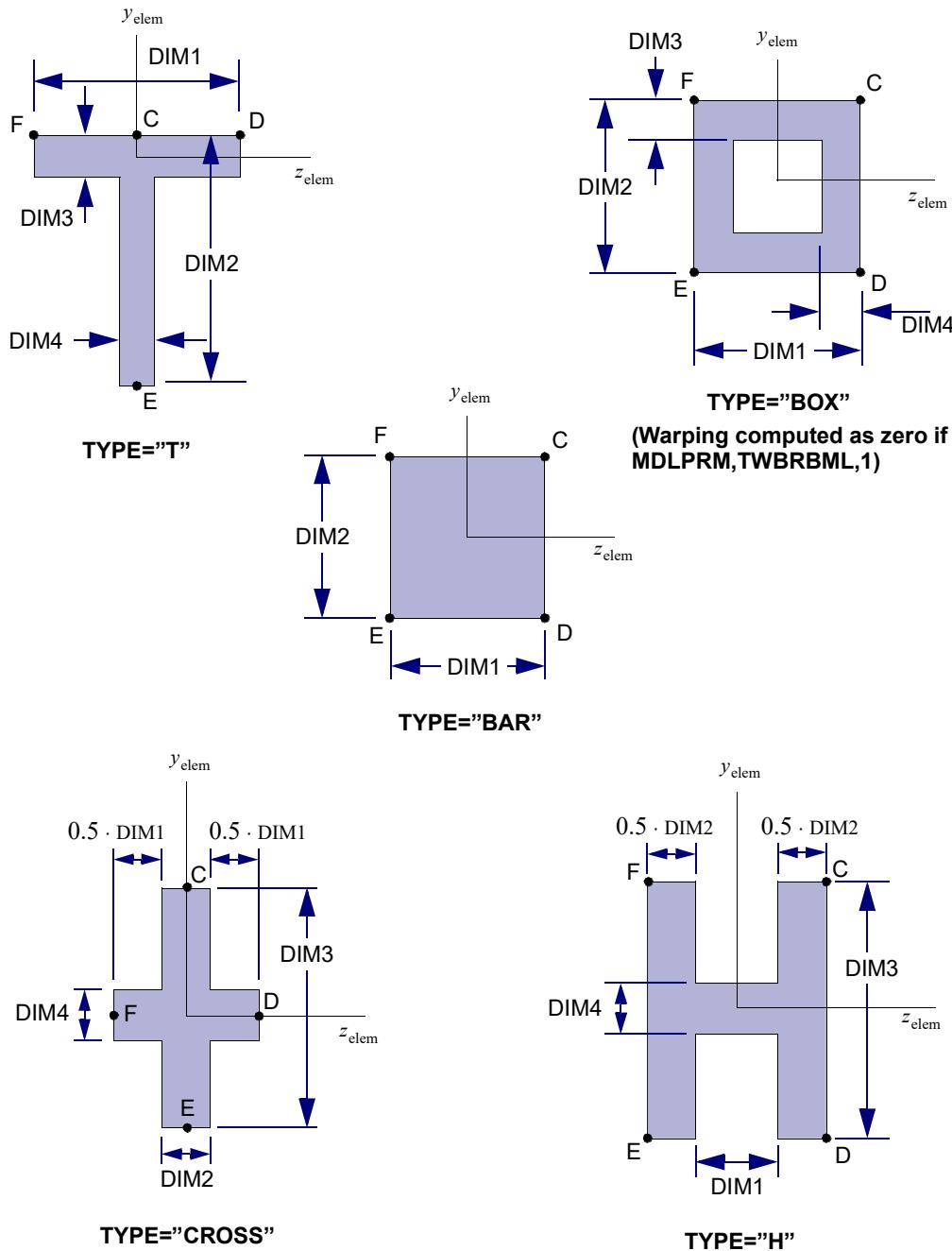
TYPE="L"

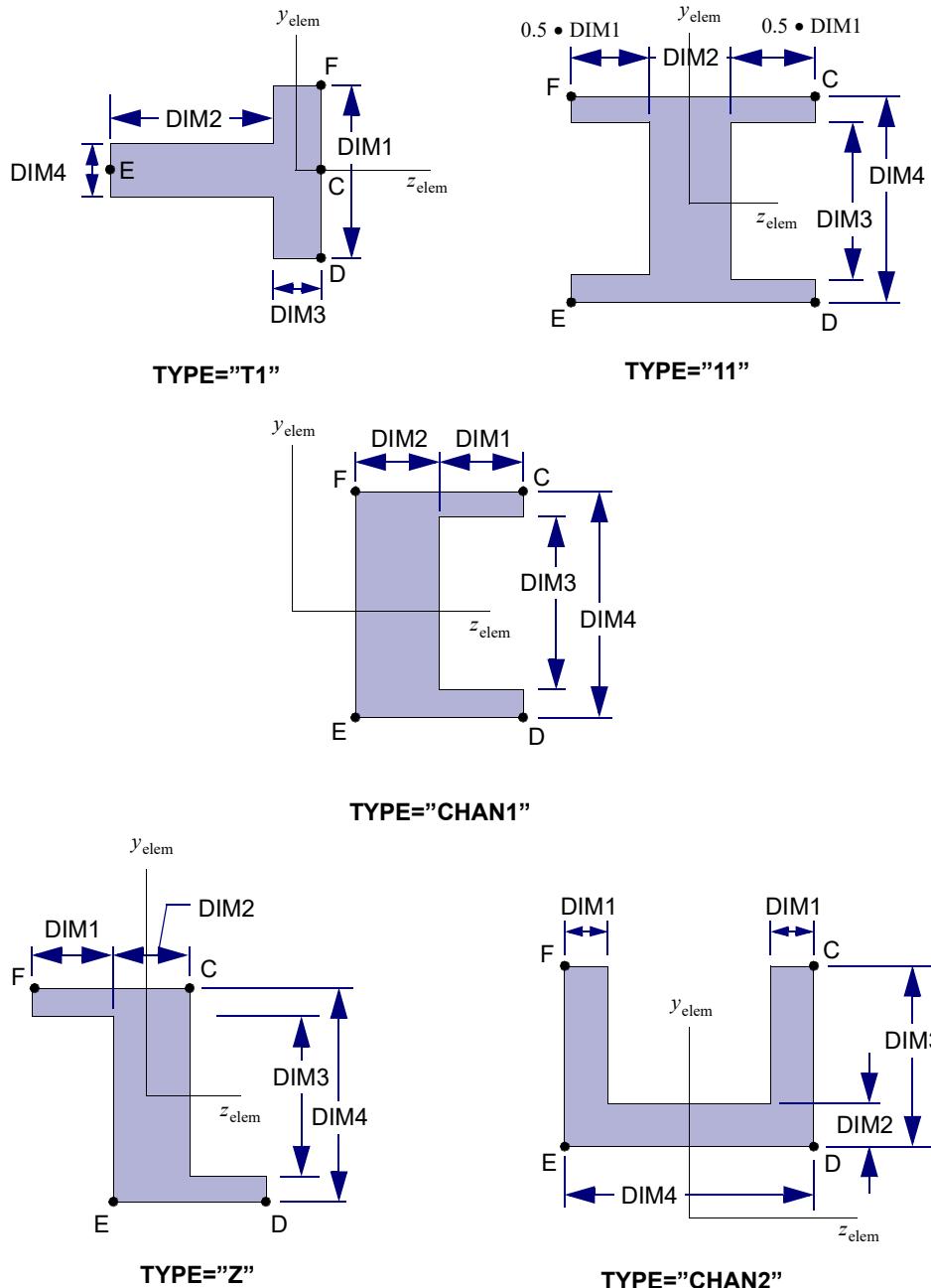


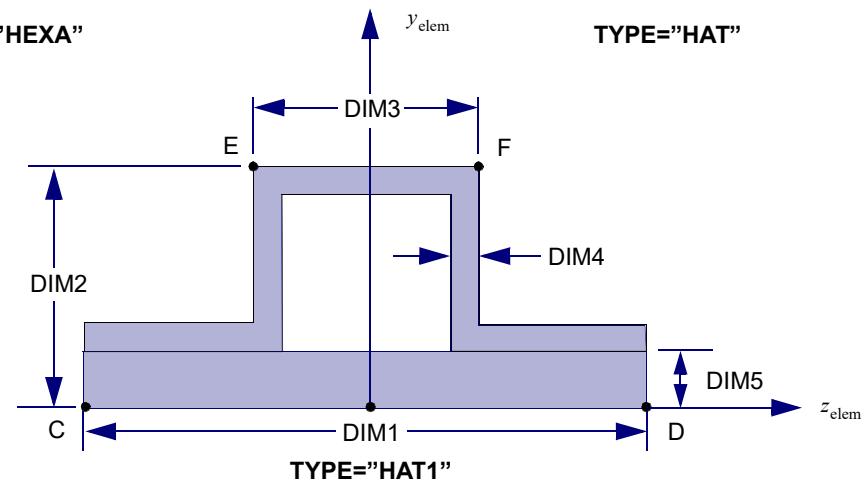
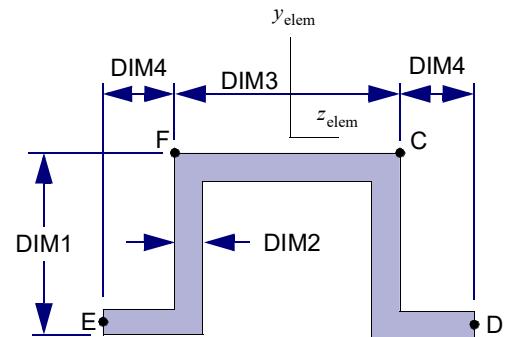
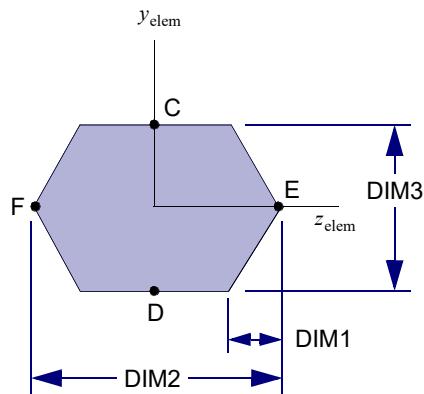
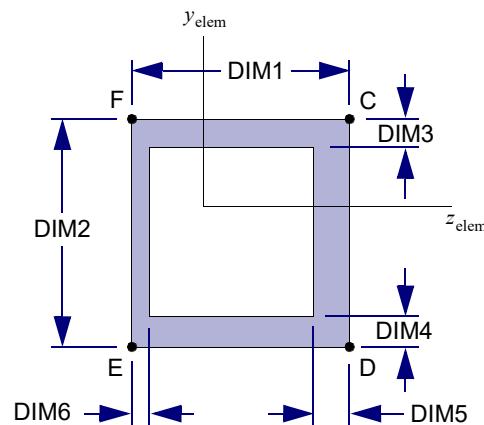
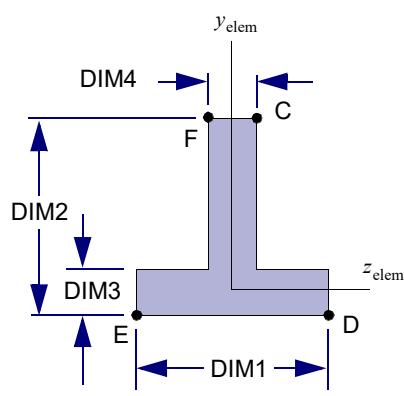
TYPE="I"



TYPE="CHAN"







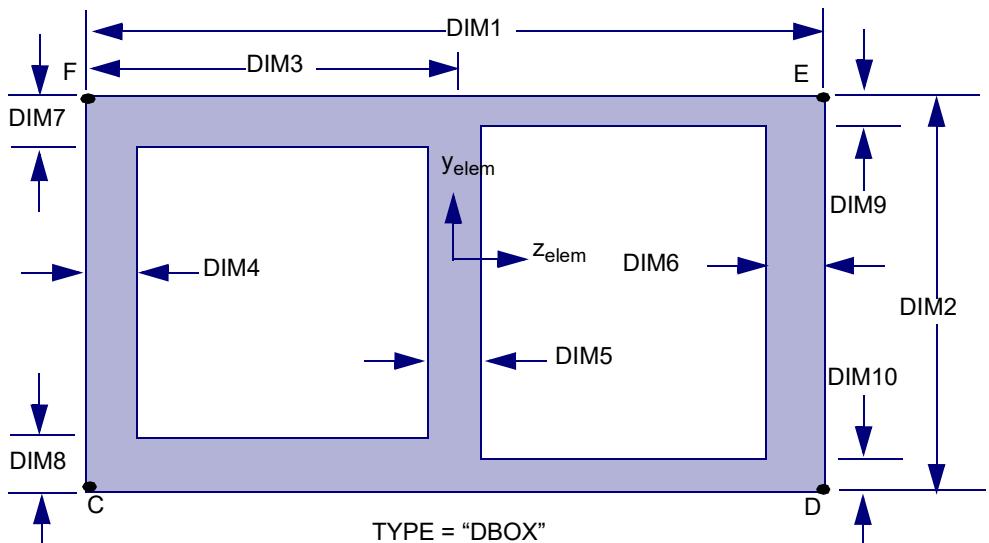


Figure 9-126 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"

PBELT**Belt Property- SOL 700**

Defines the properties of a belt element referenced by a CROD entry. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PBELT	PID	LOAD	UNLOAD	DENSITY	DAMP1	DAMP2	SLACK	PRESTRESS	

Example:

PBELT	9	12	12	2.0E-5	0.1	0.1			

Field	Content
PID	Unique belt property number. (Integer > 0; required)
LOAD	TABLED1 ID defining the force as a function of strain during loading. The strain at time n is specified as engineering strain. (Integer > 0; required) $\text{strain}(n) = (\text{length}(n) - \text{length}(0)) / \text{length}(0)$
UNLOAD	TABLED1 ID defining the force as a function of strain during unloading. The strain at time is specified as engineering strain. (Integer > 0; required) $\text{strain}(n) = (\text{length}(n) - \text{length}(0)) / \text{length}(0)$
DENSITY	Density of the belt elements as mass per unit length. (Real > 0.0; required)

Field	Content
DAMP1	<p>A damping force is added to the internal force of the belt elements to damp out high frequency oscillations. (Real > 0.0; default=0.1)</p> <p>The damping force is equal to:</p> $F_{damp} = DAMP1 \cdot mass \cdot (dvel/dt)$ <p>Where</p> $F_{damp} = \text{damping force}$ $DAMP1 = \text{damping coefficient}$ $\text{mass} = \text{mass of belt element}$ $dvel = \text{velocity of elongation}$ $dt = \text{time step}$
DAMP2	<p>The maximum damping force: $DAMP2 * F_{belt}$. (Real > 0.0; default=0.1)</p> <p>Where</p> $F_{belt} = \text{Internal force in the belt element}$
SLACK	<p>TABLED1 ID defining the slack as a function of time. (Integer > 0; default=not used)</p> <p>The slack must be specified as engineering strain and will be subtracted from the element strain at time as:</p> $\text{strain}(n) = \text{strain}(n) - SLACK(n)$ <p>The force in the element is zero until the element strain exceeds the slack.</p>
PRESTRESS	<p>TABLED1 ID defining a prestress strain as a function of time. (Integer > 0; default=not used)</p> <p>The prestress strain must be specified as engineering strain and will be added to the element strain at time as:</p> $\text{strain}(n) = \text{strain}(n) + PRESTRESS(n)$

Remarks:

1. The loading and unloading curves must start at (0.0, 0.0).

2. During loading, the loading curve is applied to determine the force in the belt element. At unloading, the unloading curve is shifted along the strain axis until it intersects the loading curve at the point from which unloading commences. The unloading table is applied for unloading and reloading, until the strain again exceeds the intersection point. Upon further loading, the loading table is applied. For subsequent unloading, the sequence is repeated.
3. Belt elements are tension only elements.
4. Instantaneous slack of an element can also be initialized per element using the TICEL entry with the keyword SLACK and a corresponding VALUE.

PBEMN1

Nonlinear Property Extensions for a PBEAM or PBEAML Entry

Specifies additional nonlinear properties for elements that point to a PBEAM or PBEAML entry in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
PBEMN1	PID	MID		SECT					
	“C2”	BEH2	INT2						

Example:

PBEMN1	27	93							
--------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number of an existing PBEAM entry. (Integer > 0)
MID	Material ID. Remark 9. (Integer ≥ 0)
SECT	Section integration. SECT = “S” a smeared cross section is used for integration. SECT = “N” a numerically integrated cross section is used. See Remark 10. (Character Default S or blank)
C2	Keyword indicating that items following apply to elements with two end grids. (Character)
BEH2	Element structural behavior. See Remark 5. (Character Default BEAM)
INT2	Integration scheme. See Remarks 5. and 6. (Character Default LC)

Remarks:

1. The PID above must point to an existing PBEAM or PBEAML Bulk Data entry and is honored only in SOL 400.
2. Tapering of the CBEAM is ignored. Only section properties at end A are used.
3. MID if blank (or 0) use the MID value on the PBEAM or PBEAML entry. If > 0 it will override the MID value on the PBEAM entry.
4. The MID entry may point to MAT1 entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Implicit Structural Materials
MAT1
MATVE
MATVP

Implicit Structural Materials
MATEP
MATF
MATS1

5. BEH2 refers to the nonlinear structural behavior of the BEAM element. An underlined item delineates a default.

Structural Classification of Elements				
Element Structural Type	BEHAV CODE	Integration Code	Element Type	# Nodes
Beam	BEAM	LC	BEAM	2
		LCC	BEAM	2
		LCO	BEAM	2
		LS	BEAM	2

6. Integration codes in Remark 5. are:

INT CODE	Integration Type
LC	Linear/Cubic
LCC	Linear/Cubic Closed section
LCO	Linear/Cubic Open section
LS	Linear-shear

7. Integration code LCO requires appropriate scalar point SA and SB entries on the CBEAM entry or a fatal message will result.
8. Normal buckling modes for beams with nonlinear extensions can be computed through the ANALYSIS=BUCK step. The buckle modes can be evaluated through a linear perturbation step about a linear or nonlinear prestressed state. The current limitation is that lateral buckling mode computations (eg. Lateral-torsional buckling seen in compression flanges of open section beams) are not supported for these elements.
9. The structural element damping coefficient, GE, is not supported on elements which reference PBEMN1.

10. Smeared cross section support for integration codes are as follows:

Primary BEAM Property	INT CODE	SECT	Interpolation	Usage	COMMENTS
PBEAM	LC	S	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Both Thin-Walled and Solid Sections with elastic materials. Euler-Bernoulli Beam	Smeared section properties only are supported. Only linear elastic materials via MAT1 card can be provided.
	LS	S	Linear Interpolation for axial displacements, transverse displacements and rotations	Both Thin-Walled and Solid Sections with elastic materials. Captures transverse shear effects - useful for deep beams.	Smeared section properties only are supported. Only linear elastic materials via MAT1 card can be provided.
	LC or LS	N			Not Supported
	LCC or LCO	S			Not Supported
	LCC or LCO	N			Not Supported
PBEAML	LC or LS	S			Not Supported
	LC	N	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Solid Sections. Euler-Bernoulli Beam	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported

Primary BEAM Property	INT CODE	SECT	Interpolation	Usage	COMMENTS
	LS	N	Linear Interpolation for axial displacements, transverse displacements and rotations	Solid Sections. Captures transverse shear effects - useful for deep beams.	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported
	LCC or LCO	S			Not Supported
	LCC	N	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Thin-Walled Closed Sections without warping. Euler-Bernoulli Beam	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported
	LCO	N	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Thin-Walled Open Sections with warping. Euler-Bernoulli Beam	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported

PBEND**Curved Beam or Pipe Element Property**

Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBEND	PID	MID	A	I1	I2	J	RB	THETAB	
	C1	C2	DI	D2	E1	E2	F1	F2	
	K1	K2	NSM	RC	ZC	DELTAN			

Example:

PBEND	39	1	0.8	0.07	0.04	0.04	10.		
	0.5	0.4	-0.5	0.4					
	0.6	0.6				0.1			

Alternate Format and Example for Elbows and Curved Pipes:

PBEND	PID	MID	FSI	RM	T	P	RB	THETAB	
			NSM	RC	ZC				
PBEND	39	1	1	0.5	0.02	10.	10.		
					0.1				

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 1. and 2. (Integer > 0)
A	Area of the beam cross section. (Real > 0.0)
I1, I2	Area moments of inertia in planes 1 and 2. (Real > 0.0)
J	Torsional stiffness. (Real > 0.0)
FSI	Flag selecting the flexibility and stress intensification factors. See Remark 3. (Integer = 1, 2, or 3)
RM	Mean cross-sectional radius of the curved pipe. (Real > 0.0)
T	Wall thickness of the curved pipe. (Real \geq 0.0; RM + T/2 < RB)
P	Internal pressure. (Real)
RB	Bend radius of the line of centroids. (Real. Optional, see CBEND entry.)
THETAB	Arc angle of element. (Real, in degrees. Optional, see CBEND entry.)
Ci, Di, Ei, Fi	The r,z locations from the geometric centroid for stress data recovery. See Remark 8. (Real)

Descriptor	Meaning
K1, K2	Shear stiffness factor K in K*A*G for plane 1 and plane 2. (Real)
NSM	Nonstructural mass per unit length. (Real)
RC	Radial offset of the geometric centroid from points GA and GB. See Figure 9-128 . (Real)
ZC	Offset of the geometric centroid in a direction perpendicular to the plane of points GA and GB and vector v. See Figure 9-128 . See Remark 9. (Real)
DELTAN	Radial offset of the neutral axis from the geometric centroid, positive is toward the center of curvature. See Figure 9-128 . See Remark 9. (Real; Default is described in Remark 5.)

Remarks:

1. For structural problems, MID must reference a MAT1 material entry.
2. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
3. When the alternate PBEND entry with FSI is used the following options are available:
Using the notation of Dodge and Moore from ORNL-TM-3658, 1972 we define:

The bend characteristic parameter λ as:

$$\lambda = \frac{R_B T}{R_M^2 \sqrt{(1 - \nu^2)}}.$$

The internal loading parameter ψ as

$$\psi = \frac{P R_B^2}{E R_M T}.$$

And the radius ratio parameter γ as

$$\gamma = \frac{R_B}{R_M}.$$

For $T \neq 0$

$$A = 2\pi R_M T$$

$$I_1 = I_2 = \frac{1}{2} A (R_M^2 + \frac{1}{4} T^2)$$

For $T = 0$

$$A = \pi R_M^2$$

$$I_1 = I_2 = \frac{1}{4} A R_M^2$$

For $T \neq 0$

$$\Delta N = R_B \left\{ 1 - \frac{1}{2} \left[\sqrt{(1 - \gamma^{-2} - \frac{1}{4} (\frac{T}{R_B})^2 + (\frac{T}{R_B}) \gamma^{-1})} + \sqrt{(1 - \gamma^{-2} - \frac{1}{4} (\frac{T}{R_B})^2 - (\frac{T}{R_B}) \gamma^{-1})} \right] \right\}.$$

For $T = 0$

$$\Delta N = \frac{1}{2} R_B [1 - \sqrt{(1 - \gamma^{-2})}]$$

Element flexibility calculations:

The shear stiffness factor represents a correction to the section area by computing an effective area $A_s = KA$. Here K is defined as:

$$K = \frac{1}{f_s}$$

f_s is the form factor obtained here from the principle of complementary virtual work. There are many other ways to obtain this factor. By complementary virtual work it is defined as:

$$f_s = \frac{A}{I_1^2} \int \frac{Q^2}{b^2} dA,$$

where Q is the first moment of area and b is the width of section cut.

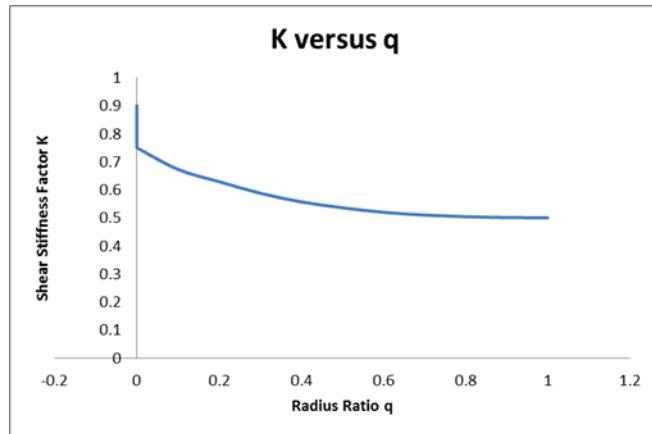
For a solid circular cross-section the above expression yields a value of $K = 0.9$, and for very thin walled pipes the above integral evaluates as $\pi T R_m^5$ and we get a value of $K = 0.5$.

For pipes with general thickness $T > 0$ we define the radius ratio as:

$$q = \frac{R_i}{R_o} = \frac{1 - \frac{1}{2} \frac{T}{R_M}}{1 + \frac{1}{2} \frac{T}{R_M}}.$$

where $T = R_o - R_i$ and R_o is the outer radius of the pipe and R_i is the inner radius of the pipe. The above integral becomes a complicated function in terms of radicals and arc sine functions of q and its explicit integration form is not warranted in view of the approximate nature for f_s . We therefore approximate with the curve fit

$$K_1 = K_2 = \frac{0.75}{1 + \frac{q}{1+q^2}}$$



The in and out of plane flexibility factors K_p used in computing the element flexibility are calculated as:

For FSI = 1

$$K_p = 1.0$$

For FSI = 2 (ASME code Section III, NB-3687.2, NB-3685.2, 1977)

$$K_p = \frac{1.65 R_M^2}{R_B T} \left[\frac{1}{1 + 6 \frac{P R_M}{T E} \left(\frac{R_M}{T} \right)^{\frac{4}{3}} \gamma^{\frac{1}{3}}} \right]$$

where it is recommended that $\lambda \geq 0.2$

For FSI = 3 (Empirical factors from Welding Research Council Bulletin 179, Dodge and Moore)

$$K_p = \frac{1.73}{\lambda} \left[\frac{1}{1 + 1.75 \lambda^{-\frac{4}{3}} \exp(-1.15 \psi^{-\frac{1}{4}})} \right]$$

where it is recommended that $0.05 \leq \lambda \leq 1.0$ and $0.0 \leq \psi \leq 0.1$. (Note in the cited Dodge and Moore the K_p coefficient has a value of 1.66 which was updated in 1991 to the current value of 1.73)

Also we should have $K_p \geq 1.0$ for all FSI.

Stress recovery stress intensification factor calculations:

Define S_1 as the in plane stress intensification factor and S_2 as the out of plane stress intensification factor.

For FSI = 1

$$S_1 = \frac{I_1}{AR_B} \left[\frac{1}{r_i} + \frac{R_B - \Delta N}{\Delta N(R_B + r_i)} \right]$$

$$S_2 = 1.0$$

If $\Delta N = 0.0$, $T = 0.0$, $R_M = 0.0$ then $S_1 = 1.0$.

r_i is the C1, D1, E1, F1 of the (r,z) recovery locations.

For FSI = 2 (ASME code Section III, NB-3687.2, NB-3685.2, 1977)

Define:

$$X_1 = 5.0 + 6\lambda^2 + 24\psi$$

$$X_2 = 17.0 + 600\lambda^2 + 480\psi$$

$$X_3 = X_1 X_2 - 6.25$$

$$X_4 = (1 - \nu^2)(X_3 - 4.5X_2)$$

and

$$S_{1t} = \sin \phi + [(1.5X_2 - 18.75) \sin 3\phi + 11.25 \sin 5\phi] / X_4$$

$$S_{1n} = \nu \lambda (9X_2 \cos 2\phi + 225 \cos 4\phi) / X_4$$

$$S_1 = S_{1t} + S_{1n}$$

$$S_{2t} = \cos \phi + [(1.5X_2 - 18.75) \cos 3\phi + 11.25 \cos 5\phi] / X_4$$

$$S_{2n} = \nu \lambda (9X_2 \sin 2\phi + 225 \sin 4\phi) / X_4$$

$$S_2 = S_{2t} + S_{2n}$$

For data recovery the circumferential angle ϕ is evaluated at 0 deg, 90 deg, 180 deg, and 270 deg. For FSI = 3 (Empirical factors from Welding Research Council Bulletin 179, Dodge and Moore)

$$S_1 = S_2 = \frac{2\lambda^{-\frac{2}{3}}(1.0 + 0.25\gamma^{-1})}{1.0 + \lambda^{-\frac{4}{3}} \exp(-\psi^{-\frac{1}{4}})}.$$

4. The transverse shear stiffness in planes 1 and 2 are $K1*A^*G$ and $K2*A^*G$, respectively. The default values for K1 and K2 on the first format are zero, which means the transverse shear flexibilities ($1/Ki*A^*G$) are set equal to zero. Transverse shear stiffness for the alternate format are automatically calculated for the curved pipe.
5. The neutral axis radial offset from the geometric centroid is default to the

$$\Delta N = \frac{II}{A \cdot RB}$$

It is recommended that the default be used whenever

$$\frac{(RB)^2 A}{II} < 15$$

in which case the default value of ΔN is within 5% of the exact expression for circular or rectangular cross sections. For the alternate format, the neutral axis offset is calculated from an analytical expression for a hollow or solid circular cross section.

The user may compute an exact value for N as follows:

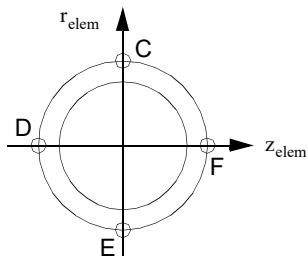
$$\Delta N = \frac{RB}{1 + \frac{(RB)^2 A}{Z}}$$

where

$$Z = \int \frac{r^2 dA}{1 + \frac{r}{RB}}$$

The integration is carried out over the cross section of the element.

6. If T is zero, a solid circular cross section of radius RM is assumed and FSI must be 1.
7. If the first format is used, third-order moments are neglected for the consistent mass matrix. These moments are zero whenever the cross section of the beam is symmetric about both the r and z axes.
8. If the circular cross-sectional property entry format is used, the stress points are automatically located at the points indicated in [Figure 9-127](#).
9. Offset vectors are treated like rigid elements and are therefore subject to the same limitations.
 - Offset vectors are not affected by thermal loads.
 - The specification of offset vectors is not recommended in solution sequences that compute differential stiffness because the offset vector remains parallel to its original orientation. (Differential stiffness is computed in buckling analysis provided in SOLs 105 and 200; SOLs 101,103 and 107 through 112 with STATSUB; and also nonlinear analysis provided in SOLs 106, 129, 153, and 159 with PARAM,LGDISP,1.)



[Figure 9-127 PBEND Circular Cross Section](#)

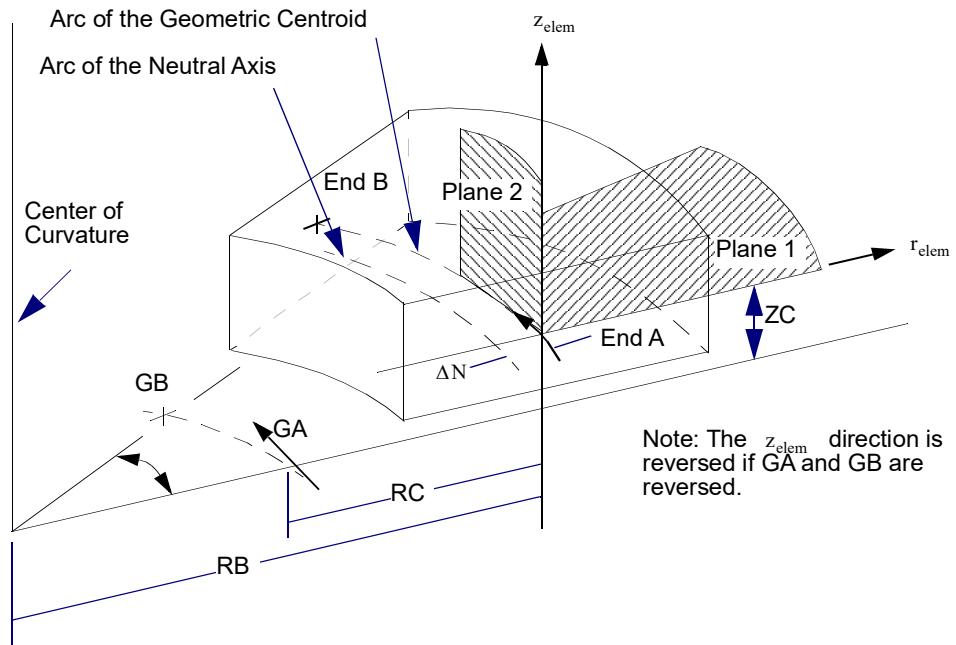


Figure 9-128 PBEND Element Coordinate System

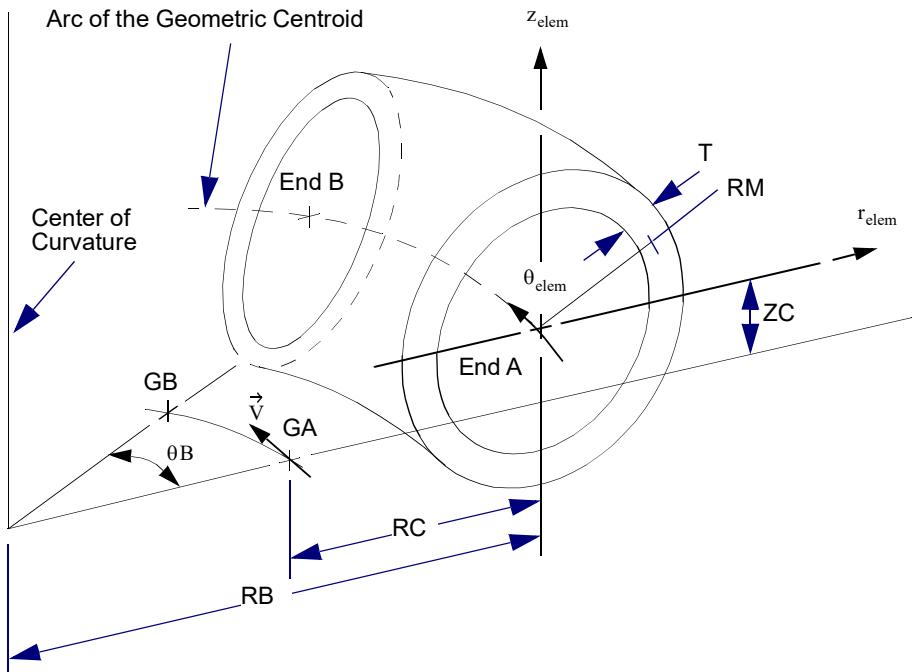


Figure 9-129 PBEND Circular Cross Section Element Coordinate System

10. For RC network solver in thermal analysis, the DELTAN is ignored.
11. PBEND is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEND property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEND entries.

PBMARB6**Arbitrary Beam/Bar Cross Section in SOL 600**

Defines arbitrary BEAM/BAR cross section for use in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
PBMARB6	PID	MID	Nseg	ORDx	ORDy	PreInt	POIS	IPRN	
	Px	Py	F1	F2	F3	F4	F5	F6	
	X1	Y1	X2	Y2	X3	Y3	X4	Y4	1st seg
	X1	Y1	X2	Y2	X3	Y3	X4	Y4	2nd seg

(more rectangular segments, if required)

	X1	Y1	X2	Y2	X3	Y3	X4	Y4	last seg
--	----	----	----	----	----	----	----	----	----------

Example: (Remark 5)

PBMARB6	20	4	2	3	3			3	
	2.0	1.0							
	0.0	1.0	0.8	0.0	0.8	1.0	0.0	1.0	
	0.0	1.0	1.0	1.0	1.0	2.0	0.0	2.0	

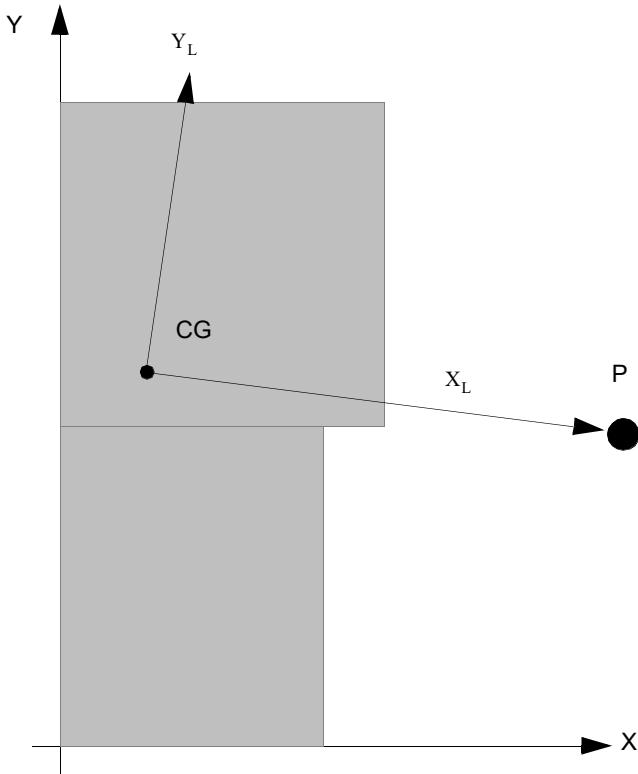
Descriptor	Meaning
PID	Property identification number. Must be unique among all property ID's. (Integer > 0; no Default)
MID	Material identification number. (Integer > 0; no Default)
Nseg	Number of quadrilateral shaped segments. See Remark 2. (Integer; no Default)
ORDx	Order of the integration rule used for each quadrilateral shaped segment in parametric - direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is 5 and the order cannot be larger than 10. (Integer; no Default)
ORDy	Order of the integration rule used for each quadrilateral shaped segment in parametric - direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is the same rule and order as in parametric -direction and the order cannot be larger than 10. (Integer; no Default)
PreInt	Enter 1 to use a pre-integrated section or 0 to use numerical integration through the analysis. If PreInt=0, the section remains elastic. (Integer; Default = 0)
POIS	For a uniform change in cross section, enter the effective Poisson's ratio. This data is not used at this time and its value will be ignored. (Real; Default = 0.0)

Descriptor	Meaning
IPRN	<p>Enter 1 to have the principal axis associated with the largest area moment of inertia to be aligned with the local x-axis.</p> <p>Enter 2 to have the principal axis associated with the smallest area moment of inertia to be aligned with the local x-axis.</p> <p>Enter 3 to have the x-axis of the coordinate system for which the section is being defined to be aligned with the local x-axis of the element (element coordinates). (Integer; Default = 3)</p>
Px	Enter the x-coordinate of a point that, when projected, lies on the positive side of the local x-axis. If the principal moments of inertia are equal, this defines the x-coordinate of a point on the positive local x-axis. This coordinate defaults to $X_{cg}+1$, where X_{cg} is the x-coordinate of the center of gravity of the section in the coordinate system in which the section was entered. The default is used when this field is blank or zero or when the user point coincides with the center of gravity.
Py	Enter the y-coordinate of a point that, when projected, lies on the positive side of the local x-axis. If the principal moments of inertia are equal, this defines the y-coordinate of a point on the positive local x-axis. This coordinate defaults to Y_{cg} , where Y_{cg} is the y-coordinate of the center of gravity of the section in the coordinate system in which the section was entered. The default is used when this field is blank or zero or when the user point coincides with the center of gravity.
	F1 Normal stiffness factor F1. (Real; Default = 1.0)
	F2 Bending stiffness factor for bending about local X axis. (Real; Default = 1.0)
	F3 Bending stiffness factor for bending about local Y Axis. (Real; Default = 1.0)
	F4 Shear stiffness factor for shear in local X direction. (Real; Default = 1.0)
	F5 Shear stiffness factor for shear in local Y direction. (Real; Default = 1.0)
	F6 Torsional stiffness factor. (Real; Default = 1.0)
	X_i, Y_i Coordinates of the 4 corners of each rectangle. Enter Nseg sets of 4 coordinates each. See Remark 3. (Real; no Default)

Remarks:

1. This option corresponds to Marc's BEAM SECT Method D.
2. No more than 100 integration points can exist in any cross section. Pre-integrated sections do not allow stress and strain output in section integration points; only generalized stresses and strains can be requested for output. Non pre-integrated sections cannot have more than 100 segments using single point integration each. For pre-integrated sections, there is no limit on the number of segments
3. The corners are given in counterclockwise order with respect to the local x-y axis.
4. This option applies only to Marc element types 98 (SOL 600 default) and 52. The difference between types 98 and 52 is that 98 has transverse shear and 52 does not have transverse shear.

5. For the example, a section is made up of two rectangles as shown in the figure below. The input for the section defines one section with two quadrilateral segments using a 3x3 Simpson integration. The lower segment has size 0.8x1 and the upper segment has size 1x1. The first principal axis defines the local x-axis and the vector from the center of gravity (CG) to the projection of point P (2.0,1.0) onto this principal axis defines its positive direction. The coordinates of corner points of the segments and the coordinates of the point P have been entered with respect to the x,y-system.



6. If this entry is used, do not enter any other properties (such as pbeam, pbeam1, pbar, pbar1, etc.) for the elements using this property.
 7. If this entry is used, the following parameters must be included in the bulk data:

PARAM,MARCBEAM,0

PARAM,MROUTLAY,N (where N is the number of integration layers, suggested values are N=5 if there is no plasticity and N=11 with plasticity)

PARAM,MARCSLHT,N (N should be the same as specified for MROUTLAY)

PBMNUM6

Four Specific Numerically Integrated BEAM/BAR Cross Section in SOL 600

Defines four specific numerically integrated BEAM/BAR cross section for use in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
PBMNUM6	PID	MID	ITYPE	ORDx	ORDy	PreInt	POIS		
	A	B	C						
	F1	F2	F3	F4	F5	F6			

Example:

PBMNUM6	30	6	2	3	3				
	7.0	2.5							

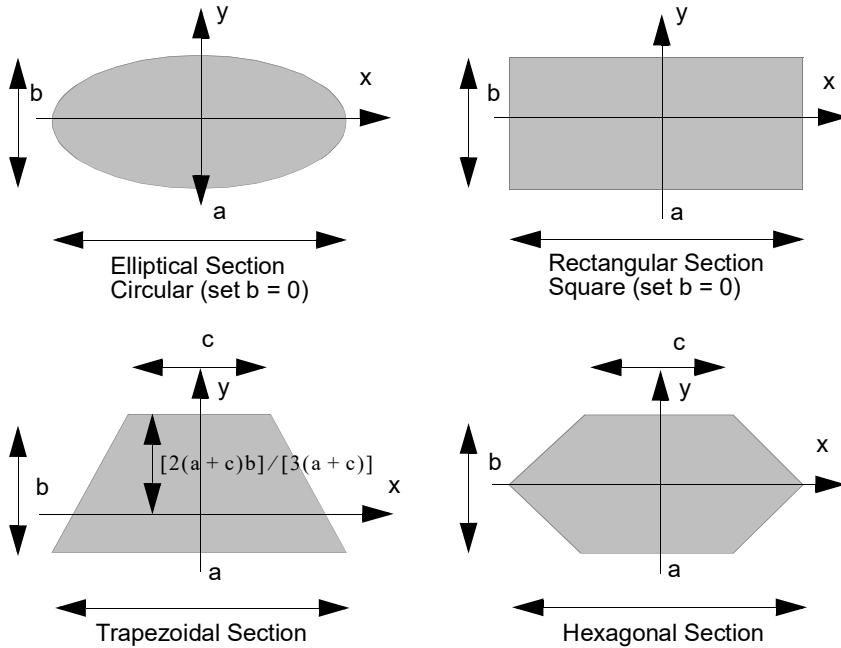
Descriptor	Meaning
PID	Property identification number. (Integer > 0; no Default) Must be unique among all property ID's
MID	Material identification number. (Integer > 0; no Default)
ITYPE	Type of section, see figure below. (Integer; no Default) <ul style="list-style-type: none"> 1 Elliptical 2 Rectangular 3 Trapezoidal 4 Hexagonal
ORDx	For an elliptical section, enter the number of subdivisions in radial direction. The default is 3. For a rectangular or trapezoidal section, enter the order of the integration rule used in local x-direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is 5 and the order cannot be larger than 10. For a hexagonal section, enter the order of the integration rule used in local x-direction over each trapezoidal half. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is 5 and the order cannot be larger than 10. (Integer; no Default)

Descriptor	Meaning
ORDy	For an elliptical section, enter the number of subdivisions in circumferential direction of a 90° sector. The default is 2. For a rectangular or trapezoidal section, enter the order of the integration rule used in local y-direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is what was entered in the first field and the order cannot be larger than 10.
	For a hexagonal section, enter the order of the integration rule used in local y-direction over each trapezoidal half. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton- Cotes rule is used. If the number is negative, a Gauss rule is used. The default is half the order in x-direction plus 1 and the order cannot be larger than 10. For non-Gauss rules, the points in the top row of the lower trapezoid coinciding with the points in the bottom row of the upper trapezoid (i.e., the points coinciding at $y = 0$) are merged together.
PreInt	Enter 1 to use a pre-integrated section or 0 to use numerical integration through the analysis. If PreInt=0, the section remains elastic. (Integer; Default = 0)
POIS	For a uniform change in cross section, enter the effective Poisson's ratio. This data is not used at this time and its value will be ignored. (Real; Default = 0.0)
A	First dimension of the cross section (Real; no Default) <ul style="list-style-type: none"> Ellipse A is the diameter of the circle or the length of the ellipse in local x. Rectangle A is the length of the square or the rectangle in local x. Trapezoid A is the width of the trapezoid in local x on minus local y side. Hexagon A is the width of the hexagon in local x at $y = 0$.
B	Second dimension of the cross section (Real; Default B=A) <ul style="list-style-type: none"> Ellipse B is the height of the ellipse in local y. Rectangle B is the height of the rectangle in local y. Trapezoid B is the height of the trapezoid in local y. Hexagon B is the height of the hexagon in local y.
C	Third dimension of the cross section if required (Real; Default = 0.0) <ul style="list-style-type: none"> Ellipse C is not used. Leave blank or enter 0. Rectangle C is not used. Leave blank or enter 0. Trapezoid C is the width of the trapezoid in local x on the plus local y side. Hexagon C is the width of the hexagon in local x on either local y side.
F1	Normal stiffness factor F1 (Real; Default = 1.0)
F2	Bending stiffness factor for bending about local X axis (Real; Default = 1.0)
F3	Bending stiffness factor for bending about local Y Axis (Real; Default = 1.0)
F4	Shear stiffness factor for shear in local X direction (Real; Default = 1.0)

Descriptor	Meaning
F5	Shear stiffness factor for shear in local Y direction (Real; Default = 1.0)
F6	Torsional stiffness factor. (Real; Default = 1.0)

Remarks:

1. This option corresponds to Marc's BEAM SECT Method C.
2. No more than 100 integration points can exist in any cross section. Pre-integrated sections do not allow stress and strain output in section integration points; only generalized stresses and strains can be requested for output. Non pre-integrated sections cannot have more than 100 segments using single point integration each. For pre-integrated sections, there is no limit on the number of segments
3. This option applies only to Marc element types 98 (SOL 600 default) and 52. The difference between types 98 and 52 is that 98 has transverse shear and 52 does not have transverse shear.
4. If this entry is used, do not enter any other properties (such as pbeam, pbeam1, pbar, pbar1, etc.) for the elements using this property.
5. The second continuation entry may be eliminated if F1 through F6 take the default values.
6. For the rectangle, if B=A or B=0.0 the cross section is a square.
7. For the Ellipse, if B=A or B=0.0, the cross section is a circle.



