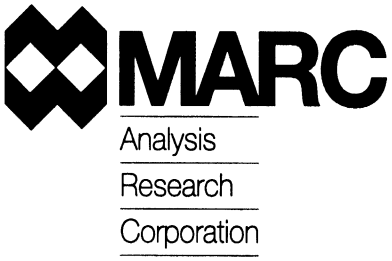


MARC K6.2

Documentation Updates

MARC Analysis Research Corporation
260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306 USA

Phone: (415) 329-6800
FAX: (415) 323-5892

Document Title: **MARC K6.2 Documentation Updates**
Part Number: UP-3500-01
Revision Level: K6.2
Revision Date: August, 1995

PROPRIETARY NOTICE

RESTRICTED RIGHTS NOTICE

TRADEMARKS

MARC K6.2 Documentation Updates

The K6.2 Documentation updates include changes motivated by modifications to the input, new input options, and corrections to the manual. The modified and new input is summarized below. Please refer to the complete manual pages.

Volume A

Volume B

Element Changes

Element 9	This element will have either two or three degrees of freedom, depending upon the other elements in the analysis.

Volume C

Modified Input

Parameter Cards	**Motivation**
ELASTIC Option	Can reduce storage.
FOLLOW FOR	May enter total values.
JOULE	Improvements if highly temperature dependent properties.
RADIATION	Alternative method for view factor calculation.

Model Definition	Motivation
USDATA	Was shown as parameter option, actually it was a model definition option.
SOLVER	Control over number of iterations with iterative solver.
CONTROL	Tolerances for absolute testing and for rigid plastic flow have changed.
CONTACT	New control parameter.
DAMAGE	New control for elastomer damage model using user subroutine UELDAM.

Modified Input

Load Incrementation	Motivation
SOLVER	Control over number of iterations with iterative solver.
CONTROL	Tolerances for absolute testing and for rigid plastic flow have changed.
RELEASE	Allow contact forces to be gradually reduced to zero over a series of increments.

New Input

Model Definition	Motivation
SDRC	SDRC Universal file may be created for postprocessing with SDRC.
UCONTACT	Controls call to user subroutine UCONTACT.

Load Incrementation	Motivation
NODE RELEASE	Control over number of iterations with iterative solver.
ACTUATOR	Allows actuator length to be changed.
GAP CHANGE	Allows gap tolerance to be changed.
VELOCITY CHANGE	Allow velocities to be changed for convective terms in heat transfer analysis.

Volume D

Modified User Subroutines

ANKOND	New argument for Joule heating.
UCOORD	Arguments have changed.

New User Subroutine

UACTUAT	Controls length of actuator.

Material Curve Program

Documentation on the usage of the material curve fitting program for rubber materials and viscoelastic materials.

Update Packet

Volume A: User Information Manual K6.2 Update Packet

The following pages with a • by the page number have been either modified or added to the *Volume A, User Information Manual* to reflect changes in the K6.2 release.

Chapter 5

Replacing Pages:	New Pages:	Moved Pages:
A 5-47 through A 5-50	A 5-47 through A 5-50	
A 5-75 through A 5-76	A 5-75 through A 5-76	

Chapter 6

A 6-5 through A 6-8	A 6-5 through A 6-8	
A 6-13 through A 6-14	A 6-13 through A 6-14	
A 6-19 through A 6-22	A 6-19 through A 6-22	
A 6-27 through A 6-28	A 6-27 through A 6-28	
A 6-57 through A 6-58	A 6-57 through A 6-58	

Volume A
User
Information

K6.2
Updates

and cohesive friction (or constant shear friction), where friction forces are defined by the expression

$$f_t \leq -mk_y t = -m\frac{\sigma_y}{\sqrt{3}}t \qquad \text{(A 5.2-69)}$$

where

f_t is the tangential force being applied
σ_y is the flow stress of the material being deformed
m is the friction factor
t is the tangent unit vector, in the direction of the relative velocity

The distributed load method of implementation may be used with continuum elements using either friction model.

Whenever shell elements are used in conjunction with the adhesive model, it becomes obvious that contact pressures are not available in the analysis. In such cases, friction is then applied as a lumped nodal force, following the same as Equation A 5.2-59. This method may also be used optionally with the continuum elements.