8. If this entry is used, the following parameters must be included in the bulk data:

PARAM,MARCBEAM,0

PARAM,MROUTLAY,N

(where N is the number of integration layers, suggested values are
N=5 if there is no plasticity and N=11 with plasticity)

PARAM,MARCSLHT,N

(N should be the same as specified for MROUTLAY)

PBMSECT

Arbitrary Cross-Section for CBEAM

Defines the shape of arbitrary cross-section for CBEAM element.

Format:

1	2	3	4	5	6	7	8	9	10
PBMSECT	PID	MID	FORM						
Data description for arbitrary section									

Example:

PBMSECT	1	10	GS						
OUTP=10,INP=20									
PBMSECT	1	10	CP						
OUTP=10,BRP=20,T=1.0,T(11)=[1.2,PT=(123,204)], NSM=0.01									

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
FORM	Cross-section form. (Character) See Remark 1.

Remarks:

- Options for FORM are

GS	General Section
OP	Open Profile
CP	Closed Profile

- Keywords for describing the arbitrary cross-section:

For GS, OP and CP:

- OUTP = value(Integer > 0); points to ID of a SET1 or SET3 that defines the OUTer Perimeter for FORM=GS or the center line for FORM=CP (or OP) by traversing through all the POINTs in the SET.
- OUTM = value(Integer > 0), points to the ID of BEGIN BULK ARBMODEL. OUTM is designed specifically for arbitrary beam cross section with finite element discritization already available. Note that OUTM must not appear together with other keyword, such as OUTP or INP, on a PBMSECT.

For GS only:

INP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that defines a INner Perimeter by traversing through all the POINTs in the SET.

For OP and CP:

BRP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that specifies a BRanch. The rules for BRP are:

- BRP must start from and/or end on OUTP.
- Segment length, defined as distance between two neighboring points on BRP or OUTP, is preferably to be longer than the segment thickness. Closely spaced points creates denser mesh for the cross section and does not increase accuracy of properties significantly.

T(id) = [value(Real > 0.0),PT=(pid1,pid2)]; specifies the thickness of a segment in profile. PT=(pid1,pid2) defines the start and end points of line segment(s). The rules for T(id) are:

- For CP and OP, it is a requirement to have a T(id) without PT=(pid1,pid2) to serve as a default thickness for all segments which do not have its thickness specified. This requirement is valid even when the thickness for every segment is specified.
- pid1 and pid2 under PT=() do not need to be neighboring points of a SET. However, pid1 and pid2 must hold its position as in SETx for OUTP and/or BRP.
- Same segment may be covered by different T(id). T(id) are processed sequentially as specified. The thickness of the last T(id) for a segment is used as the thickness of a segment. Use 'PARAM,ARBMNOW,1' to turn off segment property overwrite.
- For CP, T(id) of segment that close the profile must have (pid1=ID of last point in OUTP, pid2=ID of first point in OUTP). Note that, if input for pid1 and pid2 are reversed, T(id) will cover all segments in OUTP except the one that close the profile.

- Core(id) = [PCID,PT=(pid1,pid2)]; specifies the composite layup for CORE part of composite. PCID is the ID of a PCOMPi/PCOMPG Bulk Data entry. PT=(pid1,pid2) defines the start and end points of line segment(s) which utilizes PCID.
- Core=PCID is acceptable input and is used as default which is applicable to all segments that are not specifically defined with Core(id).
 - Thickness continuity of a ply must be maintained. Thickness change from segment to segment is not allowed.
 - For CP, the closing segment should always use the default Core=pcid.
 - No ply should go over the profile line which is defined by OUTP and BRP. If situation arises, split a ply into two in such a way the no ply go over the profile line.
- Layer(id) = [PCID,SETID]; specifies the composite layup for additional Layer(s) that wraps around Core. PCID is the ID of a PCOMP/PCOMPG Bulk Data entry. SETID selects a SET1/SET3 with POINT IDs.
- NSM = value(Real > 0.0), specifies non-structural mass per unit length.
- (id) = integer (>0) identifies INP, BRP or T which is not required if a single entity appears in the PBMSECT entry. For T, the T(id) can be used to identify the particular thickness to be designed in SOL 200.
3. Stress data recovery points are selected automatically from all points of a PBMSECT with GS form. The points with maximum and/or minimum coordinates in X and Y axes are more likely to be picked. For PBMSECT with CP or OP form, the stress data recovery points are selected from points with computed coordinates that actually encircle the profile. Similar to GS form, the points with extreme coordinates are more likely to be selected.
 4. Only the POINT entry ID should be listed under SET1 or SET3 entries which, in turn, are referenced by OUTP, INP and BRP. In addition, the POINT entry for defining an arbitrary beam cross section must have the CP and X3 fields left blank.

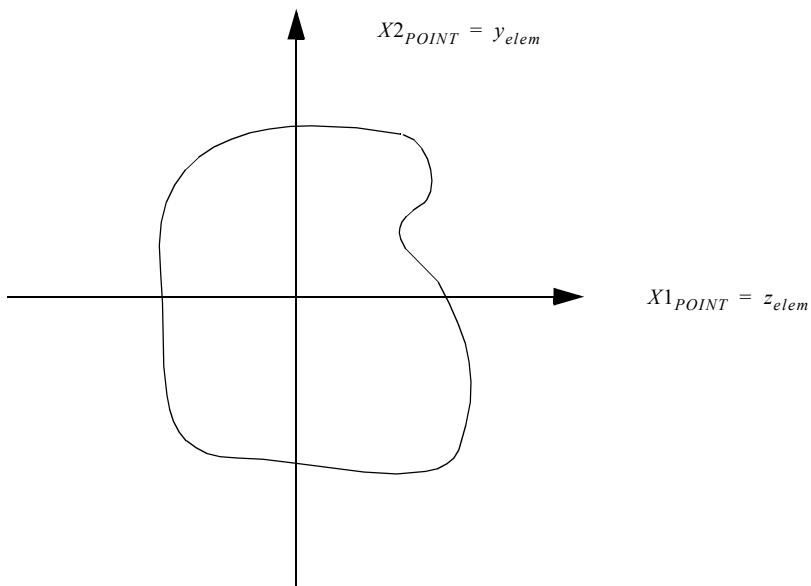


Figure 9-130 Arbitrary Cross-Section Definition

5. Current implementation of PBMSECT supports constant section beam only.
6. Note that keyword Core can be abbreviated as 'C'. Similarly, keyword Layer can be abbreviated as 'L'.
7. If Core and/or Layer appears in PBMSECT, the PBMSECT ID can not be referenced on CBEAM. Instead, it should be referenced on CBEAM3. Note, however, that a CBEAM3 cannot be used with any other of the above options except the Core and/or Layer option. The use of any other option results in a FATAL. In addition, it is recommended that CBEAM3 referencing composite PBMSECT has 3 nodes and 3 warping DOFs.'
8. If OUTM=arbid is utilized on PBMSECT, element connection, grid location, PSHELL and material Bulk Data entries must be provided after 'BEGIN ARBMODEL=arbid'.
9. Note that the 'arbid' used under 'BEGIN ARBMODEL' is considered global and can be referenced by PBMSECT with OUTM=arbid in different 'BEGIN SUPER' Bulk Data Section for Part Superelements (SE).
10. PBMSECT with Core or Layer must be utilized along with 'PARAM,ARBMSTYP,TIMOSHEN' in the Bulk Data Section.
11. The entry computes, based on an internally generated finite element analysis using a 2D mesh of the cross-section, the following:

$$A = \int dy dz$$

$$I_1 = \int y^2 dy dz$$

$$I_2 = \int z^2 dy dz$$

$$I_{12} = \int yz dy dz$$

where the above integrals are evaluated by numerical integration.

For a beam cross-section, the warping function, ϕ , satisfies the equation

$$\frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0$$

with boundary

$$\left(\frac{\partial \phi}{\partial y} + z \right) n_y + \left(\frac{\partial \phi}{\partial z} - y \right) n_z = 0$$

where n_y and n_z are the direction cosine of the normal to the boundary.

Then, the torsion constant is defined as

$$J = I_1 + I_2 - \int \left[\frac{\partial \phi}{\partial z} \frac{\partial \phi}{\partial y} \right] \begin{Bmatrix} -y \\ z \end{Bmatrix} dA$$

The load equilibrium of the beam cross-section can be resolved into two Poisson equations for the shear forces in the y and z direction as:

$$\nabla^2 f_y = -y/I_1$$

$$\nabla^2 f_z = -z/I_2$$

then, the shear stiffness factor is defined as

$$K_1 = \left[\frac{A}{I_2} \int z f_z dA \right]^{-1}$$

$$K_2 = \left[\frac{A}{I_1} \int y f_y dA \right]^{-1}$$

The warping constant is defined as

$$C_w = \int \phi^2 dA - \left[y_{sc} z_{sc} \right] [I] \begin{Bmatrix} y_{sc} \\ z_{sc} \end{Bmatrix}$$

The shear center is defined as

$$Q_1 = \int z dA \quad Q_2 = \int y dA$$

$$N_1 A = Q_1 / A \quad N_2 A = Q_2 / A$$

12. PBMSECT is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBMSECT property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.

PBRSECT**Arbitrary Cross-Section for CBAR**

Defines the shape of arbitrary cross-section for CBAR element.

Format:

1	2	3	4	5	6	7	8	9	10
PBRSECT	PID	MID	FORM	NSM					
Data description for arbitrary section									

Example:

PBRSECT	1	10	GS						
OUTP=10,INP=20									
PBRSECT	1	10	CP						
OUTP=10,BRP=20,T=1.0,T(11)=[1.2,PT=(123,204)]									

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
FORM	Cross-section form. (Character) See Remark 1.
NSM	Non-structural mass per unit length. (Real ≥ 0.0 ; Default = 0.0)

Remarks:

- Options for FORM are

GS	General Section
OP	Open Profile
CP	Closed Profile

- Keywords for describing the arbitrary cross-section:

For GS, OP and CP:

OUTP = value(Integer > 0); points to ID of a SET1 or SET3 that defines the OUTer Perimeter for FORM=GS or the center line for FORM=CP (or OP) by traversing through all the POINTs in the SET.

For GS only:

INP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that defines a INner Perimeter by traversing through all the POINTs in the SET.

For OP and CP:

BRP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that specifies a BRanch.
The rules for BRP are:

- BRP must start from and/or end on OUTP.
- Segment length, defined as distance between two neighboring points on BRP or OUTP, is preferably to be longer than the segment thickness. Closely spaced points creates denser mesh for the cross section and does not increase accuracy of properties significantly.

T(id) = [value(real > 0.0),PT=(pid1,pid2)]; specifies the thickness of a segment in profile. PT=(pid1,pid2) defines the end points of a straight line segment. The rules for T(id) are:

- For CP and OP, it is a requirement to have a T(id) without PT=(pid1,pid2) to serve as a default thickness for all segments which do not have its thickness specified. This requirement is valid even when the thickness for every segment is specified.
- pid1 and pid2 under PT=() do not need to be neighboring points of a SET. However, pid1 and pid2 must hold its position as in SETx for OUTP and/or BRP.
- Same segment may be covered by different T(id). T(id) are processed sequentially as specified. The thickness of the last T(id) for a segment is used as the thickness of a segment. Use 'PARAM,ARBMNOW,1' to turn off segment property overwrite.
- For CP, T(id) of segment that close the profile must have (pid1=ID of last point in OUTP, pid2=ID of first point in OUTP). Note that, if input for pid1 and pid2 are reversed, T(id) will cover all segments in OUTP except the one that close the profile.

NSM = value(Real > 0.0), specifies non-structural mass per unit length.
(id) = integer (>0) identifies INP, BRP or T which is not required if a single entity appears in the PBRSECT entry. For T, the T(id) can be used to identify the particular thickness to be designed in SOL 200.

3. Stress data recovery points are selected automatically from all points of a PBRSECT with GS form. The points with maximum and/or minimum coordinates in X1 and/or X2 axes are more likely to be picked. For PBRSECT with CP or OP form, the stress data recovery points are selected from points with computed coordinates that actually encircle the profile. Similar to GS form, the points with extreme coordinates are more likely to be selected.
4. Only the POINT entry ID should be listed under SET1 or SET3 entries which, in turn, are referenced by OUTP, INP and BRP. In addition, the POINT entry for defining an arbitrary beam cross section must have the CP and X3 fields left blank.
5. See Remark 11. of the PBMSECT entry for the theory used to compute the cross-sectional properties.

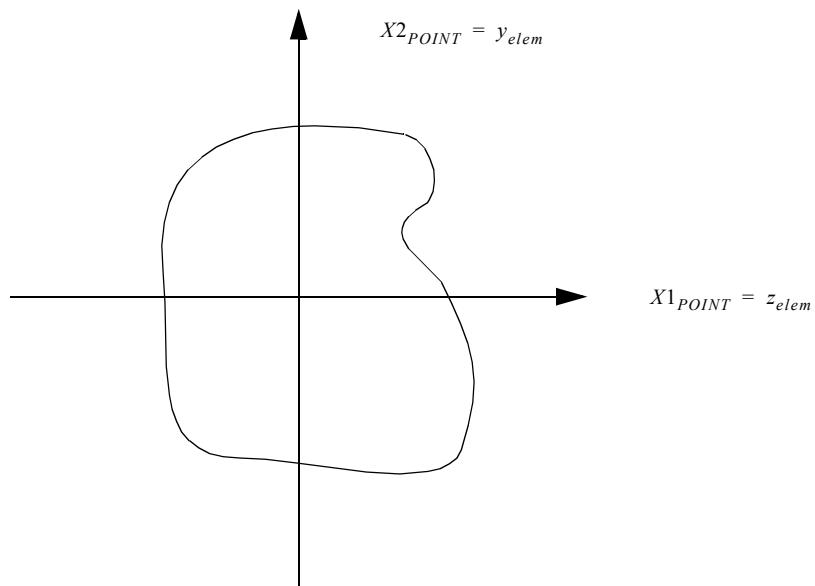


Figure 9-131 Arbitrary Cross-Section Definition

6. PBRSECT is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBRSECT property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBAR, PBARL, PBRSECT entries.

PBSH2DT

2-D Linear/Nonlinear Connection Properties using Table Lookup

Defines frequency-dependent properties of a two-dimensional element (CBUSH2D) using TABLED1 lookup.

Format:

1	2	3	4	5	6	7	8	9	10
PBSH2DT	PID	"K"	K11	K22	K12	K21			
		"B"	B11	B22	B12	B21			
		"M"	M11	M22	M12	M21			

Descriptor	Meaning
PID	Property id of a PBUSH2D entry (see Remark 1.)
K11	TABLED1 used to define frequency-dependent stiffness in the T1 direction (Integer >0 or blank)
K22	TABLED1 used to define frequency-dependent stiffness in the T2 direction (Integer >0 or blank)
K12	TABLED1 used to define frequency-dependent cross coupling stiffness (Integer >0 or blank)
K21	TABLED1 used to define frequency-dependent cross coupling stiffness (Integer >0 or blank)
B11	TABLED1 used to define frequency-dependent damping in the T1 direction (Integer >0 or blank)
B22	TABLED1 used to define frequency-dependent damping in the T2 direction (Integer >0 or blank)
B12	TABLED1 used to define frequency-dependent cross coupling damping (Integer >0 or blank)
B21	TABLED1 used to define frequency-dependent cross coupling damping (Integer >0 or blank)
M11	TABLED1 used to define frequency-dependent mass in the T1 direction (Integer >0 or blank)
M22	TABLED1 used to define frequency-dependent mass in the T2 direction (Integer >0 or blank)
M12	TABLED1 used to define frequency-dependent cross coupling mass (Integer >0 or blank)
M21	TABLED1 used to define frequency-dependent cross coupling mass (Integer >0 or blank)

Remarks:

1. A PBUSH2D with the same PID must exist. The values from the PBUSH2D will be used as the nominal stiffness, damping, and mass. Linear statics, normal modes, reduction, and superelement processing will use the nominal values. The values from the selected TABLED1 entries will be used in any frequency-dependent loop.
2. Any field left blank indicates that the associated stiffness, damping, or mass is not frequency-dependent and the nominal values will be used for that term in the solution.

PBUSH**Generalized Spring-and-Damper Property**

Defines the nominal property values for a generalized spring-and-damper structural element.

Format:

1	2	3	4	5	6	7	8	9	10
PBUSH	PID	"K"	K1	K2	K3	K4	K5	K6	
		"B"	B1	B2	B3	B4	B5	B6	
		"GE"	GE1	GE2	GE3	GE4	GE5	GE6	
		"RCV"	SA	ST	EA	ET			
		"M"	M						
		"T"	ALPHA	TREF	COINL				

Example 1:

Stiffness and structural damping are specified.

PBUSH	35	K	4.35	2.4				3.1	
		GE	.06						
		RCV	7.3	3.3					

Example 2:

Damping force per unit velocity are specified.

PBUSH	35	B	2.3						
-------	----	---	-----	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
"K"	Flag indicating that the next 1 to 6 fields are stiffness values in the element coordinate system. (Character)
Ki	Nominal stiffness values in directions 1 through 6. See Remarks 2. and 3. (Real; Default = 0.0)
"B"	Flag indicating that the next 1 to 6 fields are force-per-velocity damping. (Character)
Bi	Nominal damping coefficients in direction 1 through 6 in units of force per unit velocity. See Remarks 2., 3., and 9. (Real; Default = 0.0)
"GE"	Flag indicating that the next fields, 1 through 6 are structural damping constants. See Remark 7. (Character)
GEi	Nominal structural damping constant in directions 1 through 6. See Remarks 2. and 3. and 9. (Real; GE1 default = 0.0, GE2-6, defaults described in Remark 9.)

Descriptor	Meaning
"RCV"	Flag indicating that the next 1 to 4 fields are stress or strain coefficients. See Remarks 4. and 5.(Character).
SA	Stress recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ST	Stress recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)
EA	Strain recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ET	Strain recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)
"M"	Flag indicating that the following entries are mass properties for the CBUSH element. If inertia properties (Iij)are desired CONM2 should be used.
M	Lumped mass of the CBUSH. (Real \geq 0.0; Default=0.0)
"T"	Flag indicating that the following entries are thermal properties for the CBUSH element. See Remark 15.. (Character)
ALPHA	Thermal expansion coefficient for the CBUSH. (Real; Default=0.0)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0, See Remark 15.)
COINL	Length of a CBUSH with coincident grids. (Real; Default=0.0, COINL \geq 0.0)

Remarks:

1. Ki, Bi, or GEi may be made frequency dependent for both direct and modal frequency response by use of the PBUSHT entry.
2. The nominal values are used for all analysis types except frequency response and nonlinear analysis. For modal frequency response, the normal modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency.
3. If PARAM,W4 is not specified, GEi is ignored in transient analysis.
4. The element stresses are computed by multiplying the stress coefficients with the recovered element forces. $\sigma_i = F_i \cdot SA$ or $\sigma_i = M_i \cdot ST$
5. The element strains are computed by multiplying the strain coefficients with the recovered element displacements. $\varepsilon_i = U_i \cdot EA$ or $\varepsilon_i = \theta_i \cdot ET$
6. The "K", "B", "GE", "RCV", or "M" entries may be specified in any order.
7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.
8. Applicable fields refer to directions in the element's coordinate system.

9. For upward compatibility, if ONLY GE1 is specified on a PBUSH entry and GEi, i = 2 – 6 are all blank on the PBUSH entry, then the single structural damping GE1 is applied to all defined Ki for the PBUSH entry. In this case, GEi, i = 2 - 6, blank is not equivalent to 0.0 but a special flag.

For any PBUSH entry that has any GEi, i = 2 – 6 numerically specified including a 0.0, the fields GEi are all treated separately and any GEi, i = 2 - 6 field with no entry specified is defaulted to 0.0. In this case, for any GEi, i = 2 – 6, blank is equivalent to 0.0 and the results are radically different from the no GEi, i = 2- 6 specified as described above.

Thus for the entry:

1	2	3	4	5	6	7	8	9	10
PBUSH	3303000	K	653.	4000.	460.	10.+3	10.+3	10.+3	
		GE	.05						

The GE=.05 will be replicated to fields 5 through 9.

While for the entry:

1	2	3	4	5	6	7	8	9	10
PBUSH	3303000	K	653.	4000.	460.	10.+3	10.+3	10.+3	
		GE	.05	0.0					

The element x-direction will have a GE=.05 while the other 5 directions will have an associated GE=0.0

The entry:

1	2	3	4	5	6	7	8	9	10
PBUSH	3303000	K	653.	4000.	460.	10.+3	10.+3	10.+3	
		GE	.05		.02				

is equivalent to the full entry:

1	2	3	4	5	6	7	8	9	10
PBUSH	3303000	K	653.	4000.	460.	10.+3	10.+3	10.+3	
		GE	.05	.0	.02	.0	.0	.0	

IFPSTAR introduced an incompatibility with the above rules in Nastran versions 2014 to 2017. For Nastran version 2018 IFPSTAR, the above original rules have been restored. However, the previous incompatible IFPSTAR results can be recovered with bulk data entry MDLPRM, GEV1417, 1 or NASTRAN, SYSTEM(754)=1 in which case the last example becomes equivalent to the full entry:

1	2	3	4	5	6	7	8	9	10
PBUSH	3303000	K	653.	4000.	460.	10.+3	10.+3	10.+3	
		GE	.05	.05	.02	.05	.05	.05	

10. For SOL 600, it is not necessary to enter PBUSH if a PBUSHT with the same ID is used in the model. Omitting the PBUSH entry if a PBUSHT with the same ID is entered.

11. For SOL 600, the defaults for SA, ST, EA and ET are 1.0E-10.
12. The mass properties are assumed located at the point defined by S (or S1, S2, S3 if OCID>-1) on the CBUSH entry. They are assumed to be in the element coordinate system. Define $\alpha = S$ for the case OCID= -1 (default). See CBUSH entry. For the case OCID> -1, define r_A, r_B to be the vector from grid GA to (S1,S2,S3) and from grid GB to (S1,S2,S3) respectively, then

$$\alpha = \frac{|r_A|}{|r_A| + |r_B|}$$

$$|r_A| = \sqrt{(GA1 - S1)^2 + (GA2 - S2)^2 + (GA3 - S3)^2}$$

$$|r_B| = \sqrt{(GB1 - S1)^2 + (GB2 - S2)^2 + (GB3 - S3)^2}$$

Then the diagonal contribution to GA for the mass matrix in the element coordinate system is:

$$M11A = M22A = M33A = (1 - \alpha) * M$$

The diagonal contribution to GB for the mass matrix in the element coordinate system is:

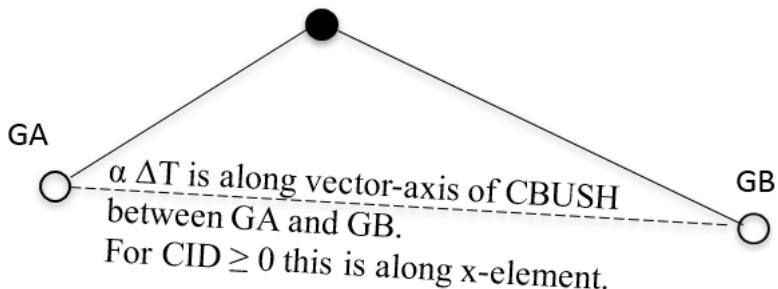
$$M11B = M22B = M33B = (\alpha) * M$$

13. PBUSH is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBUSH property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBUSH entries.
14. The CBUSH is a self-equilibrating element which means that for a CBUSH of finite length, its translational stiffness terms couple with rotational stiffness for rotational degrees of freedom. However, this coupling is dependent on the orientation of the CBUSH element. Thus there may be times when no explicit rotational stiffness is specified along a direction for which translational stiffness was specified that there is also no coupled rotational stiffness for that direction.

In modal analysis, a CBUSH of finite length having rotational degrees of freedom with no rotational stiffness (coupled or explicit) and no associated mass on the grids may cause massless mechanisms. A massless mechanism means the natural frequency for this rotational mode approaches the limit of zero rotational stiffness divided by zero mass, which is an undefined quantity. If the elements lie along a global coordinate axis, the mass term is identically zero, which leads to very large negative or positive eigenvalues and is usually beyond any reasonable search region. If the elements are skewed from the global axes, the eigenvalues may be computed at any value (including negative) because of the indeterminacy caused by numerical truncation.

15. The thermal expansion for the CBUSH will be calculated if the user supplies the thermal expansion coefficient ALPHA and TEMPERATURE(LOAD) is requested. See TEMPERTURE Case Control command for thermal loading rules.

Bush Location



PBUSH1D

Rod Type Spring-and-Damper Property

Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBUSH1D	PID	K	C	M		SA	SE		
	"SHOCKA"	TYPE	CVT	CVC	EXPVT	EXPVC	IDTS		
			IDECS	IDECS	IDESTD	IDECS			
	"SPRING"	TYPE	IDT	IDC	IDTDU	IDCDU			
	"DAMPER"	TYPE	IDT	IDC	IDTDV	IDCDV			
	"GENER"	TYPE	IDT	IDC	IDTDU	IDCDU	IDTDV	IDCDV	

Example:

PBUSH1D	35	3000.	200.	300.					
	SHOCKA	TABLE	2.2	1.2	1.		200		

The continuation entries are optional. The four options, SHOCKA, SPRING, DAMPER, and GENER can appear in any order.

Descriptor	Meaning	Default
PID	Property identification number. (Integer > 0).	Required
K	Stiffness. (Real ≥ 0).	See Remark 1.
C	Viscous damping. (Real ≥ 0).	See Remarks 1. and 2.
M	Total mass of the element. (Real ≥ 0).	Blank
SA	Stress recovery coefficient [1/area]. (Real ≥ 0).	Blank
SE	Strain recovery coefficient [1/length]. (Real ≥ 0).	Blank

Descriptor	Meaning	Default
SHOCKA	Character string specifying that the next 10 fields are coefficients of the following force versus velocity/displacement relationship. (Character). $F(u, v) = C_v \cdot S(u) \cdot \text{sign}(v) \cdot v ^{\text{EXPV}}$ <p>The force F, the displacement u, and the velocity v, are in the axial direction of the damper. The axis of the damper is defined by the two connecting grid points GA and GB on the CBUSH1D Bulk Data entry. The displacement u and the velocity v, are the relative displacement and the relative velocity with respect to the grid point GA. The scale factor S(u) must be defined with a table or with an equation.</p>	
TYPE	Character string indicating the type of definition. (Character). For TYPE = EQUAT, the fields IDETS, IDECS, IDETSD, and IDECSD are identification numbers of DEQATN entries. For TYPE = TABLE the field IDTS is an identification number of a TABLEDi entry. If no character string is provided (blanks), TYPE = TABLE is set.	TABLE
CVT	Viscous damping coefficient C_V for tension $v > 0$, force per unit velocity. (Real).	Required for SHOCKA
CVC	Viscous damping coefficient C_V for compression $v > 0$, force per unit velocity. (Real).	CVT
EXPVT	Exponent of velocity EXPV for tension $v > 0$. (Real).	1.
EXPVC	Exponent of velocity EXPV for compression $v < 0$. (Real).	EXPVT
IDTS	Identification number of a TABLEDi entry for tension and compression if TYPE = TABLE. The TABLEDi entry defines the scale factor S, versus displacement u.	Required for SHOCKA and TYPE=TABLE
IDECS	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the scale factor S, versus displacement u, for tension $u > 0$.	Required for SHOCKA and TYPE=EQUAT
IDETSD	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the scale factor S, versus displacement u, for compression $u < 0$.	IDECS
	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the scale factor S, with respect to the displacement u, for tension $u > 0$.	Required for SHOCKA and TYPE=EQUAT

Descriptor	Meaning	Default
IDECSID	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the scale factor S, with respect to the displacement u, for compression $u < 0$.	IDETSD
SPRING	Character string specifying that the next 5 fields define a nonlinear elastic spring element in terms of a force versus displacement relationship. (Character).	
	$F(u) = F_T(u)$	
	Tension is $u > 0$ and compression is $u < 0$.	
DAMPER	Character string specifying that the next 5 fields define a nonlinear viscous element in terms of a force versus velocity relationship. (Character).	
	$F(v) = F_T(v)$	
	Tension is $v > 0$ and compression is $v < 0$.	
GENER	Character string specifying that the next 7 fields define a general nonlinear elastic spring and viscous damper element in terms of a force versus displacement and velocity relationship. (Character). For this element, the relationship can only be defined with TYPE=EQUAT.	
	$F(u, v) = F_T(u, v)$	
	Tension is $u > 0$ and compression is $u < 0$. For SPRING, DAMPER, and GENER, the remaining fields are	
TYPE	Character string indicating the type of definition. (Character). For TYPE = EQUAT the following fields are identification numbers of DEQATN entries. For TYPE = TABLE the following field is an identification number of a TABLEDi entry. For "GENER", TYPE = blank or "EQUAT" is allowed..	Required for SPRING or DAMPER
IDT	Identification number of a DEQATN entry for tension if TYPE = EQUAT. Identification number of a TABLEDi entry for tension and compression if TYPE = TABLE.	Required for SPRING, DAMPER, and GENER
IDC	Identification number of a DEQATN entry for compression if TYPE = EQUAT. Is ignored for TYPE = TABLE.	IDT

Descriptor	Meaning	Default
IDTDU	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the displacement u, for tension $u > 0$. For SPRING and GENER only.	Required if TYPE=EQUAT
IDCDU	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the displacement u, for compression $u < 0$. For SPRING and GENER only.	IDTDU
IDTDV	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the velocity v, for tension $v > 0$. For DAMPER and GENER only.	Required if TYPE=EQUAT
IDCDV	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the velocity v, for compression $v < 0$. For DAMPER and GENER only.	IDCDT

Remarks:

1. For a linear spring K a default value of 0.0 is allowed by not specifying the K entry. For a linear damper C a default value of 0.0 is allowed by not specifying the C entry. If any nonlinearity is specified (SHOCKA, SPRING, DAMPER, GENER) $K > 0$ is required. If K is not provided in the presence of any nonlinearity, the element will behave as a linear element.
2. The damping C and mass M are ignored in static solution sequences.
3. The parameters defined on the continuation entries are used in nonlinear solution sequences only.
4. The linear parameters K and C are used in all solution sequences unless parameters on continuation entries are defined and a nonlinear solution sequence is used. Then, the parameters K and C are used for initial values in the first iteration of the first load step and the parameters from continuation entries overwrite the linear parameters thereafter. When SHOCKA, SPRING or GENER are specified, K is overwritten. When SHOCKA, DAMPER or GENER is specified, C is overwritten.
5. PBUSH1D may only be referenced by CBUSH1D elements in the residual structure which do not attach to omitted degrees-of-freedom.
6. The continuation entries SHOCKA, SPRING, DAMPER and GENER may be specified in any order. If more than one continuation entry is defined, then the forces of SHOCKA, SPRING, etc. are added. Multiple continuation entries of the same kind are not allowed, for example, multiple SPRING continuation entries.
7. For TYPE = TABLE, values on the TABLEDi entry are for tension and compression. If table values $f(u)$ are provided only for positive values $u > 0$, then it is assumed that $f(-u) = -f(u)$.
8. For TYPE = EQUAT, the equations for tension and compression can be different. If the identification numbers for compression are left blank, it is assumed that the equation for tension is also valid for compression.

9. PBUSH1D is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBUSH1D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBUSH1D entries.

10. The linear FORCE (STRESS) output is:

AXIAL-VELOCITY, AXIAL-STRESS

For nonlinear FORCE (STRESS) output (Continuation entry on PBUSH1D.) the following addition output where applicable is:

AXIAL-VELOCITY, AXIAL-STRESS, AXIAL-STRAIN, PLASTIC-STRAIN,
STATUS

All seven header titles appear in both linear and nonlinear.

11. While computing Single Point Forces of Constraint (SPCFORCES), the constraint relationships exist in stiffness for linear spring. However, the nonlinear elements treat the constraints as loads on set. As a result, the SPCFORCES results are different between linear and nonlinear elements.

PBUSH2D

2-D Linear/Nonlinear Connection Properties

Defines linear and nonlinear properties of a two-dimensional element (CBUSH2D entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBUSH2D	PID	K11	K22	B11	B22	M11	M22		
	SQUEEZE	BDIA	BLEN	BCLR	SOLN	VISCO	PVAPCO		
	NPORT	PRES1	THETA1	PRES2	THETA2	OFFSET1	OFFSET2		

Example:

PBUSH2D	1000	50.0	150.0	0.02	0.02				
	SQUEEZE	1.0	2.0	0.05	LONG	2.1	300.0		
	2	100.0	30.0	120.0	90.0	0.01	0.0		

OR (to include cross coupling terms)

PBUSH2D	PID	K11	K22	B11	B22	M11	M22		
	CROSS	K12	K21	B12	B21	M12	M21		

Descriptor	Meaning
PID	Property identification number (Integer > 0; Required).
K11	Nominal stiffness in T1 rectangular direction (Real; Required).
K22	Nominal stiffness in T2 rectangular direction (Real; Required).
B11	Nominal damping in T1 rectangular direction (Real; Default = 0.0).
B22	Nominal damping in T2 rectangular direction (Real; Default = 0.0).
M11	Nominal acceleration-dependent force in T1 direction (Real; Default = 0.0).
M22	Nominal acceleration-dependent force in T2 direction (Real; Default = 0.0).
'SQUEEZE'	Indicates that squeeze-film damper will be specified (Character; Required).
BDIA	Inner journal diameter. (Real > 0.0; Required)
BLEN	Damper length. (Real > 0.0; Required).
BCLR	Damper radial clearance (Real > 0.0; Required).
SOLN	Solution option: LONG or SHORT bearing (Character; Default = LONG).
VISCO	Lubricant viscosity (Real > 0.0; Required).
PVAPCO	Lubricant vapor pressure (Real; Required).
NPORT	Number of lubrication ports: 1 or 2 (Integer; no Default).
PRES1	Boundary pressure for port 1 (Real \geq 0.0; Required if NPORT= 1 or 2).

Descriptor	Meaning
THETA1	Angular position for port 1 ($0.0 \leq \text{Real} < 360.0$; Required if NPORT= 1 or 2). See Remark 2.
PRES2	Boundary pressure for port 2 ($\text{Real} \geq 0.0$; Required if NPORT= 2).
THETA2	Angular position for port 2 ($0.0 \leq \text{Real} < 360.0$; Required if NPORT= 2). See Remark 2.
OFFSET1	Offset in the SFD direction 1, see Remark 3. (Real; Default = 0.0).
OFFSET2	Offset in the SFD direction 2, see Remark 3. (Real; Default = 0.0)
CROSS	Indicates that cross coupling terms will be provided.
K12, K21	Cross coupling stiffness terms.
B12, B21	Cross coupling damping terms.
M12, M21	Cross coupling mass terms.

Remarks:

1. Either SQUEEZE or CROSS option can be used. Both options cannot be specified.
2. The angular measurement is counterclockwise from the displacement x-axis for the XY plane, the y-axis for the YZ plane, and the z-axis for the XZ plane.
3. Offsets are measured from GB relative to GA. For example, if direction 2 is in the vertical direction and a gravity load is placed on GA, OFFSET2 will be a positive value (GB ‘moves’ toward GA in the positive direction 2).
4. PBUSH2D is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBUSH2D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBUSH2D entries.
5. PBUSH2D may be used in conjunction with THPAD obtained from ROMAC for rotordynamic applications. Also see [ELEMUDS](#) and [THPAD](#) bulk data entries.
6. The template used for specifying matrices using cross entries is:

$$[I]_{2X2} = \begin{bmatrix} I_{11} & I_{12} \\ I_{21} & I_{22} \end{bmatrix}$$

where I=K,B or M

PBUSHT

Frequency Dependent or Nonlinear Force Deflection Spring and Damper Property

Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.

Format:

1	2	3	4	5	6	7	8	9	10
PBUSHT	PID	“K”	TKID1	TKID2	TKID3	TKID4	TKID5	TKID6	
		“B”	TBID1	TBID2	TBID3	TBID4	TBID5	TBID6	
		“GE”	TGEID1	TGEID2	TGEID3	TGEID4	TGEID5	TGEID6	
		“KN”	TKNID1	TKIND2	TKNID3	TKIND4	TKIND5	TKIND6	
			FDC	FUSE	DIR	OPTION	LOWER	UPPER	
			FSRS	LRGR					

Example:

PBUSHT	35	K	72						
		B	18						

Descriptor	Meaning
PID	Property identification number that matches the identification number on a PBUSH entry. (Integer > 0)
“K”	Flag indicating that the next 1 to 6 fields are stiffness frequency table identification numbers. (Character)
TKIDI	Identification number of a TABLEDI entry that defines the stiffness vs. frequency relationship in directions 1 through 6. (Integer ≥ 0 ; Default = 0)
“B”	Flag indicating that the next 1 to 6 fields are force per velocity frequency table identification numbers. (Character)
TBIDI	Identification number of a TABLEDI entry that defines the force per unit velocity damping vs. frequency relationship in directions 1 through 6. (Integer ≥ 0 ; Default = 0)
“GE”	Flag indicating that the next field is a structural damping frequency table identification number. (Character)
TGEIDI	Identification number of a TABLEDI entry that defines the non-dimensional structural damping vs. frequency relationship. (Integer ≥ 0 ; TGEID1 default = 0, for TGEID2-6 see defaults described in Remark 6.)
“KN”	Flag indicating that the next 1 to 6 fields are nonlinear force-deflection table identification numbers. (Character)
TKNIDI	Identification number of a TABLEDI entry that defines the force vs. deflection relationship in directions 1 through 6. (Integer ≥ 0 ; Default = 0)

Descriptor	Meaning
FDC	Force deflection curve rule. Specifies a dependence between displacement components. See Remark 16. for use with the FUSE option. (Character or blank default blank)
"NR" or blank	Implies there is no force deflection rule. T1-R6 directions are all independent of each other.
"TRXY"	A radial dependence exists between u_x and u_y element displacements. Either K1 or K2 may be specified on the PBUSH entry. If both are specified then K1 will be used. On the PBUSHT "KN" entry either TKNID1 or TKNID2 may be specified. If both are specified then TKNID1 is used. The expression
	$u_r = \sqrt{(u_x)^2 + (u_y)^2}$
	is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using u_r . Under the OPTION=RULTLD, the true forces f_x and f_y in the springs "x" and "y" are used for the fuse calculation. New K1 is returned from slope of load-deflection curve at slope at current u_r .
"TRXZ"	A radial dependence exists between u_x and u_z element displacements. Either K1 or K3 may be specified on the PBUSH entry. If both are specified then K1 is used. On the PBUSHT "KN" entry either TKNID1 or TKNID3 may be specified. If both are specified then TKNID1 is used. The expression
	$u_r = \sqrt{(u_x)^2 + (u_z)^2}$
	is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using u_r . Under the OPTION=RULTLD, the true forces f_x and f_z in the springs "x" and "z" are used for the fuse calculation. New K1 is returned from slope of load-deflection curve at slope at current u_r .
"TRYZ"	A radial dependence exists between u_y and u_z element displacements. Either K2 or K3 may be specified on the PBUSH entry. If both are specified then K2 is used. On the PBUSHT "KN" entry either TKNID2 or TKNID3 may be specified. If both are specified then TKNID2 is used. The expression
	$u_r = \sqrt{(u_y)^2 + (u_z)^2}$
	is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using u_r . Under the OPTION=RULTLD, the true forces f_y and f_z in the springs "y" and "z" are used for the fuse calculation. New K2 is returned from slope of load-deflection curve at slope at current u_r .

Descriptor	Meaning
	"TS"
	A spherical dependence exists between u_x , u_y , and u_z element displacements. Either K1, K2 or K3 may be specified on the PBUSH entry. If all three are specified K1 is used. IF only K2 and K3 are specified then K2 is used. On the PBUSH "KN" entry TKNID1, or TKNID2 or TKNID3 may be specified. If all three are specified then TKNID1 is used. If only TKNID2 and TKNID3 are specified then TKNID2 is used. The expression $u_s = \sqrt{(u_x)^2 + (u_y)^2 + (u_z)^2}$ is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using u_s . Under the OPTION=RULTLD, the true forces f_x , f_y and f_z in the springs "x", "y", and "z" are used for the fuse calculation. New K1 is returned from slope of load-deflection curve at slope at current u_s
FUSE	0 The element remains active irrespective of failure level. (Integer 0, 1, or 2, Default = 0) 1 The element is deactivated if maximum failure as specified in OPTION is reached. Elements remains for postprocessing. 2 The element is deactivated if maximum failure as specified in OPTION is reached. Element is removed from postprocessing.
DIR	The fuse direction component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks; Default = 0) 0 Any direction is allowed to fuse. Same if user entered 123456. 1 through 6 may be placed in the field with no embedded blanks. See Remark 13. for use with FDC.
OPTION	Specifies a failure mode. (Character - Default="RELDIS") "ULTLD" The specified failure load in compression or tension will be used to define failure using load computed from the appropriate radial dependence of displacement relationship. "RELDIS" The specified max relative \pm displacement will be used to define failure. "RULTLD" The specified failure load in compression or tension will be used to define failure using the actual current physical load in each spring.
LOWER	Lower failure bound. (Real; Default=0.0) If OPTION = "ULTLD" then LOWER specifies a lower failure load If OPTION = "RELDIS" then LOWER specifies a minimum relative displacement before failure.
UPPER	Upper failure bound. (Real; Default=0.0)

Descriptor	Meaning
	If OPTION = "ULTLTD" then UPPER specifies an upper failure load.
	If OPTION = "RELDIS" then UPPER specifies a maximum relative displacement before failure.
FSRS	Fuse Stiffness Retention Factor is a factor which scales the stiffness so that the stiffness does not instantly drop to a zero value. (Real ≥ 0.0 ; Default = 1.-5)
LRGR	Specifies if large rotation is to occur at end A. (Integer ≥ 0 ; Default = 0). See Remark 14.

Remarks:

1. The "K", "B", and "GE" fields are associated with same entries on the PBUSH entry.
2. PBUSH may only be referenced by CBUSH elements in the residual structure which do not attach to any omitted degrees-of-freedom of ASET and/or dependent DOFs of MPC/Rigid elements. Otherwise, the solution is an approximation based on nominal properties.
3. For nonlinear analysis the nominal values are used at the beginning of the analysis to compute initial loading of the CBUSH element and meaningful values should be supplied for stiffness. As the nonlinear analysis proceeds, the user supplied load deflection curves are used to compute new stiffness values. For frequency dependent modal frequency response the system modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency.
4. The "K", "B", "GE" or "KN" fields may be specified in any order.
5. The PBUSH is used in SOL 108 and SOL 111 when any "K", "B", or "GE" is specified. It is used in SOL 106, SOL 129 or SOL 400 when "KN" is specified. It is ignored in all other solution sequences.
6. For upward compatibility, if ONLY TGEID1 is specified on a PBUSH entry and TGEIDI_i, i = 2 – 6 are all blank on the PBUSH entry, then the single structural damping table is applied to all defined GEi for the PBUSH entry. In this case, TGEIDI_i, i = 2 – 6, blank is not equivalent to 0 but a special flag.

A PBUSH entry that has any TGEIDI_i, i = 2 – 6 numerically specified including a 0, all the fields TGEIDI_i are then treated separately and any TGEIDI_i, i=2 - 6 field with no entry specified is defaulted to a 0 value and the results are radically different from the no TGEIDI_i, i=2 – 6 specified as described above.

Thus for the entry:

1	2	3	4	5	6	7	8	9	10
PBUSHT	3303000	K	33030001						
		GE	33030002						

The GE table ID 3303002 will be replicated to fields 5 through 9. While:

1	2	3	4	5	6	7	8	9	10

PBUSHT	3303000	K	33030001					
		GE	33030002	0				

The 0 in FIELD 5 of the above PBUSHT above negates table 3303002 being replicated, hence, fields 5 through 9 will not have any associated table lookup.

7. For nonlinear analysis, only the “KN” line and optionally its two continuation lines are used.
8. For frequency responses, only the “K”, “B” and/or “GE” fields are used.
9. The continuations entries to “KN” are optional.
10. When the FDC field specifies one of the optional radial or spherical rules, stiffness entries not involved in the rule may still have their own independent force-deflection curves.
11. If FUSE=0, the remaining entries are ignored. If FUSE > 0, the “OPTION”, “UPPER”, and “LOWER” fields must be specified.
12. UPPER > LOWER required.
13. The following relationships exists between FDC and DIR

FDC = “TRXY”	1 or 2 or 12 in the DIR field applies to the radial rule	a 3 in the DIR field will be ignored.
FDC = “TRXZ”	1 or 3 or 13 in the DIR field applies to the radial rule	a 2 in the DIR field will be ignored.
FDC = “TRYZ”	2 or 3 or 23 in the DIR field applies to the radial rule	a 1 in the DIR field will be ignored.
FDC = “TS”	Any unique combination of the integers 1 through 3 with no embedded blanks in the DIR field applies to the spherical rule	

14. LRGR = 0 (Default) the element coordinate system is rotated with the rotation of grid A for both the CID and the V vector. LRGR = 1 will suppress large rotation at end A. The initial CID and the V vector will remain unchanged. LRGR = 2 a mid-increment method is used to rotate the element system for the V vector. LRGR = 2 is deactivated if CID is not blank. When CID is blank, LRGR = 2 is recommended for SOL400 when large rotations are present as the mid-increment method “averages” the rotations at both ends of the CBUSH and allows rotation when end A might have a rotational constraint.
15. When the fuse criteria is reached in one of the stiffness components, the entire element is assumed to fail. The component stiffnesses are then computed to be $K = K \cdot FSRS$ at initial failure.
16. The figures below depict, using FDC=TRYZ as a typical example, the OPTION="ULTLD" and OPTION="RULTLD". Under the OPTION="ULTLD" the resultant Fr load when FUSED will be $F_r = \sqrt{F_y^2 + F_z^2} \approx \text{LIMIT}$. Under the OPTION="RULTLD" the resultant Fr load when FUSED will in general be $F_r = \sqrt{F_y^2 + F_z^2} > \text{LIMIT}$ with one of the F_y or $F_z \approx \text{LIMIT}$. For FDC=TRYZ and

DIR=123, the T1 direction is not included in the force direction curve rule, thus possible fusing in the T1 direction is always based on the actual T1 spring displacement and F_x if active will be $F_x \approx \text{LIMIT}$. For FDC=NR fusing is always calculated based on each individual spring displacement and the actual individual spring loading.

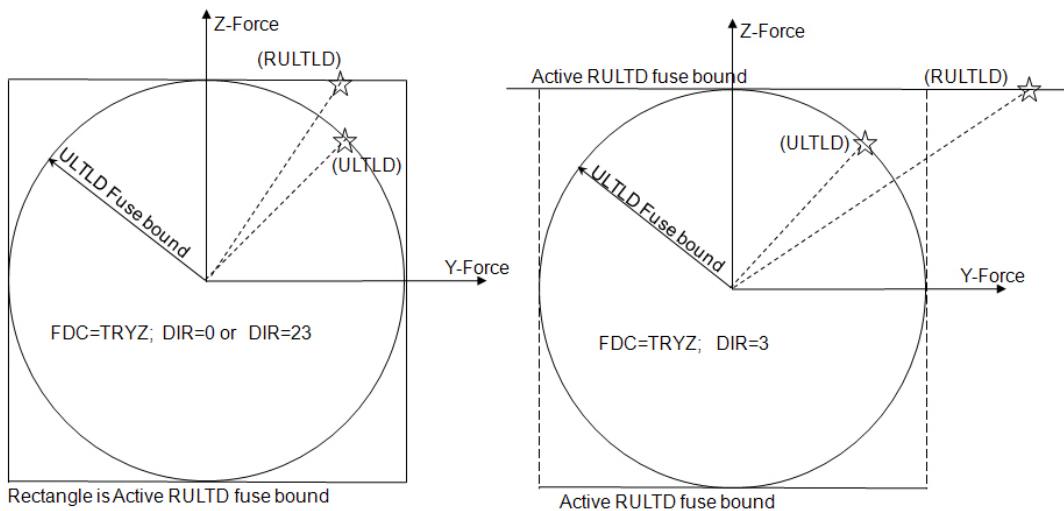


Figure 9-132 FUSE Models

PCOHE**Interface Cohesive Zone Modeling Element Properties**

Defines the properties of a fully nonlinear element used to simulate the onset and progress of delamination in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
PCOHE	PID	MID	INT	T	OUTPUT	SECANT			

Example:

PCOHE	700	701							
-------	-----	-----	--	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification. (Integer > 0)
MID	Identification number of a MCOHE entry. (Integer > 0)
INT	Integration scheme. 0 or blank Gauss integration, 1 for Newton-Coates/Lobatto integration. See Remark 3. (Integer 0 or 1; Default 0)
T	Thickness. See Remark 2. (Real > 0; Default = 1.0)
OUTPUT	Location selection for stress/strain output. If OUTPUT=GRID or blank, output is at the corner grid points. If OUTPUT=GAUSS output at the Gauss points. If INT=1, OUTPUT will be set to GRID. (Character or Blank; Default Grid)
SECANT	Tangent matrix scheme. 0 or blank a secant-type matrix is used to set up the element stiffness matrix, 1 a tangent-type matrix is used to set up the element stiffness matrix. (Integer 0 or 1; Default 0)

Remarks:

1. PCOHE can be referenced by CIFQUAD, CIFHEX, CIFPENT, and CIFQDX entries.
2. The thickness T applies only to CIFQUAD elements.
3. For initially very stiff interface elements, the Newton-Coates/Lobatto integration scheme may be preferred.
4. PCOHE is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOHE property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PCOHE entries

PCOMP

Layered Composite Element Property

Defines the properties of an n-ply composite material laminate.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2	
	MID3	T3	THETA3	SOUT3	-etc.-				

Example of multiple plies per line format:

PCOMP	181	-0.224	7.45	10000.0	HOFF				
	171	.056	0.	YES			45.		
			-45.				90.		

Example of single ply per line format:

PCOMP	181	-0.224	7.45	10000.	HOFF				
	171	.056	0.	YES					
	171	.056	45.	YES					
	171	.056	-45.	YES					
	171	.056	90.	YES					

Descriptor	Meaning
PID	Property identification number. (0 < Integer < 10000000)
Z0	Distance from the reference plane to the bottom surface. See Remarks 10. and 11. (Real; Default = -0.5 times the element thickness.)
NSM	Nonstructural mass per unit area. (Real)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0)
FT	Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed) See Remark 7. “HILL” for the Hill theory. “HOFF” for the Hoffman theory. “TSAI” for the Tsai-Wu theory. “STRN” for the Maximum Strain theory. “HFFAIL” for the Hashin failure criterion “HTAPE” for the Hashin tape criterion “HFABR” for the Hashin fabric criterion
TREF	Reference temperature. See Remark 3. (Real; Default = 0.0)

Descriptor	Meaning
GE	Damping coefficient. See Remarks 4. and 12. (Real; Default = 0.0)
LAM	Laminate Options. (Character or blank, Default = blank). See Remarks 13. and 14. <ul style="list-style-type: none"> “Blank” All plies must be specified and all stiffness terms are developed. “SYM” Only plies on one side of the element centerline are specified. The plies are numbered starting with 1 for the bottom layer. If an odd number of plies are desired, the center ply thickness (T1) should be half the actual thickness. “MEM” All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed. “BEND” All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed. “SMEAR” All plies must be specified, stacking sequence is ignored MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set as blanks). “SMCORE” All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties. The stiffness matrix is computed by placing half the face sheet thicknesses above the core and the other half below with the result that the laminate is symmetric about the mid-plane of the core. Stacking sequence is ignored in calculating the face sheet stiffness.
MID _i	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to MAT1, MAT2, MAT8, or MATDIGI Bulk Data entries. See Remarks 1. 15. and 18. (Integer > 0 or blank, except MID1 must be specified.)
T _i	Thicknesses of the various plies. See Remarks 1. (Real or blank, except T1 must be specified.)
THETAI	Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0)
SOUT _i	Stress or strain output request. See Remarks 5. and 6. (Character: “YES” or “NO”; Default = “NO”)

Remarks:

1. The default for MID₂, ..., MID_n is the last defined MID_i. In the example above, MID1 is the default for MID2, MID3, and MID4. The same logic applies to Ti.
2. At least one of the four values (MID_i, Ti, THETAI, SOUT_i) must be present for a ply to exist. The minimum number of plies is one.
3. The TREF specified on the material entries referenced by plies are not used. Instead TREF on the PCOMP entry is used for all plies of the element. If not specified, it defaults to “0.0.”

If the PCOMP references temperature dependent material properties, then the TREF given on the PCOMP will be used as the temperature to determine material properties.

TEMPERATURE Case Control commands are ignored for deriving the equivalent PSHELL and MAT2 entries used to describe the composite element.

If for a nonlinear static analysis the parameter COMPMATT is set to YES, the temperature at the current load step will be used to determine temperature-dependent material properties for the plies and the equivalent PSHELL and MAT2 entries for the composite element. The TREF on the PCOMP entry will be used for the initial thermal strain on the composite element and the stresses on the individual plies. If the parameter EPSILONT is also set to INTEGRAL, TREF is not applicable.

4. GE given on the PCOMP entry will be used for the element and the values supplied on material entries for individual plies are ignored. The user is responsible for supplying the equivalent damping value on the PCOMP entry. If PARAM,W4 is not specified GE is ignored in transient analysis. When GEij values are present on the MAT2 entry, Nastran will ignore the GE value given on the first continuation entry field (6) of the MAT2 and the GE entry given in field (8) of the PCOMP entry and use the given GEij values.
5. Stress and strain output for individual plies are available in all superelement static, normal modes, buckling, frequency response (direct and modal), transient response (direct and modal) and nonlinear static analysis and requested by the STRESS and STRAIN Case Control commands. Composite lamina stress/strain are available in random analysis by XYPILOT command.
6. For Nastran conventional elements, if PARAM,NOCOMPS is set to -1, stress and strain output for individual plies will be suppressed and the homogeneous stress and strain output will be printed. See also Remark [10](#).
7. In order to get failure index output the following must be present:
 - a. STRESS or STRAIN Case Control commands,
 - b. SB, FT, and SOUTi on the PCOMP Bulk Data entry,
 - c. Xt, Xc, Yt, Yc, and S on all referenced MAT8 Bulk Data entries if stress allowables are used, or Xt, Xc, Yt, S, and STRN=1.0 if strain allowables are used.
 - d. -1 - failure in the fiber direction.
-2 - failure in the matrix direction
-12 - failure in the inplane shear
 - e. For Hashin Failure criterion, HF_i (*i*=1,2,3,4,10,11) must be present in all referenced MAT8 bulk entry.
 - f. For Hashin Tape criterion, HT_i (*i*=1,2,3,4,5,6,10,11,12) must be present in all referenced MAT8 bulk entry.
 - g. For Hashin Fabric criterion, HFB_i (*i*=1,2,3,4,5,6,10,11,12) must be present in all referenced MAT8 bulk entry.

- h. Hashin failure theory output is captured with following code.
 - 1 - failure in the tensile first fiber mode.
 - 1 - failure in the compressive first fiber mode.
 - 2 - failure in the tensile second fiber mode.
 - 2 - failure in the compressive second fiber mode.
 - 3 - failure in the tensile matrix mode.
 - 3 - failure in the compressive matrix mode.
8. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. Any sorted echo request will also cause printout and/or punch of the derived PSHELL and MATi entries in User Information Message 4379 and/or the punch file. Use the NASTRAN system cell (361) PRTPCOMP=1 to print equivalent PSHELL/MAT2 Bulk Data entries to the .f06 file. Use the ECHO=PUNCH Case Control command to write them to the .pch file.
9. The failure index for the bonding material is calculated as Failure Index = $(\tau_{1z}, \tau_{2z})/SB$.
10. If the value specified for Z0 is not equal to -0.5 times the thickness of the element and PARAM,NOCOMPS,-1 is specified, then the homogeneous element stresses are incorrect, while element forces and strains are correct. For correct homogeneous stresses, use ZOFFS on the corresponding connection entry.
11. Use of Z0 to offset a laminate does not change the reference plane. Z0 offsets the bottom of the plies from the reference plane. An unsymmetric layup or the use of Z0 to specify an unsymmetric layup, is not recommended in buckling analysis or the calculation of differential stiffness. Also, Z0 should not be used to specify an unsymmetric layup. The presence of coupling between bending and extension generally increases deflections. Hence, coupling decreases the effective stiffness of a laminate, reduces buckling loads and vibration frequencies significantly.
12. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.
13. The SYM option for the LAM option computes the complete stiffness properties while specifying half the plies. The MEM, BEND, SMEAR and SMCORE options provide special purpose stiffness calculations. SMEAR ignores stacking sequence and is intended for cases where this sequence is not yet known, stiffness properties are smeared. SMCORE allows simplified modeling of a sandwich panel with equal face sheets and a central core.
14. Element output for the SMEAR and SMCORE options is produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress/strain results and prints results for the equivalent homogeneous element.
15. Temperature-dependent ply properties only available in SOL 106 and SOL 400. See PARAM,COMPMATT for details.
16. For SOL 600, the default for SOUT is YES for the top and bottom layer and NO for all layers.
17. For SOL 600, LAM=BLANK if SMEAR is specified on the SOL 600 Executive Control statement. Other LAM options are not available using SOL 600. The default option for SOL 600 is to use complete through the thickness integration for all layers. This is achieved by not entering SMEAR on the SOL 600 entry. Options to speed up complete through the thickness integration by making certain assumptions such as no plasticity in the layers are available using the PCOMP entry. Please note, the meaning of SMEAR for SOL 600 and for other MSC Nastran solution sequences is not the same.

18. PCOMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSHELL, PCOMP, PCOMPG entries.
19. For PCOMP, the non-default Zo option should not be used in conjunction with MEM, BEND, SMEAR or SMCORE as wrong results may occur. These four options provide special purpose stiffness calculations wherein stacking sequence effects are ignored (membrane-bending coupling terms are set to zero) at the preliminary design stage level. A better choice for offsets is to use the ZOFF option on the connectivity entries (CQUAD4, CTRIA3 etc).
A User Fatal Message is issued if the user specifies a non-default value of Zo for any of the above smearing options. The fatal message may be changed to a User Warning Message by specifying a positive value for system cell 668 (e.g. nastran system(668)=1) to allow the program to continue.
20. In layer composite output, the traditional element model outputs stress or strain separately by using STRESS or STRAN Case Control Command, respectively.
The advanced element model does not recognize STRAIN Case Control Command. As long as user gives STRESS Case Control Command, both stress and strain are outputted together (similar to NLSTRESS).

PCOMPA

Additional Data for Layered Composite Element Property - SOL 700

Defines additional properties of a multi-ply laminate composite material. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPA	PID	FORM	SHFACT	REF	STRDEF	DT1D	STRNOUT	CLT	
+	SPINCOR								

Example:

PCOMPA	1	BLT						1	
+	YES								

Descriptor	Meaning
PID	Unique property number referring to a PCOMP property number. (Integer > 0; Required)
FORM	Element formulation. (Character; See Remark 1.)
SHFACT	Shear correction factor, see Remark 4. (Real ≥ 0.0 ; default=0.83333)
REF	Reference surface. (Character; default=MID) <ul style="list-style-type: none"> TOP Reference surface is the top of the surface. MID Reference surface is the central surface. BOT Reference surface is the bottom surface.
STRDEF	Definition in stress-strain output. (Character; default=FIBER) <ul style="list-style-type: none"> FIBER Stresses defined in the fiber and matrix directions. ELEM Stresses defined in the element coordinate system
DT1D	Time step skip for one-dimensional failure modes. See Remark 2. (Character; default=NO) <ul style="list-style-type: none"> YES Skip one-dimensional failure modes. NO Normal time-step calculation.
STRNOUT	Strain output option. See Remarks 3. and 4. (Character; default=YES) <ul style="list-style-type: none"> YES Total strain is calculated. NO No strain is stored in memory.
ICLT	Option to use Classical Lamination Theory. See Remark 4. (Integer > 0; default=0) <ul style="list-style-type: none"> 1 Use the Classical Lamination Theory. 0 Use the integration technique.
SPINCOR	Spin correction. See Remark 4. (Character; default=NO)

Descriptor	Meaning
NO	No SPINCOR applied
YES	SPINCOR applied

Remarks:

1. For CQUAD4 elements, the default formulation is Key-Hoff. For CTRIA3 elements, the default formulation is C0-TRIA.
2. If the failure mode is such that fiber and shear strength or matrix and shear strength are lost in all layers, the element is not included in the time-step calculation. If the element fails completely, the element is omitted from the time-step calculations, irrespective of the value entered in this field.
3. If the STRNOUT field is NO, the strain cannot be output.
4. If ICLT is set to 1, the analysis is performed with classical lamination theory. In this case, it is not possible to request the total strain output. The (transverse) shear correction factor input is ignored since it is calculated inside Nastran. There is no update of the cross-sectional properties due to failure. The failure flag only indicates that the failure condition is satisfied. Additional output for element variables is available, namely the stress resultants (NXN, NYN, NXZ, MXN, MYN, MZN, QXZ, and QZZ). Also the ABD-Q matrices of each element can be requested for output. These data are only stored in the first layer. The variable names are AijM, BijM, DijM, and QsijM for the components of the A-, B-, D- and Q-matrices, respectively. For example, to request the A11 of the A-matrix, the variable name is A11M01.
5. The options for SPINCOR are:

- | | |
|-----|-----------------------------------|
| NO | No SPINCOR correction is applied. |
| YES | A SPINCOR correction is applied. |

When SPINCOR = NO, slight asymmetric forces are applied to the shell element's grid points. This approach is, in general, acceptable up to about 10° in plane shear angle.

The SPINCOR option is required for fabric models and is turned on by default to accurately keep track of the fiber directions.

PCOMPF**Integration Procedure Used in Conjunction with PCOMP or PCOMPG**

Defines the integration procedure for through the thickness integration of composite shells in SOL 600 and SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPF	INT	PID1	THRU	PID2	BY	N			

Alternate Formats (SOL 400):

1	2	3	4	5	6	7	8	9	10
PCOMPF	INT	PID1	PID2	PID3	THRU	PID4	PID5	THRU	
	PID6	PID7	TO	PID8	PID9	PID10	PID11	PID12	
	THRU	PID13	BY	N					

1	2	3	4	5	6	7	8	9	10
PCOMPF	INT	ALL							

Examples:

PCOMPF	2	100	THRU	200	BY	10			
--------	---	-----	------	-----	----	----	--	--	--

Example of Application to Single PID (SOL 400):

PCOMPF	1		23	TO	25		33		
	44	THRU	54	BY	2				
	100								

Descriptor	Meaning
INT	1 Conventional through the thickness integration of each layer, allows all available material behavior through the thickness. (Default) 2 Linear elastic material, fast-integrated through the thickness - thermal strains and temperature dependent material properties are not allowed. 3 Linear elastic material, fast integrated through the thickness.
PIDI	Property identification number. (0 < Integer < 10000000) corresponds to a matching PCOMP or PCOMPG entry.
N	Property identification number increment. See Remark 7.. (Integer or blank)

Remarks:

1. If nonlinear behavior is set on a MATS1 (or other option), but INT is > 1, then the nonlinear material behavior is ignored.
2. In SOL 600 only, if temperature dependent behavior is specified on a MATT1 or similar option and INT=2, the material values specified on the MATT1, MATT2, MATT8 option are ignored (the values on MAT1, MAT2 and MAT8 are used). In SOL 400, temperature loading or temperature dependent behavoir should not be used with INT=2, a user fatal will be issued in this case. INT=3 can be used instead.
3. If more than one PCOMP exists with different INT values, and there is an overlap in PID's, that is to say a given PID could have been assigned different values of INT, then a user fatal will be issued.
4. With the 'THRU" and "THRU", "BY" forms, blank fields are allowed for readability. Any combination of a list of ID's and "THRU" and "THRU", "BY" is allowed. The "THRU" and "BY" lists may have missing ID's. That is the list of ID's in a THRU range need not be continuous.
5. In SOL 600 only, if all composite shells are to use the same INT value, it may be entered with PARAM,MFASTCMP instead of PCOMP.
6. In SOL 600 only, if a "THRU" or "THRU", "BY" range is entered, all items associated with the range must be on the same line (or for large field a line and the continuation entry of that line).
7. For automatic generation of property identification numbers, the default increment value is 1 if property identification numbers are increasing or -1 if property identification numbers are decreasing (i.e., the user need not specify BY and the increment value).

PCOMPQF**Frequency Dependent Composite structural damping Property**

Defines the frequency dependent properties for a PCOMP/PCOMPG Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPQF	PID						GE		

Examples:

PCOMPQF	33						15		
---------	----	--	--	--	--	--	----	--	--

Descriptor	Meaning
PID	Property identification number that matches the identification number on PCOMP/PCOMPG entry. (Integer > 0)
GE	Identification number of a TABLEDi entry that defines the non-dimensional structural damping coefficient vs. frequency relationship. (Integer > 0; Default = 0)

Remarks:

1. The Fields 8 of this entry corresponds to Field 8 of a PCOMP or PCOMG entry. The value in Field 8 of the PCOMP or PCOMPG entry is replaced by the table referenced in the corresponding field of this entry.
2. If the PCOMP/PCOMG MIDi fields point to MAT2 entries with the 2nd continuation fields containing GEij entries, then this entry is ignored for that MIDi entry and GE frequency dependency must be indicated by use of the MAT2F entry with same ID as the MIDi entry.
3. IF GE=0.0 on corresponding PCOMP or PCOMPG then GE table must be blank or 0.

PCOMPG

Layered Composite Element Property (Alternate to PCOMP Entry)

Defines global (external) ply IDs and properties for a composite material laminate.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPG	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	GPLYID1	MID1	T1	THETA1	SOUT1	GEFLG			
	GPLYID2	MID2	T2	THETA2	SOUT2				

Example of single ply per line format:

PCOMPG	181	-0.224	7.45	10000.	HOFF				
	1001	171	.056	0.	YES				
	101	171	.07	45.	YES				
	2002	171	.056	-45.	YES				
	102	171	0.55	90.	YES				

Descriptor	Meaning
PID	Property identification number. (0 < Integer < 10000000)
Z0	Distance from the reference plane to the bottom surface. See Remark 11. (Real; Default = -0.5 times the element thickness.)
NSM	Nonstructural mass per unit area. (Real)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0)
FT	Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed) See Remark 8. “HILL” for the Hill theory. “HOFF” for the Hoffman theory. “TSAI” for the Tsai-Wu theory. “STRN” for the Maximum Strain theory. “HFAIL” for the Hashin failure criterion “HTAPE” for the Hashin tape criterion “HFABR” for the Hashin fabric criterion
TREF	Reference temperature. See Remark 4. (Real; Default = 0.0)
GE	Damping coefficient. See Remark 5. (Real; Default = 0.0)
LAM	Laminate Options. (Character or blank, Default = blank). See Remarks 14. “Blank” All plies must be specified and all stiffness terms are developed.

Descriptor	Meaning
"MEM"	All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed.
"BEND"	All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed.
"SMEAR"	All plies must be specified, stacking sequence is ignored MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set to zero.
"SMCORE"	All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties. The stiffness matrix is computed by placing half the face sheet thicknesses above the core and the other half below with the result that the laminate is symmetric about the mid-plane of the core. Stacking sequence is ignored in calculating the face sheet stiffness.
GPLYID _i	User-defined Global (External) Ply ID. See Remark 1. (Integer > 0)
MID _i	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to MAT1, MAT2, or MAT8 Bulk Data entries. See Remarks 2. and 15. (Integer > 0 or blank, except MID1 must be specified.)
T _i	Thicknesses of the various plies. See Remark 2. (Real or blank, except T1 must be specified.)
THETAI _i	Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0)
SOUT _i	Stress or strain output request. See Remarks 6. and 7. (Character: "YES" or "NO"; Default = "NO")
GEFLG	Ply structural damping flag may be only entered only once and if entered must be entered on the first ply. (Integer -2,or -1; Default=0) See Remark 19.

Remarks:

1. The global ply identification number must be unique with respect to other plies in the entry. The plies are defined in stacking sequence starting with the bottom layer.
2. The default for MID2, ..., MIDn is the last defined MIDi. In the example above, MID1 is the default for MID2, MID3, and MID4. The same logic applies to Ti.
3. The global ply ID (GPLYID_i) and at least one of the four values (MID_i, Ti, THETAI_i, SOUT_i) must be present for a ply to exist. The minimum number of plies is one.
4. The TREF specified on the material entries referenced by plies are not used. Instead TREF on the PCOMPG entry is used for all plies of the element. If not specified, it defaults to "0.0."

If the PCOMPG references temperature dependent material properties, then the TREF given on the PCOMPG will be used as the temperature to determine material properties.

TEMPERATURE Case Control commands are ignored for deriving the equivalent PSHELL and MAT2 entries used to describe the composite element.

If for a nonlinear static analysis the parameter COMPMATT is set to YES, the temperature at the current load step will be used to determine temperature-dependent material properties for the plies and the equivalent PSHELL and MAT2 entries for the composite element. The TREF on the PCOMPG entry will be used for the initial thermal strain on the composite element and the stresses on the individual plies. If the parameter EPSILONT is also set to INTEGRAL, TREF is not applicable.

5. GE given on the PCOMPG entry will be used for the element and the values supplied on material entries for individual plies are ignored. The user is responsible for supplying the equivalent damping value on the PCOMPG entry. If PARAM,W4 is not specified GE is ignored in transient analysis. When GEij values are present on the MAT2 entry, Nastran will ignore the GE value given on the first continuation entry field (6) of the MAT2 and the GE entry given in field (8) of the PCOMPG entry and use the given GEij values.
6. Stress and strain output for individual plies are available in all superelement static, frequency response (direct and modal), transient response (direct and modal) and normal modes analysis and requested by the STRESS and STRAIN Case Control commands. Composite lamina stress/strain are available in random analysis by XYPILOT command.
7. For Nastran conventional elements, if PARAM,NOCOMPS is set to -1, stress and strain output for individual plies will be suppressed and the homogeneous stress and strain output will be printed. See also Remark 11.
8. In order to get failure index output the following must be present:
 - a. STRESS or STRAIN Case Control commands,
 - b. SB, FT, and SOUTi on the PCOMPG Bulk Data entry,
 - c. Xt, Xc, Yt, Yc, and S on all referenced MAT8 Bulk Data entries if stress allowables are used, or Xt, Xc, Yt, S, and STRN=1.0 if strain allowables are used.
 - d. -1 - failure in the fiber direction
-2 - failure in the matrix direction
-12 - failure in the inplane shear.
 - e. For Hashin Failure criterion, HFi ($i=1,2,3,4,10,11$) must be present in all referenced MAT8 bulk entry.
 - f. For Hashin Tape criterion, HTi ($i=1,2,3,4,5,6,10,11,12$) must be present in all referenced MAT8 bulk entry.
 - g. For Hashin Fabric criterion, HFBi ($i=1,2,3,4,5,6,10,11,12$) must be present in all referenced MAT8 bulk entry.

- h. Hashin failure theory output is captured with following code.
 - 1 - failure in the tensile first fiber mode.
 - 1 - failure in the compressive first fiber mode.
 - 2 - failure in the tensile second fiber mode.
 - 2 - failure in the compressive second fiber mode.
 - 3 - failure in the tensile matrix mode.
 - 3 - failure in the compressive matrix mode.
- 9. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. Any sorted echo request will also cause printout and/or punch of the derived PSHELL and MATi entries in User Information Message 4379 and/or the punch file. Use the NASTRAN system cell (361) PRTPCOMP=1 to print equivalent PSHELL/MAT2 Bulk Data entries to the .f06 file. Use the ECHO=PUNCH Case Control command to write them to the .pch file.
- 10. The failure index for the bonding material is calculated as Failure Index = $(\tau_{1z}, \tau_{2z})/SB$.
- 11. If the value specified for Z0 is not equal to -0.5 times the thickness of the element and PARAM,NOCOMPS,-1 is specified, then the homogeneous element stresses are incorrect, while lamina stresses and element forces and strains are correct. For correct homogeneous stresses, use ZOFFS on the corresponding connection entry. Use of Z0 to offset a laminate does not change the reference plane. Z0 offsets the bottom of the plies from the reference plane.
- 12. An unsymmetric layup or the use of Z0 to specify an unsymmetric layup, is not recommended in buckling analysis or the calculation of differential stiffness. Also, Z0 should not be used to specify an unsymmetric layup. The presence of coupling between bending and extension generally increases deflections. Hence, coupling decreases the effective stiffness of a laminate, reduces buckling loads and vibration frequencies significantly.
- 13. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.
- 14. Element output for the SMEAR and SMCORE options is produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress/strain results and prints results for the equivalent homogeneous element.
- 15. Temperature-dependent ply properties only available in SOL 106 and SOL400. See PARAM,COMPMATT for details.
- 16. For SOL 600, the default for SOUT is YES for the top and bottom layer and NO for all layers.
- 17. For SOL 600, LAM=BLANK if SMEAR is specified on the SOL 600 Executive Control statement. Other LAM options are not available using SOL 600. The default option for SOL 600 is to use complete through the thickness integration for all layers. This is achieved by not entering SMEAR on the SOL 600 entry. Options to speed up complete through the thickness integration by making certain assumptions such as no plasticity in the layers are available using the PCOMPF entry. Please note, the meaning of SMEAR for SOL 600 and for other MSC Nastran solution sequences is not the same.
- 18. PCOMPG is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOMPG property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSHELL, PCOMP, PCOMPG entries.

19. Defines structural damping matrix data for composites. This entry if used MUST appear on the first ply entry ONLY and all subsequent plies follows depending if -1 or -2 described below

GEFLG Value	Meaning
0	The GE value specified in Field 8 of the PCOMPG parent entry is used for computation of all structural damping.
-1	The GEi value associated with the MID1 of this ply and the MIDI with MID(i > 1) for all subsequent plies will be used in computing the K4 structural damping properties for the composite. For any MIDI for which there is no associated GE value on the MAT1, MAT2, or MAT8 entry, the GE value specified in Field 8 of the PCOMPG parent entry will be used. The resulting smeared value GE will be placed in field 8 of the PCOMPG and used for computing structural damping.
-2	The GEi value associated with the MID1 of this ply and the MIDI with MID(i > 1) for all subsequent plies will be used in computing the K4 structural damping properties for the composite. For any MIDI for which there is no associated GE value on the MAT1, MAT2, or MAT8 entry then that ply will not contribute to the computation of the K4 structural damping of the composite. The resulting smeared value GE will be placed in field 8 of the PCOMPG and used for computing structural damping.

When GEFLG is used, the smeared GE value will appear in the second continuation entry field (6) of the four internally generated equivalent MAT2 entries

If GEFLG appears with extended MAT2 entries the GEFLG is ignored for that PCOMPG: See Remark of MAT2 entry.

PCOMPLS**Layered Solid Composite Element Property**

Defines global (external) ply IDs and properties for a composite material laminate in SOL 600, SOL 400, all linear solution sequences between SOL 101 and SOL 112, and analysis only for SOL 200.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPLS	PID	DIRECT	CORDM	SB	ANAL				
	“C8”	BEH8	INT8	BEH8H	INT8H				
	“C20”	BEH20	INT20	BEH20H	INT20H				
	ID1	MID1	T1	THETA1					
	ID2	MID2	T2	THETA2					

Example:

PCOMPLS	782	1							
	1001	171	.3	12.3					
	100	175	.7	77.7					

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
DIRECT	The layer direction for BEHi = SLCOMP. See Remark 5. for direction definition. A positive value implies that the composite layer inputs is a fractional percent of the total element thickness in the ply direction and is recommended. A negative value implies that the composite layer input is the actual thickness of that ply. (Integer ± 1 , ± 2 , or ± 3 ; Default +1)
CORDM	Identification number of the material coordinate system. See Remark 10. (Integer; Default = 0, which is basic)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). (Real ≥ 0.0)
ANAL	Analysis type. ANAL=‘IS’ - Implicit structural elements are being referred to. ANAL=‘IH’ - Implicit heat analysis elements are being referred to. ANAL=‘ISH’ - Implicit structural and heat elements are being referred to. (Character Default ISH)
C8	Keyword indicating that two items following apply to elements with eight corner grids. (Character)
C20	Keyword indicating that two items following apply to elements with eight corner grids and twelve mid-side grids. (Character)
BEHi	Element structural behavior. See Remarks 4. and 7. (Character default: SLCOMP for BEH8 and BEH20)
INTi	Integration scheme. See Remark 9. (Character default: L for INT8, Q for INT20)

BEHiH	Element heat behavior. See Remarks 4. and 8. (Character Default: SLCOMP for BEH8H and BEH20H)
INTiH	Integration scheme. See Remark 9. (Character Default: L for INT8H, Q for INT20H)
IDi	Global Ply ID. Must be unique with respect to other plies in this entry. See Remark 2. (Integer > 0)
MIDi	Material ID for the ply. See Remark 3. (Integer > 0)
Ti	Either fractional percent of the total element thickness or actual thickness of that ply depending on \pm value of DIRECT. See Remarks 5. and 6.. (Real > 0.0)
THETAi	Orientation angle of the ply in the plane of the plies. Measured relative to the projection z-axis defined by CORDM on the plane defined by DIRECT. See Remark 1. (Real; Default = 0.0)

Remarks:

1. The PCOMPLS can only be referenced by a CHEXA entry. To view the composite results of large models using solid composite elements in PATRAN or SimXpert, the following environment variable setting is required when starting the session: DRANAS_NAST_MEM=2048MB.
2. Global Ply ID is intended as a unique ply identifier for ply alignment across all PCOMPG, PLCOMP, and PCOMPLS entries.
3. The MIDi entry may point to MAT1, MAT9, MATORT, MATHE, MATUSR or MATDIGI entries. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Implicit Structural Materials					
MAT1	MAT9	MATORT	MATHE	MATUSR	MATDIGI
MATVE	<MATVE>	<MATVE>	MATVE		
MATVP	MATVP	MATVP			
MATEP	MATEP	MATEP			
MATF	MATF	MATF			
MATS1		MATSORT			
<MATVE> refers to the ALTERNATE format for type ORTHO.					

Heat Materials	
MAT4	MAT5

If heat analysis is being performed and the user wishes to override standard MSC Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, MATG and MATDIGI entries.

If MATDIGI is being used, the ANAL field must be set to IS.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

4. The keyword entries, between themselves, may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
5. The following table describes layer orientation for BEHi=SLCOMP.

For INT8=L and INT20=Q, a total of 510 layers are allowed for any one element.

For INT8=ASTN, a total of 2040 layers are allowed for any one element.

Note the ply numbering starts from the bottom to the top parallel to the positive thickness direction.

Layer orientation			
DIRECT	Normal to Layer Plane	Layers run parallel from face (ply numbering starts here)	to face (ends here)
1	Element T direction	G1-G2-G3-G4	G5-G6-G7-G8
2	Element R direction	G1-G4-G8-G5	G2-G3-G7-G6
3	Element S direction	G1-G2-G6-G5	G4-G3-G7-G8

6. The ply thickness of the element is computed using isoparametric coordinates of the element in the DIRECT direction and the element nodes are mapped between -1 and +1. The ply thickness is entered in one of two ways:
 - a. Relative thickness where the numbers entered, are a fractional percent of the total thickness. This is the preferred method. For this method, the sum of all the fractional percents of thickness must sum to 1.0.
 - b. Absolute thickness where the layer thickness is entered directly. Using this option, the code sums the total user input thickness across all plies and then figures the fractional percent of each individual ply as in method 6a.

7. In the following table, BEHi refers to the structural behavior of 3D-solid elements. An underlined item delineates a default.

Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Solid continuum composite	SLCOMP	<u>L</u>	HEX	8
		ASTN*	HEX	8
		<u>Q</u>	HEX	20

*Only DIRECT=1 or -1 is allowed

8. In the following table, BEHiH refers to the heat behavior of 3D-solid elements. An underlined item delineates a default.

Heat Classification of Elements				
Element Heat Type	BEHiH CODE	Integration Code	Element Type	# Nodes
Solid continuum composite	SLCOMP	<u>L</u> <u>Q</u>	HEX HEX	8 20

9. Integration codes in Remark 7. and 8. are:

INT CODE	Integration Type
L	Linear
ASTN	Assumed STrain enhanced formulation solid shell
Q	Quadratic

10. The material coordinate system CORDM may be the basic system (0 or blank) or the element coordinate system (-1) or any defined coordinate system (> 0) (Default = 0).

THETAi in conjunction with CORDM is used to define the ply orientation.

- a. For BEH = SLCOMP and INT = ASTN, the X axis of the CORDM system is projected onto the layer plane to form the local X axis. The normal direction of the element (thickness direction = local Z axis) is crossed with the local X axis to yield the local Y axis. THETAi is then measured positive counter-clockwise about the local Z axis. If the X axis of the CORDM is in the same direction as the thickness direction, then the analysis will stop with an error.
- b. For BEH = SLCOMP and INT = ASTN, the Material Coordinate System is a function of how NLMOPTS, INLAM, CPROJ is set.

If CPROJ is ON then the projection scheme (see Figure #) is activated. With the activation of projection scheme the X axis of the CORDM system is projected onto the layer plane to form the local X axis. The normal direction of the element (thickness direction = local Z axis) is crossed with the local X axis to yield the local Y axis. THETAI is then measured positive counter-clockwise about the local Z axis. If the X axis of the CORDM is in the same direction as the thickness direction, then the analysis will stop with an error.

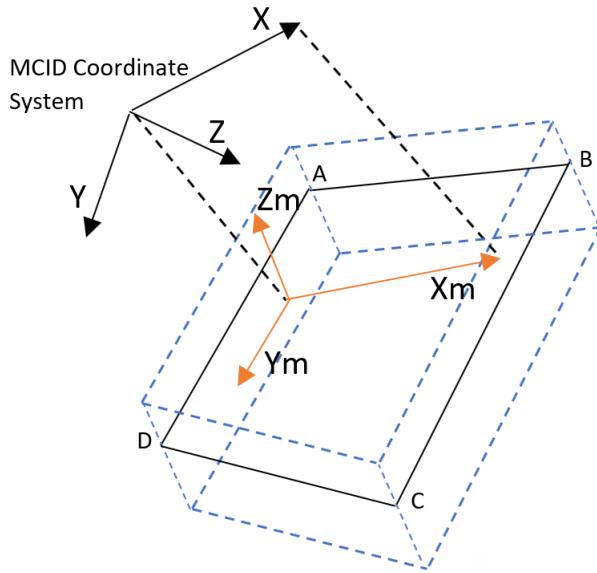


Figure 9-133 Projection Scheme

In Figure 9-133, plane ABCD is the layer plane of interest. Z_m is normal to the layer plane (or is the thickness direction). X_m is the projection of the X axis of the MCID coordinate system on the layer plane. $Z_m \times X_m$ gives the Y_m . Angle theta is measured counter-clockwise from X_m axis about the Z_m axis.

If CPROJ is OFF (default), then the CORDM axes directly form the local X, Y and Z axes of the layer plane. THETAI is then measured positive counter-clockwise about the local Z axis.

11. DIRECT, SB, ANAL, "C8" and "C20" are not supported in SOL 600 will be ignored if entered.
12. For SOL 600, PCOMPLS is made into PSOLID and PCOMP internally. Existing PSOLID and PCOMP entries with the same PID may be overwritten. The combination PSOLID and PCOMP is preferred over PCOMPLS for SOL 600, either option (but not both) may be used.
13. PCOMPLS is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOMPLS property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PCOMPLS, PLCOMP, PCOMPG entries.

14. It is a general limitation in SOL 400 analysis = HSTAT and SOL 101 APP HEAT won't support Gradient and Flux calculation in z direction for layered solid composite (material laminate) element.

PCONEAX

Conical Shell Element Property

Defines the properties of a conical shell element described on a CCONEAX entry.

Format:

1	2	3	4	5	6	7	8	9	10
PCONEAX	ID	MID1	T1	MID2	I	MID3	T2	NSM	
	Z1	Z2	PHI1	PHI2	PHI3	PHI4	PHI5	PHI6	
	PHI7	PHI8	PHI9	PHI10	PHI11	PHI12	PHI13	PHI14	

Example:

PCONEAX	2	4	1.0	6	16.3	8	2.1	0.5	
	0.001	-0.002	23.6	42.9					

Descriptor Meaning

ID	Property identification number. (Unique Integer > 0)
MIDi	Material identification number for membrane, bending, and transverse shear. (Integer ≥ 0)
T1	Membrane thickness. (Real > 0.0 if MID1 = 0)
T2	Transverse shear thickness. (Real > 0.0 if MID3 = 0)
I	Moment of inertia per unit width. (Real)
NSM	Nonstructural mass per unit area. (Real)
Z1, Z2	Fiber distances from the middle surface for stress recovery. (Real)
PHIi	Azimuthal coordinates (in degrees) for stress recovery. (Real)

Remarks:

1. PCONEAX is allowed only if an AXIC entry is also present.
2. PCONEAX entries may reference MAT1 or MAT2 material entries. However, only orthotropic material properties are consistent with axisymmetry. Therefore, G13 and G23 values on the MAT2 entry referenced by MID1 or MID2 and the G12 value on the MAT2 entry referenced by MID3 should be set to 0.0. In addition, the MID3 entry, if it references a MAT2 material matrix, should be of size 2 x 2.
3. If either MID1 = 0 or blank or T1 = 0.0 or blank, then both must be zero or blank.
4. If either MID2 = 0 or blank or I = 0.0 or blank, then both must be zero or blank.
5. If either MID3 = 0 or blank or T2 = 0.0 or blank, then both must be zero or blank.
6. A maximum of 14 azimuthal coordinates (PHIi) for stress recovery may be specified.
7. For a discussion of the conical shell problem, see Section 5.3.3 of the *MSC Nastran Reference Guide*.

8. The following elastic relationships are assumed:

- In-plane forces per unit width

$$\{F\} = T1[G_1]\{\varepsilon\}$$

where $\{\varepsilon\}$ is the vector of strains in the middle surface.

- Bending moments per unit width

$$\{M\} = I[G_2]\{\chi\}$$

where $\{\chi\}$ is the vector of curvatures.

- Transverse shear forces per unit width

$$\{V\} = T2[G_3]\{\gamma\}$$

where $\{\gamma\}$ is the vector of transverse shear strains.

$[G_1]$, $[G_2]$ and $[G_3]$ are the stress-strain matrices defined by MID1, MID2, and MID3, respectively.

Pconv**Convection Property Definition**

Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
Pconv	Pconid	MID	Form	Expf	Ftype	Tid			
	Chlen	Gidin	Ce	E1	E2	E3			

Examples:

Pconv	53	2	0	.25					
Pconv	4				1	101			
Pconv	38	21			2	54			
	2.0	235	0	1.0	0.0	0.0			

Alternate Format and Examples:

1	2	3	4	5	6	7	8	9	10
Pconv	Pconid	MID	Form	Expf	"3"	H1	H2	H3	
	H4	H5	H6	H7	H8				
Pconv	20				3	10.0			
Pconv	7				3	10.32	10.05	10.09	
	10.37								

Descriptor	Meaning
Pconid	Convection property identification number. (Integer > 0)
MID	Material property identification number. See Remarks 2. (Integer > 0)
Form	Type of formula used for free convection. (Integer 0, 1, 10, 11, 20, or 21; Default = 0)
Expf	Free convection exponent as implemented within the context of the particular form that is chosen. See Remark 3. (Real ≥ 0.0 ; Default = 0.0)
Ftype	Formula type for various configurations of free convection. See Remarks 2. and 5. (Integer ≥ 0 ; Default = 0)
Tid	Identification number of a TABLEHT entry that specifies the two-variable tabular function of the free convection heat transfer coefficient. See Remark 5. (Integer ≥ 0 or blank)
Chlen	Characteristic length. See Remarks 6. and 8. (Real > 0.0 or blank)

Descriptor	Meaning
GIDIN	Grid ID of the referenced inlet point. See Remarks 7. and 8. (Integer > 0 or blank)
CE	Coordinate system for defining the direction of boundary-layer flow. See Remarks 7. and 8. (Integer ≥ 0 ; Default = 0)
Ei	Component of the vector for defining the direction of boundary-layer flow in coordinate system CE. See Remarks 7. and 8. (Real or blank)
Hi	Free convection heat transfer coefficient. See Remark 5. (Real for H1 and Real or blank for H2 through H8; Default for H2 through H8 is H1)

Remarks:

- Every surface to which free convection is to be applied must reference a PCONV entry. PCONV is referenced on the CONV Bulk Data entry.
- MID is used to supply the convection heat transfer coefficient (H) for FTYPE=0, or the thermal conductivity (K) for FTYPE=2. MID is ignored for FTYPE=1 and FTYPE=3 and may be blank.
- EXPF is the free convection temperature exponent.
 - If FORM = 0, 10, or 20, EXPF is an exponent of $(T - TAMB)$, where the convective heat transfer is represented as

$$q = H \cdot u_{CNTRLND} \cdot (T - TAMB)^{EXPF} \cdot (T - TAMB).$$
 - If FORM = 1, 11, or 21,

$$q = H \cdot u_{CNTRLND} \cdot (T^{EXPF} - TAMB^{EXPF})$$

where T represents the elemental grid point temperatures and TAMB is the associated ambient temperature.
- FORM specifies the formula type and the reference temperature location used in calculating the convection film coefficient if FLMND = 0.
 - If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperatures (average).
 - If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).
 - If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperatures).
- FTYPE defines the formula type used in computing the convection heat transfer coefficient h.
 - If FTYPE = 0, h is specified in the MAT4 Bulk Data entry referenced by MID.
 - If FTYPE = 1, h is computed from $h = f(T_w, T_a)$, where f is a two-variable tabular function specified in the TABLEHT Bulk Data entry referenced by TID, T_w is the wall temperature, and T_a is the ambient temperature.
 - If FTYPE = 2, h is computed from $Nu = f(T_w, T_a)$, where $Nu_L = hL/K$ or $Nu_x = hX/K$ is the Nusselt number, f is a two-variable tabular function specified in the TABLEHT Bulk Data entry referred by TID, T_w is the wall temperature, and T_a is the ambient temperature.