Table A 5.2-2 Continuum Elements

Numerical Method		
Friction Method	**Distributed**	**Point**
Adhesive	possible	possible (optional)
Cohesive	possible	not available

Table A 5.2-3 Shell Elements

Numerical Method		
Friction Method	**Distributed**	**Point**
Adhesive	distributed	optional
Cohesive	not available	possible

NOTE

When friction forces are treated as distributed forces, the analyst should take into consideration that no friction will be applied at an interface that has a single node in contact.

Equation A 5.2-68 and Equation A 5.2-69 are inequalities whenever two contacting surfaces stick to each other, and equalities whenever surfaces slide. The character of the contact constraints changes depending on whether there is sticking or slipping. In order to avoid this distinction, an approximation was made within the CONTACT option in such a way that there is always slip. However, the friction forces decrease to zero when the amount of slip also decreases to zero. The slip or sliding velocity below which such decrease comes into effect is given by the user. MARC is able to calculate a value if there are moving rigid bodies. This modeling is quite robust and does not affect the quality of the solution provided the sliding velocity referred to is small (a couple of orders of magnitude) compared to typical sliding velocities in the analysis. In this way, only nodes that would otherwise stick are affected.

Sometimes the two standard friction models implemented are insufficient for an appropriate friction description. To address this problem, a user subroutine UFRIC has been created that permits the user to constantly monitor the interface conditions and modify friction. For instance, it is possible to switch models, and make friction dependent on location, pressure, temperature, amount of sliding, etc.

When a deformable-body contacts a rigid-body, the friction coefficient of the rigid surface is used. When deformable-deformable contact occurs, the friction coefficient is averaged. The CONTACT TABLE option may also be used if complex situations occur; in which case, the friction coefficient from the CONTACT TABLE option is used.

In metal forming analyses, the user often would like to remove the rigid surfaces and evaluate the mechanical springback. The RELEASE option may be used to release the nodes of a deformable-body which are in contact with a rigid-body. Either the MOTION CHANGE option must be used to move the rigid surface away, or the CONTACT TABLE option must be used to ensure that the nodes do not recontact the surface they were released from.

The CONTACT option has also been made compatible with the coupled thermal-mechanical analysis capabilities of MARC. This feature produces a simultaneous solution of both heat transfer and mechanics problems. Enhancements were made to the contact algorithms that lead to an easy modeling of realistic coupled problems. The concept of bodies also exists. In this case, if the bodies are deformable, a fully coupled analysis is done. A rigid-body, described so far by its geometry, is assigned a constant temperature. A new modeling possibility was introduced in which the body is rigid, but a heat transfer analysis is performed in it. In these cases, the body is defined as a set of HEAT TRANSFER elements. The same boundary node determination, as well as surface determination, is done as for deformable bodies. Mechanical contact between a deformable-body and these rigid bodies is also done by means of ties. However, a rigid-body motion is given to the full heat transfer mesh, controlled in the same way a rigid-body normally is.

Heat fluxes (FILMS) are automatically created on all the boundaries of the deformable-bodies. Two film coefficients are assigned to each body. One corresponds to free convection and is used whenever the boundary is free. The other is a contact film, and is used whenever the boundary is in contact with another body. In these cases the heat flux across the interface is given by

$$q = HD\,(TD - T) \qquad \text{(A 5.2-70)}$$

where

T is the surface temperature

HD is the film coefficient between the two surfaces

TD is the temperature of the same contact location, as obtained from interpolation of nodal temperatures of the body being contacted.

When a deformable-body contacts a rigid-body, the coefficients associated with the rigid-body are used. When two deformable bodies are in contact, the average film coefficient is used. The CONTACT TABLE option may also be used to specify the film coefficient. As with all other coupled problems, heat generated by plastic deformation can be calculated and applied as a volumetric flux. The heat generated by friction is also calculated, and applied as a surface flux provided that friction forces are based on nodal stresses rather than nodal forces.

Two user subroutines are available to facilitate the creation of more sophisticated boundary flux definitions (such as radiation and convections with variations in space, temperature, pressure, etc.) UHTCON allows the user to specify a film coefficient while the surface is in contact, and UHTCOE allows the user to specify a film coefficient while the surface is free.

Nonlinear Support

The MARC program provides two options for the modeling of support conditions: springs and elastic foundations. In a nonlinear problem, the spring stiffness and the equivalent spring stiffness of the elastic foundation can be modified through a user subroutine. In the nonlinear spring option, the incremental force in the spring is

$$\Delta F = K(\Delta u_2 - \Delta u_1) \qquad \text{(A 5.2-71)}$$

where K is the spring stiffness, Δu_2 is the displacement increment of the degree of freedom at the second end of the spring, and Δu_1 is the displacement increment of the degree of freedom at the first end of the spring.