- If FTYPE = 3, h is the free convection heat transfer coefficient applied to grid point Gi of the referenced HBDY surface element.
6. CHLEN specifies the characteristic length used to compute the average heat transfer coefficient \bar{h} . The following table lists typical values of CHLEN for various convection configurations.

Convection Configuration	Characteristic Length CHLEN
Free convection on a vertical plate or cylinder	Height of the plate or cylinder
Free convection from horizontal tubes	Diameter of the pipes
Free convection from horizontal square plates	Length of a side
Free convection from horizontal rectangular plates	Average length of four sides
Free convection from horizontal circular disks	0.9d, where d is the diameter of the disk.
Free convection from horizontal unsymmetric plates	A/P, where A is the surface area and P is the perimeter of the surface.

7. GIDIN, CE and Ei are used to define the distance from the leading edge of heat transfer. GIDIN specifies the referenced grid ID where heat transfer starts. CE and Ei define the direction of boundary-layer flow. If CE field is blank, the default is CE=0 for basic coordinate system. If E1, E2, and E3 fields are blank, the defaults are Ei = < 1.0, 0.0, 0.0 >, i.e. the flow is in the x direction.
8. CHLEN, GIDIN, CE, and Ei are required only for free convection from flat plates with FTYPE = 2. In this case, if the heat transfer coefficient is spatial dependent, GIDIN must be specified. Otherwise, CHLEN has to be defined for the computation of average heat transfer coefficient \bar{h} . For free convection from tubes (CHBDYP elements with TYPE="ELCY", "TUBE" or "FTUBE"), CHLEN, GIDIN, CE, and Ei need not be specified, because MSC Nastran will use the average diameter of tubes as the characteristic length while computing Nu. CHLEN, GIDIN, CE, and Ei are ignored for $FTYPE \neq 2$.
9. For RC network solver in thermal analysis, the FORM, EXPF, FTYPE, TID, CHLEN, GIDIN, CE, E1, E2 and E3 are ignored.

PCONVM

Forced Convection Property Definition

Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
PCONVM	PCONID	MID	FORM	FLAG	COEF	EXPR	EXPPI	EXPO	

Example:

PCONVM	3	2	1	1	.023	0.80	0.40	0.30	
--------	---	---	---	---	------	------	------	------	--

Descriptor Meaning

PCONID	Convection property identification number. (Integer > 0)
MID	Material property identification number. (Integer > 0)
FORM	Type of formula used for convection. (Integer = 0, 1, 10, 11, 20, or 21; Default = 0)
FLAG	Flag for mass flow convection. (Integer = 0 or 1; Default = 0)
COEF	Constant coefficient used for forced convection. (Real > 0.0)
EXPR	Reynolds number convection exponent. (Real \geq 0.0; Default = 0.0)
EXPPI	Prandtl number convection exponent for heat transfer into the working fluid. (Real \geq 0.0; Default = 0.0)
EXPO	Prandtl number convection exponent for heat transfer out of the working fluid. (Real \geq 0.0; Default = 0.0)

Remarks:

1. Every surface to which forced convection is applied must reference a PCONVM entry. PCONVM is referenced on the CONVM entry.
2. MID specifies material properties of the working fluid at the temperature of the point FLMND. FLMND is specified on the CONVM entry.
3. The material properties are used in conjunction with the average diameter and mass flow rate (mdot). MID references the material properties and supplies the fluid conductivity (k), heat capacity (cp), and viscosity (μ) needed to compute the Reynolds (Re) and Prandtl (Pr) numbers as follows:

$$\text{Re} = 4 \cdot |\text{mdot}| / (\pi \cdot \text{diameter} \cdot \mu)$$

$$\text{Pr} = \text{cp} \cdot \mu / k$$

4. FORM controls the type of formula used in determination of the forced convection film coefficient h. There are two cases:

- If FORM = 0, 10, or 20 than $h = \text{coef} \cdot \text{Re}^{\text{EXPR}} \cdot \text{Pr}^{\text{EXPPI}}$.

- If FORM = 1, 11, or 21 then the above h is multiplied by k and divided by the average hydraulic diameter.
 - FORM also specifies the reference temperature used in calculating material properties for the fluid if FLMND = 0.
 - If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperature (average).
 - If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).
 - If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperature).
5. In the above expression, EXPP is EXPPI or EXPPO, respectively, for heat flowing into or out of the working fluid. This determination is performed internally.
 6. FLAG controls the convective heat transfer into the downstream point (the second point as identified on the CHBDYi statement is downstream if mdot is positive).
 - FLAG = 0, no convective flow (stationary fluid).
 - FLAG = 1, convective energy flow that is consistent with the Streamwise Upwind Petrov Galerkin (SUPG) element formulation.
 7. No phase change or internal heat generation capabilities exist for this element.
 8. For RC network solver in thermal analysis, the FORM, FLAG, COEF, EXPR, EXPPI and EXRPO are ignored.

PCONV1**Thermal Convection Calculation Properties**

Defines the properties required to calculate convective heat transfer. Can exist in a simple mode with convection coefficient defined in the MID or in advanced mode where the H value is calculated using the geometric parameters and referenced material.

Format:

1	2	3	4	5	6	7	8	9	10
PCONV1	PID		CorrID	MID	Mdot	Velocity	Length or Diameter	Flow Cross Section	
			Length function type	Flow Cross Sec type	Mdot f	Velocity f	Length or Diameter f	Flow Cross Sec f	
	C1	C2	C3	C4	C5	C6	C7	C8	
	C9	C10	C11	C12	C13	C14	C15	C16	
	C17	C18	C19	C20	C21	C22	C23	C24	

Example:

PCONV1	2		701	2				1.0	
--------	---	--	-----	---	--	--	--	-----	--

Descriptor	Meaning
PID	Property identification number, referenced by CHBDYP and/or CONV. (Integer > 0)
Corr ID	ID of convection type in correlation library (Integer ≥ 0 ; Default 0)
MID	Material identification number for convection calculation (Integer > 0)
Mdot	Mass flow rate (Real; Default 0.0)
Velocity	Fluid velocity (Real ≥ 0.0 ; Default 0.0)
Length or Diameter	Geometry, based on Corr ID (Real ≥ 0.0 ; Default 0.0)
Flow Cross Section	Flow Cross Section (Real ≥ 0.0 ; Default 0.0)
Length function type	Geometry function type, 1=time 2=temperature (Integer > 0; Default 2)
Flow Cross Sec type	Flow Cross Section function type, 1=time 2=temperature (Integer > 0; Default 2)
Mdot f	Mass flow rate TABLEDj table ID (Integer ≥ 0 ; Default 0)
Velocity f	Fluid velocity TABLEDj table ID (Integer ≥ 0 ; Default 0)
Length or Diameter f	Geometric function TABLEMj or TABLEDj table ID (Integer ≥ 0 ; Default 0)
Flow Cross Sec f	Flow Cross Section function TABLEMj or TABLEDj table ID (Integer ≥ 0 ; Default 0)
C1 thru C24	Geometric properties based on chosen convection correlation (Real; Default 0.0)

Remarks:

1. This entry is for RC Network solver only.
2. The PCONV1 entry contains the properties for a CONV and CHDBYP, and can be used for connecting with a PRJCON. PID must be unique to both the PCONVID in PCONV and the PID in PHBDY. This will be the ID referenced by CONV, PRJCON, and CHBDYP.
3. MATID must reference a MAT4 fluid material.
4. For Corr. ID and C1 thru C24, please reference *MSC SINDA User's Guide and Library Reference* or *P/Thermal User's Guide*.

PDAMP**Scalar Damper Property**

Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.

Format:

1	2	3	4	5	6	7	8	9	10
PDAMP	PID1	B1	PID2	B2	PID3	B3	PID4	B4	

Example:

PDAMP	14	2.3	2	6.1					
-------	----	-----	---	-----	--	--	--	--	--

Describer Meaning

PIDI Property identification number. (Integer > 0)

Bi Force per unit velocity. (Real)

Remarks:

1. Damping values are defined directly on the CDAMP2 and CDAMP4 entries, and therefore do not require a PDAMP entry.
2. A structural viscous damper, CVISC, may also be used for geometric grid points.
3. Up to four damping properties may be defined on a single entry.
4. For a discussion of scalar elements, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. PDAMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PDAMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PDAMP entries.

PDAMP5

Scalar Damper Property for CDAMP5

Defines the damping multiplier and references the material properties for damping. CDAMP5 is intended for heat transfer analysis only.

Format:

1	2	3	4	5	6	7	8	9	10
PDAMP5	PID	MID	B						

Example:

PDAMP5	2	3	4.0						
--------	---	---	-----	--	--	--	--	--	--

Describer Meaning

- PID Property identification number. (Integer > 0)
MID Material identification number of a MAT4 or MAT5 entry. (Integer > 0)
B Damping multiplier. (Real > 0.0)

Remarks:

1. B is the mass that multiplies the heat capacity CP on the MAT4 or MAT5 entry.
2. RC network solver does not support PDAMP5 for thermal analysis.

PDAMPT

Frequency-Dependent Damper Property

Defines the frequency-dependent properties for a PDAMP Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
PDAMPT	PID1	TBID1							

Example:

PDAMPT	12	34							
--------	----	----	--	--	--	--	--	--	--

Descriptor Meaning

- PID Property identification number that matches the identification number on a PDAMP entry. (Integer > 0)
- TBID1 Identification number of a TABLEDi entry that defines the damping force per-unit velocity versus frequency relationship. (Integer ≥ 0 ; Default = 0)

Remarks:

1. PDAMPT may only be referenced by CDAMP1 or CDAMP3 elements in the residual structure, which do not attach to any omitted degrees-of-freedom.
2. The PDAMPT entry is ignored in all solution sequences except frequency response analysis.

PDISTB

Element property distributions.

Defines element distributions of property data.

Format:

1	2	3	4	5	6	7	8	9	10
PDISTB	ID	TYPE	PID	PNAME	LOCTNUM	SMULT	DELTA		
	EID1	V1	EID2	V2	EID3	V3	EID4	V4	
	EID5	V5	EID6	V6	EID7	V7	EID8	V8	
	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-			

Example:

PDISTB	7008	PCOMP G	700	GE		.9			
	205	.013	224	.015	362	.135			
PDISTB	70014	PCOMP G	700	T	1				
	205	.013	224	.015	362	.135			
PDISTB	70034	PCOMP G	700	T	3				
	205	.013	224	.015	362	.135	400	.067	

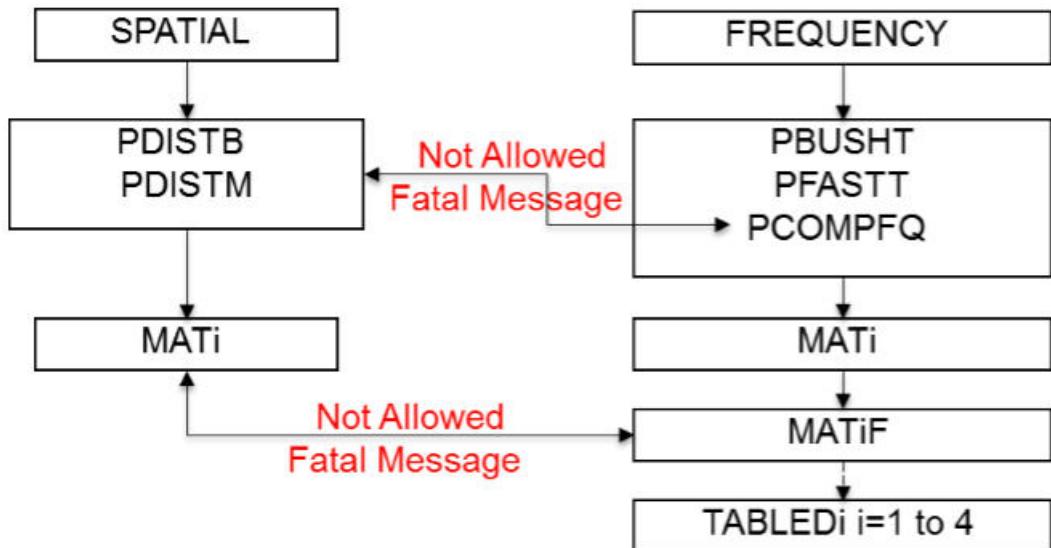
Descriptor Meaning

ID	Unique identification number. (Integer > 0, no default)
TYPE	Name of a property entry, PBAR, PBEAM, PBEAM3, PCOMP, PCOMPG, PROD, PSHEAR, PSHELL and PTUBE (Character-Required).
PID	Property identification number that matches the identification number of an existing Property entry. (Integer > 0, no default)
PNAME	Property name, such as "T", "I12" (Character). See Remark 8..
LOCNUM	The Numerical order of Ply in PCOMP or PCOMPG. The Station number on a PBEAM or PBEAM3. (Integer ≥ 0).
SMULT	Scale factor of distribution entries. (Real, Default=1.0)
DELTA	Delta value of distribution entries. (Real, Default=0.0)
EIDi	Element ID associated with Property PID. (Integer > 0, no default)
Vi	The value of item to be changed. (Real)

Remarks:

1. The ID of the PDISTB is for user convenience of grouping identification and is not used for case control selection.

2. The PID must point to an existing “PSHELL”, “PCOMPG”, and “PCOMP” entry. If this entry is used, Nastran will check that all elements have a unique element ID across all element types. If a non-unique element ID is found a fatal message will be issued.
3. If no distribution is given, the value associated with the named property entry will be used.
4. If an EID occurs on this entry for an element not associated with referred to property the EID is ignored.
5. If an EID is referred to twice on this entry or another PDISTB entry with the same PID and PNAME, Or in the case of a PCOMP or PCOMPG with same PID, PNAME, and LOCNUM referral a fatal message will be issued.
6. $\text{VALUE}_i = \text{SMULT}^* V_i + \text{DELTA}$
7. The PID referred to may not have an associated PCOMPFQ entries. A fatal message will be issued if there is such an association. Additionally a fatal message will be issued if the associated materials are associated with a MATiF entry.



8. For most Primary property entries, the QRG symbol is used for valid PNAME. See values in table below. For more difficult entries such as a PCOMP the PNAME for example of the thickness of PLY 3 would be entered as PNAME=T and LOCTNUM=3. Note on PCOMPG it is the numerical PLY listed not the GPLYID. Thus for example if the second ply listed on a PCOMPG entry had GPLYID=4 and the thickness is to be changed then PNAME=T and LOCTNUM=2 not 4.

Property	PNAME	Corresponding QRG Description
Current Supported Items are:		
PSHELL	T	T - Membrane thickness
	I12	$12I/T^{**3}$ Bending moment of inertia ratio
	TS	TS/T Transverse shear thickness ratio
	NSM	Nonstructural mass.
	Z1, Z2	Fiber distances for stress calculations
PCOMP (G)	Z0	Distance from the reference plane to the bottom surface
	NSM	Nonstructural mass.
	SB	Allowable shear stress of the bonding material
	TREF	Reference temperature
	GE	Damping coefficient
	T	Thicknesses of the various plies (Ply Identified by LOCTNUM)
	THETA	Orientation angle of the various plies. (Ply Identified by LOCTNUM)
PBEAM	LOCTNUM=1 corresponds to END A; LOCTNUM=12 corresponds to END B; LOCTNUM=2-11 Intermediate stations left to right.	
	A	Area of bar cross section (Identified by LOCTNUM)
	I1, I2, I12	Area moments of inertia (Identified by LOCTNUM)
	J	Torsional constant (Identified by LOCTNUM)
	NSM	Nonstructural mass. (Identified by LOCTNUM)
	K1, K2	Shear stiffness factor K in K^*A^*G for plane 1 and plane 2.
PROD	A	Area of bar cross section
	J	Torsional constant
	NSM	Nonstructural mass.
PBAR	A	Area of bar cross section
	I1, I2, I12	Area moments of inertia
	J	Torsional constant
	NSM	Nonstructural mass.
	K1, K2	Area factor for shear

Property	PNAME	Corresponding QRG Description
Current Supported Items are:		
PTUBE	OD	Outside diameter of tube
	T	Thickness of tube
	NSM	Nonstructural mass.
	OD2	Diameter of tube at second grid point
PSHEAR	T	Thickness of shear panel.
	NSM	Nonstructural mass.
	F1, F2	Effectiveness factor for extensional stiffness along edges 2-3 and 1-4.
PBEAM3	LOCNUM=1	corresponds to END A; LOCNUM=2 corresponds to END B; LOCNUM=3 “middle” station C
	A	Area of the beam cross-section (Identified by LOCNUM)
	IZ, IY, IYZ	Area moments of inertia (Identified by LOCNUM)
	J	Torsional constant (Identified by LOCNUM)
	NSM	Nonstructural mass. (Identified by LOCNUM)
	KY, KZ	Shear effectiveness factors for local y- and z-directions.
	CW	Warping coefficient (Identified by LOCNUM).

PDISTBM

Element property material distributions.

Defines element distributions of property data.

Format:

1	2	3	4	5	6	7	8	9	10
PDISTBM	ID	TYPE	PID	MIDNUM					
	EID1	MIDV1	EID2	MIDV2	EID3	MIDV3	EID4	MIDV4	
	EID5	MIDV5	EID6	MIDV6	EID7	MIDV7	EID8	MIDV8	
	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-			

Example:

PDISTBM	70013	PSHELL	700	2					
	205	16							

Descriptor Meaning

ID	Unique identification number. (Integer > 0, no default)
TYPE	Name of a property entry, PBAR, PBEAM, PBEAM3, PCOMP, PCOMPG, PROD, PSHEAR, PSHELL, PSOLID, and PTUBE (Character-Required).
PID	Property identification number that matches the identification number of an existing Property entry. (Integer > 0, no default)
MIDNUM	MIDI name (Integer ≥ 0 , Default=1) See Remark 6.
EIDi	Element ID associated with Property PID. (Integer > 0, no default)
MIDVi	The replacement value of the MIDi. (Integer>0, no default)

Remarks:

1. The PID must point to an existing “PSHELL”, “PCOMPG”, “PCOMP”, and “PSOLID”, entry. If this entry is used, Nastran will check that all elements have a unique element ID across all element types. If a non-unique element ID is found a fatal message will be issued.
2. If no distribution is given, the value associated with the named property entry will be used.
3. If an EID occurs on this entry for an element not associated with referred to property entry the EID is ignored.
4. If an EID is referred to twice on this entry or another PDISTBM entry with the same PID and MIDNUM referral a fatal message will be issued.
5. The MIDVi the user supplies must be a valid MATi entry unique across all existing MATi entries. The MATi referred to may not have associated MATiF entries. A fatal message will be issued if there is such an association.

6. The MIDNUM is property dependent. For example a PSHELL has MID1 through MID4, so a MIDNUM=3 means that the MID3 field of the PSHELL is being changed. For PCOMP or PCOMPG the MIDNUM refers to the MIDi of the Numerical order of ply.

PDUMi**Dummy Element Property**

Defines the properties of a dummy element ($3 \leq i \leq 7$). Referenced by the CDUMi entry.

Format:

1	2	3	4	5	6	7	8	9	10
PDUMi	PID	MID	A1	A2	A3	A4	A5	A6	
	A7	-etc.-							

Example:

PDUM3	108	2	2.4	9.6	1.E4	15.		3.5	
	5		2						

Descriptor Meaning

- | | |
|-----|---|
| PID | Property identification number. (Integer > 0) |
| MID | Material identification number. (Integer > 0) |
| Aj | Additional fields. (Real or Integer) |

Remark:

1. The additional fields are defined in the user-written element subroutines.
2. The fields on this entry are required to be defined on the corresponding ADUMi entry. This entry requires a license for "USER MODIFIABLE Nastran". Other than the PID field, all field checking is the responsibility of the user supplied code.

PELAS**Scalar Elastic Property**

Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).

Format:

1	2	3	4	5	6	7	8	9	10
PELAS	PID1	K1	GE1	S1	PID2	K2	GE2	S2	

Example:

PELAS	7	4.29	0.06	7.92	27	2.17	0.0032		
-------	---	------	------	------	----	------	--------	--	--

Descriptor Meaning

PIDI	Property identification number. (Integer > 0)
Ki	Elastic property value. (Real)
GEi	Damping coefficient, g_e . See Remarks 5. and 6. (Real)
Si	Stress coefficient. (Real)

Remarks:

1. Be careful using negative spring values.
2. Spring values are defined directly on the CELAS2 and CELAS4 entries, and therefore do not require a PELAS entry.
3. One or two elastic spring properties may be defined on a single entry.
4. For a discussion of scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. If PARAM,W4 is not specified, GEi is ignored in transient analysis. See [Parameters, 783](#).
6. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.
7. If PELAS is used in conjunction with PELAST, Ki > 0, and the initial slope of the nonlinear force-displacement relationship defined by the PELAST should agree with Ki.
8. PELAS is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PELAS property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PELAS entries.

PELAS1

Defines a spring property designated by a force-deflection curve for SOL 700. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PELAS1	PID	TID							

Example:

PELAS1	22	33							
--------	----	----	--	--	--	--	--	--	--

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

PID Property identification number. (Integer, no Default, > 0)

TID Identification number of a TABLED1 entry which defines the force deflection curve.
(Integer, no Default, > 0)

Remarks:

1. Unlike PELAST, when PELAS1 is used, no PELAS entry is made.
2. All PELAS and PELAS1 ID's must be unique.
3. This entry may only be referenced by a CELAS1D entry.

PELAST

Frequency Dependent or Nonlinear Elastic Property

Defines the frequency dependent or nonlinear properties for a PELAS Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
PELAST	PID	TKID	TGEID	TKNID					

Example:

PELAST	44	38							
--------	----	----	--	--	--	--	--	--	--

Describer Meaning

- | | |
|-------|--|
| PID | Property identification number that matches the identification number on a PELAS entry. (Integer > 0) |
| TKID | Identification number of a TABLEDi entry that defines the force per unit displacement vs. frequency relationship. (Integer > 0; Default = 0) |
| TGEID | Identification number of a TABLEDi entry that defines the nondimensional structural damping coefficient vs. frequency relationship. (Integer > 0; Default = 0) |
| TKNID | Identification number of a TABLEDi entry that defines the nonlinear force vs. displacement relationship. (Integer > 0; Default = 0) |

Remarks:

1. The PELAST entry may only be referenced by CELAS1 or CELAS3 elements in the residual structure which do not attach to any omitted degrees-of-freedom.
2. For frequency dependent modal frequency response the modes are computed using the nominal Ki values as specified on the PELAS entry.
3. The nominal values are used for all analysis types except frequency response and nonlinear analyses. For frequency dependent modal frequency response the system modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency. For nonlinear analysis the nominal values for Ki should agree with the initial slope of the nonlinear force-displacement relationship defined by the PELAST, or the results will be unpredictable.

4. The following table summarizes the usage PELAST entry in various solution sequences

Field	Frequency Response	NONLINEAR (See Remark 6)	Linear (Non-Frequency Response)
TKID	Used	Ignored	Ignored
TGEID	Used	Ignored	Ignored
TKNID	Ignored	Used	Ignored

5. PELAST is used in SOL 108 and SOL 111 when TKID or TGEID is specified. It is used in SOL 106, SOL 129 and SOL 400 when TKIND is specified. It is ignored in all other solution sequences.
6. This entry is not available in SOL 600 and if entered will cause the job to terminate. If PARAM,MRPELAST,1 is entered this entry will be ignored in SOL 600.

PERMEAB

Air Bag Permeability

Defines the permeability of a COUPLE and/or GBAG (sub)surface.

Permeability is the velocity of gasflow through a (sub)surface and is defined as a linear or tabular function of the pressure difference over the surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PERMEAB	PID	PERMC	PERMT	FLOW	PENV	RHOENV	SIEENV	CP	

Example:

PERMEAB	201	0.5		OUT	1.E5	1.128	2.21E5	1001.	
---------	-----	-----	--	-----	------	-------	--------	-------	--

Descriptor	Meaning
PID	Unique identification number of a PERMEAB entry. (Integer > 0, Required)
PERMC	Permeability is a linear function of the pressure difference. permeability = PERM – C*abs (Pinside – PENV) For Pinside > PENV: outflow For Pinside < PENV: inflow See Remark 3. (Real ≥ 0)
PERMT	Permeability is a tabular function of the pressure difference: table contains: permeability versus Pinside – PENV For Pinside > PENV: outflow For Pinside < PENV: inflow See Remark 3. (Integer ≥ 0)
FLOW	Defines the allowed directions of the flow. (Character, Default = BOTH) BOTH In- and outflow are allowed. IN Only inflow allowed. OUT Only outflow allowed.
PENV	Environmental pressure. (Real > 0, Required)
RHOENV	Environmental density. (Real > 0, Required)
SIEENV	Environmental specific internal energy. (Real > 0, Required)
CP	Environmental specific heat at constant pressure. See Remark 5. (Real > 0)

Remarks:

1. The PERMEAB entry can be referenced from a LEAKAGE entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver or Roe solver using an EOSGAM (ideal gas) equation of state.

3. Either PERM-C or PERM-T must be specified.
4. The values for the environment p_{env} (PENV), ρ_{env} (RHOENV), e_{env} (SIEENV) must be defined consistent with an ideal-gas equation of state:

$$p_{env} = (\gamma_{env} - 1)\rho_{env}e_{env}$$

The γ_{env} is calculated and is used when inflow occurs. Inflow occurs when $p_{env} > p_{inside}$.

5. CP is only required if updating of Euler or gasbag gas constants is done, for example if hybrid inflators are defined.

PERMGBG

Air Bag Permeability

Defines a permeable area of a COUPLE and/or GBAG surface, connected to another GBAG.

The velocity of the gas flow through the surface is defined as a linear or tabular function of the pressure difference. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PERMGBG	PID	PERMC	PERMT	FLOW	GBID				

Example:

PERMGBG	12		10		2				
---------	----	--	----	--	---	--	--	--	--

Descriptor	Meaning
PID	Unique identification number of a PERMEAB entry. It can be referenced from either a LEAKAGE to model the flow between GBAGs, or from a LEAKAGE to model the flow between an Eulerian air bag and a GBAG. (Integer > 0; Required)
PERMC	Permeability is a linear function of the pressure difference. permeability = PERM – C*abs (Pinside – Pgbid) The gas flow is from the higher to the lower pressure. See Remark 3. (Real > 0)
PERMT	Permeability is a tabular function of the pressure difference: table contains: permeability versus Pinside – Pgbid The gas flow is from the higher to the lower pressure. See Remark 3. (Integer > 0)
FLOW	Defines the allowed directions of the flow. (Character; Default = BOTH) BOTH In- and outflow are allowed. IN Only inflow allowed into the GBAG or the coupling surface that references this entry. OUT Only outflow allowed into the GBAG or the coupling surface that references this entry.
GBID	Number of a GBAG entry. This GBAG is the one that is connected to the GBAG or coupling surface that references this entry. (Integer > 0; Required)

Remarks:

1. The PERMGBG entry can be referenced from a LEAKAGE entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver, using an EOSGAM (ideal gas) equation of state.

3. Either PERMC or PERMT must be specified.

PEULER

Eulerian Element Properties

Defines the properties of Eulerian element. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PEULER	PID	MID	TYPE						

Example:

PEULER	100	25							
--------	-----	----	--	--	--	--	--	--	--

Describer Meaning

PID	Unique property number. (Integer > 0, Required)
MID	Number of a MATDEUL entry defining the constitutive model. (Integer ≥ 0 , Required)
TYPE	The type of Eulerian material being used. (Character, Default = HYDRO)
	HYDRO Hydrodynamic material with no shear strength + void.
	1STORDER Single material, 1 st order accurate Riemann solution-based fluids- & gases Euler solver.
	2NDORDER Single material, 2 nd order accurate Riemann solution-based fluids- & gases Euler solver.
	STRENGTH Structural material with shear strength + void.
	MMHYDRO Multimaterial hydrodynamic material with no shear strength + void.
	MMSTREN Structural multimaterial with shear strength + void.

Remarks:

1. Make the property number unique with respect to all other property numbers.
2. The elements that reference this property use the Eulerian formulation.
3. If TYPE is set to HYDRO, only one material number for all the Eulerian elements of TYPE is used and a hydrodynamic yield model is chosen.
4. If the TYPE is set to either 1st Order or 2nd Order, only one material for all Eulerian elements of TYPE is used and the Riemann solution-based solver is chosen.
5. If TYPE is set to STRENGTH, only one material number for all the Eulerian elements of TYPE is used and a nonhydrodynamic yield model is chosen.
6. If TYPE is set to MMHYDRO, different material numbers for all Eulerian elements of TYPE are used and a hydrodynamic behavior is chosen for each material.
7. If TYPE is set to MMSTREN, different material numbers for all Eulerian elements of TYPE are used and a yield model is chosen for each material.

8. In a multimaterial Euler calculation, the options MMSTREN and MMHYDRO cannot be mixed; they are mutually exclusive.
9. If the material number is blank or zero, the corresponding elements are void. Note that this is not allowed in the Riemann solution-based Euler solvers, as they will not handle void elements. If you define void elements and select either the 1st Order or 2nd Order scheme, an error message will be issued and the analysis will stop.
10. Initial conditions are defined on the TICEL Bulk Data entry.

PEULER1

Eulerian Element Properties

Eulerian element properties. The initial conditions of these elements are defined in geometric regions. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PEULER1	PID		TYPE	SID	SID2				

Example:

PEULER1	100		HYDRO	300					
---------	-----	--	-------	-----	--	--	--	--	--

Descriptor Meaning

PID	Unique property number. (Integer > 0; Required)												
TYPE	They type of Eulerian material(s) being used. (Character; Default = HYDRO) <table> <tr> <td>HYDRO</td><td>Hydrodynamic material + void.</td></tr> <tr> <td>1STORDER</td><td>Single material, 1st order accurate Riemann solution-based fluids- & gases solver.</td></tr> <tr> <td>2NDORDE</td><td>Single material, 2nd order accurate Riemann solution-based fluids- & gases solver.</td></tr> <tr> <td>STRENGTH</td><td>Structural material with shear strength + void.</td></tr> <tr> <td>MMHYDRO</td><td>Multimaterial hydrodynamic material with no shear strength + void.</td></tr> <tr> <td>MMSTREN</td><td>Structural multimaterial with shear strength + void.</td></tr> </table>	HYDRO	Hydrodynamic material + void.	1STORDER	Single material, 1 st order accurate Riemann solution-based fluids- & gases solver.	2NDORDE	Single material, 2 nd order accurate Riemann solution-based fluids- & gases solver.	STRENGTH	Structural material with shear strength + void.	MMHYDRO	Multimaterial hydrodynamic material with no shear strength + void.	MMSTREN	Structural multimaterial with shear strength + void.
HYDRO	Hydrodynamic material + void.												
1STORDER	Single material, 1 st order accurate Riemann solution-based fluids- & gases solver.												
2NDORDE	Single material, 2 nd order accurate Riemann solution-based fluids- & gases solver.												
STRENGTH	Structural material with shear strength + void.												
MMHYDRO	Multimaterial hydrodynamic material with no shear strength + void.												
MMSTREN	Structural multimaterial with shear strength + void.												
SID	Number of a TICEUL1 entry specifying the materials and geometric grouping criteria. (Integer > 0; Required)												
SID2	Number of a EULFOR1 entry defining acceleration field. (Integer>=0 Default=0)												

Remarks:

1. Remarks 1 through 6 of the PEULER definition apply also here.
2. Initial conditions and/or material assignments are defined on the TICEUL1 Bulk Data entry.

PFAST

CFAST Fastener Property

Defines the CFAST fastener property values.

Format:

1	2	3	4	5	6	7	8	9	10
PFAST	PID	D	MCID	MFLAG	KT1	KT2	KT3	KR1	
	KR2	KR3	MASS	GE	ALPHA	TREF	COINL		

Example:

PFAST	7	1.1	70		100000.	46000.	12300.		
-------	---	-----	----	--	---------	--------	--------	--	--

Descriptor Meaning

PID	Property identification number. (Integer > 0)
D	Diameter of the fastener. See Remark 2. (Real > 0)
MCID	Specifies the element stiffness coordinate system. See Remark 1. (Integer ≥ -1 or blank; Default = -1)
MFLAG	Defines if the coordinate system defined by MCID is absolute or relative. See Remark 1. (Integer 0 or 1; Default = 0) If MFLAG = 0, MCID defines a relative coordinate system. See Remark 1a. If MFLAG = 1, MCID defines an absolute coordinate system. See Remark 1c.
KTi	Stiffness values in directions 1 through 3. (Real)
KRi	Rotational stiffness values in directions 1 through 3. (Real; Default = 0.0)
MASS	Lumped mass of fastener. (Real; Default = 0.0)
GE	Structural damping. (Real; Default = 0.0)
ALPHA	Thermal expansion coefficient for the CFAST. (Real; Default=0.0)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0, See Remark 8.)
COINL	Length of a CFAST with coincident grids. (Real; Default=0.0, COINL ≥ 0.0)

Remarks:

1.

- a. If MCID ≥ 0 and MFLAG = 0 (Default), then the KT1 stiffness will be applied along the x_{elem} axis direction of the fastener defined as

$$\hat{e}_1 = \frac{\hat{x}_B - \hat{x}_A}{\|\hat{x}_B - \hat{x}_A\|}$$

The T2 direction defined by MCID will be used to define the orientation vector \vec{v} of the fastener. Then the element z_{elem} axis will be defined as

$$\hat{\vec{e}}_3 = \frac{\hat{\vec{e}}_1 \times \vec{v}}{\|\hat{\vec{e}}_1 \times \vec{v}\|}$$

The KT3 stiffness will lie along the z_{elem} axis. The element y_{elem} axis is defined as

$$\hat{\vec{e}}_2 = \hat{\vec{e}}_3 \times \hat{\vec{e}}_1$$

The KT2 stiffness will lie along the y_{elem} axis

This option allows the user to define orthotropic material properties normal to the axis of the fastener that will “slide” with the curve of the patches.

- b. If MCID = -1, MFLAG is ignored, and the following element system is defined: the x_{elem} axis direction of the fastener defined as

$$\hat{\vec{e}}_1 = \frac{\vec{x}_B - \vec{x}_A}{\|\vec{x}_B - \vec{x}_A\|}$$

Relative to the basic system, find the smallest component j of the element x_{elem} axis unit vector. If two such components are equal, take the first one. Form a unit vector in the basic system. For example, assuming the $j = 3$ component of $\hat{\vec{e}}_1$ was the smallest.

$$b_j = b_3 = \begin{Bmatrix} 0 \\ 0 \\ 1 \end{Bmatrix}$$

Form the following orthogonal vector:

$$\hat{\vec{e}}_2 = \vec{b}_j - \frac{\hat{\vec{e}}_1 \cdot \vec{b}_j}{\hat{\vec{e}}_1 \cdot \hat{\vec{e}}_1} \hat{\vec{e}}_1$$

$$\hat{\vec{e}}_2 = \frac{\hat{\vec{e}}_2}{\|\hat{\vec{e}}_2\|}$$

Form $\hat{\vec{e}}_3$ as

$$\hat{\vec{e}}_3 = \hat{\vec{e}}_1 \times \hat{\vec{e}}_2$$

- c. If MCID ≥ 0 and MFLAG = 1, then the MCID will be used to compute stiffness. KT1 will be applied along the MCID T1 axis, KT2 along the MCID T2 axis, and KT3 along the MCID T3 axis. The element forces will be computed in the coordinate system defined in Remark 1b.
- d. If the length of GA - GB is zero, then the element x_{elem} axis is defined to lie along the projected normal to patch A.

2. The diameter D is used along with the piercing points of GA and GB to determine the location of fictitious grid points to form a fictitious hexa volume that determines the elements and physical grids used for the fastener element. Four points are positioned at $\pm a$ positions parallel to the element axis where $a = f(D)$. The stiffness contribution of the fastener depends on both the stiffness values specified and the diameter D, because the $\pm a$ positions are used along with the surface shape functions of the fictitious hexa to weight the contribution of the physical grids used to the grids GA and GB of the fastener element.
3. The CFAST element (see Figure 9-134), for stiffness and structural damping calculations, is designed to satisfy rigid body equilibrium requirements. When $\hat{x}_B - \hat{x}_A$ has finite length, internal rigid links connect grids GA and GB. This may result in coupling between translational and rotational degrees-of-freedom even when no rotational stiffness (KR1-KR3) are specified. For mass calculations, half the specified mass value is placed directly onto the projected grid A and grid B translational degrees-of-freedom.

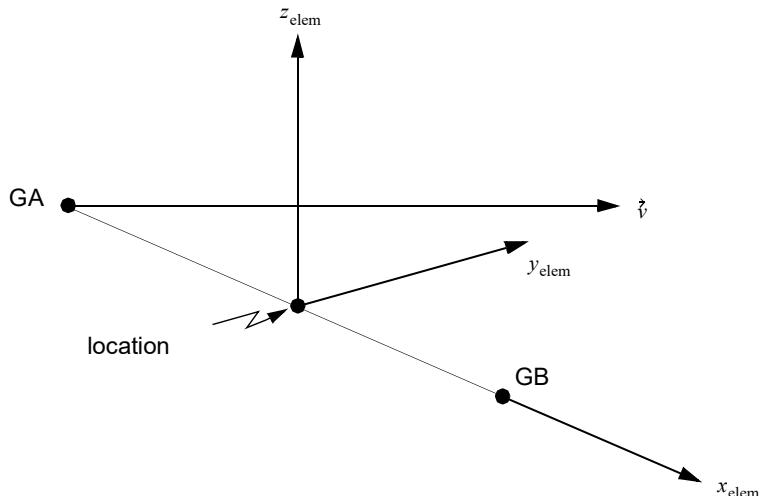


Figure 9-134 CFAST Element

4. The CFAST element lies midway between GA and GB.
5. Values for KT_i and KR_i are specified at the user's discretion. Assuming a short stubby beam where shear is dominate, possible values might be:

$$KT1 = \frac{EA}{L}$$

$$KT2 = \frac{G_2 A_s}{L}$$

$$KT3 = \frac{G_3 A_s}{L}$$

$$KR1 = \frac{GJ}{L}$$

$$KR2 = \frac{EI}{L} + \frac{G_2 A_s L}{3}$$

$$KR3 = \frac{EI}{L} + \frac{G_3 A_s L}{3}$$

where:

$$A = \pi D^2 / 4$$

$$I = \pi D^4 / 64$$

$$J = \pi D^4 / 32$$

$$L = |\vec{x}_B - \vec{x}_A|$$

$$A_s = A_s = A / \alpha_s$$

$$\alpha_s = 4 / 3$$

E , G_2 , G_3 , and G are the material properties of the fastener.

The fastener stiffness is not, however, independent of the surrounding structure. The values of stiffness specified should not overwhelm the stiffness of the local structure or max ratio's will occur. One possible way to estimate the local stiffness S is by the relationship.

$$S = \frac{t_p E_p E}{E_p + E}$$

where t_p is a shell thickness and E_p is the modulus of the shell.

6. The element force and strain are computed as follows:

$$\{f_e\} = [K_e] \{u_e\} \text{ for statics}$$

$$\{f_e\} = ([K_e] + i(g + g_e)[K_e])(\{u_e\}_{\text{real}} + i\{u_e\}_{\text{imag}}) \text{ for frequency}$$

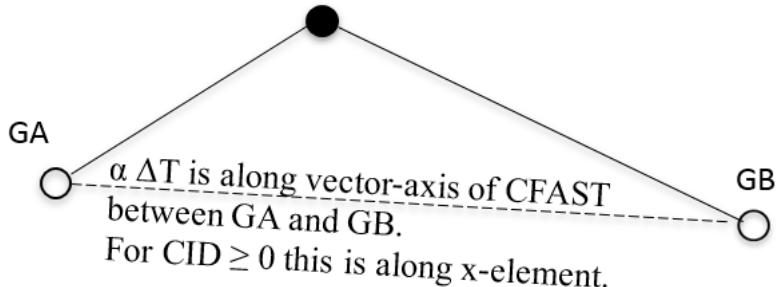
$$\{f_e\} = [K_e] \{u_e\} + \left(\frac{g}{w3} + \frac{g_e}{w4} \right) [K_e] \{v_e\} \text{ for transient}$$

where $[K_e]$ is the 6×6 element stiffness matrix, $\{u_e\} = \{u_b\} - \{u_a\}$ relative displacement in the element coordinate system, and $\{v_e\} = \{v_b\} - \{v_a\}$ relative velocity in the element coordinate system. The subscripts a and b stand for end A and end B of the fastener. g is defined by param,g; $w3$ is defined by param,w3, $w4$ is defined by param,w4; and g_e is the GE entry of the PFAST. $\{u_e\}$ is the strain output. Stress output is the same as force output.

7. PFAST is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PFAST property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PFAST entries.

8. The thermal expansion for the CFAST will be calculated if the user supplies the thermal expansion coefficient ALPHA and TEMPERATURE(LOAD) is requested. See TEMPERTURE Case Control command for thermal loading rules.

Fast Location



PFASTT

Frequency Dependent and material nonlinear CFAST Property

Defines the frequency dependent and material nonlinear properties for a PFAST Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
PFASTT	PID	"K"	TKID1	TKID2	TKID3	TKID4	TKID5	TKID6	
		"GE"	TGEID						
		"KN"	TKNID1	TKNID2	TKNID3	TKNID4	TKNID5	TKNID6	
			FDC	FUSE	DIR	OPTION	LOWER	UPPER	
			FSR	LRGR					

Examples:

PFASTT	33	"GE"	158						
--------	----	------	-----	--	--	--	--	--	--

Descriptor	Meaning	
PID	Property identification number that matches the identification number on PFAST entry. (Integer > 0)	
"K"	Flag indicating that the next 1 to 6 fields are stiffness frequency table identification numbers. (Character)	
TKID _i	Identification number of a TABLED _i entry that defines the stiffness vs. frequency relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)	
"GE"	Flag indicating that the next field is a structural damping frequency table identification number. (Character)	
TGEID	Identification number of a TABLED _i entry that defines the nondimensional structural damping coefficient vs. frequency relationship. (Integer ≥ 0; Default = 0)	
"KN"	Flag indicating that the next 1 to 6 fields are nonlinear force-deflection table identification numbers. (Character)	
TKNID _i	Identification number of a TABLED _i entry that defines the force vs. deflection relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)	
FDC	Force deflection curve rule. Specifies dependence between displacement components. See PBUSHT FDC entry of detail of rules.	
FUSE	0	The element remains active irrespective of failure level. (Integer 0, 1, or 2, Default = 0)
	1	The element is deactivated if maximum failure as specified in OPTION is reached. Elements remains for post processing.
	2	The element is deactivated if maximum failure as specified in OPTION is reached. Element is removed from post processing.

Descriptor	Meaning
DIR	The fuse direction. (0 < Integer < 6; Default = 0). 0 or 123456 imply any direction may fuse. Values of 1 through 6 may be placed in the field with no embedded blanks. See Remark 13. of PBUSHT.
OPTION	<p>Specifies a failure mode. (Character - Default="RELDIS")</p> <p>"ULTLD" The specified failure load in compression or tension will be used to define failure using load computed from the appropriate radial dependence of displacement relationship.</p> <p>"RELDIS" The specified max relative + displacement will be used to define failure.</p> <p>"RULTLD" The specified failure load in compression or tension will be used to define failure using the actual current physical load in each spring.</p>
LOWER	<p>Lower failure bound. (Real; Default=0.0)</p> <p>If OPTION = "ULTLD" then LOWER specifies a lower failure load</p> <p>If OPTION = "RELDIS" then LOWER specifies a minimum relative displacement before failure.</p>
UPPER	<p>Upper failure bound. (Real; Default=0.0)</p> <p>If OPTION = "ULTLD" then UPPER specifies an upper failure load.</p> <p>If OPTION = "RELDIS" then UPPER specifies a maximum relative displacement before failure.</p>
FSR	Fuse Stiffness Retention Factor is a factor which scales the stiffness so that the stiffness does not instantly drop to a zero value. (Real > 0.0; Default = 1.-5)
LRGR	Specifies if large rotation is to occur at end A. (Integer > 0; Default = 0). Remark 14. of the PBUSHT.

Remarks:

1. For frequency dependent modal frequency response the modes are computed using the nominal Ki values as specified on the PFAST entry.
2. The nominal values are used for all analysis types except frequency response. For frequency dependent modal frequency response the system modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency.
3. The PFAST entry may only be referenced by CFAST elements in the residual structure of SOL400 which do not attach to any omitted degrees-of-freedom.
4. For nonlinear analysis the nominal values for Ki on the PFAST should agree with the initial slope of the nonlinear force-displacement relationship defined by the PFAST, or the results will be unpredictable.

PFTG

Fatigue Properties.

Defines fatigue properties of elements.

Format:

1	2	3	4	5	6	7	8	9	10
PFTG	ID		FINISH	KFINISH	KF	SCALE	OFFSET		
	SHAPE	KTREAT	DIAM	T1	T2	SPTFLG			

Examples:

PFTG	3		POLISH			1.2			
------	---	--	--------	--	--	-----	--	--	--

Descriptor	Meaning
ID	Unique ID referenced by a FTGDEF bulk data entry. (Integer>0)
FINISH	Material Surface Finish. This is a result of manufacturing process. Value can be NONE, POLISH, GROUND, MACHINE, POOR, ROLLED, CAST, KROUGH, KSURFC (Character; Default=NONE). See Remark 1.
KFINISH	Roughness factor for FINISH = KROUGH (0.0 < Real < 1.0, no Default). Surface roughness in microns for FINISH = KSURFC (Real > 0.0; no Default; 0.0 ≤ Real ≤ 1.0 for materials not listed in Remark 2.).
KF	Fatigue strength reduction factor. (Real ≥ 0.0, Default=1.0). See Remark 3.
SCALE	Factor used to scale the resulting FE stresses of entities associated to this property set (Real; Default=1.0).
OFFSET	Offset used to offset the resulting FE stresses of entities associated to this property set (Real; Default=0.0).
SHAPE	Shape factor (Real ≥ 1.0; No default). See Remark 4.
KTREAT	Treatment factor (Real ≥ 0.0; Default = 1.0).
DIAM	Spot weld nugget diameter. Used in the fatigue analysis of spot welds only. (Real > 0.0 or blank, No Default). See Remark 5. and 6.
T1/T2	Top (T1) and bottom (T2) thickness of shells connecting spot welds. Used in the fatigue analysis of spot welds only. (Real > 0.0 or blank, No Default). Both should be left blank if either one is left blank. See Remark 5.
SPTFLG	Flag to indicate that a lookup table is to be used to define the spot weld nugget diameter. 0 or 1, Default = 0, no lookup). Only used if CWELD elements are used to define spot welds. See Remark 6.

Remarks:

1. The KF field can be used in lieu of or in addition to the FINISH & KTREAT field to modify the fatigue limit by multiplying the original fatigue limit by this value. POOR = Poor Machined. ROLLED = As Rolled. CAST = As Cast. KROUGH and KSURFC require that a KFINISH be entered. A material CODE on the MATFTG entry must be supplied to use anything other than NONE or POLISH, otherwise an error is issued.
2. If KFINISH = KSURFC, the user should enter a value for surface roughness R_z in μm . This is the average surface roughness according to the German standard DIN 4768. The Surface Roughness Factor K_r will then be calculated based on the strength and type of material (for example stronger materials are in general more sensitive to surface finish, and cast materials less so). The method for calculating K_r is taken from the FKM guideline *Analytical Strength Assessment of Components in Mechanical Engineering*.

If $R_z \leq 1\mu\text{m}$, $K_r = 1$.

Otherwise:

$$K_r = 1 - a_r \log(R_z) \log(2R_m/R_{m,N,min})$$

R_m is the UTS in MPa

$R_{m,N,min}$ and a_r are constants.

Table 9-30 Constants for Derivation of Surface Roughness Factor K_r from Roughness R_z

Material	Steel	GS*	GGG†	GT‡	GG**	Wrought Al Alloys	Cast Al alloys
CODE from MATFTG	13,15,16-99	9-12, 15	5-8	2-4	1	100-105	106
a_r	0.22	0.20	0.16	0.12	0.06	0.22	0.20
$R_{m,N,min}$	400	400	400	350	100	133	133

*GS = cast steel and heat treatable cast steel, for general purpose

†GG = cast iron with lamellar graphite (grey cast iron)

‡GGG = nodular cast iron

**GT = malleable cast iron

3. Fatigue strength reduction factor can take into account notch effects, size effects, and loading type influence. $K_f = C_{\text{notch}} \cdot C_{\text{size}} \cdot C_{\text{loading}}$ where the latter three are correction factors for each effect, respectively.
4. Setting the shape factor activates the Seeger Heuler plastic limit load correction. Leave this field blank if no plastic limit load correction is required.
5. If T1, T2, and DIAM are specifically supplied, they are used directly in the fatigue analysis of spot welds. If either T1 or T2 are blank, the thicknesses are automatically determined from the PSHELL entries that connect the spot welds.

When performing optimization using SOL 200, it is necessary to leave T1 and T2 blank if the connecting shell thicknesses are being used as design variable, otherwise the fatigue analysis of the spot welds will not use correct thicknesses from one design cycle to another.

6. If DIAM is left blank, the following rule is used:

- The diameter is derived based on the minimum thickness of the two sheets either side of the weld by performing a lookup on a table. This table is provided in the `spotweld.sys` file in the `msc20xx/<mach_arch>/nCode/nsyy` directory of a standard MSC Nastran installation. This is true for element types CBAR/CBEAM and CHEXA used to define spot welds. If using CWELD elements the diameter is automatically extracted from the PWELD entry, if SPTFLG=0. If SPTFLG=1, then the lookup table is used for CWELDs.

In the case of a table lookup to determine DIAM, the thicknesses of the top and bottom sheets must be within the range of the lookup table, which is up to 3mm by default. Otherwise a fatal error is issued. Also the diameter is set as a function of the thickness of the thinnest sheet joined by the spot weld. No interpolate between the data points in the `spotweld.sys` file is done; rather, the thickness of the thinnest sheet from each spot weld is compared to the table, and the value of thickness that is nearest to but less than or equal to the thickness of the sheet is identified. The corresponding diameter from the table is assigned to that spot weld.

Example `spotweld.sys` file:

```
SPOT WELD DEFINITION FILE
NUGGET DIAMETER=BY_THICKNESS
0.3,3.5
0.8,4.0
1.2,5.0
2.0,5.5
3.0,6.0
```

Note that there may be rounding errors in the extraction of sheet thicknesses, so when defining a `spotweld.sys` file, it may be a good idea to reduce the sheet thickness values by a small tolerance in order to avoid anomalous results

Table 9-31 Availability of Settings for Different Analysis Types

Fatigue Property	Analysis / Material Type Category			
	SN	EN	SpotWeld	SeamWeld
FINISH	✓	✓	✓	✓
KFINISH	✓	✓†		
KF	✓	✓†		
SCALE	✓	✓	✓	✓
OFFSET	✓	✓	✓	✓
SHAPE		✓		

Table 9-31 Availability of Settings for Different Analysis Types

Fatigue Property	Analysis / Material Type Category			
	SN	EN	SpotWeld	SeamWeld
KTREAT	✓	✓†		
DIAM			✓	
T1/T2			✓	
SPTFLG			✓	

✓ Available; † Not available for Multi-mean or Multi R-ratio curves

PGAP**Gap Element Property**

Defines the properties of the gap element (CGAP entry).

Format:

1	2	3	4	5	6	7	8	9	10
PGAP	PID	U0	F0	KA	KB	KT	MU1	MU2	
	TMAX	MAR	TRMIN						

Example:

PGAP	2	.025	2.5	1.E6		1.E6	0.25	0.25	
------	---	------	-----	------	--	------	------	------	--

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
U0	Initial gap opening. See Figure 9-136 . (Real; Default = 0.0)
F0	Preload. See Figure 9-136 . (Real \geq 0.0; Default = 0.0)
KA	Axial stiffness for the closed gap; i.e., $U_a - U_b > U0$. See Figure 9-136 . (Real > 0.0)
KB	Axial stiffness for the open gap; i.e., $U_a - U_b < U0$. See Figure 9-136 . See Remark 2. (Real \geq 0.0; Default = $10^{-14} \cdot KA$)
KT	Transverse stiffness when the gap is closed. See Figure 9-137 . It is recommended that $KT \geq (0.1 \cdot KA)$. (Real \geq 0.0; Default = $MU1 \cdot KA$)
MU1	Coefficient of static friction (μ_s) for the adaptive gap element or coefficient of friction in the y transverse direction (μ_y) for the nonadaptive gap element. See Remark 3. and Figure 9-137 . (Real \geq 0.0; Default = 0.0)
MU2	Coefficient of kinetic friction (μ_k) for the adaptive gap element or coefficient of friction in the z transverse direction (μ_z) for the nonadaptive gap element. See Remark 3. and Figure 9-137 . (Real \geq 0.0 for the adaptive gap element, $MU2 \leq MU1$; Default = $MU1$)
TMAX	Maximum allowable penetration used in the adjustment of penalty values. The positive value activates the penalty value adjustment. See Remark 4. (Real; Default = 0.0)
MAR	Maximum allowable adjustment ratio for adaptive penalty values KA and KT. See Remark 5. ($1.0 < \text{Real} < 10^6$; Default = 100.0)
TRMIN	Fraction of TMAX defining the lower bound for the allowable penetration. See Remark 6. ($0.0 \leq \text{Real} \leq 1.0$; Default = 0.001)

Remarks:

1. [Figure 9-135](#), [Figure 9-136](#), and [Figure 9-137](#) show the gap element and the force-displacement curves used in the stiffness and force computations for the element.

2. For most contact problems, KA (penalty value) should be chosen to be three orders of magnitude higher than the stiffness of the neighboring grid points. A much larger KA value may slow convergence or cause divergence, while a much smaller KA value may result in inaccurate results. The value is adjusted as necessary if TMAX > 0.0.
3. When the gap is open, there is no transverse stiffness. When the gap is closed and there is friction, the gap has the elastic stiffness (KT) in the transverse direction until the friction force is exceeded and slippage starts to occur.
4. There are two kinds of gap elements: adaptive gap and nonadaptive gap. If TMAX \geq 0.0, the adaptive gap element is selected by the program. When TMAX = 0.0, penalty values will not be adjusted, but other adaptive features will be active (i.e., the gap-induced stiffness update, gap-induced bisection, and subincremental process). The value of TMAX = -1.0 selects the nonadaptive (old) gap element. The recommended allowable penetration TMAX is about 10% of the element thickness for plates or the equivalent thickness for other elements that are connected to the gap.
5. The maximum adjustment ratio MAR is used only for the adaptive gap element. Upper and lower bounds of the adjusted penalty are defined by

$$\frac{K^{init}}{\text{MAR}} \leq K \leq K^{init} \cdot \text{MAR}$$

where K^{init} is either KA or KT.

6. TRMIN is used only for the penalty value adjustment in the adaptive gap element. The lower bound for the allowable penetration is computed by TRMIN · TMAX . The penalty values are decreased if the penetration is below the lower bound.

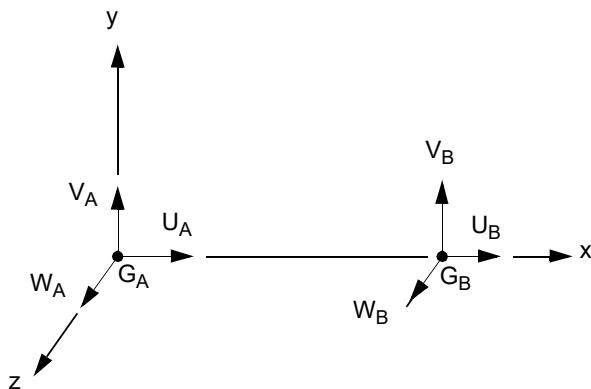


Figure 9-135 The CGAP Element Coordinate System

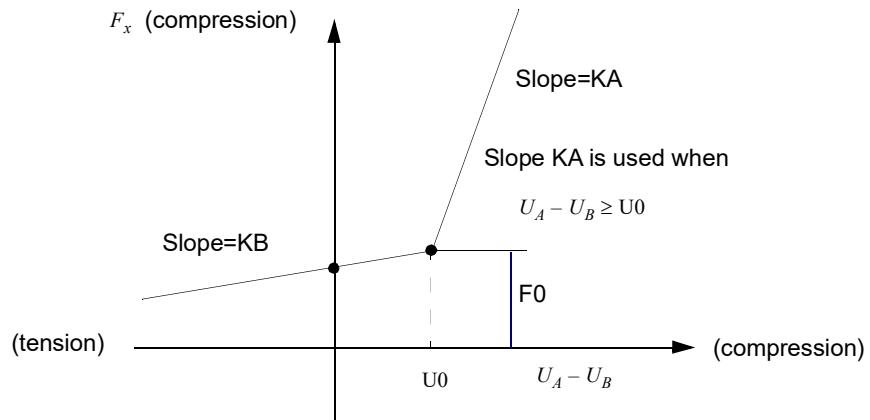


Figure 9-136 CGAP Element Force-Deflection Curve for Nonlinear Analysis

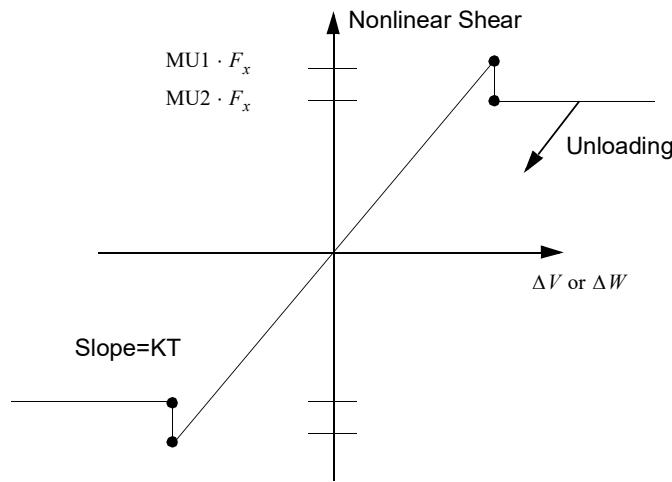


Figure 9-137 Shear Force for CGAP Element

7. If U_0 is specified negative and GA and GB are not coincident, then the direction for closing must be controlled by the use of the CID field on the CGAP entry.
8. PGAP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PGAP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PGAP entries.

PHBDY

CHBDYP Geometric Element Definition

A property entry referenced by CHBDYP entries to give auxiliary geometric information for boundary condition surface elements.

Format:

1	2	3	4	5	6	7	8	9	10
PHBDY	PID	AF	D1	D2					

Example:

PHBDY	2	.02	1.0	1.0					
-------	---	-----	-----	-----	--	--	--	--	--

Describer Meaning

PID	Property identification number. (Unique Integer among all PHBDY entries). (Integer > 0)
AF	Area factor of the surface used only for CHBDYP element TYPE = “POINT”, TYPE = “LINE”, TYPE = “TUBE”, or TYPE = “ELCYL”. For TYPE = “TUBE”, AF is the constant thickness of the hollow tube. (Real > 0.0 or blank)
D1, D2	Diameters associated with the surface. Used with CHBDYP element TYPE = “ELCYL”, “TUBE”, and “FTUBE”. (Real > 0.0 or blank; Default for D2 = D1)

Remarks:

1. The PHBDY entry is used with CHBDYP entries.
2. AF
 - For TYPE = “POINT” surfaces, AF is the area.
 - For TYPE = “LINE” or TYPE = “ELCYL” surfaces, AF is the effective width: area = AF · (length).
 - For TYPE = “FTUBE” and outer TYPE = “TUBE” surfaces
$$\text{area} = \pi \cdot \left(\frac{D1 + D2}{2}\right) \cdot \sqrt{\left(LGTH\right)^2 + \left(\frac{D1 - D2}{2}\right)^2}$$
3. D1 and D2 are used only with TYPE = “ELCYL”, TYPE = “TUBE”, and TYPE = “FTUBE” surfaces.
 - For TYPE = “ELCYL” surfaces, D1 and D2 are the two diameters associated with the ellipse.
 - For TYPE = “FTUBE” and outer TYPE = “TUBE” surfaces, D1 and D2 are the diameters associated with the first and second grid points, respectively.
 - For inner TYPE = “TUBE” surfaces, the diameters are reduced by twice the thickness ($2 \cdot AF$) .

PINTC

Properties of Geometric Interface -- Curve

Defines the properties for interface elements along curve interfaces between boundaries of multiple subdomains of p-elements.

Format:

1	2	3	4	5	6	7	8	9	10
PINTC	PID	TOL	DSCALE						

Example:

PINTC	1	0.01	1000.0						
-------	---	------	--------	--	--	--	--	--	--

Descriptor Meaning

- | | |
|--------|--|
| PID | Property identification number. (Integer > 0) |
| TOL | Tolerance for distance between interface elements and subdomain boundaries. See Remark 2. (Real > 0.0; Default = 0.01) |
| DSCALE | Scaling parameter for Lagrange multiplier functions. See Remark 3. (Real > 0.0; Default = 1000.0) |

Remarks:

1. All PIDs must be unique.
2. TOL may be specified for the distance between the interface element and the boundaries. If the distance is greater than TOL, a warning message will occur. If the distance is less than TOL, but greater than the tolerance used by the geometric evaluator for the GMCURV method, a warning will be issued from the geometric evaluator.
3. DSCALE does not need to be specified unless the interface elements are poorly conditioned. Poor DSCALE conditioning can be determined from the epsilon value of the linear equation solution. A good value for DSCALE is two or three orders of magnitude less than the elastic moduli of the subdomain boundaries.

PINTS**Properties of Geometric Interface -- Surface**

Defines the properties for interface elements along surface interfaces between boundaries of multiple subdomains of p-elements.

Format:

1	2	3	4	5	6	7	8	9	10
PINTS	PID	TOL	DSCALE						

Example:

PINTS	1	0.01	1000.						
-------	---	------	-------	--	--	--	--	--	--

Descriptor	Meaning	Type	Default
PID	Property identification number	Integer > 0	Required
TOL	Tolerance for distance between interface element and subdomain boundaries.	Real > 0	0.01
DSCALE	Scaling parameter for Lagrange multiplier functions.	Real > 0	1000.

Remarks:

1. All PIDs must be unique.
2. The TOL tolerance may be specified for the distance between the interface element and the subdomain boundaries. If the distance is greater than the TOL, a warning will be issued. If the distance is less than the TOL, but greater than the tolerance used by the geometric evaluator for the GMSURF, a warning from the geometric evaluator will be issued.
3. The DSCALE scaling parameter for the Lagrange multipliers does not need to be changed unless the interface elements are poorly conditioned. This could be determined from the epsilon value of the linear equation solution. A good value for DSCALE, which has the units of elastic modulus, is two or three orders of magnitude less than the elastic modulus of the subdomain boundaries.

PLCOMP

Plane Strain or Axisymmetric Composite Element Property

Defines global (external) ply IDs and properties for a composite material laminate.

Format:

1	2	3	4	5	6	7	8	9	10
PLCOMP	PID	DIRECT	THICKOP	SB	ANAL				
	"C4"	BEH4	INT4	BEH4H	INT4H				
	"C8"	BEH8	INT8	BEH8H	INT8H				
	ID1	MID1	T1	THETA1					
	ID2	MID2	T2	THETA2					

Example:

PLCOMP	782	1							
	1001	171	.3	12.3					
	100	175	.7	77.7					

Descriptor	Meaning
PID	Element property identification number. (Integer > 0)
DIRECT	The layer direction for BEHi=COMPS or AXCOMP. See Remark 5. for direction definition. A positive value implies that the composite layer input is a fractional percent of the total element thickness in the ply direction and is recommended. A negative value implies that the composite layer input is the actual thickness of that ply. (Integer ± 1 or ± 2 ; Default + 1)
THICKOP	An out-of-plane thickness. (Real, Default = 1.0)
ANAL	Analysis type. ANAL='IS' - Implicit structural elements are being referred to. ANAL='IH' - Implicit heat analysis elements are being referred to. ANAL= "ISH" - Implicit structural and heat elements are being referred to. (Character Default ISH).
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). (Real ≥ 0.0)
C4	Keyword indicating that two items following apply to elements with four corner grids. (Character)
C8	Keyword indicating that two items following apply to elements with four corner grids and four mid-side grids. (Character)
BEHi	Element structural behavior. See Remarks 4. and 7. (Character default: COMPS for BEH4 and BEH8)
INTi	Integration scheme. See Remark 9. (Character Default: L for INT4H, Q for INT8H and INT8H)

Descriptor	Meaning
BEHiH	Element heat behavior. See Remark 8. (Character Default: COMPS for BEH4H and BEH8H)
INTiH	Integration scheme. See Remark 9. (Character Default: L for INT4H, Q for INT8H, Q for INT8H)
IDi	Global Ply ID. Must be unique with respect to other plies in this entry. See Remark 2. (Integer > 0)
MIDi	Material ID for the ply. See Remark 3. (Integer > 0)
Ti	Either fractional percent of the total element thickness or actual thickness of that ply depending on \pm value of DIRECT. See Remarks 5. and 6. (Real > 0.0)
THETAi	Orientation angle of the ply in the plane of the plies. These angles are measured about the thickness direction of the element. (Real; Default = 0.0)

Remarks:

1. The PLCOMP can only be referenced by a CQUAD, CQUAD4, CQUAD8, or CQUADX entry. Currently it should be used ONLY in SOL400. Erroneous results will occur in other Solution sequences.
2. Global Ply ID is intended as a unique ply identifier for ply alignment across ALL PCOMPG, PLCOMP, and PCOMPLS entries.
3. The MIDi entry may point to MAT1, MAT3, MATHORT, MATHE, MATUSR or MATDIGI entries. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

Implicit Structural Materials					
MAT1	MAT3	MATORT	MATHE	MATUSR	MATDIGI
MATVE	<MATVE>	<MATVE>	MATVE		
MATVP	MATVP	MATVP			
MATEP	MATEP	MATEP			
MATF	MATF	MATF			
MATS1		MATSORT			

<MATVE> refers to the ALTERNATE format for type ORTHO

Heat Materials

MAT4 | MAT5

If heat analysis is being performed and the user wishes to override standard MSC Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

4. The keyword entries may occur, between themselves, in any order or not at all. If a keyword entry is missing, its defaults are assumed.
5. The following table describes layer orientation for BEHi=COMPS or AXCOMP. A total of 1026 plies are allowed for any one element.

Note the ply numbering starts from the bottom to the top parallel to the positive thickness direction.

Layer Orientation			
DIRECT	Normal to Layer edge	Layers run parallel from edge (ply numbering starts here)	to edge (ends here)
1	Element Y direction	G1-G2	G4-G3
2	Element X direction	G1-G4	G2-G3

6. The ply thickness of the element is computed using isoparametric coordinates of the element in the DIRECT direction and the element nodes are mapped between -1 and + 1. The ply thickness is entered in one of two ways:
 - a. Relative thickness where the numbers are a fractional percent of the total thickness. This is the preferred method. For this method, the sum of all the fractional percentages of thickness must sum to 1.0.
 - b. Absolute thickness where the layer thickness is entered directly. Using this option, the code sums the total user input thickness across all plies and then figures the fractional percent of each individual ply as in method 6a.
7. In the following table, BEHi refers to the structural behavior of 2D-solid elements. An underlined item delineates a default.

Implicit Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Plane Strain composite	COMPS	L	QUAD	4
		Q	QUAD	8
Axisymmetric composite	AXCOMP	L	QUAD	4
		Q	QUAD	8

8. In the following table, BEHiH refers to the heat behavior of 2D-solid elements. An underlined item delineates a default.

Heat Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Plane Strain composite	COMPS	<u>L</u>	QUAD	4
		<u>Q</u>	QUAD	8
Axisymmetric composite	AXCOMP	L	QUAD	4
		Q	QUAD	8

9. Integration codes in Remark 7 are:

INT CODE	Integration Type
L	Linear
Q	Quadratic

10. PLCOMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PLCOMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PCOMPLS, PLCOMP, PCOMPG, PSHELL, PLPLANE entries.

PLOAD**Static Pressure Load**

Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD	SID	P	G1	G2	G3	G4			

Example:

PLOAD	1	-4.0	16	32	11				
-------	---	------	----	----	----	--	--	--	--

Descriptor Meaning

SID Load set identification number. (Integer > 0)

P Pressure. (Real)

Gi Grid point identification numbers. (Integer > 0; G4 may be zero or blank.)

Remarks:

1. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. The grid points define either a triangular or a quadrilateral surface to which a pressure is applied. If G4 is blank, the surface is triangular.
3. In the case of a triangular surface, the assumed direction of the pressure is computed according to the right-hand rule using the sequence of grid points G1, G2, G3 illustrated in [Figure 9-138](#).

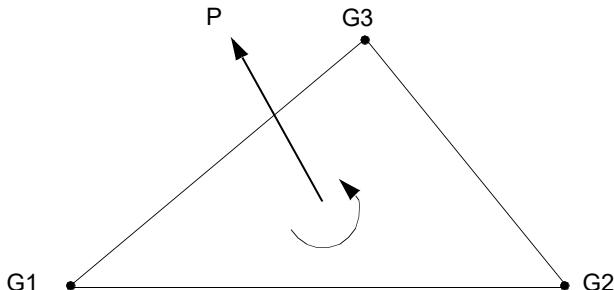
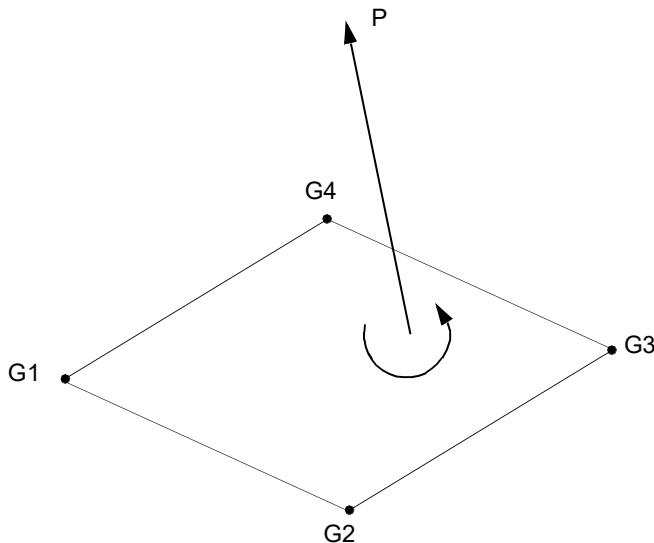


Figure 9-138 Pressure Convention for Triangular Surface of Surface Elements and/or the Faces of Solid Elements

The total load on the surface (see [Figure 9-139](#)), AP, is divided into three equal parts and applied to the grid points as concentrated loads. A minus sign in field 3 reverses the direction of the load.

4. In the case of a quadrilateral surface, the grid points G1, G2, G3, and G4 should form a consecutive sequence around the perimeter. The right-hand rule is applied to find the assumed direction of the pressure. Four concentrated loads are applied to the grid points in approximately the same manner as for a triangular surface. The following specific procedures are adopted to accommodate irregular and/or warped surfaces:
 - The surface is divided into two sets of overlapping triangular surfaces. Each triangular surface is bounded by two of the sides and one of the diagonals of the quadrilateral.
 - One-half of the pressure is applied to each triangle, which is then treated in the manner described in Remark [2](#).



[Figure 9-139](#) Pressure Convention for Quadrilateral Surface of Surface Elements and/or the Faces of Solid Elements

5. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, 400 and 600 if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153, 400 and 600) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

PLOAD1

Applied Load on CBAR, CBEAM or CBEND Elements

Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis. For the CBEND element, only distributed loads over an entire length may be defined.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	

Example:

PLOAD1	25	1065	MY	FRPR	0.2	2.5E3	0.8	3.5E3	
--------	----	------	----	------	-----	-------	-----	-------	--

Descriptor Meaning

SID	Load set identification number. (Integer > 0)
EID	CBAR, CBEAM, or CBEND element identification number. (Integer > 0)
TYPE	Load type. (Character: "FX", "FY", "FZ", "FXE", "FYE", "FZE", "MX", "MY", "MZ", "MXE", "MYE", "MZE")
SCALE	Determines scale factor for X1, X2. (Character: "LE", "FR", "LEPR", "FRPR")
X1, X2	Distances along the CBAR, CBEAM, or CBEND element axis from end A. (Real; X2 may be blank; $0 \leq X1 \leq X2$)
P1, P2	Load factors at positions X1, X2. (Real or blank)

Remarks:

1. In the static solution sequences, SID must be selected by the LOAD Case Control command.
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. If $X2 \neq X1$, a linearly varying distributed load will be applied to the element between positions X1 and X2, having an intensity per unit length of bar equal to P1 at X1 and equal to P2 at X2, except as noted in Remarks 8. and 9.
3. If X2 is blank or equal to X1, a concentrated load of value P1 will be applied at position X1.
4. If P1 = P2 and X2 \neq X1, a uniform distributed load of intensity per unit length equal to P1 will be applied between positions X1 and X2 except as noted in Remarks 8. and 9.
5. Load TYPE is used as follows to define loads:
"FX", "FY" or "FZ": Force in the x, y, or z direction of the basic coordinate system.
"MX", "MY" or "MZ": Moment in the x, y, or z direction of the basic coordinate system.

- “FXE”, “FYE” or “FZE”: Force in the x, y, or z direction of the element’s coordinate system.
- “MXE”, “MYE” or “MZE”: Moment in the x, y, or z direction of the element’s coordinate system.
6. If SCALE = “LE” (length), the xi values are actual distances along the element axis, and, if $X1 \neq X2$, then Pi are load intensities per unit length of the element.
 7. If SCALE = “FR” (fractional), the xi values are ratios of the distance along the axis to the total length, and (if $X2 \neq X1$) Pi are load intensities per unit length of the element.
 8. If SCALE = “LEPR” (length projected), the xi values are actual distances along the element axis, and (if $X2 \neq X1$) the distributed load is input in terms of the projected length of the element.

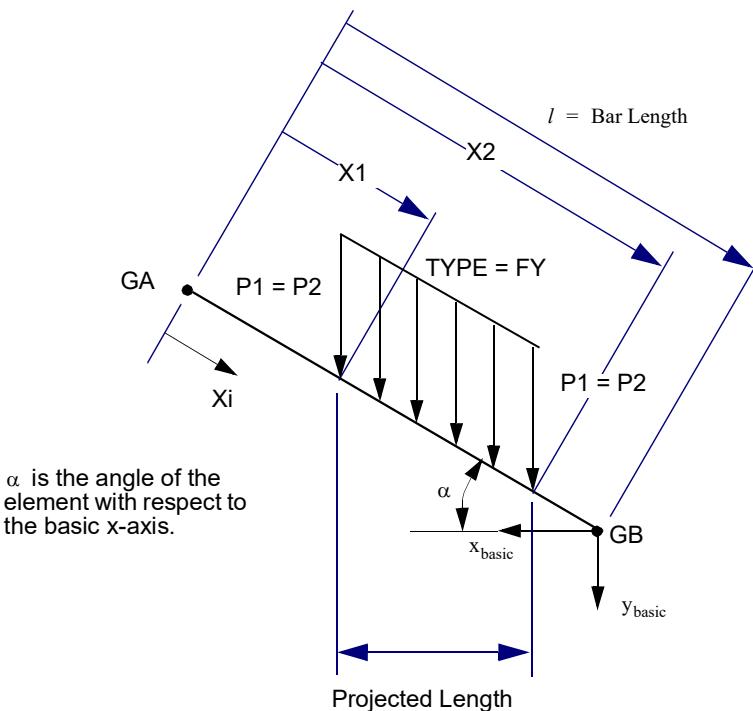


Figure 9-140 PLOAD1 Convention on Beam or Bar Elements

- If SCALE = “LE”, the total load applied to the bar is $P_1 (X_2 - X_1)$ in the y-basic direction.
- If SCALE = “LEPR”, the total load applied to the bar is $P_1 (X_2 - X_1) \cos\alpha$ in the y-basic direction.
9. If SCALE = “FRPR” (fractional projected), the Xi values are ratios of the actual distance to the length of the bar (CBAR entry), and if $X1 \neq X2$, then the distributed load is specified in terms of the projected length of the bar.
 10. Element identification numbers for CBAR, CBEAM, and CBEND entries must be unique.
 11. For the CBEND element, the following coordinate equivalences must be made for the element coordinates

$$R_{elem} \equiv X_{elem}$$

$$\theta_{elem} \equiv Y_{elem}$$

12. Only distributed loads applied over the entire length of the CBEND element may be applied.
13. Projected loads are not applicable to the CBEND element.
14. Loads on CBEAM elements defined with PLOAD1 entries are applied along the line of the shear centers.
15. If a CBARAO or PLOAD1 entry is specified and stress and/or force output is requested, then the stresses and/or forces will be calculated at each location X_i and output as a separate line. The force and stress locations $X_i=0$ and $X_i = l$ will always be output. This output format will be used for all beam and bar elements.
16. If on the TYPE field of the PLOAD1 entry, the element coordinate system direction (e.g. TYPE = FYE) option is selected, then the projection (i.e. SCALE = FRPR or LEPR) option is ignored and the result is the same as the SCALE = FR (or LE) option.

PLOAD2

Uniform Normal Pressure Load on a Surface Element

Defines a uniform static pressure load applied to CQUAD4, CSHEAR, or CTRIA3 two-dimensional elements.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	EID8	-etc.-						

Example:

PLOAD2	21	-3.6		4	16		2		
--------	----	------	--	---	----	--	---	--	--

Alternate Format and Example:

PLOAD2	SID	P	EID1	"THRU"	EID2				
PLOAD2	1	30.4	16	THRU	48				

Describer Meaning

- SID Load set identification number. (Integer > 0)
 P Pressure value. (Real)
 EIDi Element identification number. (Integer ≥ 0 or blank; for the "THRU" option, EID1 < EID2.)

Remarks:

1. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. At least one positive EID must be present on each PLOAD2 entry.
3. If the alternate form is used, all elements EID1 through EID2 must be two-dimensional.
4. The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element entry. Refer to the PLOAD entry.
5. All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.
6. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution

sequences, SOLs 106, 129, 153, 159, 400 and 600, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153, 400 and 600) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

7. The PLOAD2 entry may not be applied to p-elements. The PLOAD4 must be used.

PLOAD4**Pressure Load on Surface and Faces of Solid Elements**

Defines a pressure load on a face of a CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element. See Remark 17.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4	G1	G3 or G4	
	CID	N1	N2	N3	SORL	LDIR			

Example:

PLOAD4	2	1106	10.0	8.0	5.0		48		
	6	0.0	1.0	0.0					

Alternate Format and Example (See Remark 8.):

PLOAD4	SID	EID1	P1	P2	P3	P4	"THRU"	EID2	
	CID	N1	N2	N3	SORL	LDIR			

PLOAD4	2	1106	10.0	8.0	5.0		THRU	1143	
	6	0.0	1.0	0.0					

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
EID EID1 EID2 } EID2	Element identification number. (Integer > 0; for the "THRU" option, EID1 < EID2)
P1, P2, P3, P4	Load per unit surface area (pressure) at the corners of the face of the element. (Real or blank; Default for P2, P3, and P4 is P1.)
G1	Identification number of a grid point connected to a corner of the face. Required data for solid elements only. (Integer > 0 or blank)
G3	Identification number of a grid point connected to a corner diagonally opposite to G1 on the same face of a CHEXA, CPENTA or CPYRAM element. Required data for quadrilateral faces of CHEXA, CPENTA and CPYRAM elements only. For CPYRAM element triangle faces, G1 and G3 are adjacent corner nodes on the quadrilateral face, and the load is applied on the triangular face which includes those grids. G3 must be omitted for a triangular surface on a CPENTA element.
G4	Identification number of the CTETRA grid point located at the corner; this grid point may not reside on the face being loaded. This is required data and is used for CTETRA elements only. (Integer > 0)

Descriptor	Meaning
CID	Coordinate system identification number. See Remark 2. (Integer ≥ 0 ; Default = 0)
N1, N2, N3	Components of vector measured in coordinate system defined by CID. Used to define the direction (but not the magnitude) of the load intensity. See Remark 2. (Real)
SORL	The character string SURF or LINE. SURF means the surface load acting on the surface of the element and LINE means the consistent edge loads acting on the edges of the element. The default is SURF. See Remark 13.
LDIR	Denote the direction of the line load (SORL=LINE), character string X, Y, Z, TANG, or NORM. The default is NORM. See Remark 14.

Remarks:

1. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. The continuation entry is optional. If fields 2, 3, 4, and 5 of the continuation entry are blank, the load is assumed to be a pressure acting normal to the face. If these fields are not blank, the load acts in the direction defined in these fields. Note that if CID is a curvilinear coordinate system, the direction of loading may vary over the surface of the element. The load intensity is the load per unit of surface area, not the load per unit of area normal to the direction of loading.
3. For the faces of solid elements, the direction of positive pressure (defaulted continuation) is inward. For triangular and quadrilateral faces, the load intensity P1 acts at grid point G1 and load intensities P2, P3, (and P4) act at the other corners in a sequence determined by applying the right-hand rule to the outward normal.
4. For plate elements, the direction of positive pressure (defaulted continuation) is in the direction of positive normal, determined by applying the right-hand rule to the sequence of connected grid points. The load intensities P1, P2, P3, (and P4) act respectively at corner points G1, G2, G3, (and G4) for triangular and quadrilateral elements. (See plate connection entries.)
5. If P2, P3, and P4 are blank fields, the load intensity is uniform and equal to P1. P4 has no meaning for a triangular face and may be left blank in this case.
6. Equivalent grid point loads are computed by linear or bilinear interpolation of load intensity followed by numerical integration using isoparametric shape functions. Note that a uniform load intensity will not necessarily result in equal equivalent grid point loads.
7. G1 and G3 are ignored for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements.
8. The alternate format is available only for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements. The continuation entry may be used in the alternate format.

9. For triangular faces of CPENTA elements, G1 is an identification number of a corner grid point that is on the face being loaded and the G3 or G4 field is left blank. For faces of CTETRA elements, G1 is an identification number of a corner grid point that is on the face being loaded and G4 is an identification number of the corner grid point that is not on the face being loaded. Since a CTETRA has only four corner points, this point G4 will be unique and different for each of the four faces of a CTETRA element.
10. For the CQUADR and CTRIAR element, only pressure that acts normal to the element is computed properly. Surface tractions are not resolved into moments normal to the element.
11. All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.
12. If fields 3 through 5 of the continuation entry are not blank, the load is assumed to have a fixed direction. If fields 2 through 5 of the continuation entry are left blank, the load is assumed to be a pressure load. In this case, follower force effects are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, 400 and 600, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153, 400 and 600) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
13. The SORL field is ignored by all elements except QUADR and TRIAR. For QUADR or TRIAR only, if SORL=LINE, the consistent edge loads are defined by the PLOAD4 entry. P1, P2, P3 and P4 are load per unit length at the corner of the element. If all four Ps are given, then the line loads along all four edges of the element are defined. If any P is blank, then the line loads for only two edges are defined. For example, if P1 is blank, the line loads of the two edges connecting to G1 are zero. If two Ps are given, then the line load of the edge connecting to the two grid points is defined. If only one P is given, the second P value default to the first P value. For example, P1 denotes that the line load along edge G1 and G2 has the constant value of P1.
14. The direction of the line load (SORL=LINE) is defined by either (CID, N1, N2, N3) or LDIR. Fatal error will be issued if both methods are given. TANG denotes that the line load is in tangential direction of the edge, pointing from G1 to G2 if the edge is connecting G1 and G2. NORM denotes that the line load is in the mean plan, normal to the edge, and pointing outward from the element. X, Y, or Z denotes the line load is in the X, Y, or Z direction of the element coordinate system. If both (CID, N1, n2, N3) and LDIR are blank, then the default is LDIR=NORM.
15. For SOL 600, the SORL field may also be used by CQUAD4 and CTRIA3 in addition to CQUADR and CTRIAR.
16. For SOL 600, the LDIR field must be blank or a fatal error will occur. SOL 600 line loads must use the CID, N1, N2, N3 fields.
17. For SOL 400, with large displacement (PARAM,LGDISP), higher-order 6-node triangles (CTRIA6), 8-node quadrilateral (CQUAD8), 20-node hexagonal (CHEXA), or 15-node pentahedral (CPENTA), the load is not calculated correctly. Use lower-order elements if distributed loads are required in a large displacement analysis.

PLOADB3

Applied distributed load on CBEAM3 elements

Defines a distributed load to a CBEAM3 element over entire length of the beam axis.

Format:

1	2	3	4	5	6	7	8	9	10
PLOADB3	SID	EID	CID	N1	N2	N3	TYPE	SCALE	
	P(A)	P(B)	P(C)						

Example:

PLOADB3	10	1002	LOCAL	1.0			MOMENT		
	100.	90.	70.						

Descriptor Meaning

SID	Load set identification number. (Integer > 0; Required)
EID	CBEAM3 element identification number. (Integer > 0, Required)
CID	Coordinate system for load definition. (Character or Integer; Default = "BASIC") "LOCAL" Local coordinate system; "ELEMENT" Element coordinate system; "BASIC" or 0 Basic coordinate system;
n (n>0):	Any user-specified coordinate system identification number.
N1, N2, N3	Load vector components measured in coordinate system specified by CID. (Real; at least one Ni ≠ 0.0)
TYPE	Type of applied load. (Character = "FORCE", "MOMENT" or "BIMOMENT"; Required)
SCALE	Load vector scale factor. (Real; Default = 1.0)
P(j)	Magnitudes of load at j (j=A, B and C). (Real; Default = 0.0)

Remarks:

1. In the static solution sequences, SID must be selected by the LOAD Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

2. The load vector is defined by $\vec{P}_j = SCALE \cdot P_j \cdot \vec{N}$, (j = A, B, C). The orientation of load \vec{P} is determined by vector \vec{N} and the magnitude is equal to $SCALE \cdot P$ times magnitude of vector \vec{N} .

3. The distributed load is applied over the entire length of the beam axis, along the line of the shear center.

PLOADX1

Pressure Load on Axisymmetric Element

Defines surface traction to be used with CAXISYM, CQUADX, CTRIAZ, and CTRIAZ6 axisymmetric elements.

Format:

1	2	3	4	5	6	7	8	9	10
PLOADX1	SID	EID	PA	PB	GA	GB	THETA		

Example:

PLOADX1	200	35	3.5	10.5	10	30	20.		
---------	-----	----	-----	------	----	----	-----	--	--

Descriptor Meaning

SID	Load set identification number. (Integer > 0)
EID	Element identification number. (Integer > 0)
PA	Surface traction at grid point GA. (Real)
PB	Surface traction at grid point GB. (Real; Default = PA)
GA, GB	Corner grid points. GA and GB are any two adjacent corner grid points of the element. (Integer > 0)
THETA	Angle between surface traction and inward normal to the line segment. (Real; Default = 0.0)

Remarks:

1. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. PLOADX1 is intended only for the CAXISYM, CQUADX, CTRIAZ, and CTRIAZ6 elements.
3. The surface traction is assumed to vary linearly along the element side between GA and GB.
4. The surface traction is input as force per unit area. MSC Nastran converts this traction into equivalent nodal loads by integration over the surface. For AXISYM and hyperelastic QUADX and TRIAX elements this integration is over one radian. For TRIAX6 and axisymmetric harmonic QUADX and TRIAX elements this integration is over two pi radians.
5. THETA is measured counter-clockwise from the inward normal of the straight line between GA and GB, to the vector of the applied load, as shown in [Figure 9-141](#), [Figure 9-142](#), and [Figure 9-143](#). Positive pressure is in the direction of inward normal to the line segment.

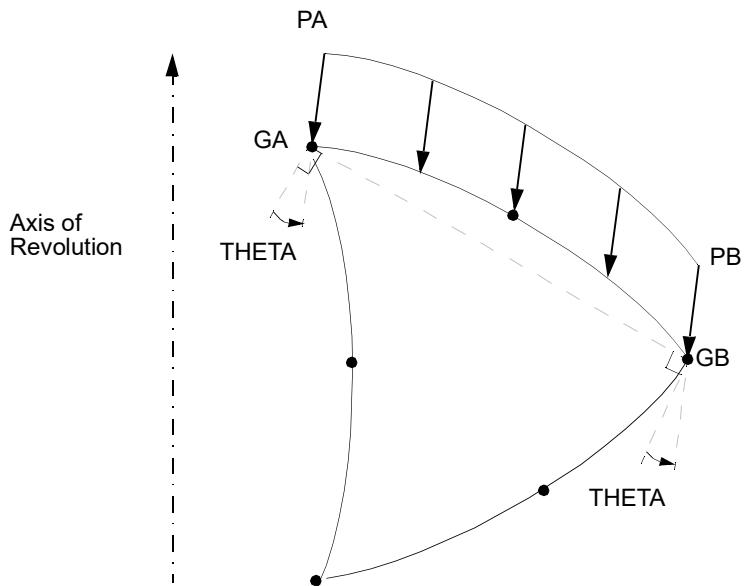


Figure 9-141 Pressure Load on CTRIA6 or CTRIAX Element

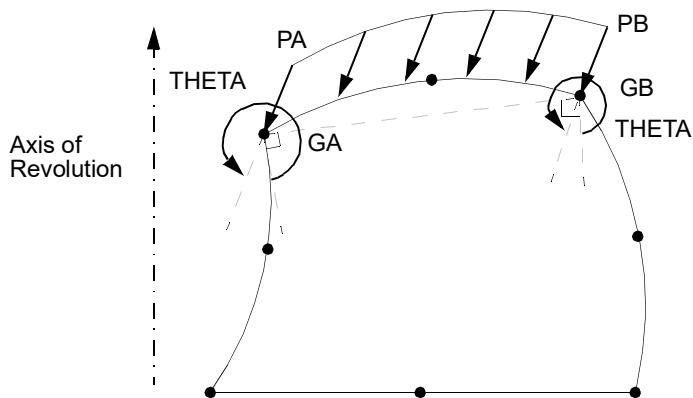


Figure 9-142 Pressure Load on CQUADX Element

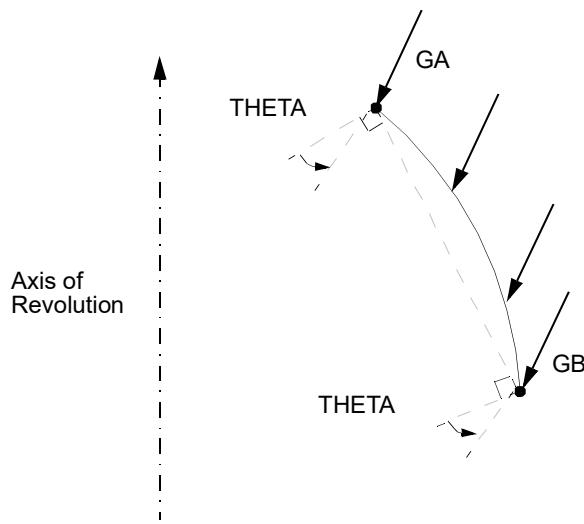


Figure 9-143 Pressure Load on CAXISYM Element

6. Axisymmetric harmonic elements may have Harmonic "N" greater than zero. For this case the "PA" and "PB" fields of the PLOADX1 entry are coefficients of $\cos(N*\theta)$. The PLOADX1 cannot supply azimuthal tractions on axisymmetric harmonic elements.