Use the SPRINGS model definition option for the input of spring data. User subroutine USPRNG may be used to specify the value of K based on the amount of previous deformation for nonlinear springs. In dynamic analysis, the SPRING option may also be used to define a dashpot.

In the elastic nonlinear foundation option, the elements in MARC may be specified as being supported on a frictionless (nonlinear) foundation. The foundation supports the structure with an increment force per unit area given by

$$\Delta P_n = K(u_n)\,\Delta u_n \qquad \text{(A 5.2-72)}$$

where K is the equivalent spring stiffness of the foundation (per unit surface area), and Δu_n is the incremental displacement of the surface at a point in the same direction as ΔP_n.

To input nonlinear foundation data, use the FOUNDATION model definition option.

To specify the value of K for the nonlinear equivalent spring stiffness based on the amount of previous deformation of the foundation, use the user subroutine USPRNG.

Nonlinear Loading

When the structure is deformed, the directions and the areas of the surface loads are changed. For most deformed structures, such changes are so small that the effect on the equilibrium equation can be ignored. But for some structures such as flexible shell structure with large pressure loads, the effects on the results can be quite significant so that the surface load effects have to be included in the finite element equations.

MARC forms both pressure stiffness and pressure terms based on current deformed configuration with the FOLLOW FOR option. The FOLLOW FOR option should be used with the LARGE DISP option and the parameter option CENTROID should not be included due to the use of the residual load correction.

type of damping is particularly useful in problems where the characteristics of the model and/or the response change strongly during analysis (e.g., problems involving opening or closing gaps).

Element damping uses coefficients on the element matrices and is represented by the equation:

$$C = \sum_{i=1}^{n} \alpha_i M_i + \left(\beta_i + \gamma_i \frac{\Delta t}{\pi} \right) K_i \qquad \text{(A 5.4-32)}$$

where

C is the global damping matrix
M_i is the mass matrix of i^{th} element
K_i is the stiffness matrix of the i^{th} element
α_i is the mass damping coefficient on the i^{th} element
β_i is the usual stiffness damping coefficient on the i^{th} element
γ_i is the numerical damping coefficient on the i^{th} element
Δt is the time increment

If the same damping coefficients are used throughout the structure, Equation A 5.4-32 is equivalent to Rayleigh damping.

The damping coefficients associated with springs (stiffness and numerical damping) and with mass points (mass damping) are zero. The damping on elastic foundations is the same as the damping on the element on which the foundation is applied. For springs, a dashpot can be added for nonlinear analysis.

Harmonic Response

Harmonic response analysis allows you to analyze structures vibrating around an equilibrium state. This equilibrium state may be unstressed or statically prestressed. Statically prestressed equilibrium states may include material and/or geometric nonlinearities. You can compute the damped response for prestressed structures at various states.

In many practical applications, components are dynamically excited. These dynamic excitations are often harmonic and usually cause only small amplitude vibrations. MARC linearizes the problem around the equilibrium state. If the equilibrium state is a nonlinear, statically prestressed situation, MARC considers all effects of the nonlinear deformation on the dynamic solution. These effects include the following:

- initial stress
- change of geometry
- influence on constitutive law

The vibration problem can be solved as a linear problem using complex arithmetic.

The analytical procedure consists of the following steps:

1. MARC calculates the response of the structure to a static preload (which may be nonlinear) based on the constitutive equation for the material response. In this portion of the analysis, the program ignores inertial effects.
2. MARC calculates the complex-valued amplitudes of the superimposed response for each given frequency, and amplitude of the boundary tractions and/or displacements. In this portion of the analysis, the program considers both material behavior and inertial effects.
3. You can apply different loads with different frequencies or change the static preload at your discretion. All data relevant to the static response is stored during calculation of the complex response.

To initiate a harmonic response analysis, use the HARMONIC parameter. To define the excitation frequency, use the HARMONIC history definition option. If you enter the HARMONIC history definition option with a set of incremental data, the program assumes those incremental data apply only for the harmonic excitation. This is true for applied boundary conditions as well as loads.