PLOTEL**Dummy Plot Element Definition**

Defines a one-dimensional dummy element for use in plotting.

Format:

1	2	3	4	5	6	7	8	9	10
PLOTEL	EID	G1	G2						

Example:

PLOTEL	29	35	16						
--------	----	----	----	--	--	--	--	--	--

Describer Meaning

- EID Element identification number. (Integer > 0)
 G1, G2 Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. This element is not used in the model during any of the solution phases of a problem. It is used to simplify plotting of structures with large numbers of colinear grid points, where the plotting of each grid point along with the elements connecting them would result in a confusing plot.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one PLOTEL element may be defined on a single entry.
4. In superelement analysis, PLOTELs, as well as other elements such as CBAR, CQUAD4, etc., will affect the formation of the superelement tree. The PLOTEL EIDs will also appear in the superelement map output; see the description of PARAM,SEMAPRPT in [Parameters, 783](#).
5. Only grid points connected by structural elements appear on structure plots. This does not include points connected only by rigid or general elements or MPCs. A plot element in parallel with elements that do not plot will cause these points to be present.

PLPLANE

Fully Nonlinear Plane Element Properties

Defines the properties of a fully nonlinear (i.e., large strain and large rotation, etc.) plane strain, plane stress, or axisymmetric element. Please refer to PSHLN2.

Format:

1	2	3	4	5	6	7	8	9	10
PLPLANE	PID	MID	CID	STR					

Example:

PLPLANE	203	204	-2	GRID					
---------	-----	-----	----	------	--	--	--	--	--

Descriptor Meaning

PID	Element property identification number. (Integer > 0)
MID	Identification number of a MATHP entry for Nastran conventional elements, or MATHE and other linear or nonlinear material entries for Nastran advanced nonlinear elements. (Integer > 0). See Remarks 7.
CID	Identification number of a coordinate system defining the plane of deformation. See Remarks 2. and 3. (Default = 0)
STR	Location of stress and strain output. (Character: "GAUS" or "GRID", Default = "GRID")

Remarks:

1. PLPLANE can be referenced by a CQUAD, CQUAD4, CQUAD8, CQUADX, CTRIA3, CTRIA6, or CTRIAX entry.
2. Plane strain hyperelastic elements must lie on the x-y plane of the CID coordinate system. Stresses and strains are output in the CID coordinate system.
3. Axisymmetric hyperelastic elements must lie on the x-y plane of the basic coordinate system. CID may not be specified and stresses and strains are output in the basic coordinate system.
4. Negative values of CID will indicate an axes combination for axisymmetric elements. The following table describes the value and axes. Inside one model, only one CID value is allowed.

Axial	Radial	CID Value
X	Y	-2
X	Z	-3
Y	X	-4

Axial	Radial	CID Value
Y	Z	-5
Z	X	-6
Z	Y	-7

5. CID = 0 is the default value which equal to CID = -2.
6. PLPLANE is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PLPLANE property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness among PLPLANE, PAXSYMH, and PSHELL entries.
7. When using Enhanced materials (SOL400), the MID field may point to additional Material entries such as a MAT1. For a full list see the PSHLN2 entry. If a material type other than the MATHP or MATHE entry is used in solution sequences other than SOL400, wrong results will occur. Note linear stress recovery is not available for any element using the PLPLANE.

PLSOLID

Fully Nonlinear Solid Element Properties

Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.

Format:

1	2	3	4	5	6	7	8	9	10
PLSOLID	PID	MID	STR						

Example:

PLSOLID	20	21							
---------	----	----	--	--	--	--	--	--	--

Descriptor Meaning

- | | |
|-----|---|
| PID | Element property identification number. (Integer > 0) |
| MID | Identification number of a MATHP entry. (Integer > 0) |
| STR | Location of stress and strain output. (Character: “GAUS” or “GRID”, Default = “GRID”) |

Remarks:

1. PLSOLID can be referenced by a CHEXA, CPENTA, or CTETRA entry.
2. Stress and strain are output in the basic coordinate system.
3. PLSOLID is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PLSOLID property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PLSOLID entries.

PLTSURF**Surface plot mesh definition**

Defines the surface plot mesh.

Format:

1	2	3	4	5	6	7	8	9	10
PLTSURF	EID	GID1	GID2	GID3	GID4	GID5	GID6	GID7	
	GID8								

Descriptor Meaning

EID Identification number of element.

GID_i Grid identification number of structure/cavity models or trim component surface element.

Remarks:

1. This element is used only to describe the surface mesh of 3D model. The connection can be 3, 4, 6 or 8 nodes and the connection sequence mirrors those of CTRIA3, CQUAD4, CTRIA6 and CQUAD8.
2. PLTSURF can be utilized to describe surface mesh on ACPEMCP for trim component coupling in PEM job.
3. For trim component, PLTSURF ID must be referenced on SET3,id,ELEM. If SET1 entry is used with PLTSURF ID, fatal error or unexpected results may be the results.
4. PLTSURF entry can also be used to define surface mesh for structure/cavity models and it behaves similarly as PLOTEL and does not participate in analysis.

PMARKER

Property Definition of a Marker Element

Defines the behavior of the marker element in the EULER domain. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PMARKER	ID	TYPE							

Example:

PMARKER	7	FIXED							
---------	---	-------	--	--	--	--	--	--	--

Describer Meaning

- | | |
|--------|--|
| ID | Marker property ID, referred by CMARKB2 and CMARKN1 entries. (Integer > 0, Required) |
| TYPE | Behavior of the marker grid points in the Euler domain: (Character, Default = FIXED) |
| FIXED | The marker will not move in the Euler domain. |
| MOVING | The marker will be moved by velocities in the Euler domain. |

Remarks:

1. The PMARKER entry will be ignored for elements that are referring to structural grid points. These structural grid points will move with the structure and the Euler velocities do not change their velocity.
2. TYPE = FIXED. This means that the marker is stationary through out the simulation and it is therefore not moving with the Euler velocity. If the marker grid is located outside the Eulerian domain(s), the Marker will still be allowed to exist. It means, however, that no variables are recorded and that the variables will appear as zero on the Time History plots.
3. TYPE = MOVING. The marker is moving along with the Eulerian material. When the grid point approaches a coupling surface there is no mechanism that prevents the marker from passing through the coupling surface. When this happens the marker enters an element that is covered and motion of the grid point will stop. It is allowed that the grid point moves from one Euler domain to the other through a porous hole or a coupling surface with interactive failure.

PMASS

Scalar Mass Property

Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries).

Format:

1	2	3	4	5	6	7	8	9	10
PMASS	PID1	M1	PID2	M2	PID3	M3	PID4	M4	

Example:

PMASS	7	4.29	6	13.2					
-------	---	------	---	------	--	--	--	--	--

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

PID _i	Property identification number. (Integer > 0)
------------------	---

M _i	Value of scalar mass. (Real)
----------------	------------------------------

Remarks:

1. Mass values are defined directly on the CMASS2 and CMASS4 entries, and therefore do not require a PMASS entry.
2. Up to four mass values may be defined by this entry.
3. For a discussion of scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
4. PMASS is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PMASS property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PMASS entries.

PMINC

Constant Spallation Model

Defines a spallation model where the minimum pressure is constant. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PMINC	PID	VALUE	FVTOL	FVTOL2					

Example:

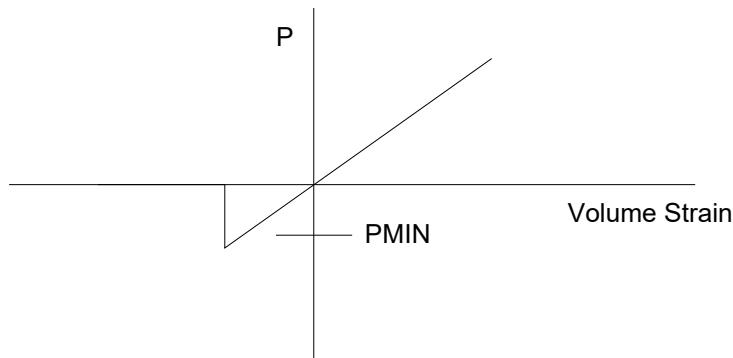
PMINC	220	-370							
-------	-----	------	--	--	--	--	--	--	--

Descriptor Meaning

PID	Unique PMINC number. Referenced from MATDEUL. (Integer > 0, Required)
VALUE	The value of the minimum pressure. (Real ≤ 0.0 , Default = 0.0)
FVTOL	Volume cutoff tolerance. (Real > 0 , 1.E-4)
FVTOL2	Maximum void fraction that is permissible under tension. (Real ≥ 0 ; Default = 0.0)

Remarks:

1. If the pressure in an element falls below the minimum pressure, the element spalls. The pressure and yield stress are set to zero.



2. The default for the volume cutoff tolerance is 1.E-4. This value should be decreased in case of large mass increase of material without any reason.
3. If an element spalls a void is created. In order to prevent getting void fractions that are too small, a void fraction is put to zero if it is smaller than FVTOL. The default for FVTOL is 1.E-4 and works only for Eulerian elements. This value should be decreased in case of large mass increase of material without any reason. Voids can be created during transport of material, because of a material failure and by unloading.

4. With FVTOL2 = 0, any void fraction in an element will lead to failure, and then no tensile stresses are possible. In simulations in which tensile conditions are present, it can be required to allow for tensile stresses in the presence of a small void fraction not exceeding a threshold. This threshold is given by FVTOL2. A good value for FVTOL2 = 2 * FVTOL = e.E-4. FVTOL 2 is only used for Eulerian materials. FVTOL 2 should be larger than FVTOL.

PMREBAI

Defines Rebar Property Information for CMREBAI Elements in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
PMREBAI	IP	TOL	IFILE	X1	X2	X3	MICRO	FACTOR	
	IM1	POS1	AREA1	SSS1	ANG1	IPOST1	IORI1		
	IM2	POS2	AREA2	SSS2	ANG2	IPOST2	IORI2		
	IM3	POS3	AREA3	SSS3	ANG3	IPOST3	IORI3		
	IM4	POS4	AREA4	SSS4	ANG4	IPOST4	IORI4		
	IM5	POS5	AREA5	SSS5	ANG5	IPOST5	IORI5		
	"HEXA"	ID1	THRU	ID2	IHE3	THRU	IHE4		
	"HEXA"	ID5	THRU	ID6	etc.				

Example (two rebar layers through matrix CHEXA elements 100-120):

PMREBAI	20		1	1.0	0.0	0.0	1	0.01	
	100	0.25	0.075	3.0	45.0	2000	1		
	101	0.275	0.050	32.0	4 -30.0	2001	1		
	HEXA	100	THRU	120					

Descriptor Meaning

- IP Rebar property ID. (Integer, Required, no Default)
- TOL Exterior tolerance. A rebar grid is considered within a matrix CHEXA element if the distance between the element and the grid is smaller than the tolerance times the average edge length of the element unless the grid is actually inside another matrix CHEXA element. (Real, Default = 0.05)
- IFILE Option to create a file for rebar verification. (Integer, Default = 0). If IFILE=1, a file named jid.marc_rebar.mfd will be created for use by Mentat.
- X1 First direction cosine of reference axis. (Real, Required, no Default)
- X2 Second direction cosine of reference axis. (Real, Required, no Default)
- X3 Third direction cosine of reference axis. (Real, Required, no Default)
- MICRO Option to activate "micro buckling" behavior. (Integer, Default = 0) If MICRO=0 micro buckling is not activated, if MICRO=1 micro buckling is activated.
- FACTOR Factor to use to reduce rebar stiffness in compression if MICRO=1. (Real, Default = 0.02)

IMi	Material identification number. (Integer, Required, no Default)
POSi	Relative position of rebar layer at edge 1 (pr/t) - the ratio of the distance between the reference surface and the rebar layer to the distance across the element. (Real, Required, no Default)
AREAi	Rebar cross sectional area. (Real, Required, no Default)
SSSi	Number of rebars per unit length in each layer. (Real, Required, no Default)
ANGi	Angle (degrees) between the rebar and the projection of the reference axis on the rebar layer plane (see the following figure). (Real, Default = 0.0, must be between -90.0 and 90.0), see Figure 9-144 .
IPOSTi	Global identification number of the rebar layer used for postprocessing. (Integer, Default = 0)
IORIi	Rebar layer orientation type, see Figure 9-145 . (Integer, Required, no Default) 1 = Thickness direction is from the 1,2,3,4 face to the 5,6,7,8 face 2 = Thickness direction is from the 1,4,8,5 face to the 2,3,7,6 face 3 = Thickness direction is from the 2,1,5,6 face to the 3,4,8,7 face
HEXA	Enter the string HEXA to indicate the line that contains matrix CHEXA element ID's through which the rebar passes. (Character, Required)
ID1, ID3, etc.	Starting matrix CHEXA element ID. (Integer ID1 is Required)
THRU	Enter string THRU to indicate that more than one matrix CHEXA element with these rebar properties apply.
ID2, ID4, etc.	Ending matrix CHEXA element ID. (Integer, ID2 is Required)

Remarks:

1. This entry is the property entry for CMREBAI elements and makes use of Marc's REBAR and INSERT capabilities for rebar membrane element types 147 and 148. Entries CMREBAR and PMREBAR makes use of Marc's REBAR only capability and use rebar elements 23 and 146.
2. If rebar layers i are not desired, the lines 2-5 may be omitted. The line for i=1 is required.
3. See the following two figures for a definition of ANG and IORI.
4. Any material valid for SOL 600 may be used to define IM.
5. For each rebar layer, the user is required to define cord material identification number, cross section area of the cords, density of the cords, and an angle (defining spatial orientation of the cords). α is the angle between the cord and the projection of a predefined reference axis on the rebar layer plane as shown in the next figure.

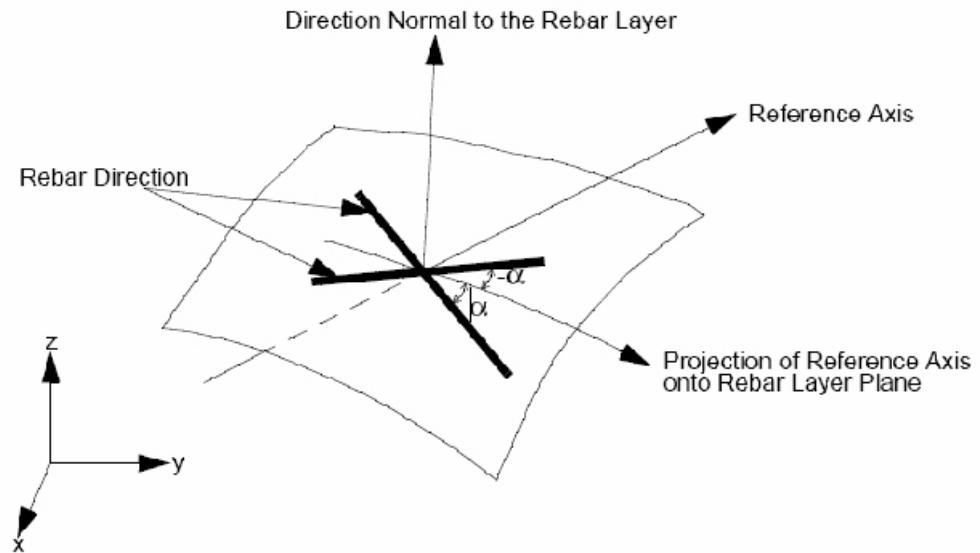


Figure 9-144 Description of rebar orientation on a single rebar layer

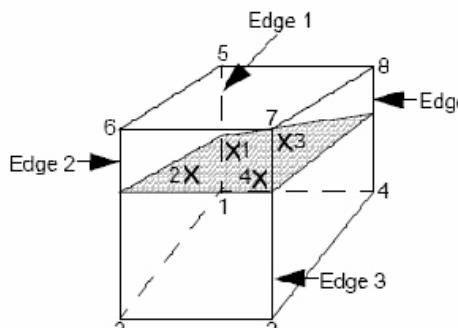
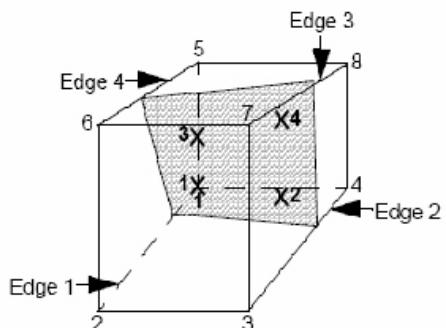
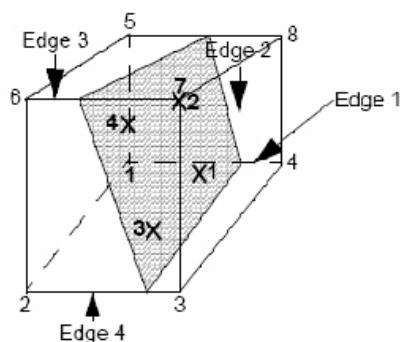
3-D**Rebar Orientation Type 1****Rebar Orientation Type 2****Rebar Orientation Type 3**

Figure 9-145 Rebar numbering and orientation type (IORI value) - only 3D supported.

PMREBAR

Defines Rebar Property Information for CMREBAR Elements in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. Enter lines 2 through 6 below to describe up to 5 “rebar layers”. If only the structure only contains two rebar layers, only enter lines 1-3 as shown in the example. SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
PMREBAR	IP		IFILE	X1	X2	X3	MICRO	FACTOR	
	IM1	POS1	AREA1	SSS1	ANG1	IPOST1	IORI1		
	IM2	POS2	AREA2	SSS2	ANG2	IPOST2	IORI2		
	IM3	POS3	AREA3	SSS3	ANG3	IPOST3	IORI3		
	IM4	POS4	AREA4	SSS4	ANG4	IPOST4	IORI4		
	IM5	POS5	AREA5	SSS5	ANG5	IPOST5	IORI5		

Example (two rebar layers):

PMREBAR	20		1	1.0	0.0	0.0	1	0.01	
	100	0.25	0.075	3.0	45.0	2000	1		
	101	0.275	0.050	32.0	4 -30.0	2001	1		

Descriptor Meaning

- IP Rebar property ID. (Integer; Required; no Default)
- IFILE Option to create a file for rebar verification. (Integer; Default = 0). If IFILE=1, a file named jid.marc_rebar.mfd will be created for use by Mentat.
- X1 First direction cosine of reference axis. (Real; Required; no Default)
- X2 Second direction cosine of reference axis. (Real; Required; no Default)
- X3 Third direction cosine of reference axis. (Real; Required; no Default)
- MICRO Option to activate “micro buckling” behavior. (Integer; Default = 0) If MICRO=0 micro buckling is not activated, if MICRO=1 micro buckling is activated.
- FACTOR Factor to use to reduce rebar stiffness in compression if MICRO=1. (Real; Default = 0.02)
- IMi Material identification number. (Integer; Required; no Default)
- POSi Relative position of rebar layer at edge 1 (pr/t) - the ratio of the distance between the reference surface and the rebar layer to the distance across the element. (Real; Required; no Default)
- AREAi Rebar cross sectional area. (Real; Required; no Default)

SSSi	Number of rebars per unit length in each layer. (Real; Required; no Default)
ANGi	Angle (degrees) between the rebar and the projection of the reference axis on the rebar layer plane (see the following figure). (Real; Default = 0.0, must be between -90.0 and 90.0), see Figure 9-146 .
IPOSTi	Global identification number of the rebar layer used for postprocessing. (Integer; Default = 0)
IORIi	Rebar layer orientation type, see Figure 9-147 . (Integer; Required; no Default) 1 = Thickness direction is from the 1,2,3,4 face to the 5,6,7,8 face 2 = Thickness direction is from the 1,4,8,5 face to the 2,3,7,6 face 3 = Thickness direction is from the 2,1,5,6 face to the 3,4,8,7 face

Remarks:

1. This entry is the property entry for CMREBAR elements and makes use of Marc's REBAR capability for element types 23 and 146. Entries CMREBAI and PMREBAI makes use of Marc's REBAR and INSERT capabilities and use membrane rebar elements 147 and 148.
2. See [Figure 9-146](#) and [Figure 9-147](#) for a definition of ANG and IORI.
3. Any material valid for SOL 600 may be used to define IM.
4. For each rebar layer, the user is required to define cord material identification number, cross section area of the cords, density of the cords, and an angle (defining spatial orientation of the cords). α is the angle between the cord and the projection of a predefined reference axis on the rebar layer plane as shown in the [Figure 9-146](#).

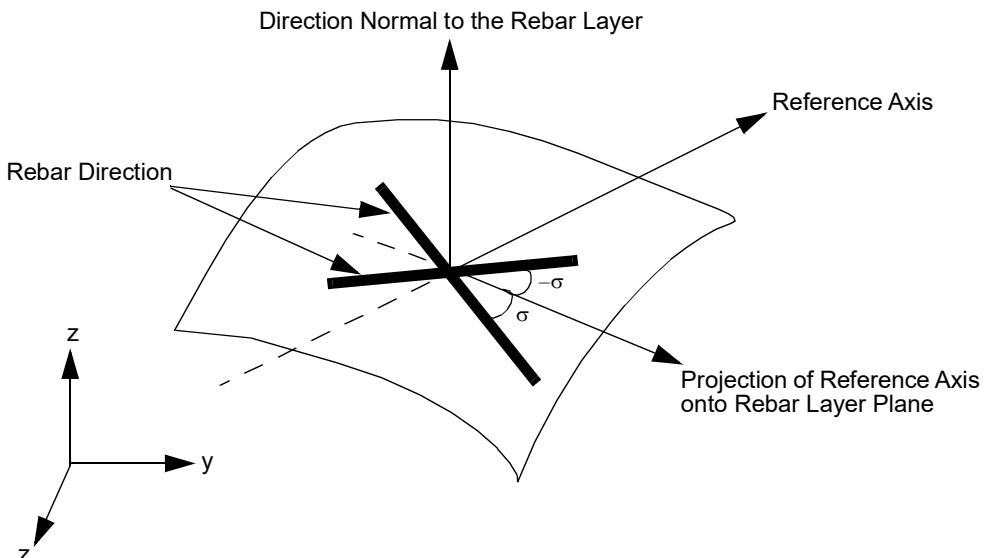


Figure 9-146 Description of rebar orientation on a single rebar layer

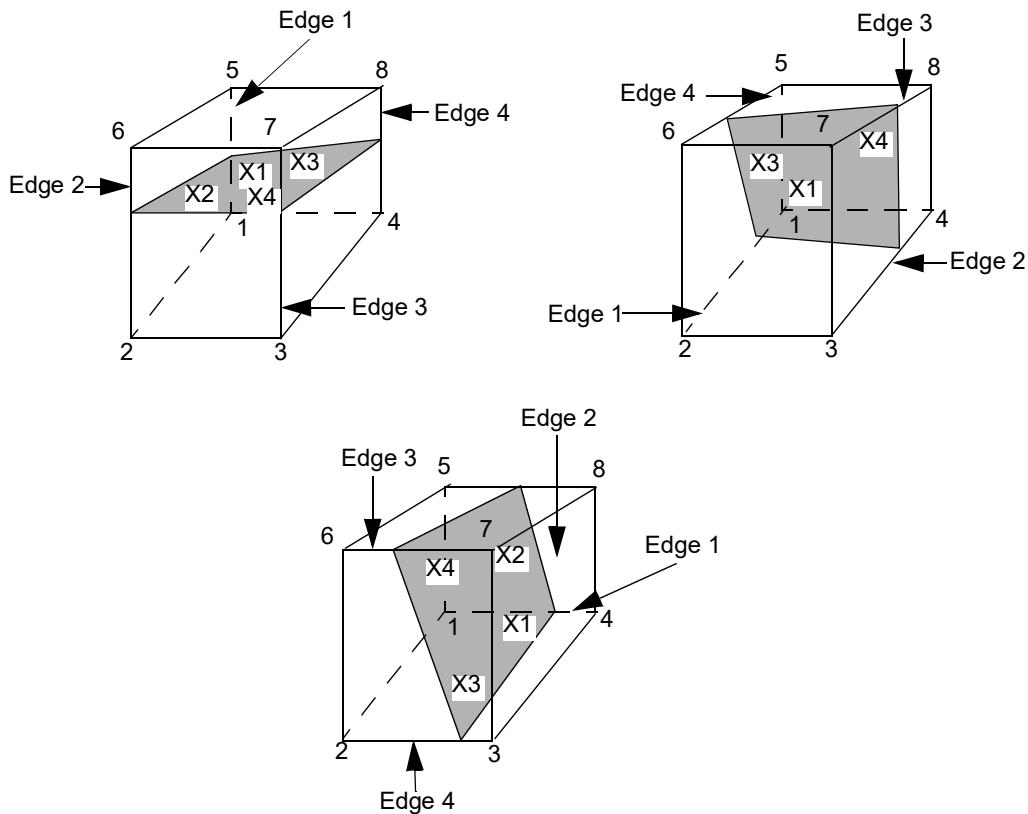


Figure 9-147 Rebar numbering and orientation type (IORI value) - only 3D supported

POINT**Edge Point for FEEDGE Entry**

Define edge point for FEEDGE or SELOC entries.

Format:

1	2	3	4	5	6	7	8	9	10
POINT	ID	CP	X1	X2	X3				

Example:

POINT	12	1	1.	2.	5.				
-------	----	---	----	----	----	--	--	--	--

Descriptor	Meaning	Type	Default
ID	Point identification number.	Integer ≥ 0	Required
CP	Identification number of coordinate system in which the location of point is defined.	Integer ≥ 0	0
X1, X2, X3	Location of the point in coordinate system CP.	Real	0.0

Remarks:

1. POINT is used to specify additional geometric points for edges and can be used by p-version elements. There are no degrees-of-freedom assigned to a point.
2. FEEDGE entries can refer to POINT entries.
3. SELOC entries can refer to POINT entries in the residual or part superelements.
4. ID of POINTs must be unique with respect to ID of GRID entries.
5. POINT entries can be referenced on SET1/SET3 for defining arbitrary beam cross section, ABCS, via PBRSECT/PBMSECT. Note that CP and X3 must be left blank for POINT entries used for ABCS.

POINTAX**Conical Shell Point**

Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested. These points are not subject to constraints via MPCAX, SPCAX, or OMITAX entries.

Format:

1	2	3	4	5	6	7	8	9	10
POINTAX	ID	RID	PHI						

Example:

POINTAX	2	3	30.0						
---------	---	---	------	--	--	--	--	--	--

Descriptor Meaning

- | | |
|-----|--|
| ID | Point identification number. (Unique Integer > 0) |
| RID | Identification number of a RINGAX entry. (Integer > 0) |
| PHI | Azimuthal angle in degrees. (Real) |

Remarks:

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

PORFCPL**Flow Between Two Coupling Surfaces Through a Hole**

Defines an interaction between two coupling surfaces through a hole. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PORFCPL	PID	SIZE		FLOW	CSID	MID			

Example:

PORFCPL	1	SMALL		BOTH	1				
---------	---	-------	--	------	---	--	--	--	--

Descriptor Meaning

PID Unique PORFCPL identification number. (Integer > 0; Required)

SIZE Defines the type of flow method that is used for mass leaving or entering the airbag volume. (Character; Default = SMALL)

SMALL The size of the hole in the surface is the same or smaller than the size of the Euler mesh used. The velocity of the gasflow through the hole is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is the method that is used if the PORHOLE is used on a GBAG entry.

LARGE The size of the hole in the surface is larger than the Euler mesh used. The velocity of the gasflow through the hole is based on the velocity method for an Eulerian air bag. If the PORHOLE is used on a GBAG entry, it will default back to method used for SIZE=SMALL.

FLOW Defines the allowed directions of the flow. (Character; Default = BOTH)

BOTH In- and outflow are allowed.

IN Only inflow allowed into the COUPLE that references this entry.

OUT Only outflow allowed into the COUPLE that references this entry.

CSID The ID of the COUPLE entry. This COUPLE is the one that is connected to the coupling surface that references this entry. (Integer > 0; Required)

MID MATDEUL ID number of the transported gas. Only used when connecting to a GBAG and Euler solver uses the multi-material and SIZE=LARGE. See Remark 2. (Integer > 0)

Remarks:

1. The PORFCPL entry can only be referenced from LEAKAGE entry.

2. For SIZE=LARGE: once gas from a GBAG flows into an Eulerian domain it is treated as Eulerian material. For the single material Euler solver only one Eulerian material is present and the material number MID can be left blank. Since GBAG material is an ideal gas it is required that Eulerian material also uses an EOSGAM (ideal gas) equation of state. When using the multi-material solver the Material number MID has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM.

PORFGBG**Flow Between Two Air Bags Through a Hole**

Defines a hole in a couple and/or GBAG (sub)surface, connected to another GBAG. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PORFGBG	FID	SIZE		FLOW	GBID	MID			

Example:

PORFGBG	1	SMALL		BOTH	1				
---------	---	-------	--	------	---	--	--	--	--

Descriptor	Meaning
FID	Unique number of a PORFGBG entry. It can be referenced from either a LEAKAGE to model the flow between GBAGs, or between Eulerian air bag and a GBAG or between Eulerian air bags. (Integer > 0; Required)
SIZE	Defines the type of flow method that is used for mass leaving or entering the air bag volume. (Character; Default = SMALL)
SMALL	The size of the hole in the surface is the same or smaller than the size of the Euler mesh used. The velocity of the gasflow through the hole is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is the method that is used if the PORHOLE is used on a GBAG entry.
LARGE	The size of the hole in the surface is larger than the Euler mesh used. The velocity of the gasflow through the hole is based on the velocity method for an Eulerian air bag. If the PORHOLE is used on a GBAG entry, it will default back to method used for SIZE=SMALL.
FLOW	Defines the allowed directions of the flow. (Character; Default = BOTH)
BOTH	In- and outflow are allowed.
IN	Only inflow allowed into the GBAG or the coupling surface that references this entry.
OUT	Only outflow allowed into the GBAG or the coupling surface that references this entry.
GBID	Number of a GBAG entry. This GBAG is the one that is connected to the GBAG or coupling surface that references this entry. (Integer > 0; Required)
MID	MATDEUL ID number of the GBAG gas. Only used when connecting a GBAG to an Eulerian air bag that uses the multi-material Euler solver and SIZE=LARGE. See Remark 3. (Integer > 0)

Remarks:

1. The PORFGBG entry can be referenced from a LEAKAGE entry.
2. When used with Euler and SIZE=SMALL, this entry can only be used with the single material hydrodynamic Euler solver, using an EOSGAM (ideal gas) equation of state.
3. For SIZE=LARGE: once gas from a GBAG enters an Eulerian domain it is treated as Eulerian material. For the single material Euler solver only one Eulerian material is present and the material number MID can be left blank. Since GBAG material is an ideal gas it is required that Eulerian material also uses an EOSGAM (ideal gas) equation of state. When using the Multi-material solver the Material number MID has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM.

PORFLOW

Porous Flow Boundary

Defines the material properties for the in- or outflow of an Eulerian mesh through a porous area of the couple surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PORFLOW	FID		TYPE1	VALUE1	TYPE2	VALUE2	TYPE3	VALUE3	
	TYPE4	VALUE4	-etc.-						

Example:

PORFLOW	120		XVEL	100.0	1				

Descriptor Meaning

FID Unique number of a PORFLOW entry. (Integer > 0; Required)

TYPE The properties on the flow boundary. (Character, Required)

MATERIAL MATDEUL ID number.

XVEL Velocity in the x-direction.

YVEL Velocity in the y-direction.

ZVEL Velocity in the z-direction.

PRESSURE Pressure.

DENSITY Density.

SIE Specific internal energy.

FLOW The type of flow boundary required.

METHOD The method used for the material transport.

VALUEi The value of the property specified in the TYPE field. (Real or Character, Required)

For TYPEi set to FLOW, the value is a character entry: either IN, OUT or BOTH, indicating that the flow boundary is defined as an inflow, outflow, or possibly an in- or outflow boundary. The default is BOTH. See Remark 4.

For TYPEi set to METHOD, the value is a character entry: either VELOCITY or PRESSURE, indicating that the material transport is based on the velocity method or the pressure method. The default is VELOCITY. See Remark 4.

Remarks:

1. Reference FID by a LEAKAGE entry.
2. Any material properties not specifically defined have the same value as the element that the (SUB)SURFACE segment is intersecting.

3. The SURFACE can be only a general coupling surface (see the COUPLE entry).
4. The different methods used to calculate the material transport through a porous (sub)surface are described in General Coupling.
5. METHOD=VELOCITY is valid for all equation of state models.
METHOD=PRESSURE is valid for EOSGAM (ideal gas) in combination with the single material hydrodynamic Euler solver.
6. Alternative methods are available to define holes and permeable sections in an air bag. See the entries: LEAKAGE, PORHOLE, PERMEAB, PORFGBG and PERMGBG.
7. In the case of material flow into a multi-material Euler mesh, the material number, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions
8. Prescribing both pressure and velocity may lead to the instabilities.

PORFLWT

Time Dependent Porous Flow Boundary

Defines a time dependent flow through a porous area of the couple surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PORFLWT	FID	TYPE							
	VELTYPE	VELOCITY	PRESTYP	PRES					
	MID	DENSTYP	DENSITY	SIETYP	SIE				

Example:

PORFLWT	2	IN							
	TABLE	101	TABLE	102					
	91	TABLE	105	TABLE	107				

Descriptor Meaning

FID Unique number of a PORFLWT entry. (Integer > 0; Required)

TYPE Type of flow boundary. (Character; Default = BOTH)

IN Inflow boundary (see Remarks 2, and 3.).

Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.

OUT Only outflow is allowed. The outflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use the material mixture as present in the adjacent Euler element.

BOTH Material is allowed to flow in or out. The switch between inflow and outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be left unspecified. If not given the pressure in the adjacent Euler element will be taken.

VELTYPE Type of velocity definition. (Character; Default = ELEMENT)

ELEMENT Value of Euler element.

CONSTANT Value is constant in time.

TABLE Value varies in time.

VELOCITY Value of inflow or outflow velocity. If VELTYPE = TABLE it refers to a TABLED1. The velocity direction is normal to the coupling surface or subsurface. A positive velocity corresponds with inflow. See Remark 6. (Integer or Real)

PRESTYP Type of pressure definition. See Remark 6. (Character)

	ELEMENT	Value of Euler element.
	CONSTANT	Value is constant in time.
	TABLE	Value varies in time.
PRES		Value of inflow or outflow pressure. If PRESTYPE = TABLE it refers to a TABLED1. (Integer or Real)
MID		MATDEUL ID of inflowing material. Input is not allowed for TYPE = OUT. When MID is specified, it is required to also define density and SIE for the inflowing material. (Integer; Default = 0)
DENSTYP		Type of density definition: (Character, Required when MID > 0.)
	ELEMENT	Value of Euler element
	CONSTANT	Value is constant in time
	TABLE	Value varies in time
DENSITY		Value of density. If DENSTYP = TABLE, it refers to a TABLED1 ID. (Integer or Real, Required when MID is given.)
SIETYPE		Type of density definition. Required when MID is given. (Integer or Real)
	ELEMENT	Value of Euler element.
	CONSTANT	Value is constant in time.
	TABLE	Value varies in time.
SIE		Value of specific internal energy. If SIETYPE = TABLE it refers to a TABLED1. Required when MID is given. (Integer or Real)

Remarks:

1. Reference FID by a LEAKAGE entry.
2. Any material properties not specifically defined have the same value as the element that the segment of the coupling surface is intersecting.
3. The surface can be only a general coupling surface (see the COUPLE entry).
4. Alternative methods are available to define holes and permeable sections in an air bag. See the entries: LEAKAGE, PORHOLE, PERMEAB, PORFGBG and PERMGBG.
5. In the case of material flow into a multi-material Euler mesh, the material number, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions

6. The boundary condition initiates/determines a wave in compressible material like gas and water. This can be either an outgoing or an ingoing wave. For stability it is important that the waves created are compatible with the flow type near the boundary. Relevant flow types are subsonic inflow, subsonic outflow, supersonic inflow and supersonic outflow. For example for subsonic inflow prescribing both pressure and velocity would initiate outgoing waves. Outgoing waves for an inflow boundary condition is known to be unstable. However, for supersonic inflow one can specify both pressure and velocity since there are no outgoing waves at a supersonic inflow boundary.

PORHOLE

Holes in Air Bag Surface

Defines a hole in a COUPLE and/or GBAG surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PORHOLE	FID	SIZE		FLOW	PENV	RHOENV	SIEENV	CP	
	MID								

Example:

PORHOLE	301								

Descriptor Meaning

PID	Unique number of a PORHOLE entry. (Integer > 0; Required)								
SIZE	Defines the type of flow method that is used for mass leaving or entering the airbag volume. (Character; Default = SMALL)								
SMALL	The size of the hole in the surface is the same or smaller than the size of the Euler mesh used. The velocity of the gasflow through the hole is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is the method that is used if the PORHOLE is used on a GBAG entry.								
LARGE	The size of the hole in the surface is larger than the Euler mesh used. The velocity of the gasflow through the hole is based on the velocity method for an Eulerian air bag. If the PORHOLE is used on a GBAG entry, it will default back to method used for SIZE=SMALL.								
FLOW	Defines the allowed directions of the flow. (Character; Default = BOTH)								
BOTH	In- and outflow are allowed.								
IN	Only inflow allowed.								
OUT	Only outflow allowed.								
PENV	Environmental pressure. (Real > 0.0; Required)								
RHOENV	Environmental density. (Real > 0.0; Required)								
SIEENV	Environmental specific internal energy. (Real > 0.0; Required)								
CP	Environmental specific heat at constant pressure. See Remark 4. (Real > 0.0)								
MID	MATDEUL ID number of the environment gas. Only used for an Eulerian air bag that uses the multi-material Euler solver and SIZE=LARGE. See Remark 6. (Integer > 0)								

Remarks:

1. The PORHOLE entry can be referenced from a LEAKAGE entry.
2. When used with Euler, this entry can only be used with the single material hydrodynamic Euler solver, using an EOSGAM (ideal gas) equation of state.
3. The values for the environment p_{env} (PENV), ρ_{env} (RHOENV), e_{env} (SIEENV) must be defined consistent with an ideal-gas equation of state:

$$p_{env} = (\gamma_{env} - 1)\rho_{env}e_{env}$$

The γ_{env} is calculated and is only used when inflow occurs. Inflow occurs when $p_{env} > p_{inside}$.

4. CP is only required if updating of Euler or gasbag gas constants is done, for example if hybrid inflators are defined.
5. For in and out flow of an uniform pressure air bag (GBAG), the material transport is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is equivalent to the PORHOLE entry.
6. When used in combination with the single material hydrodynamic Euler solver, an EOSGAM (ideal gas) equation of state is required. In that case, the material number, MID, can be left blank. When using the multi-material solver, the material number, MID, has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM.

PORHYDS

Porous Flow Boundary with a Hydrostatic Pressure Profile

Prescribes a hydrostatic pressure profile on a porous BSURF. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PORHYDS	PID								

Example:

PORHYDS	120								
---------	-----	--	--	--	--	--	--	--	--

Describer Meaning

PID Unique number of a PORHYDS entry. (Integer > 0; Required)

Remarks:

1. Reference PID by a LEAKAGE entry.
2. The surface can be only a general coupling surface (see the COUPLE entry).
3. It is required that the coupling surface refers to a HYDSTAT entry. This HYDSTAT entry will be used to prescribe a hydrostatic pressure profile on the subsurface. For example, the water level and atmospheric pressure are taken from the HYDSTYAT entry. This defines the pressure and the inflow density.
4. In contributions of the surface to the Euler elements the pressure gradient across the surface is taken into account. Therefore splitting up of the surface and creating new PORHYDS entries does not increase the accuracy of prescribed pressures. If the water level and atmospheric pressure are the same in the whole region outside the coupling surface using one PORHYDS entry is sufficient.
5. The atmospheric pressure is prescribed on those parts of the surface that are above the water level.

PORUDS

User-defined Porosity Model Specified by a User Defined Service.

Defines a porosity model of a COUPLE surface through a user-written subroutine. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PORUDS	PID	GROUP	UNAME						

Example:

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE mypor 'SCA.MDSolver.Obj.Uds.Dytran.Flow'
```

In Bulk Data:

1	2	3	4	5	6	7	8	9	10
PORUDS	12	mypor	EXPOR						

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

Remarks:

1. Only can be used for SOL 700.
2. The porosity ID (PID) must be referenced by a LEAKAGE entry.
3. UNAME can be:

Subroutine Name	Function
EXPOR	Standard user defined flow boundary on the coupling

PRAC2D**CRAC2D Element Property**

Defines the properties and stress evaluation techniques to be used with the CRAC2D structural element.

Format:

1	2	3	4	5	6	7	8	9	10
PRAC2D	PID	MID	T	IPLANE	NSM	GAMMA	PHI		

Example:

PRAC2D	108	2	0.10	0	.17	.50	180.		
--------	-----	---	------	---	-----	-----	------	--	--

Describer Meaning

PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
T	Element thickness. (Real > 0.0)
IPLANE	Plane strain or plane stress option. Use 0 for plane strain; 1 for plane stress. (Integer = 0 or 1, Default = 0)
NSM	Non-structural mass per unit area. (Real \geq 0.0; Default = 0)
GAMMA	Exponent used in the displacement field. See Remark 4. (Real; Default = 0.5)
PHI	Angle (in degrees) relative to the element x-axis along which stress intensity factors are to be calculated. See Remark 4. (Real; Default = 180.0)

Remarks:

1. PRAC2D is a primary property entry. Primary property entries are grouping entities for many applications in Nastran. Therefore it is highly recommended that the PRAC2D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PRAC2D entries.
2. PRAC2D entry may refer to MAT1, MAT2, or MAT8 material property entries.
3. For plane strain analysis, only MAT1 type data should be used.
4. Nondefault values for GAMMA and PHI have not been tested. Therefore, the default value should be used.

PRAC3D**CRAC3D Element Property**

Defines the properties of the CRAC3D structural element.

Format:

1	2	3	4	5	6	7	8	9	10
PRAC3D	PID	MID	GAMMA	PHI					

Example:

PRAC3D	108	2	.50	180.					
--------	-----	---	-----	------	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GAMMA	Exponent used in the displacement field. See Remark 3. (Real; Default = 0.5)
PHI	Angle (in degrees) relative to the element x axis along which stress intensity factors are to be calculated. See Remark 3. (Real; Default = 180.0)

Remarks:

1. PRAC3D is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PRAC3D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PRAC3D entries.
2. Either isotropic (MAT1) or anisotropic (MAT9) material entries may be referenced.
3. Nondefault values for GAMMA and PHI have not been tested. Therefore, the default value should be used.

PRESAX

Conical Shell Pressure Load

Defines the static pressure loading on a conical shell element.

Format:

1	2	3	4	5	6	7	8	9	10
PRESAX	SID	P	RID1	RID2	PHI1	PHI2			

Example:

PRESAX	3	7.92	4	3	20.6	31.4			
--------	---	------	---	---	------	------	--	--	--

Describer**Meaning**

SID Load set identification number. (Integer > 0)

P Pressure value. (Real)

RID1, RID2 Ring identification numbers. See RINGAX entry. (Integer > 0)

PHI1, PHI2 Azimuthal angles in degrees. (Real; PHI2 > PHI1)

Remarks:

1. PRESAX is allowed only if an AXIC entry is also present.
2. Load sets must be selected with the Case Control command LOAD = SID.
3. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.
4. For axisymmetric loading over 360 degrees, use PHI1 = 0.0 and PHI2 = 360.0.

PRESPT

Fluid Pressure Point

Defines the location of pressure points in the fluid for recovery of pressure data.

Format:

1	2	3	4	5	6	7	8	9	10
PRESPT	IDF	IDP1	PHI1	IDP2	PHI2	IDP3	PHI3		

Example:

PRESPT	14	141	0.0			142	90.0		
--------	----	-----	-----	--	--	-----	------	--	--

Descriptor	Meaning
IDF	Fluid point (RINGFL entry) identification number. (Integer > 0)
IDPi	Pressure point identification number. (Integer > 0)
PHIi	Azimuthal position on fluid point referenced by IDF in fluid coordinate system. (Real)

Remarks:

1. PRESPT is allowed only if an AXIF entry is also present.
2. All pressure point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. The pressure points are used primarily for the identification of output data. They may also be used as points at which to measure pressure for input to control devices.
4. One, two, or three pressure points may be defined per entry.
5. Output requests for velocity and acceleration of these degrees-of-freedom will result in derivatives of pressure with respect to time.

PRIMx

Thermal Geometric Primitives for RC Radiation

Specifies the properties of geometric primitives to be used in radiation calculations in place of elements.

Format (GEOM2):

1	2	3	4	5	6	7	8	9	10
PRIM1	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	A_mesh	B_mesh				

Example:

PRIM1	11	101	102	3	4	2			
	0.	0.	0.	1.	0.	0.			
	0.	1.	0.	3	4				

Format:

1	2	3	4	5	6	7	8	9	10
PRIM2	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	P4(1)	P4(2)	P4(3)	A_mesh	B_mesh	

Example:

PRIM2	12	102	103	3	4	2			
	0.	0.	0.	1.	0.	0.			
	0.	1.	0.	1.	1.	0.	3	4	

Format:

1	2	3	4	5	6	7	8	9	10
PRIM3	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	A_mesh	B_mesh				

Example:

PRIM3	13	103	104	3	4	2			
	0.	0.	0.	1.	0.	0.			
	0.	1.	0.	3	4				

Format:

1	2	3	4	5	6	7	8	9	10
PRIM4	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	Diam1	Diam2	Angle1	Angle2		
	A_mesh	B_mesh							

Example:

PRIM4	14	104	105	3	4	2			
	0.	0.	0.	0.0	1.0	0.			
	0.	0.	1.	1.	0.	60.	180.		
	3	4							

Format:

1	2	3	4	5	6	7	8	9	10
PRIM5	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	Diam1	Angle1	Angle2	A_mesh	B_mesh	

Example:

PRIM5	15	105	106	3	4	2			
	0.	0.	0.	0.	1.	0.			
	0.	0.	1.	1.	60.	180.	3	4	

Format:

1	2	3	4	5	6	7	8	9	10
PRIM6	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	Diam1	Diam2	Angle1	Angle2		
	A_mesh	B_mesh							

Example:

PRIM6	16	106	107	3	4	2			
	0.	0.	0.	0.	1.	0.			
	0.	0.	1.	1.	0.	60.	80.		
	3	4							

Format:

1	2	3	4	5	6	7	8	9	10
PRIM7	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	Diam1	Angle1	Angle2	Trunc1	Trunc2	
	A_mesh	B_mesh							

Example:

PRIM7	17	107	108	3	4	2			
	0.	0.	0.	0.	1.	0.			
	0.	0.	1.	1.	60.	180.	-0.5	0.5	
	3	4							

Format:

1	2	3	4	5	6	7	8	9	10
PRIM8	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	Diam1	Angle1	Angle2	Trunc1	Trunc2	
	A_mesh	B_mesh							

Example:

PRIM8	18	108	109	3	4	2			
	0.	0.	0.	0.	1.	0.			
	0.	0.	1.	1.	60.	180.	0.5	1.	
	3	4							

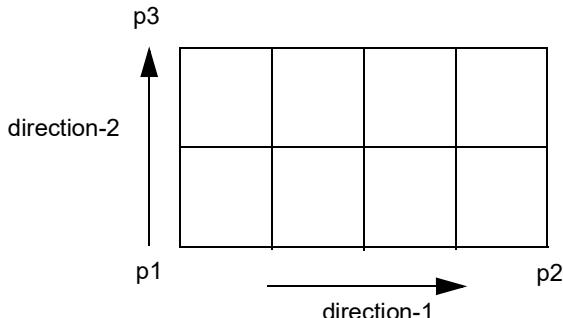
Descriptor	Meaning
PRIMID	Primitive identification number, unique to all other PRIMx (Integer > 0; Required)
SET3 ID	ID of the element collection to which this primitive describes. This collection acts as the elements that will exist in the thermal model, but the collection will be absent from the radiation model. Instead, the primitive will be used to calculate radiation, and be redistributed back onto the elements. All radiation properties for the primitive will be applied to the element collection, and must be consistent across. (Integer > 0; Required)
RADMID	ID of the radiation material properties used to describe this primitive for analysis.
Pi(a)	The position of point i in the a axis, as described in the correlating picture. For example P2(2) denotes the y coordinate of the second point. Position is always described in global coordinates. (Real; Required)

Descriptor	Meaning
Diamx	Diameter x of the primitive, if applicable and as described in the correlating picture. (Real ≥ 0.0 ; Required)
Anglex	Angle x of the primitive in degrees, if applicable and as described in the correlating picture. ($0 \leq \text{Real} \leq 360.0$; Required)
Truncx	Truncation x of the primitive, if applicable and as described in the correlating picture. (For PRIM8, Real ≥ 0 ; Required). (For PRIM7, $-0.5 * \text{Diam1} \leq \text{Real} \leq 0.5 * \text{Diam1}$)
A_mesh	Number of mesh spaces in parametric direction-1, as described in the correlating picture. (Integer > 0 ; Required)
B_mesh	Number of mesh spaces in parametric direction-2, as described in the correlating picture. (Integer > 0 ; Required)

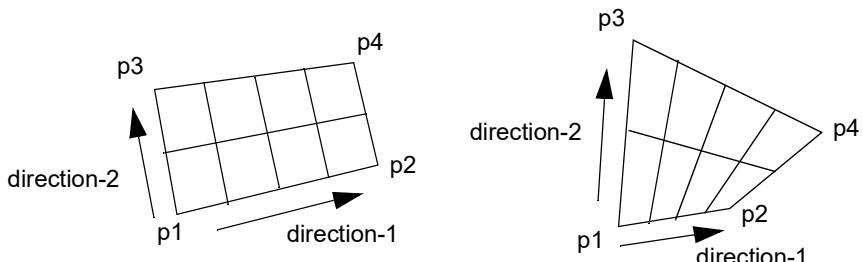
Remarks:

1. This entry is for RC Network solver only.
2. Set 3 should include all the elements which belong to this primitive. It will cause wrong results if only partial of the elements are included.
3. About the primitives

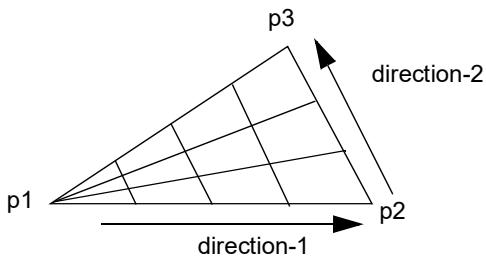
Prim1: Rectangle



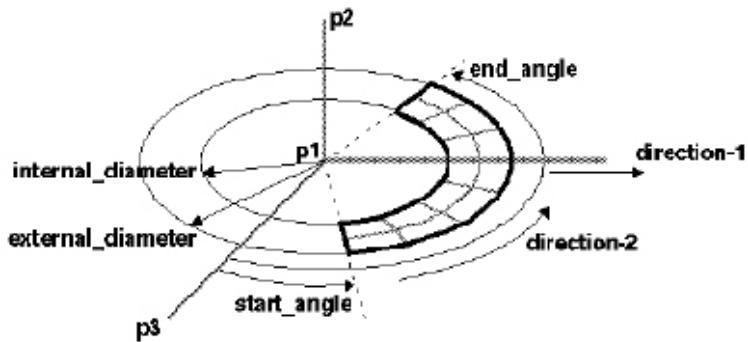
Prim2: Quad



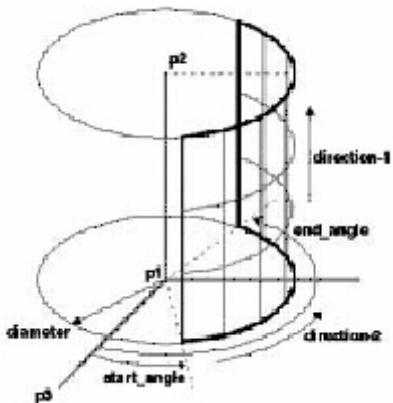
Prim3: Triangle



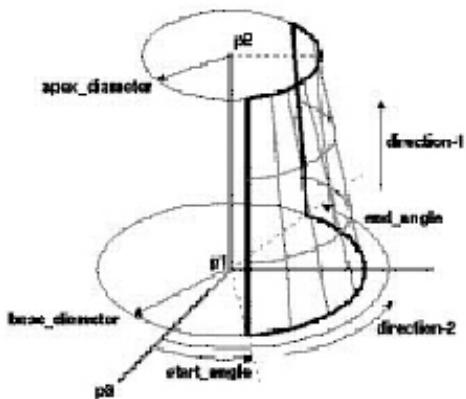
Prim4: Disc



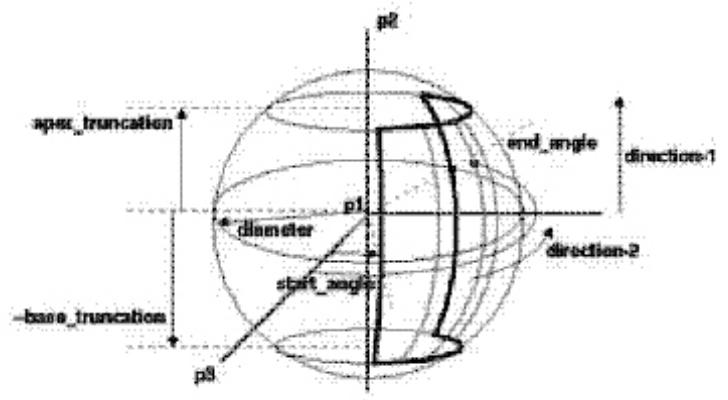
Prim5: Cylinder



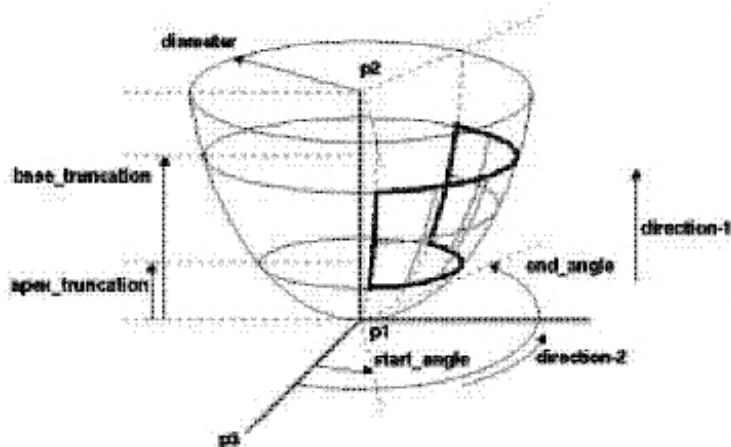
Prim6: Cone



Prim7: Sphere



Prim8: Parabolic



PRJCON**Thermal RC Element Contact**

Specifies a thermal connection between two regions of elements. The connection is automatically determined geometrically as a projection of the slave region on to the master, and the strength of the connection is calculated based on the properties given.

Format: (HEAT1)

1	2	3	4	5	6	7	8	9	10
PRJCON	BID								
	"HEAT1"	SET3 MASTER	SET3 SLAVE	h					

Example:

PRJCON	1								
	HEAT1	1	2	1.2					

Format: (HEAT2)

1	2	3	4	5	6	7	8	9	10
PRJCON	BID								
	"HEAT2"	SET3 MASTER	SET3 SLAVE	ID					

Example:

PRJCON	1								
	HEAT2	1	2	1001					

Format: (HEAT3)

1	2	3	4	5	6	7	8	9	10
PRJCON	BID								
	"HEAT3"	SET3 MASTER	SET3 SLAVE	F	Emis Master	Emis Slave			

Example:

PRJCON	1								
	HEAT3	1	2	1.	0.85	0.5			

Format: (HEAT4)

1	2	3	4	5	6	7	8	9	10
PRJCON	BID								
	"HEAT4"	SET3 MASTER	SET3 SLAVE	F	RADC ID Master	RADC ID Slave			

Example:

PRJCON	1								
	HEAT4	1	2	1.	1001	1002			

Descriptor	Meaning
BID	BCBODY identification number (Integer > 0; Required)
HEATx	Indicates the start of HEAT elements (Character)
SET3 Master	ID of the master element collection for connection (Integer > 0; Required)
SET3 Slave	ID of the slave element collection for connection (Integer > 0; Required)
h	Convection correlation (Real ≥ 0.0 ; Required for HEAT1)
PCONID	PCONID of the property to be used for h value. (Integer > 0; Required for HEAT2)
F	View factor between parts ($0.0 \leq \text{Real} \leq 1.0$; Required for HEAT3 & HEAT4)
Emis Master	Emissivity of master collection (Real ≥ 0.0 ; Required for HEAT3)
Emis Slave	Emissivity of slave collection (Real ≥ 0.0 ; Required for HEAT3)
RADC id Master	RADMID of the material to be used for master emissivity value. (Integer > 0; Required)
RADC id Slave	RADMID of the material to be used for slave emissivity value. (Integer > 0; Required)

Remarks:

1. This “HEATx” and the later parameters are for RC Network solver only.
2. For HEAT2, PCONID must refer to a PCONV1 type and not a PCONV or PCONVM.
3. RC Network Solver uses a projection method to determine the connection (not the “nearest neighbor method”). In most of the cases, the projection method is more accurate than the nearest neighbor method.

PROD**Rod Property**

Defines the properties of a rod element (CROD entry).

Format:

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J	C	NSM			

Example:

PROD	17	23	42.6	17.92	4.2356	0.5			
------	----	----	------	-------	--------	-----	--	--	--

Describer	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 2. and 3. (Integer > 0)
A	Area of the rod. (Real)
J	Torsional constant. (Real)
C	Coefficient to determine torsional stress. (Real; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real)

Remarks:

1. PROD is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PROD property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PROD entries.
2. For structural problems, MID must reference a MAT1 material entry.
3. For heat transfer problems, MID must reference a reference MAT4 or MAT5 entry.
4. The formula used to calculate torsional stress is

$$\tau = \frac{CM_0}{J}$$

where M_0 is the torsional moment.

PRODN1

Nonlinear Property Extensions for a PROD Entry

Specifies additional nonlinear properties for elements that point to a PROD entry in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
PRODN1	PID	MID			ANAL				
	“C2”	BEH2	INT2	BEH2H	INT2H				

Example:

PRODN1	22	98							
--------	----	----	--	--	--	--	--	--	--

Descriptor Meaning

PID	Property identification number of an existing PROD entry. (Integer > 0)
MID	Material identification number. Remarks 7.(Integer ≥ 0)
ANAL	Analysis type. ANAL='IS' - Implicit structural elements are being referred to. ANAL='IH' - Implicit heat analysis elements are being referred to. ANAL='ISH' - Implicit structural and heat elements are being referred to. (Character Default ISH).
C2	Keyword indicating that items following apply to elements with two end grids. (Character)
BEH2	Element structural behavior. See Remark 4. (Character Default ROD)
INT2	Integration scheme. See Remarks 4. and 5. (Character Default L)
BEH2H	Element heat behavior. See Remark 4. (Character Default ROD)
INT2H	Integration scheme. See Remarks 4. and 5. (Character Default L)

Remarks:

1. The PID above must point to an existing PROD Bulk Data entry and is honored only in SOL 400.
2. MID if blank (or 0) use the MID value on the PROD entry. If > 0 it will override the MID value on the PROD.
3. The MID entry may point to the MAT1 entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Caution: The MATVE, MATVP, MATEP, and MATF entries are only associated with a CROD element if the CROD element refers to a PRODN1 entry.

The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

Implicit Structural Materials	
MAT1	
MATVE	
MATVP	
MATEP	
MATF	
MATS1	

Heat Materials	
MAT4	
	MAT5

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness of the MAT4 and MAT5 entries.

4. BEH2/BEH2H refers to the nonlinear structural/heat behavior of the ROD element. An underlined item delineates default.

Structural/Heat Classification of Elements				
Element Structural/Heat Type	BEH2/BEH2H CODE	Integration Code	Element Type	# Nodes
Rod	ROD	L	ROD	2

5. Integration codes in Remark 4. are:

INT CODE	Integration Type
L	Linear

6. Any J, C, or NSM value on the PROD will be ignored.
7. The structural element damping coefficient, GE, is not supported on elements which reference PRODN1.

PSEAM**CSEAM Property**

Defines the PSEAM property values.

Format:

1	2	3	4	5	6	7	8	9	10
PSEAM	PID	MID	TYPE	W	T	IN			

Example:

PSEAM	7	1		16.					
-------	---	---	--	-----	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
TYPE	“KEYWORD” type of Seam Weld generated. (Character; Default = LINE)
W	Width of the SEAM. See Remark 1. (Real > 0.)
T	Thickness of the SEAM. See Remark 2. (Real > 0. or blank)
IN	Integration scheme. See Remark 6. of PSOLID entry. (Integer 0 or 2; Default = 2) If IN = 0, 2x2x2 reduced shear integration with bubble functions. If IN = 2, 2x2x2 reduced shear integration.

Remarks:

1. The length of the SEAM is the distance between GS and GE. The width W of the SEAM is measured perpendicular to the length and lies in the plane of the patches A and B (see [Figure 9-148](#)). The width is also used to find the projection of the SEAM on the two patches A and B.

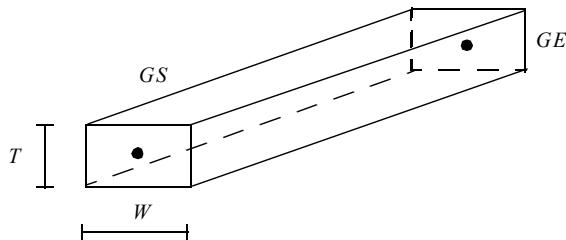


Figure 9-148 Dimensions of a CSEAM Element

2. If left blank, the thickness will be computed as $T = (T_A + T_B)/2$ where T_A is the thickness of patch A and T_B is the thickness of patch B.

3. PSEAM is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PSEAM property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSEAM entries.

PSET**p-Version Element Polynomial Distribution**

Describes polynomial order distribution and is selected by the ADAPT Case Control command.

Format:

1	2	3	4	5	6	7	8	9	10
PSET	SID	POLY1	POLY2	POLY3	CID	SETTYP	ID		

Example:

PSET	127	1	2	1			12		
------	-----	---	---	---	--	--	----	--	--

Descriptor	Meaning	Type	Default
SID	ID selected in the ADAPT Case Control command.	Integer > 0	Required
CID	Coordinate system used to specify polynomial values in different directions. See Remark 1.	Integer ≥ 0	Remark 2.
POLYi	Polynomial order in 1, 2, 3 directions of the CID system.	Integer > 0	Remark 3.
SETTYP	Type of set provided ("SET" or "ELID")	Character	"SET"
ID	SET ID or element ID with this p-value specification.	Integer > 0	999999

Remarks:

1. CID facilitates the specification of the p-order in curvilinear systems. For example, when modeling a thin cylinder, the user can restrict the p-order through the thickness of all elements to be 2 or 3 without specifically checking the connectivity of each element.
2. If the CID system is blank, the element's topology is used to establish the 1, 2, 3 directions. The 1 direction is from the first to the second grid of the element, the 2 direction is from the first to the fourth, and, the 3 direction is from the first to the fifth. If CID is not blank then the following algorithm will be used to determine the p-order of each edge: a vector will be defined in the CID system from the first to the second grid of every edge. (Curvilinear systems are evaluated at the midpoint of this vector.) The p-level of each edge is now determined by the nearest integer to

$$p = \sqrt{(n_1 \cdot \text{POLY1})^2 + (n_2 \cdot \text{POLY2})^2 + (n_3 \cdot \text{POLY3})^2}$$

where (n_1, n_2, n_3) are the components of this unit vector in the CID system.

3. The default value for POLY2 and POLY3 is POLY1.
4. Any overlap of the PSET specification will result in a warning message and the use of the PSET with the highest pi entry.

5. Whenever SETTYP = "SET", a SET command must be defined in the SETS DEFINITION section of the Case Control Section.
6. SET = 999999 is a reserved set that includes all elements.
7. Whenever there are more than one PSET entries for a given element, then:
 - If CID on the PSET entries are the same, the entry with the maximum POLYi will be used.
 - If CID on the PSET entries are different, a fatal message is issued.

PSHEAR**Shear Panel Property**

Defines the properties of a shear panel (CSHEAR entry).

Format:

1	2	3	4	5	6	7	8	9	10
PSHEAR	PID	MID	T	NSM	F1	F2			

Example:

PSHEAR	17	23	42.6	17.92	4.236	0.5			
--------	----	----	------	-------	-------	-----	--	--	--

Descriptor Meaning

- | | |
|-----|--|
| PID | Property identification number. (Integer > 0) |
| MID | Material identification number of a MAT1 entry. (Integer > 0) |
| T | Thickness of shear panel. (Real ≠ 0.0) |
| NSM | Nonstructural mass per unit area. (Real) |
| F1 | Effectiveness factor for extensional stiffness along edges 1-2 and 3-4. See Remark 2.
(Real ≥ 0.0; Default = 0.0) |
| F2 | Effectiveness factor for extensional stiffness along edges 2-3 and 1-4. See Remark 2.
(Real ≥ 0.0; Default = 0.0) |

Remarks:

1. PSHEAR is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PSHEAR property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSHEAR entries.
2. The effective extensional area is defined by means of equivalent rods on the perimeter of the element. If $F1 \leq 1.01$, the areas of the rods on edges 1-2 and 3-4 are set equal to $(F1 \cdot T \cdot PA)/(L12 + L34)$ where PA is the panel surface area-half the vector cross product area of the diagonals-and L12, L34 are the lengths of sides 1-2 and 3-4. Thus, if $F1 = 1.0$, the panel is fully effective for extension in the 1-2 direction. If $F1 > 1.01$, the areas of the rods on edges 1-2 and 3-4 are each set equal to $0.5 \cdot F1 \cdot T^2$.

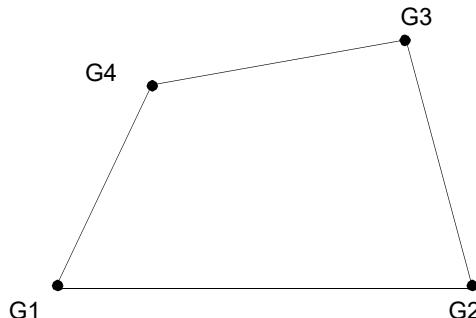


Figure 9-149 Extensional Area for Shear Panel

Thus, if $F_1 = 30$, the effective width of skin contributed by the panel to the flanges on edges 1-2 and 3-4 is equal to $15T$. The significance of F_2 for edges 2-3 and 1-4 is similar.

3. Poisson's ratio coupling for extensional effects is ignored.
4. The parameter entry MDLPRM,SHEARP,GARVEY(default) selects the standard Garvey shear panel. MDLPRM,SHEARP,HARDER selects the Harder shear panel: See Remark 4. of the CSHEAR entry.

PSHEARN

Nonlinear Property Extensions for a PSHEAR Entry

Specifies nonlinear properties for elements that point to a PSHEAR entry in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
PSHEARN	PID	MID			ANAL				
	"C4"	BEH4	INT4						

Example:

PSHEARN	22	18							
---------	----	----	--	--	--	--	--	--	--

Descriptor Meaning

PID	Property identification number of an existing PSHEAR entry. (Integer > 0)
MID	Material identification number. Remark 9. (Integer ≥ 0 or blank)
ANAL	Analysis type. ANAL = 'IS' - Implicit structural elements are being referred to. (Character: Default IS)
C4	Keyword indicating that two items following apply to elements with four corner grids. (Character)
BEH4	Element structural behavior. See Remark 5. (Character Default MB)
INT4	Integration scheme. See Remarks 4. and 6. (Character Default L)

Remarks:

1. The PID must point to an existing PSHEAR Bulk Data entry and is honored only in SOL 400.
2. Only large membrane rotation is supported. Stringer effectiveness is ignored and only membrane action is considered.
3. MID if blank (or 0) use the MID value on the PSHEAR entry. If > 0 is will override the MID value on the PSHEAR.
4. The MID entry for nonlinear structures may point to the MAT entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Caution: The MATVE, MATVP, MATEP, MATF and MATSMA entries are only associated with a CSHEAR element if the CSHEAR element refers to a PSHEARN entry.

Implicit Structure Materials
MAT1
MATVE
MATVP
MATEP
MATF
MATS1
MATSMA

5. BEH4 refers to the nonlinear structural behavior of the SHEAR element. An underlined item delineates default.

Structural Classification of Elements				
Element Structural Type	BEH2/BEH2H CODE	Integration Code	Element Type	# Nodes
SHEAR	MB	<u>L</u>	SHEAR	4

6. Integration codes in Remark 4. are:

INT CODE	Integration Type
L	Linear

7. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation.
8. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available.
9. The structural element damping coefficient, GE, is not supported on elements which reference PSHEARN.

PSHELL**Shell Element Property**

Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	$12I/T^{**3}$	MID3	TS/T	NSM	
	Z1	Z2	MID4						

Example:

PSHELL	203	204	1.90	205	1.2	206	0.8	6.32	
	.95	-.95							

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID1	Material identification number for the membrane. (Integer ≥ 0 or blank)
T	Default membrane thickness for T_i on the connection entry. If T is blank then the thickness must be specified for T_i on the CQUAD4, CTRIA3, CQUAD8, and CTRIA6 entries. (Real or blank) Average thickness if TFLAG = 1 ($T_i = T_i \cdot T$) .
MID2	Material identification number for bending. (Integer ≥ -1 or blank)
$12I/T^{**3}$	Bending moment of inertia ratio, $12I/T^3$. Ratio of the actual bending moment inertia of the shell, I , to the bending moment of inertia of a homogeneous shell, $T^3/12$. The default value is for a homogeneous shell. (Real > 0.0 ; Default = 1.0)
MID3	Material identification number for transverse shear. If MID2 is blank or -1, then MID3 must be blank. (Integer > 0 or blank)
TS/T	Transverse shear thickness ratio, T_s/T . Ratio of the shear thickness, (T_s), to the membrane thickness of the shell, T . The default value is for a homogeneous shell. (Real > 0.0 ; Default = .833333)
NSM	Nonstructural mass per unit area. (Real)
Z1, Z2	Fiber distances for stress calculations. The positive direction is determined by the right-hand rule, and the order in which the grid points are listed on the connection entry. See Remark 11. for defaults. (Real or blank)
MID4	Material identification number for membrane-bending coupling. See Remarks. (Integer > 0 or blank, must be blank unless MID1 > 0 and MID2 > 0 , may not equal MID1 or MID2.)

Remarks:

1. PSHELL is a primary property entry. Primary properties are used to internally group entities. Therefore it is highly recommended that PSHELL entries use unique identification numbers (PIDs) with respect to all other property entries to avoid unexpected grouping results. PSHELL, PCOMP, and PCOMPG entries must have unique PIDs.
2. The structural mass is calculated from the density using the membrane thickness and membrane material properties. If MID1 is blank, then the density is obtained from the MID2 material.
3. The results of leaving an MIDi field blank (or MID2 = -1) are:

MID1	No membrane or coupling stiffness
MID2	No bending, coupling, or transverse shear stiffness
MID3	No transverse shear flexibility
MID4	No bending-membrane coupling unless ZOFFS is specified on the connection entry. See Remark 6.
MID2=-1	See Remark 12.

Note: **MID1 and MID2 must be specified if the ZOFFS field is also specified on the connection entry.**

4. The continuation entry is not required.
5. The structural damping (GE on the MATi entry) is obtained from the MID1 material. If MID1 is blank, then it is obtained from the MID2 material. If PARAM,SHLDAMP,DIFF is set or is anything other than SAME, then the structural damping K^4 matrix is computed using the GE entries on the MATi entries according to rules in the following table (Table 32). If a single PSHELL corresponds to row eight (8) of Table 32, then all PSHELLs in the model will follow the rule of row eight (8). Rows 1-7 is an attempt to maintain upward compatibility, if a user inadvertently places a SHLDAMP,DIFF in the model.

Note: Large values of damping associated with an MID4 entry, when using PARAM,SHLDAMP,DIFF, can cause structural instability in transient dynamics.

Table 32 SHELL Structural Damping Rules

SHELL Structural Damping Rules					
Row	MID1	MID2	MID3	MID4	K ⁴ based on
1*	v	v			
2	v				GE on MID1
3	v	-1			
4	v	v			
5		v			GE on MID2
6		v	v		
7	v1	v2	v3	v4	GE on MID1 if $n = m$ and $ge_1 = ge_2 = \dots = ge_m$ or $m = 1$ and $ge_1 \neq 0$ or $m = 0$ where: n is total number of non blank MIDi (vi) m is total number of non zero ge_i
8	v1	v2	v3	v4	Otherwise: $GE = ge_1 \cdot \text{membrane-stiff} +$ $ge_2 \cdot \text{bending-stiff} +$ $ge_3 \cdot \text{transverse shear-stiff} +$ $ge_4 \cdot \text{bending-membrane-stiff}$
$* v \rightarrow \text{MIDi values the same}; vi \rightarrow \text{MIDi values different or blank}$ $ge_i \rightarrow \text{GE value from a MATj entry associated with MIDi}$					
If for row eight (8), a $ge_i = 0$, it is replaced by $ge_i = 1.0E-8$					

PSHELL rules with extended MAT2 entries

The PSHELL has four MIDi entries: MID1-Membrane; MID2-Bending; MID3-Transverse Shear; MID4-Membran-Bending Coupling.

The Damping for shell elements is determined by the GE value associated to the material entry specified by MID1 (or MID2 if MID1 is blank).

PSHELL allows any of its MIDi fields to use any combination of MAT1, MAT2, or MAT8 material entries.

The MAT2 entry has a feature consisting of six (6) structural damping coefficients called GE11, GE12, GE13, GE22, GE23, and GE33 in a *second continuation entry*. When any one or more of these GEij entries are non-zero, the GE entry in field six (6) of the first continuation entry is ignored for that material.

PSHELL rules with extended MAT2 entries

MAT2 entries with any GEij are called *extended* MAT2 entries. The damping is computed for such a material as $G_{ij} = G_{ij} * GE_{ij}$ where G_{ij} are the elastic material coefficients on the parent MAT2 entry. An element structural damping matrix is then generated using these scaled G_{ij} .

Any PSHELL with an *extended* MAT2 in the MID1 field assumes that every MID2 and MID4 present will perform a $G_{ij} = G_{ij} * GE_{ij}$ calculation to form the appropriate structural damping matrix. The MID3 calculation is of the form $J_{11} = J_{11} * GE_{11}$, $J_{12} = J_{12} * GE_{12}$, and $J_{22} = J_{22} * GE_{22}$.

1	For the case where MID1 is <i>extended</i> (is a reference to a MAT2 with GEij defined) but MID2, MID3, and/or MID4 contain a MAT1, a non- <i>extended</i> MAT2, or a MAT8 entry the calculation is of the form $G_{ij} = G_{ij} * GE$ where GE is the structural damping coefficient of the associated MAT1, MAT2, or MAT8 entry.
2	For the case where MID1 is a MAT1, a non- <i>extended</i> MAT2, or a MAT8, but any one of the MID2, MID3, or MID4 fields have <i>extended</i> MAT2 entries, the structural damping coefficient GE of the MATi entry is used to simply scale the element stiffness as K4 matrix contribution ? GE * K.
3	For case two (2) above, the user has the option of using PARAM,SHLDAMP,DIFF in which case any MID2, MID3, or MID4 containing an <i>extended</i> MAT2 entry force the use of the extended fields. HOWEVER, remember the PARAM,SHLDAMP,DIFF is global – meaning EVERY PSHELL scales each of its MIDI material by the appropriate GEij or GE to form an element structural damping matrix.

6. The following should be considered when using MID4.
 - The MID4 field should be left blank if the material properties are symmetric with respect to the middle surface of the shell. If the element centerline is offset from the plane of the grid points but the material properties are symmetric, the preferred method for modeling the offset is by use of the ZOFFS field on the connection entry. Although the MID4 field may be used for this purpose, it may produce ill-conditioned stiffness matrices (negative terms on factor diagonal) if done incorrectly.
 - Only one of the options MID4 or ZOFFS should be used; if both methods are specified the effects are cumulative. Since this is probably not what the user intends, this may result in unexpected answers. Note that the mass properties are not modified to reflect the existence of the offset when the ZOFFS and MID4 methods are used. If the weight or mass properties of an offset plate are to be used in an analysis, the RBAR method must be used to represent the offset. See [Shell Elements \(CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR\)](#) (p. 123) in the *MSC Nastran Reference Guide*.
 - The effects of MID4 are not considered in the calculation of differential stiffness. Therefore, it is recommended that MID4 be left blank in buckling analysis.
7. This entry is referenced by the CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR entries via PID.
8. For structural problems, MIDI must reference a MAT1, MAT2, or MAT8 material property entry

9. If the transverse shear material MID3 or the membrane-bending coupling term MID4 references a MAT2 entry, then G33 must be zero. If MID3 references a MAT8 entry, then G1Z and G2Z must not be zero.
10. For heat transfer problems, MIDi must reference a MAT4 or MAT5 material property entry.
11. The default for Z1 is -T/2, and for Z2 is +T/2. T is the local plate thickness defined either by T on this entry or by membrane thicknesses at connected grid points, if they are input on connection entries.
12. For plane strain analysis, set MID2=-1 and set MID1 to reference a MAT1 entry. In-plane loads applied to plain strain elements are interpreted as line-loads with a value equal to the load divided by the thickness. Thus, if a thickness of "1.0" is used, the value of the line-load equals the load value. Pressure can be approximated with multiple line loads where the pressure value equals the line-load divided by the length between the loads. For SOL 600, plain strain models must be in the basic coordinate system X-Y plane.
13. For a material nonlinear property, MID1 must reference a MATS1 entry and be the same as MID2, unless a plane strain (MID2 = -1) formulation is desired. Also, MID3 cannot reference a MATS1 entry.
14. If transverse shear flexibility is specified for a model with curved shells where the loading is dominated by twist and shell normals are turned off (e.g., PARAM,SNORM,-1), then results may be inaccurate and may diverge when the mesh is refined. PARAM,SNORM should be set for this unique model condition.
15. For 3D contact analysis, MID2 must be non-zero and for 2D contact analysis, MID1 must be non-zero.

PSHELL1

Properties of shell elements for SOL 700 only

Defines the properties of shell elements with variable shell thickness for SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PSHELL1	PID	MID	FORM	QUAD	NUMB	SHFACT	REF	SPINCOR	
+	T1	T2	T3	T4	TRANSHR	SHRLCK			

Example:

PSHELL1	7	2	BLT	GAUSS	5	0.9			
	10.0	10.0	10.0	10.0					

Descriptor	Meaning
PID	Unique property number. (Integer; Required)
MID	Material number. See Remark 2. (Integer; Required)
FORM	Shell formulation. See Remark 2. (Character; Required)
	HUGHES Hughes-Liu
	BLT Belytschko-Lin-Tsay
	KEYHOFF Key-Hoff
	C0-TRIA C0 triangle
	MEMB Membrane element (no bending)
	DUMMY Dummy element (only applicable for FSI coupling)
QUAD	Type of quadrature. (Character; Default = GAUSS)
	GAUSS Gauss quadrature
	LOBATTO Lobatto quadrature
NUMB	The number of integration points through the thickness. (Integer; Default = 3 for quadrature). 1 for membrane element. Maximum value is 5.
SHFACT	Shear factor. (Real; Default = 0.83333)
REF	Reference surface. (character, default=MID)
	TOP Reference surface is the top surface.
	MID Reference surface is the central surface.
	BOT Reference surface is the bottom surface.
SPINCOR	Spin correction. See Remark 5. (Character; default=NO)
	NO No SPINCOR applied

Descriptor	Meaning
YES	SPINCOR is applied
T1 to T4	Element thickness at the grid points. See Remark 4. (Real; Default = 0.0)
TRANSHR	Method of transverse-shear calculation. (Character; See Remark 6.)
LINEAR	Linear transverse shear
CONSTANT	Constant transverse shear
CONAPX	Approximated constant transverse shear
SHRLCK	Shear-lock avoidance. (Character; See Remark 6.)
AVOID	Avoid shear lockup
NOAVOID	No avoid

Remarks:

- For constant thickness shell element with three-point Gauss integration, PSHELL entry is recommended.
- For CQUAD4 elements, the default of FORM for formulation is KEYHOFF.
For CTRIA3 elements, the default of FORM for formulation is CO-TRIA.
- Property identification number (PID) must be unique.
- If the thickness T1 is set to blank or 0.0, the thickness of the shell must be defined on the CTRIA3 and CQUAD4 entry.
- The options for SPINCOR are:

- NO No SPINCOR correction is applied.
 YES SPINCOR correction is applied.

When SPINCOR = NO, slight asymmetric forces are applied to the shell element's grid points. This approach is, in general, acceptable up to about 10° in plane shear angle.

- The following defaults apply:

	BLT	HUGHES	KEYHOFF
TRANSHR	Not Available	Not Available	LINEAR
SHRLCK	NOAVOID	Not Available	AVOID

PSHLN1

Nonlinear Property Extensions for a PSHELL or PCOMP(G) Entry

Specifies additional nonlinear properties for shell elements that point to a PSHELL or PCOMP(G) entry in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
PSHLN1	PID	MID1	MID2		ANAL				
	“C3”	BEH3	INT3	BEH3H	INT3H				
	“C4”	BEH4	INT4	BEH4H	INT4H				
	“C6”	BEH6	INT6	BEH6H	INT6H				
	“C8”	BEH8	INT8	BEH8H	INT8H				

Example:

PSHLN1	22	98							
--------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number of an existing PSHELL or PCOMP(G) entry. (Integer > 0)
MID1	Membrane material identification number. See Remark 17. (Integer ≥ 0 or blank)
MID2	Bending material identification number. See Remark 17. (Integer ≥ 0 or blank)
ANAL	Analysis type. ANAL = ‘IS’ - Implicit structural elements are being referred to. ANAL = ‘IH’ - Implicit heat analysis elements are being referred to. ANAL = ‘ISH’ - Implicit structural and heat elements are being referred to. (Character Default ISH)
C3	Keyword indicating that two items following apply to elements with three corner grids. (Character)
C4	Keyword indicating that two items following apply to elements with four corner grids. (Character)
C6	Keyword indicating that items following apply to elements with three corner grids and three midside grids. (Character)
C8	Keyword indicating that two items following apply to elements with four corner grids and four midside grids. (Character)
BEHi	Element structural behavior. See Remark 12. (Character Default: DCTN for BEH3, DCT for BEH4, and DCT for BEH8, MB for BEH6)
INTi	Integration scheme. See Remarks 11. and 13. (Character Default: LDK for INT3, L for INT4, QRI for INT8, Q for INT6)

Descriptor	Meaning
BEHiH	Element heat behavior. See Remark 12. (Character Default: DCT for BEH3H, BEH4H, and BEH8H, MB for BEH6H)
INTiH	Integration scheme. See Remarks 11. and 13. (Character Default: L for INT3H, L for INT4H, Q for INT8H and INT6H)

Remarks:

1. The PID must point to an existing PSHELL, PCOMP, or PCOMPG Bulk Data entry and is honored only in SOL 400.
2. The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
3. MID1 if blank (or 0) use the MID1 value on the PSHELL. If > 0 it will override the MID1 value on the PSHELL. MID1 is ignored for PCOMP/PCOMPG.
4. MID2:
 - a. If BEHi = DCT or DCTN: If blank (or 0) use the MID2 value on the PSHELL. If > 0 it will override the MID2 value on the PSHELL. If MID2 = -1 on the PSHELL entry it must be replaced with a positive value MID2 entry or the PSHELL should be replaced with a PLPLANE entry and a PSHLN2 entry should be used instead of a PSHLN1 entry.
 - b. If BEHi = MB: MID2 on both the PSHELL and PSHLN1 entries are ignored.
 MID2 is ignored for PCOMP/PCOMPG.
5. The MID1 or MID2 entries, were applicable, may point to MAT1, MAT2, MAT8, MATHE and MATSMA entries. The table below shows associated nonlinear entries. The association is established through the material entries having the same values as the MID1 or MID2 entries.
 Caution: The primary MATHE and the secondary MATVE, MATVP, MATEP, MATF, and MATS8 entries are only associated with a Shell (CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR, or CTRIA6) element if the Shell element refers to a PSHLN1 entry.
 The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

Implicit Structural Materials			
MAT1	MAT2	MAT8	MATHE
MATVE	<MATVE>	<MATVE>	MATVE
MATVP	MATVP		
MATEP	MATEP	MATEP	
MATF	MATF	MATF	
MATS1		MATS8	

<MATVE> refers to the ALTERNATE format for type ORTHO

6. If MID3 is not specified on the PSHELL, BEH4 = DCTN with INT4 = LDK should be used and any CQUAD8 elements using this PSHELL should have a new PSHELL with MID3 specified. If not a fatal message will be issued. MID3 is not used for PCOMP/PCOMPG.
7. If MID2 is not specified on the PSHELL or overridden with a nonzero value on the PSHLN1, BEH4 = MB. and BEH8 = MB. If not, a fatal message will be issued.

Heat Materials

MAT4

MAT5

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

8. NSM is not currently supported when this entry is used.
9. When this entry points to a PCOMP or a PCOMPG, special restrictions should be noted for some of the PCOMP/PCOMPG entries:
 - NSM, FT and GE are not supported. If a failure theory for any ply defined on the PCOMP/PCOMG entry is required, a MATF entry should be specified for the associated material.
 - The allowable inter-laminar bond strength SB is supported. When used in conjunction with the parabolic shear option (NLMOPTS, TSHEAR), the provided value of SB is used to calculate the ply shear bond index for thick shells using the relation - Bond index = max(inter-laminar shear stress)/SB.
 - LAM=BLANK and LAM=SYM (only for PCOMP) are supported. No smearing i.e., conversion of PCOMP/PCOMG into equivalent PSHELL is supported. Conventional integration through the thickness and across all layers is used. LAM=MEM, BEND, SMEAR and SMCORE are treated in a manner similar to LAM=BLANK, i.e., the sequence of the plies and the data given for each ply is used to carry out the conventional thickness integration.
 - SOUTi is not supported for individual plies. If STRESS output is requested for a particular shell element, then integration point stresses and elastic strains in the material coordinate system for all plies of the element are printed. If the parabolic shear option is used, then integration point values of the inter-laminar stresses and the bond index are also printed.
 - Layer Composite (PCOM/PCOMG) is not supported to CTRAI6 by PSHLN1.
10. If BEHi = MB is selected on PSHLN1, any ZOFF entry on the element connection entry will be ignored with a user warning.

11. In the following table, BEHi refers to the nonlinear structural behavior of shell elements. An underlined item delineates a default.

Implicit Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Doubly-curved thick shell	DCT	L	QUAD	4
		<u>QRI</u>	QUAD	8
		LRIH	QUAD	4
Doubly-curved thin shell	DCTN	<u>LDK</u>	TRIA	3
		LDK	QUAD	4
Membrane three-dimensional	MB	L	QUAD	4
		Q	QUAD	8
		L	TRIA	3
		Q	TRIA	6

12. In the following table, BEHiH refers to the nonlinear heat behavior of shell elements. An underlined item delineates a default.

Heat Classification of Elements				
Element Heat Type	BEHiH CODE	Integration Code	Element Type	# Nodes
Doubly-curved thick shell	DCT	L	QUAD	4
		Q	QUAD	8
		L	TRIA	3
Membrane three-dimensional	MB	L	QUAD	4
		Q	QUAD	8
		L	TRIA	3
		Q	TRIA	6

13. Integration codes in Remark 11. are:

INT CODE	Integration Type
L	Linear
LRIH	Linear Reduced Integration Hourglass control (assumed strain)
Q	Quadratic
QRI	Quadratic Reduced Integration
LDK	Linear Discrete Kirchhoff

14. For different MID cards in PSHELL card, advanced elements follow smeared formulation and only section forces and section moments output.
15. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation if BEH=PSTRS.
16. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available if BEH=PSTRS.
17. Structural damping is supported for elements which reference PSHLN1. The structural damping coefficient, GE, is supported for elements using the MAT1 entries.

PSHLN2

Nonlinear Property Extensions for a PLPLANE Entry

Specifies additional nonlinear properties for plane strain, plane stress, or axisymmetric elements that point to a PLPLANE entry in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
PSHLN2	PID	MID	DIRECT	T	ANAL				
	"C3"	BEH3	INT3	BEH3H	INT3H				
	"C4"	BEH4	INT4	BEH4H	INT4H				
	"C6"	BEH6	INT6	BEH6H	INT6H				
	"C8"	BEH8	INT8	BEH8H	INT8H				

Example:

PSHLN2	22	98							
--------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number of an existing PLPLANE entry. (Integer > 0)
MID	Material identification number. Remarks 13. (Integer > 0)
DIRECT	The layer direction for BEHi=COMPS or AXCOMP. See Remark 6. (Integer = 1 or 2; Default = 1)
T	A thickness. (Real; Default = 1.0)
ANAL	Analysis type. ANAL = 'IS' - Implicit structural elements are being referred to. ANAL = 'IH' - Implicit heat analysis elements are being referred to. ANAL = 'ISH' - Implicit structural and heat elements are being referred to. (Character Default ISH)
C3	Keyword indicating that two items following apply to elements with three corner grids. (Character)
C4	Keyword indicating that two items following apply to elements with four corner grids. (Character)
C6	Keyword indicating that two items following apply to elements with three corner grids and three midside grids. (Character)
C8	Keyword indicating that two items following apply to elements with four corner grids and four midside grids. (Character)
BEHi	Element structural behavior. See Remark 7. (Character Default: PLSTRN for BEH3, BEH4, BEH6, and BEH8)
INTi	Integration scheme. See Remarks 7. and 10. (Character Default: L for INT3, INT4, Q for INT6 and INT8)

Descriptor	Meaning
BEHiH	Element heat behavior. See Remark 9. (Character Default: PLSTRN for BEH3H, BEH4H, BEH6H, and BEH8H.)