The small amplitude vibration problem can be written with complex arithmetic as follows

$$[K + i\omega D - \omega^2 M]\,\bar{u} = \bar{P} \qquad \text{(A 5.4-33)}$$

where

$\bar{u} = u_{re} + iu_{im}$ complex response vector
$\bar{P} = P_{re} + iP_{im}$ complex load vector

The notation is further defined below:

$$K = \Sigma K_{el} + \Sigma K_{sp} \qquad \text{(A 5.4-34)}$$

where

K_{el} are element stiffness matrices
K_{sp} are the spring stiffness matrices

$$M = \Sigma M_{el} + \Sigma M_{mp} \qquad \text{(A 5.4-35)}$$

where

M_{el} are element mass matrices
M_{mp} are masspoint contributions

$$D = \Sigma D_{el} + \Sigma D_{d} + \Sigma\,(\alpha K + \beta M + 2\omega\gamma K) \qquad \text{(A 5.4-36)}$$

The material identification number specified in the COMPOSITE option, is cross-referenced with the material identification number supplied in the ISOTROPIC, ORTHOTROPIC, ANISOTROPIC, TEMPERATURE EFFECTS, ORTHO TEMP, WORK HARD, and RATE EFFECTS options. The ISOTROPIC, ORTHOTROPIC and ANISOTROPIC model definition options allow the user to input material constants such as Young's modulus, Poisson's ratio, shear modulus, etc. The TEMPERATURE EFFECTS and ORTHO TEMP options allow for input of temperature dependency of these material constants. Material constants for a typical layer are as follows:

t_i	thickness of the i^{th} layer
Young's moduli,	E_{xx}, E_{yy}, E_{zz}, ν_{xy}, ν_{yz}, ν_{zx},
Poisson's ratios, and shear moduli	G_{xy}, G_{yz}, G_{zx}
ρ	density
a_{xx}, a_{yy}	coefficients of thermal expansion
σ_y	yield stress
Mat	material identifier associated with temperature-dependent properties and work hardening data

User subroutines ANELAS, HOOKLW, ANEXP, and ANPLAS can be used for the anisotropic behavior of elastic constants, coefficient of thermal expansion, and yield condition.

There are seven given classes of strain-stress relations. The class of a particular element depends on the number of direct (NDI) and shear (NSHEAR) components of stress. Table A 6.2-1 lists the seven classes of elements.

Material Preferred Direction

Every element type in MARC has a default orientation (that is, a default coordinate system) within which element stress-strain calculations take place. This system is also assumed to be the coordinate system of material symmetry. This is especially important for non-isotropic materials (orthotropic, anisotropic, or composite materials). With the ORIENTATION option, the user specifies the orientation of the material axes of symmetry (relationship between the element coordinate system and the global coordinate system, or the 0^o ply angle line, if COMPOSITE) in one of four different ways:

1. as a specific angle offset from an element edge,
2. as a specific angle offset from the line created by two intersecting planes,
3. as a particular coordinate system specified by user-supplied unit vectors, or
4. as specified by user subroutine ORIENT. This is accomplished by the specification of an orientation type, an orientation angle, or one or two user defined vectors.

For the first option (EDGE I-J orientation type), the intersecting plane is defined by the surface normal vector and a vector parallel to the vector pointing from element node I to element node J. The intersection of this plane with the surface tangent plane defines the 0^o orientation axis. (See Figure A 6.2-2.) The orientation angle is measured in the tangent plane positive about the surface normal.