INTiH	Integration scheme. See Remarks 9. and 10. (Character Default: L for INT3H, L for INT4H, Q for INT6H and INT8H)

Remarks:

1. The PID must point to an existing PLPLANE Bulk Data entry and is honored only in SOL 400. Since these are additional nonlinear properties to the PLPLANE, the PLPLANE must still have an associated MATHP.
2. It is REQUIRED to override the MID value on the PLPLANE entry.
3. The element must lie in the x-y plane of the basic system. The CID field of the PLPLANE entry is not valid for this entry.
4. The MID entry may point to MAT1, MAT3, MATHORT, MATHE, MATG or MATSMA entry and MUST be used to override the MID field on a PLPLANE entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Caution: The primary MAT1, MAT3, MAT8, MATORT, MATHE, and MATG entries and the secondary MATVE, MATVP, MATEP, MATF, MATS1, MATS3, MATS8, and MATSORT entries are only associated with a 2D or axisymmetric (CQUAD4, CQUAD, CQUAD8, CTRIA3, or CTRIA6) or (CTRIAX or CQUADX) element if the element refers to a PSHLN2 entry.

Implicit Structural Materials							
MAT1	MAT2	MAT3	MAT8	MATORT	MATHE	MATG	MATHP
MATVE	<MATVE>	<MATVE>	<MATVE>	<MATVE>	MATVE		
MATVP	MATVP	MATVP		MATVP			
MATEP	MATEP	MATEP	MATEP	MATEP			
MATF	MATF	MATF	MATF	MATF			
MATS1		MATS3	MATS8	MATSORT			

<MATVE> refers to the ALTERNATE format for type ORTHO
MAT1 applicable to all BEHi codes of 7 below, except COMPS and AXCOMP.

MAT3 axisymmetric orthotropic applicable only to BEHi=AXSOLID code of 7 below.

MAT8 orthotropic applicable only to BEHi=PSTRS code of 7 below.

MATG for BEH4=COMPS or AXCOMP, INT4=L only.
MATG has an associated field IDMEM that points to a MAT1.

With MATHE, MATHP, and MAT1 (with MATS1 or MATEP) BEH3=IPS or IAX with INT3=L is recommended.

Heat Materials

MAT4 MAT5

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries. The CTRIA6 remains a valid nonlinear heat transfer element BUT cannot be used in conjunction with this entry because it lies in an x-z plane and not an x-y plane.

5. The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
6. The following table describes layer orientation for BEHi=COMPS or AXCOMP.

Layer Orientation			
DIRECT	Normal to Layer edge	Layers run parallel from edge	to edge
1	Element Y direction	G1-G2	G4-G3
2	Element X direction	G1-G4	G2-G3

7. In the following table, BEHi refers to the nonlinear structural behavior of 2D-solid elements. An underlined item delineates a default.

Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Plane Stress	PSTRS	L <u>Q</u> QRI LRIH Q	QUAD QUAD QUAD QUAD TRIA	4 8 8 4 6
Plane Strain	PLSTRN	<u>L</u> <u>L</u> <u>Q</u> QRI LRIH Q	QUAD TRIA QUAD QUAD QUAD TRIA	4 3 8 8 4 6
Plane Strain composite	COMPS	L	QUAD	4
Axisymmetric Solid	AXSOLID	L L LT Q QRI QT LRIH Q	QUAD TRIA QUAD QUAD QUAD QUAD QUAD TRIA	4 3 4 8 8 8 4 6
Axisymmetric Composite	AXCOMP	L	QUAD	4
Incompressible Plane Strain	IPS	L	TRIA	3
Incompressible Axisymmetric	IAX	L	TRIA	3
Only BEH4 = COMPS or AXCOMP with INT4 = L may use the MATG, additionally they should not use a MAT1, MAT2, MAT3, MAT8, MATORT, or MATHE as they will suffer from hourgassing.				

8. Note for this entry with a BEHi=COMPS or AXCOMP, the THETA/MCID value on the element connection entry will be ignored.
9. In the table below, BEHiH refers to the nonlinear heat behavior of 2D-solid elements. An underlined item delineates a default.

Heat Classification of Elements				
Element Heat Type	BEHiH CODE	Integration Code	Element Type	# Nodes
Planar Solid	PLSTRN	L L Q Q	QUAD TRIA QUAD TRIA	4 3 8 6
Axisymmetric Solid	AXSOLID	L L Q Q	QUAD TRIA QUAD TRIA	4 3 8 6

10. Integration codes in Remarks 7. and 9. are:

INT CODE	Integration Type
L	Linear
LRIH	Linear Reduced Integration Hourglass Control (assumed strain)
Q	Quadratic
QRI	Quadratic Reduced Integration
QT	Quadratic with Twist
LT	Linear with Twist

11. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation.
12. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available.
13. The structural element damping coefficient, GE, is not supported on elements which reference PSHLN2.

PSLDN1

Nonlinear Property Extensions for a PSOLID Entry

Specifies additional nonlinear properties for solid elements that point to a PSOLID entry in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
PSLDN1	PID	MID	DIRECT		ANAL				
	“C4”	BEH4	INT4	BEH4H	INT4H				
	“C5”	BEH5	INT5	BEH5H	INT5H				
	“C6”	BEH6	INT6	BEH6H	INT6H				
	“C8”	BEH8	INT8	BEH8H	INT8H				
	“C10”	BEH10	INT10	BEH10H	INT10H				
	“C13”	BEH13	INT13	BEH13H	INT13H				
	“C15”	BEH15	INT15	BEH15H	INT15H				
	“C20”	BEH20	INT20	BEH20H	INT20H				

Example:

PSLDN1	22	55	2						
--------	----	----	---	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number of an existing PSOLID entry. (Integer > 0)
MID	Material identification number. Remark 8.(Integer ≥ 0)
DIRECT	The layer direction for BEHi=SLCOMP. See Remark 5. (Integer, 1, 2, OR 3; Default = 1)
ANAL	Analysis type. ANAL = ‘IS’ - Implicit structural elements are being referred to. ANAL = ‘IH’ - Implicit heat analysis elements are being referred to. ANAL = ‘ISH’ - Implicit structural and heat elements are being referred to. (Character Default ISH)
C4	Keyword indicating that two items following apply to elements with four corner grids. (Character)
C5	Keyword indicating that two items following apply to elements with five corner grids. (Character)
C8	Keyword indicating that two items following apply to elements with eight corner grids. (Character)
C10	Keyword indicating that two items following apply to elements with four corner grids and six midside grids. (Character)
C13	Keyword indicating that two items following apply to elements with five corner grids and eight midside grids. (Character)
C15	Keyword indicating that items following apply to elements with six corner and nine midside grids. (Character)

Descriptor	Meaning
C20	Keyword indicating that two items following apply to elements with eight corner grids and twelve midside grids. (Character)
BEHi	Element structural behavior. See Remark 6. (Character default: SOLID for BEH4, BEH6, BEH8, BEH10, BEH15, and BEH20)
INTi	Integration scheme. See Remark 7. (Character default: L for INT4, INT6, and INT8; Q for INT10, INT15, and INT20)
BEHiH	Element heat behavior. See Remark 6. (Character Default: SOLID for BEH4H, BEH6H, BEH8H, BEH10H, BEH15, and BEH20H)
INTiH	Integration scheme. See Remark 7. (Character Default: L for INT4H, INT6H, and INT8H; Q for INT10H, INT15H, and INT20H)

Remarks:

1. The PID must point to an existing PSOLID Bulk Data entry and is honored only in SOL 400.
2. The MID entry may point to MAT1, MAT9, MATORT, MATHE, or MATG entries and can be used to override the MID field on a PSOLID entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Caution: The primary MATORT, MATHE, and MATG entries and the secondary MATVE, MATVP, MATEP and MATF entries are only associated with a 3D Solid (CHEXA, CPENTA, CPYRAM and CTETRA) element if the Solid element refers to a PSLDN1 entry.

The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

Implicit Structure Materials					
MAT1	MAT9	MATORT	MATHE	MATG	MATHP
MATVE	<MATVE>	<MATVE>	MATVE		
MATVP	MATVP	MATVP			
MATEP	MATEP	MATEP			
MATF	MATF	MATF			
MATS1		MATSORT			

<MATVE> refers to the ALTERNATE format for type ORTHO
MATG for BEH8=SLCOMP, INT8=L only.

MATG has an associated field IDMEM that points to a MAT1
BEH4 = ISOL, INT4 = L is currently limited to MAT1 (with possible association with MATS1 or
MATEP), MATHE and MATHP

Heat Materials

MAT4 MAT5

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

3. The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
4. The following table describes layer orientation for BEH8=SLCOMP, INT8=L.

Layer Orientation			
DIRECT	Normal to Layer plane	Layers run parallel from face	to face
1	Element T direction	G1-G2-G3-G4	G5-G6-G7-G8
2	Element R direction	G1-G4-G8-G5	G2-G3-G7-G6
3	Element S direction	G1-G2-G6-G5	G4-G3-G7-G8

5. In the following table, BEHi refers to the nonlinear structural behavior of the solid element. An underlined item delineates a default.

Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
SOLID	SOLID	L	HEX	8
		<u>Q</u>	HEX	20
		QRI	HEX	20
		LRIH	HEX	8
		<u>Q</u>	TET	10
		LRIH	TET	10
		<u>L</u>	TET	4
		<u>L</u>	PEN	6
		<u>Q</u>	PEN	15
		<u>L</u>	PYR	5
Solid continuum composite	SLCOMP	<u>L</u>	PYR	5
		ASTN*	PYR	13

Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Solid Shell	SLCOMP	ASTN	HEX	8
Incompressible solid	ISOL	L	TET	4
Only BEH8=SLCOMP,INT=L may use a MATG, additionally it should not use a MAT1, MAT9, MATORT, or MATHE as it will suffer from hour-glassing.				
It is recommended that for BEH4 = ISOL, INT4 = L that NLMOPTS,LRGSTRN,2 be flagged				
*Only DIRECT=1 is allowed.				

6. In the following table, BEHiH refers to the heat behavior of the solid element. An underlined item delineates a default.

Heat Classification of Elements				
Element Heat Type	BEHiH CODE	Integration Code	Element Type	# Nodes
SOLID	SOLID	L	HEX	8
		<u>Q</u>	HEX	20
		<u>Q</u>	TET	10
		<u>L</u>	TET	4
		<u>L</u>	PEN	6
		<u>Q</u>	PEN	15
		<u>L</u>	PYR	5
		<u>Q</u>	PYR	13

7. Integration codes in Remark 5. are:

INT CODE	Integration Type
L	Linear
LRIH	Linear Reduced Integration Hourglass control (assumed strain)
ASTN	Assumed STraN enhanced formulation solid shell
Q	Quadratic
QRI	Quadratic Reduced Integration

8. Structural damping is supported for elements which reference PSLDN1.
- The structural damping coefficient, GE, is supported for elements using the MAT1 or MAT9 entries.
 - Extended damping coefficients GEij are supported for elements using the MAT9 entries.

PSOLID

Properties of Solid Elements

Defines the properties of solid elements (CHEXA, CPENTA, CPYRAM and CTETRA entries).

Format:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	CORDM	IN	STRESS	ISOP	FCTN		

Example:

PSOLID	2	100	6	TWO	GRID	REDUCED	PORO		
--------	---	-----	---	-----	------	---------	------	--	--

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Identification number of a MAT1, MAT4, MAT5, MAT9, MAT10, MATHP, MATPE1, or MATHE entry. (Integer > 0)
CORDM	Identification number of the material coordinate system. See Remarks 3. and 4. (Integer; Default = 0, which is the basic coordinate system; see Remark 3.)
IN	Integration network. See Remarks 6., 7., 8., and 10. (Integer; Character, or blank)
STRESS	Location selection for stress output. See Remarks 9. and 10. (Integer; Character, or blank)
ISOP	Integration scheme. See Remarks 6., 7., 8., and 10. (Integer; Character, or blank)
FCTN	Fluid element flag. (Character: “FFLUID” indicates a fluid element with frequency dependent rigid absorber properties, “PFLUID” indicates a fluid element, “SMECH” indicates a structural element, “PORO” indicates an element of poroelastic medium, see Remark 12.; Default = “SMECH.”)

Remarks:

1. PSOLID is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PSOLID property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PCOMPLS entries.
2. Isotropic (MAT1 or MAT4), anisotropic (MAT5 or MAT9), or fluid (MAT10) material properties may be referenced. If FCTN = “PFLUID” or “FFLUID”, then MID must reference a MAT10 entry. PFLUID and FFLUID are not available for SOL 600 and SOL 700.

3. See the CHEXA, CPENTA, or CTETRA entry for the definition of the element coordinate systems. The material coordinate system (CORDM) may be the basic system (0 or blank), any defined system (Integer > 0), or the standard internal coordinate system of the element based on eigenvalue techniques to insure non bias in the element formulation designated as element coordinate system (-1) or the element system (-2). The default value for CORDM is zero unless it is overridden by the NASTRAN statement with the CORDM keyword. When MAT9 entry is used, the coordinate system used effects the stiffness calculation of the element. See [Executing MSC Nastran, 1](#).
4. For the CPYRAM entry the element coordinate system is the same as the basic coordinate system. CORDM values of 0, -1, -2 or blank are treated to be identical to the basic coordinate system.
5. Advanced nonlinear solid elements identified through an additional PSLDN1 entry, or SPROPMAP > -1 in NLMOPTS, or IDAMP>0 in NLSTEP, and so on, in SOL 400 do not support: IN, ISOP, and FCTN. Also, in this case, CORDM=-1 is only supported for CHEXA elements. Fatal error is issued if CORRDM < 0 is used for CPENTA and CTETRA.

The following Remarks, DO NOT APPLY TO SOL 600 or SOL 700.

6. For CHEXA and CPENTA elements with no midside nodes, reduced shear integration with bubble functions (ISOP = blank or "REDUCED" and IN = blank or "BUBBLE") is the default. This is recommended because it minimizes shear locking and Poisson's ratio locking and does not cause modes of deformation that lead to no strain energy. The effects of using nondefault values are as follows:
 - a. IN = "THREE" or 3 produces an overly stiff element.
 - b. If IN = "TWO" or 2 and the element has midside nodes, modes of deformation may occur that lead to no strain energy.
 - c. Standard isoparametric integration (ISOP = "FULL" or 1 and IN = "TWO" or 2; or "THREE" or 3) produces an element overly stiff in shear. This type of integration is more suited to nonstructural problems.
 - d. In SOL105 the default reduced integration scheme often produces spurious modes in which case it is recommended that ISOP="FULL" and IN="THREE" be used.
7. IN = "BUBBLE" is not allowed for CTETRA elements or for CHEXA and CPENTA elements with midside nodes.
8. If you use IN = "BUBBLE" for CTETRA elements, NASTRAN internally switch to IN=2 if you have 4-noded CTETRA element and IN=3 greater than 4 nodes.
9. Stress output may be requested at the Gauss points (STRESS = "GAUSS" or 1) of linear CHEXA and CPENTA elements (i.e. elements with no midside nodes). Gauss point output is available for the linear and quadratic CTETRA and CPYRAM elements (i.e. elements with or without midside nodes).
10. The following tables indicate the allowed options and combination of options. If a combination not found in the table is used, then a warning message will be issued and default values will be assigned for all options.
11. The gauss point locations for the solid elements are documented in [Nonlinear Analysis, 535](#) in the *MSC Nastran Reference Guide*.

12. The entries with FCTN="PORO" must select MATPE1 identification number in MID field.

Table 9-32 CHEXA Entry Options

CHEXA	Integration	IN	STRESS (Default: GRID)	ISOP (Default: See Remarks 5 and 7.)	Nonlinear Capability			
					SOLs 106, 129, 400	SOL 600	SOL 700	
8 Node	2x2x2 Reduced Shear with Bubble Function (default)	BUBBLE or Blank or 0 (Default)	Blank or GRID or GAUSS or 1	Blank or REDUCED (Default*)	Yes	Yes**	Yes	
	2x2x2 Reduced Shear Only	TWO or 2						
	2x2x2 Standard Isoparametric	FULL or 1			Yes	No		
	3x3x3 Reduced Shear Only	THREE or 3	Blank or GRID	Blank or REDUCED	No	Yes	Yes	
	3x3x3 Standard Isoparametric			FULL or 1		Yes**	No	
9-20 Node	2x2x2 Reduced Shear Only	TWO or 2	Blank or GRID	Blank or REDUCED	No	Yes	No	
	2x2x2 Standard Isoparametric			FULL or 1		Yes	No	
	3x3x3 Reduced Shear Only (default)	Blank or THREE or 3 (Default)		Blank or REDUCED (Default*)		No	No	
	3 x3x3 Standard Isoparametric			FULL or 1		Yes	No	
p-elements	Reduced (p-order) Bubble	0 or 1	Not applicable	0	No	No	No	
	Bubble, P+ISOP Integration	1		-10 ≤ ISOP ≤ -10				
	No Bubble, P+ISOP Integration	2 or 3						

*REDUCED is the default only for structural elements (FCTN="SMECH").

** Requires PARAM,MRALIAS

Table 9-33 CPENTA Entry Options

CPENTA	Integration	IN	STRESS (Default: GRID)	ISOP (Default: See Remarks 5 and 7.)	Nonlinear Capability		
					SOLs 106, 129, 400	SOL 600	SOL 700
6 Node	2x3 Reduced Shear with Bubble Function (Default)	Blank or 0 or BUBBLE (Default)	GAUSS or 1 or Blank or GRID	Blank or REDUCED (Default*)	Yes		
	2x3 Reduced Shear Only					Yes**	Yes
	2x3 Standard Isoparametric			FULL or 1		Yes	No
	3x7 Reduced Shear Only	THREE or 3	Blank or GRID	Blank or REDUCED	No	No	No
	3x7 Standard Isoparametric			FULL or 1		No	No
7-15 Node	2x3 Reduced Shear Only	TWO or 2	Blank or GRID	Blank or REDUCED	No	Yes**	No
	2x3 Standard Isoparametric			FULL or 1		Yes	No
	3x7 Reduced Shear Only (default)	Blank or THREE or 3 (Default)		Blank or REDUCED (Default*)		No	No
	3x7 Standard Isoparametric			FULL or 1		No	No

** Requires PARAM,MRALIAS

p-elements Bubble Function	2x3 for p=1, 1, 1 3x7 for p=2, 2, 2 (p+1)x(p) for all other	0 or 1	Not applicable	0	No		
	2x3 for p=1, 1, 1 3x7 for p=2, 2, 2			0			
	3x7 for p=1, 1, 1			1			
	(p+ISOP+1) x (p+ISOP) for all other			-10 ≤ ISOP ≤ -1 or 2 ≤ ISOP ≤ 10		No	No
p-elements Standard Isoparametric; (no bubble function)	2x3 for p=1, 1, 1 3x7 for p=2, 2, 2	2 or 3		0			
	3x7 for p=1, 1, 1			1			
	(p+ISOP+1) x (p+ISOP) for all other			-10 ≤ ISOP ≤ -1 or 2 ≤ ISOP ≤ 10			

*REDUCED is the default only for structural elements (FCTN="SMECH").

Table 9-34 CTETRA Entry Options

CTETRA	Integration	IN	STRESS (Default: GRID)	ISOP	Nonlinear Capability		
					SOLs 106, 129, 400	SOL 600	SOL 700
4 Node	1-Point Standard Isoparametric (Default)	Blank or TWO or 2 (Default)	GAUSS or 1 or Blank or GRID	Blank or FULL	Yes	Yes**	Yes
	5-Point Standard Isoparametric	THREE or 3	Blank or GRID		No	Yes	No
5-10 Node	5-Point Standard Isoparametric	Blank or THREE or 3 (Default)	GAUSS or 1 or Blank or GRID	Blank or FULL	Yes	Yes	No
p-elements	1-Point; P=1,1,1 5-Point; P=2,2,2 P+1 Cubic Point; P>2	0 or 1	Not applicable	0	No	No	No
	5-Point; P=1,1,1 P+1 Cubic for all other			1			
	P+ISOP Cubic			-10 ≤ ISOP ≤ -1 or 2 = ISOP ≤ 10			

**Requires PARAM,MRALIAS

Table 9-35 CPYRAM Entry Options

CPYRAM	Integration	IN	STRESS (Default: GRID)	ISOP	Nonlinear Capability		
					SOL 400	SOL 600	SOL 700
5 Node	5-Point Standard Isoparametric (Default)	Not applicable	GAUSS or 1 or Blank or GRID	Not applicable	Yes	No	No
	5-Point Standard Isoparametric				Yes	No	No
13 Node	8-Point Standard Isoparametric	Not applicable	GAUSS or 1 or Blank or GRID	Not applicable	Yes	No	No

PSPRMAT

Spring and damper elements for translation and rotation - SOL 700

Defines linear and nonlinear spring element property for translation. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PSPRMAT	PID								
	K	FAILMTF	FAILMCF	LOAD	UNLOAD				

Example:

PSPRMAT	1								
	20.0E3								

Descriptor	Meaning
PID	Property ID. PID is referenced on the CSPR entry. (Integer > 0; required.)
K	Elastic stiffness (force/displacement). (Real > 0; See Remarks 1. and 2.)
FAILMTF	Tensile failure force. See Remarks 1. and 2. (Real > 0; default=Not failure)
FAILMCF	Compressive failure force. See Remarks 1. and 2. (Real > 0; default=Not failure)
LOAD	Number of a TABLED1 entry defining the variation of force (y-value) with displacement (x-value) during loading. (Integer > 0; See Remarks 1., 3. and 4.)
UNLOAD	Number of a TABLExx entry defining the variation of force (y-value) with displacement (x-value) during unloading. (Integer > 0; See Remarks 3. and 4.)

Remarks:

- Only one set of K, FAILMTF and FAILMCF or LOAD and UNLOAD is allowed. If K is defined, LOAD and ULOAD are not allowed. If LOAD is defined, K, FAILMTF and FAILMCF are not allowed.
- For linear spring, K is required and FAILMTF and FAILMCF are optional.
- For nonlinear spring, LOAD is required and UNLOAD is optional. If UNLOAD is not defined, unloading path is identical as LOAD.
- Input for loading and unloading must be consistent. Both curves must be either completed defined or have only positive values (start from (0.,0.). When only positive values are defined, the curves are automatically mirrored.

PSSHL Properties for Solid Shell (CSSHL) Elements in SOL 600

Defines the properties for Solid Shell (CSSHL) elements in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
PSSHL	PID	MID	IT	SF	MCID				

Example:

PSSHL	11	33		.8333					
-------	----	----	--	-------	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number. (Integer > 0; Required)
MID	Identification of a MATxxx entry. All MAT entries available in SOL 600 can be specified except for hyperelastic materials. (Integer > 0)
IT	Transition thickness - Enter only if a solid shell is attached to a standard shell (such as CQUAD4), in which case TT is the thickness of the standard shell. (Real; Default = 0.0)
SF	Transverse shear factor - Leave blank if transverse shear is not to be considered. (Real or blank, if entered SF must range between 0.0 and 1.0)
CORDM	Identification number of a material coordinate system. See Remark 3. (Integer; Default = 0)

Remarks:

1. PSSHL entries should have unique identification numbers with respect to all other property entries.
2. MID may reference isotropic, orthotropic or anisotropic materials with or with plasticity, however hyperelastic and foam materials are not available.
3. See the CHEXA entry for the definition of the element coordinate system. The material coordinate system (CORDM) may be the basic system (0 or blank) or any defined system (Integer > 0), or the element coordinate system (-1). The default value for CORDM is zero.

PTUBE**Tube Property**

Defines the properties of a thin-walled cylindrical tube element (CTUBE entry).

Format:

1	2	3	4	5	6	7	8	9	10
PTUBE	PID	MID	OD	T	NSM	OD2			

Example:

PTUBE	2	6	6.29	0.25					
-------	---	---	------	------	--	--	--	--	--

Describer Meaning

PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 3. and 4. (Integer > 0)
OD	Outside diameter of tube. (Real > 0.0)
T	Thickness of tube. (Real; T ≤ OD / 2.0)
NSM	Nonstructural mass per unit length. (Real)
OD2	Diameter of tube at second grid point (G2) on CTUBE entry. (Real; Default = OD)

Remarks:

1. If T is zero, a solid circular rod is assumed.
2. PTUBE is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PTUBE property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PTUBE entries.
3. For structural problems, MID must reference a MAT1 material entry.
4. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
5. Tapered OD tubes with constant wall thickness are available for heat transfer only. The effective diameter is given by:

$$D_{\text{effective}} = T + \frac{D_2 - D_1}{\log_e\left(\frac{D_2 - T}{D_1 - T}\right)}$$

where:

$$D_1 = \text{OD}$$

$$D_2 = \begin{cases} \text{OD2 if } \text{OD2} \neq 0 \\ \text{OD if } \text{OD2} = 0 \text{ or blank} \end{cases}$$

PVAL**p-Version Element Polynomial Order Distribution**

Describes polynomial order distribution and is selected by the ADAPT Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
PVAL	ID	POLY1	POLY2	POLY3	CID	SETTYP	ID		

Example:

PVAL	127	1	2	1					
------	-----	---	---	---	--	--	--	--	--

Descriptor	Meaning	Type	Default
ID	ID selected in ADAPT Bulk Data entry.	Integer > 0	Required
CID	Coordinate system used to specify polynomial values in different directions. See Remark 1.	Integer ≥ 0	Remark 2.
POLYi	Polynomial order in 1, 2, 3 directions of the CID system.	Integer > 0	Remark 3.
SETTYP	Type of set provided (SET or ELID). See Remark 6.	Character	"SET"
ID	SET ID or Element ID with these p value specifications. See Remark 6.	Integer > 0	999999

Remarks:

1. CID facilitates the specification of the p-order in curvilinear systems. For example, when modeling a thin cylinder, the user can restrict the p-order through the thickness of all elements to be 2 or 3 without specifically checking the connectivity of each element.
2. If the CID system is blank, the element's topology is used to establish the 1, 2, 3 directions. The 1 direction is from the first to the second grid of the element, the 2 direction is from the first to the fourth, and, the 3 direction is from the first to the fifth. If CID is not blank then the following algorithm will be used to determine the p-order of each edge: a vector will be defined in the CID system from the first to the second grid of every edge. (Curvilinear systems are evaluated at the mid point of this vector). The p-level of each edge is now determined by the nearest integer to:

$$p = \sqrt{(n_1 \cdot \text{POLY1})^2 + (n_2 \cdot \text{POLY2})^2 + (n_3 \cdot \text{POLY3})^2}$$

where (n_1, n_2, n_3) are the components of this unit vector in the CID system.

3. For accuracy and efficiency the recommended minimum p-order is 3. The default value for POLY2 and POLY3 is POLY1.

4. Each finite element has to have a unique PVAL for PSTRUPTID, PMINID, PMAXID. Any overlap of the PVAL specification will result in a warning message and the use of the PVAL with the highest p_i entry.
5. The intermediate PVAL entries generated will have an identification number starting with ADGEN.
6. Whenever SETTYP = "SET", a SET command must be defined under the SETS DEFINITION command in the Case Control Section.
7. SET = 999999 is a reserved set that includes all elements.
8. If there are more than one PVAL entries for a given element, then
 - If CID on the PVALs are the same, the entry with the maximum POLYi will be used.
 - If CID on the PVALs are different, a fatal message is issued.

PVISC

Viscous Damping Element Property

Defines properties of a one-dimensional viscous damping element (CVISC entry).

Format:

1	2	3	4	5	6	7	8	9	10
PVISC	PID1	CE1	CR1		PID2	CE2	CR2		

Example:

PVISC	3	6.2	3.94						
-------	---	-----	------	--	--	--	--	--	--

Describer	Meaning
------------------	----------------

- | | |
|------------------|--|
| PID _i | Property identification number. (Integer > 0) |
| CE1, CE2 | Viscous damping values for extension in units of force per unit velocity. (Real) |
| CR1, CR2 | Viscous damping values for rotation in units of moment per unit velocity. (Real) |

Remarks:

1. Viscous properties are material independent; in particular, they are temperature independent.
2. One or two viscous element properties may be defined on a single entry.
3. PVISC is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PVISC property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PVISC entries.

PVISC1

Nonlinear Damper Property- SOL 700

Defines the properties of a nonlinear damper where the damping constant varies with the velocity. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
PVISC1	PID	TABLE							

Example:

PVISC1	8	236							
--------	---	-----	--	--	--	--	--	--	--

Field	Content
PID	Unique property number. (Integer > 0; required)
TABLE	TABLED1 ID defining the variation of the force (y-value) with velocity (x-value). (Integer > 0; required)

Remarks:

1. This entry defines the properties of a nonlinear damper. Use the PVISC entry to define linear dampers.
2. The values in the table are interpolated to get the force for a particular velocity.

PWELD

Connector Element Property

Defines the property of connector (CWELD) elements.

Format:

1	2	3	4	5	6	7	8	9	10
PWELD	PID	MID	D			MSET		TYPE	
	LDMIN	LDMAX							

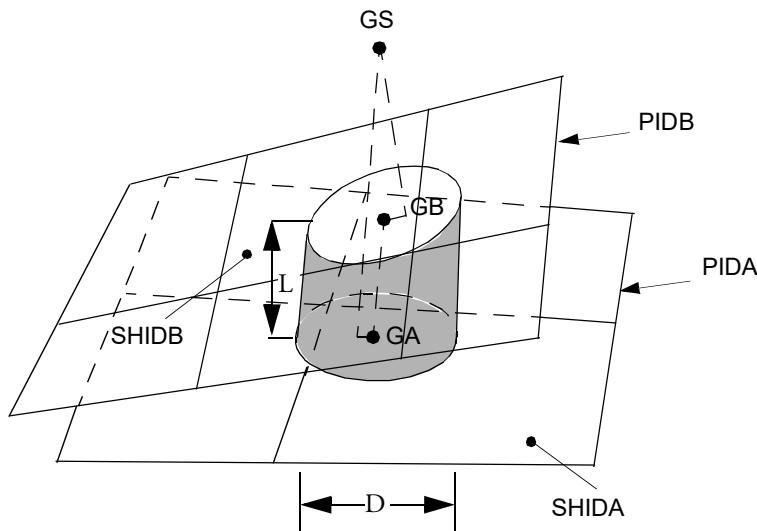
Example:

PWELD	100	3	1.0						
-------	-----	---	-----	--	--	--	--	--	--

Descriptor	Meaning	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number. See Remark 1.	Integer > 0	Required
D	Diameter of the connector. See Remark 1.	Real > 0	Required
MSET	Active ONLY for "PARAM,OLDWELD,YES". Flag to eliminate m-set degrees-of-freedom (DOFs). The MSET parameter has no effect in a nonlinear SOL 400 analysis. =OFF m-set DOFs are eliminated, constraints are incorporated in the stiffness, see Remark 2. =ON m-set DOFs are not eliminated, constraints are generated.	Character	OFF
TYPE	Character string indicating the type of connection, see Remarks 3. and 4. =blank general connector ="SPOT" spot weld connector	Character	Blank
LDMIN	Active ONLY for "PARAM,OLDWELD,YES". Smallest ratio of length to diameter for stiffness calculation, see Remark 4.	Real or blank	0.2
LDMAX	Active ONLY for "PARAM,OLDWELD,YES". Largest ratio of length to diameter for stiffness calculation, see Remark 4.	Real or blank	5.0

Remarks:

1. The material MID, the diameter D, and the length are used to calculate the stiffness of the connector in 6 directions. MID can only refer to the MAT1 Bulk Data entry. The length is the distance of GA to GB (see [Figure 9-150](#)).



[Figure 9-150 Length and Diameter of the CWELD Connector](#)

2. This remark is now valid only if "PARAM,OLDWELD,YES" is present. The parameter MSET is active only for the formats ELEMID and GRIDID (see [CWELD, 1684](#) for the format descriptions). MSET = "OFF" incorporates constraints at the element stiffness matrix level avoiding explicit m-set constraint equations. For the formats PARTPAT and ELPAT, constraints are always eliminated on the element level. MSET = "ON" generates explicit m-set constraints. For example, if a patch-to-patch connection is specified with the formats "GRIDID" or "ELEMID" on the CWELD entry, and MSET=ON is specified, 2x6 explicit constraints are generated that connect the 6 degrees-of-freedom of GA to the translational degrees-of-freedom of grid points GAi and the 6 degrees-of-freedom of GB to GBi. The 2x6 degrees-of-freedom of GA and GB are put into the m-set. The constraints are labeled "RWELD". The identification numbers of the generated RWELD constraint elements start with an offset of 100,001,001 by default. The offset number can be changed with PARAM, OSWELM. For MSET = "OFF" or blank, the 2x6 constraint equations are built into the stiffness matrix of the CWELD element, thereby condensing the 2x6 degrees-of-freedom of GA and GB.
3. TYPE = "SPOT" is good for the formats PARTPAT, ELPAT, or ELEMID with patch to patch connections. For point to patch connections or another formats of patch to patch connections, TYPE is always set to blank.

If "PARAM,OLDWELD,YES" is present, the effective length for the stiffness of the CWELD element is set to $L_e = 1/2 \cdot (t_A + t_B)$ regardless of the distance GA to GB. t_A and t_B are the shell thicknesses of shell A and B, respectively. The effective length is used to avoid excessively stiff or soft connections due to mesh irregularities.

For the new CWELD, the locations of GA/GB (if GA/GB is not specified by the user or the SWLDPRM parameter MOVGAB is equal to 1) will be adjusted so that the distance GA to GB is equal to $L = 1/2 \cdot (t_A + t_B)$.

4. If TYPE=blank and "PARAM,OLDWELD,YES" is present, the effective length L_e of the CWELD is equal to the true length L, the distance of GA to GB, as long as the ratio of the length L to diameter D is in the range $LDMIN \leq L/D \leq LDMAX$. If L is below the range, the effective length is set to $L_e = LDMIN \cdot D$ and if L is above the range, the effective length is set to $L_e = LDMAX \cdot D$.

For the new CWELD with TYPE=blank, LDMIN and LDMAX are ignored. If the distance of GA to GB (patch to patch connections) or GS to GA (point to patch connections) is less than 1.0e-4, the locations of GA/GB (if GA/GB is not specified by the user or the SWLDPRM parameter MOVGAB is equal to 1) will be adjusted so that the distance is equal to 1.0e-4.

5. PWELD is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PWELD property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PWELD entries.
6. The new CWELD will not contribute to MASS by default even if its associated MATi entry has a non-zero density. To react to a non-zero density "SWLDPARM,WMASS,1" is required. If mass is computed, the PARAM,COUPMASS effects the mass calculation.

Entries Q-S

QBDY1

Boundary Heat Flux Load for CHBDYj Elements, Form 1

Defines a uniform heat flux into CHBDYj elements.

Format:

1	2	3	4	5	6	7	8	9	10
QBDY1	SID	Q0	EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	EID8	-etc.-						

Example:

QBDY1	109	1.-5	721						
-------	-----	------	-----	--	--	--	--	--	--

Alternate Format and Example:

QBDY1	SID	Q0	EID1	"THRU"	EID2				
QBDY1	109	1.-5	725	THRU	735				

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
Q0	Heat flux into element. (Real)
EIDi	CHBDYj element identification numbers. (Integer ≠ 0 or "THRU". For "THRU" option EID2 > EID1.)

Remarks:

1. QBDY1 entries must be selected with the Case Control command LOAD = SID in order to be used in static analysis. The total power into an element is given by the equation:

$$P_{in} = (\text{Effective area}) \cdot Q0$$

2. QBDY1 entries must be referenced on a TLOADi Bulk Data entry through the EXCITEID specification for use in transient analysis. The total power into an element is given by the equation:

$$P_{in}(t) = (\text{Effective area}) \cdot Q0 \cdot F(t - \tau)$$

where the function of time $F(t - \tau)$ is specified on a TLOADi entry.

3. The sign convention for Q0 is positive for heat input.
4. RC network solver does not support QBDY1 for thermal analysis.

QBDY2

Boundary Heat Flux Load for CHBDYj Elements, Form 2

Defines grid point heat flux into CHBDYj elements.

Format:

1	2	3	4	5	6	7	8	9	10
QBDY2	SID	EID	Q01	Q02	Q03	Q04	Q05	Q06	
	Q07	Q08							

Example:

QBDY2	109	721	1.-5	1.-5	2.-5	2.-5			
-------	-----	-----	------	------	------	------	--	--	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
EID	Identification number of an CHBDYj element. (Integer > 0)
Q0i	Heat flux at the i-th grid point on the referenced CHBDYj element. (Real or blank)

Remarks:

1. QBDY2 entries must be selected with the Case Control command LOAD=SID in order to be used in static analysis. The total power into each point i on an element is given by

$$P_i = AREA_i \cdot Q0i$$

2. QBDY2 entries must be referenced on a TLOADi Bulk Data entry through the EXCITEID specification for use in transient analysis. All connected grid points will have the same time function but may have individual delays. The total power into each point i on an element is given by

$$P_i(t) = AREA_i \cdot Q0i \cdot F(t - \tau_i)$$

where $F(t - \tau_i)$ is a function of time specified on a TLOADi entry.

3. The sign convention for Q0i is positive for heat flux input to the element.
4. RC network solver does not support QBDY2 for thermal analysis.

QBDY3**Boundary Heat Flux Load for a Surface**

Defines a uniform heat flux load for a boundary surface.

Format:

1	2	3	4	5	6	7	8	9	10
QBDY3	SID	Q0	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

Example:

QBDY3	2	20.0	10	1	THRU	50	BY	2	
-------	---	------	----	---	------	----	----	---	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
Q0	Thermal heat flux load, or load multiplier. Q0 is positive for heat flow into a surface. (Real)
CNTRLND	Control point for thermal flux load. (Integer ≥ 0 ; Default = 0)
EIDi	CHBDYj element identification numbers. (Integer $\neq 0$ or “THRU” or “BY”)

Remarks:

1. QBDY3 entries must be selected in Case Control (LOAD = SID) to be used in steady state. The total power into a surface is given by the equation:
 - if CNTRLND ≤ 0 then $P_{in} = (\text{Effective area}) \cdot Q0$
 - if CNTRLND > 0 then $P_{in} = (\text{Effective area}) \cdot Q0 \cdot u_{\text{CNTRLND}}$
where u_{CNTRLND} is the temperature of the control point and is used as a load multiplier.
2. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time $F(t - \tau)$ defined on the TLOADi multiplies the general load, with τ specifying time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
3. The CNTRLND multiplier cannot be used with any higher-order elements.
4. When using “THRU” or “BY”, all intermediate CHBDYE, CHBDYG, or CHBDYP elements must exist.
5. For RC network solver in thermal analysis, CNTRLND can only be used as the ID in CONTROLT as thermostats controller.

QHBDY

Boundary Heat Flux Load

Defines a uniform heat flux into a set of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
QHBDY	SID	FLAG	Q0	AF	G1	G2	G3	G4	
	G5	G6	G7	G8					

Example:

QHBDY	2	AREA4	20.0		101	102	104	103	
-------	---	-------	------	--	-----	-----	-----	-----	--

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
FLAG	Type of face involved (must be one of the following: "POINT", "LINE", "REV", "AREA3", "AREA4", "AREA6", "AREA8")
Q0	Magnitude of thermal flux into face. Q0 is positive for heat into the surface. (Real)
AF	Area factor depends on type. (Real > 0.0 or blank)
Gi	Grid point identification of connected grid points. (Integer > 0 or blank)

Remarks:

1. The continuation entry is optional.
2. For use in steady state analysis, the load set is selected in the Case Control Section (LOAD = SID).
3. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time $F(t - \tau)$ defined on the TLOADi entry multiplies the general load. 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.
4. The heat flux applied to the area is transformed to loads on the points. These points need not correspond to an HBDY surface element.
5. The flux is applied to each point i by the equation

$$P_i = AREA_i \cdot Q0$$
6. The number of connected points for the types are 1 (POINT), 2 (LINE, REV), 3 (AREA3), 4 (AREA4), 4-6 (AREA6), 5-8 (AREA8).

7. The area factor AF is used to determine the effective area for the POINT and LINE types. It equals the area and effective width, respectively.. For axisymmetric elements, the magnitude of the area factors should account for an integration over the entire circumference (i.e., 2π radians). Note that the REV option can be used to apply the heat flux along the edges of axisymmetric elements. AF is not used for the other types, which have their area defined implicitly and must be left blank.
8. The type of face (FLAG) defines a surface in the same manner as the CHBDYi Bulk Data entry. For physical descriptions of the geometry involved, see the CHBDYG discussion.

QSET**Generalized Degree-of-Freedom**

Defines generalized degrees-of-freedom (q-set) to be used for dynamic reduction or component mode synthesis.

Format:

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

Example:

QSET	15	123456	1	3	9	2	105	6	
------	----	--------	---	---	---	---	-----	---	--

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

Remarks:

- Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
- Degrees-of-freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
- When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
- If Modules are present then this entry may only be specified in the main Bulk Data section.

QSET1

Generalized Degree-of-Freedom (Alternate Form of QSET Entry)

Defines generalized degrees-of-freedom (q-set) to be used for generalized dynamic reduction or component mode synthesis.

Format:

1	2	3	4	5	6	7	8	9	10
QSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Example:

QSET1	123456	1	7	9	22	105	6	22	
	52	53							

Alternate Format and Example:

QSET1	C	ID1	"THRU"	ID2					
QSET1	0	101	THRU	110					

Descriptor	Meaning
C	Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2.)

Remarks:

- Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
- Degrees-of-freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
- When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
- If Modules are present then this entry may only be specified in the main Bulk Data section.

QVECT

Thermal Vector Flux Load

Defines thermal vector flux from a distant source into a face of one or more CHBDYi boundary condition surface elements.

Format:

1	2	3	4	5	6	7	8	9	10
QVECT	SID	Q0	TSOUR	CE	E1 or TID1	E2 or TID2	E3 or TID3	CNTRLND	
	EID1	EID2	-etc.-						

Example:

QVECT	10	20.0	1000.0		1.0	1.0	1.0	101	
	20	21	22	23					

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
Q0	Magnitude of thermal flux vector into face. (Real or blank)
TSOUR	Temperature of the radiant source. (Real or blank)
CE	Coordinate system identification number for thermal vector flux. See Remark 9. (Integer ≥ -1 or blank)
Ei	Vector components (direction cosines in coordinate system CE) of the thermal vector flux. (Real; Default = 0.0)
TIDI	TABLEDi entry identification numbers defining the components as a function of time. (Integer > 0)
CNTRLND	Control point. (Integer ≥ 0 ; Default = 0)
EIDI	Element identification number of a CHBDYE, CHBDYG, or CHBDYP entry. (Integer > 0) Key word "THRU" can be used to assist the listing with ascending order of EIDI

Remarks:

1. The continuation entry is required.
2. If the coordinate system CE is not rectangular, then the thermal vector flux is in different directions for different CHBDYi elements. The direction of the thermal vector flux over an element is aligned to be in the direction of the flux vector at the geometric center of the element. The geometric center is measured using the grid points and includes any DISLIN specification on the VIEW entry for TYPE=LINE CHBDYi elements. The flux is presumed to be uniform over the face of each element; i.e., the source is relatively distant.

3. For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD = SID). The total power into an element is given by:

- If CNTRLND = 0 then, $P_{in} = -\alpha A(\hat{e} \cdot \hat{n}) \cdot Q0$.
- If CNTRLND > 0 then, $P_{in} = -\alpha A(\hat{e} \cdot \hat{n}) \cdot Q0 \cdot u_{CNTRLND}$.

where

α = face absorptivity (supplied from a RADM statement).

A = face area as determined from a CHBDYi surface element.

\hat{e} = vector of direction cosines E1, E2, E3.

\hat{n} = face normal vector. See CHBDYi entries.

$\hat{e} \cdot \hat{n}$ = 0 if the vector product is positive, (i.e., the flux is coming from behind the face).

$u_{cntrlnd}$ = temperature value of the control point used as a load multiplier.

4. If the absorptivity is constant, its value is supplied by the ABSORP field on the RADM entry. If the absorptivity is not a constant, the thermal flux is assumed to have a wavelength distribution of a black body at the temperature TSOUR.

- For a temperature-dependent absorptivity, the element temperature is used to determine α .
- For a wavelength-dependent absorptivity, the integration of the flux times α is computed for each wavelength band. The sum of the integrated thermal fluxes over all the wavelength bands is Q0. The wave bands are specified with the RADBND entry.
- The user has the responsibility of enforcing Kirchhoff's laws.

5. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time $F(t - \tau)$ defined on the TLOADi entry multiplies the general load. τ provides any required time delay. $F(t - \tau)$ is a function of time specified on the TLOADi entry. The value of F is calculated for each loaded grid point. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.

The total power into an element is given by:

- If CNTRLND = 0 then, $P_{in} = -\alpha A(\hat{e}(t) \cdot \hat{n}) \cdot Q0 \cdot F(t - \tau)$.
- If CNTRLND > 0 then,

$$P_{in} = -\alpha A(\hat{e}(t) \cdot \hat{n}) \cdot F(t - \tau) \cdot Q0 \cdot u_{CNTRLND}.$$

6. If the referenced face is of TYPE = ELCYL, the power input is an exact integration over the area exposed to the thermal flux vector.

7. If the referenced face is of TYPE = REV, the thermal flux vector must be parallel to the axis of symmetry if an axisymmetric boundary condition is to be maintained.

8. When applied to a surface element associated with a radiation enclosure cavity, any incident energy that is not absorbed ($\alpha < 1.0$) is lost from the system and is not accounted for in a reflective sense ($\alpha + \rho = 1.0$).

9. If a heat flux normal to the surface is desired, set CE to -1. This allows a nondirectional temperature dependent heat load on the CHBDYi. the RADMT scale factor times Q0 equals to the total power. Remember that the absorptivity must fall between 0.0 and 1.0, (see the RADMT Bulk Data entry).
10. For RC network solver in thermal analysis, CNTRLND can only be used as the ID in CONTROLT as thermostats controller.