Table A 6.2-1 Classes of Stress-Strain Relations

Class 1	NDI = 1,NSHEAR = 0 Beam Elements 5, 8, 13, 16, 23, 46, 47, 48, 52, 64, 77, 79 $\{\varepsilon\} = \{1/E_{xx}\}\{\sigma\}$
Class 2	NDI = 2,NSHEAR = 0 Axisymmetric Shells 15 and 17 $\begin{Bmatrix}\varepsilon_{xx}\\ \varepsilon_{yy}\end{Bmatrix} = \begin{bmatrix}1/E_{xx} & -\nu_{yx}/E_{yy}\\ -\nu_{xy}/E_{xx} & 1./E_{yy}\end{bmatrix}\begin{Bmatrix}\sigma_{xx}\\ \sigma_{yy}\end{Bmatrix}$ $\nu_{yx} = \nu_{xy}E_{yy}/E_{xx}$
Class 3	NDI = 1,NSHEAR = 1 Beam Elements 14, 45, 76, 78 $\begin{Bmatrix}\varepsilon\\ \gamma\end{Bmatrix} = \begin{bmatrix}1./E_{xx} & 0.\\ 0. & 1./G_{xy}\end{bmatrix}\begin{Bmatrix}\sigma\\ \tau\end{Bmatrix}$
Class 4	NDI = 1,NSHEAR = 1 Plane Stress, Plates and Thin Shells 49 and 72 $\begin{Bmatrix}\varepsilon_{xx}\\ \varepsilon_{yy}\\ \gamma_{xy}\end{Bmatrix} = \begin{bmatrix}1./E_{xx} & -\nu_{yx}/E_{yy} & 0.\\ -\nu_{xy}/E_{xx} & 1./E_{yy} & 0.\\ 0. & 0. & 1./G_{xy}\end{bmatrix}\begin{Bmatrix}\sigma_{xx}\\ \sigma_{yy}\\ \tau_{xy}\end{Bmatrix}$ $\nu_{yx} = \nu_{xy}E_{yy}/E_{xx}$
Class 5	NDI = 2,NSHEAR = 1 Thick Axisymmetric Shells 1 and 89 $\begin{Bmatrix}\varepsilon_{mm}\\ \varepsilon_{\theta\theta}\\ \gamma_{T}\end{Bmatrix} = \begin{bmatrix}1./E_{mm} & -\nu_{\theta m}/E_{\theta\theta} & 0.\\ -\nu_{m\theta}/E_{mm} & 1./E_{\theta\theta} & 0.\\ 0. & 0. & 1/G_{m\theta}\end{bmatrix}\begin{Bmatrix}\sigma_{mm}\\ \sigma_{\theta\theta}\\ \tau_{T}\end{Bmatrix}$ $\nu_{m\theta} = \nu_{\theta m}E_{mm}/E_{\theta\theta}$

Table A 6.2-1 Classes of Stress-Strain Relations (Continued)

Class 6

NDI = 3,NSHEAR = 1

Plain Strain, Axisymmetric with No Twist

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} 1./E_{xx} & -\nu_{yx}/E_{yy} & -\nu_{zx}/E_{zz} & 0. \\ -\nu_{xy}/E_{xx} & 1./E_{yy} & -\nu_{zy}/E_{zz} & 0. \\ -\nu_{xz}/E_{xx} & -\nu_{yz}/E_{yy} & 1./E_{zz} & 0. \\ 0. & 0. & 0. & 1./G_{xy} \end{bmatrix} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{zx} \end{Bmatrix}$$

$$\nu_{yx} = \nu_{xy}E_{yy}/E_{xx} \qquad \nu_{zy} = \nu_{yz}E_{zz}/E_{yy} \qquad \nu_{xz} = \nu_{zx}E_{xx}/E_{zz}$$

Class 7

NDI = 2,NSHEAR = 3

Thick Shell, Element 22, 75

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} 1./E_{xx} & -\nu_{yx}/E_{xx} & 0. & 0. & 0. \\ -\nu_{xy}/E_{xx} & 1./E_{yy} & 0. & 0. & 0. \\ 0. & 0. & 1./G_{xy} & 0. & 0 \\ 0. & 0. & 0. & 1./G_{yz} & 0. \\ 0. & 0. & 0. & 0. & 1./G_{zx} \end{bmatrix} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix}$$

$$\nu_{yx} = \nu_{xy}E_{yy}/E_{xx}$$

Class 8

NDI = 3,NSHEAR = 3

Three-Dimensional Brick Elements, Axisymmetric with Twist

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} 1./E_{xx} & -\nu_{yx}/E_{zz} & -\nu_{zx}/E_{zz} & 0. & 0. & 0. \\ -\nu_{xy}/E_{xx} & 1./E_{yy} & -\nu_{zy}/E_{zz} & 0. & 0. & 0. \\ -\nu_{xz}/E_{xx} & -\nu_{yz}/E_{yy} & 1./E_{zz} & 0. & 0. & 0. \\ 0. & 0. & 0. & 1./G_{xy} & 0. & 0. \\ 0. & 0. & 0. & 0. & 1./G_{yz} & 0. \\ 0. & 0. & 0. & 0. & 0. & 1./G_{zx} \end{bmatrix} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix}$$

$$\nu_{yx} = \nu_{xy}E_{yy}/E_{xx} \qquad \nu_{zy} = \nu_{yz}E_{zz}/E_{yy} \qquad \nu_{xz} = \nu_{zx}E_{xx}/E_{zz}$$

For the second option (global plane orientation type), the intersecting plane is the chosen global coordinate plane. The intersection of this plane with the surface tangent plane defines the 0^o orientation axis. (See Figure A 6.2-3.)

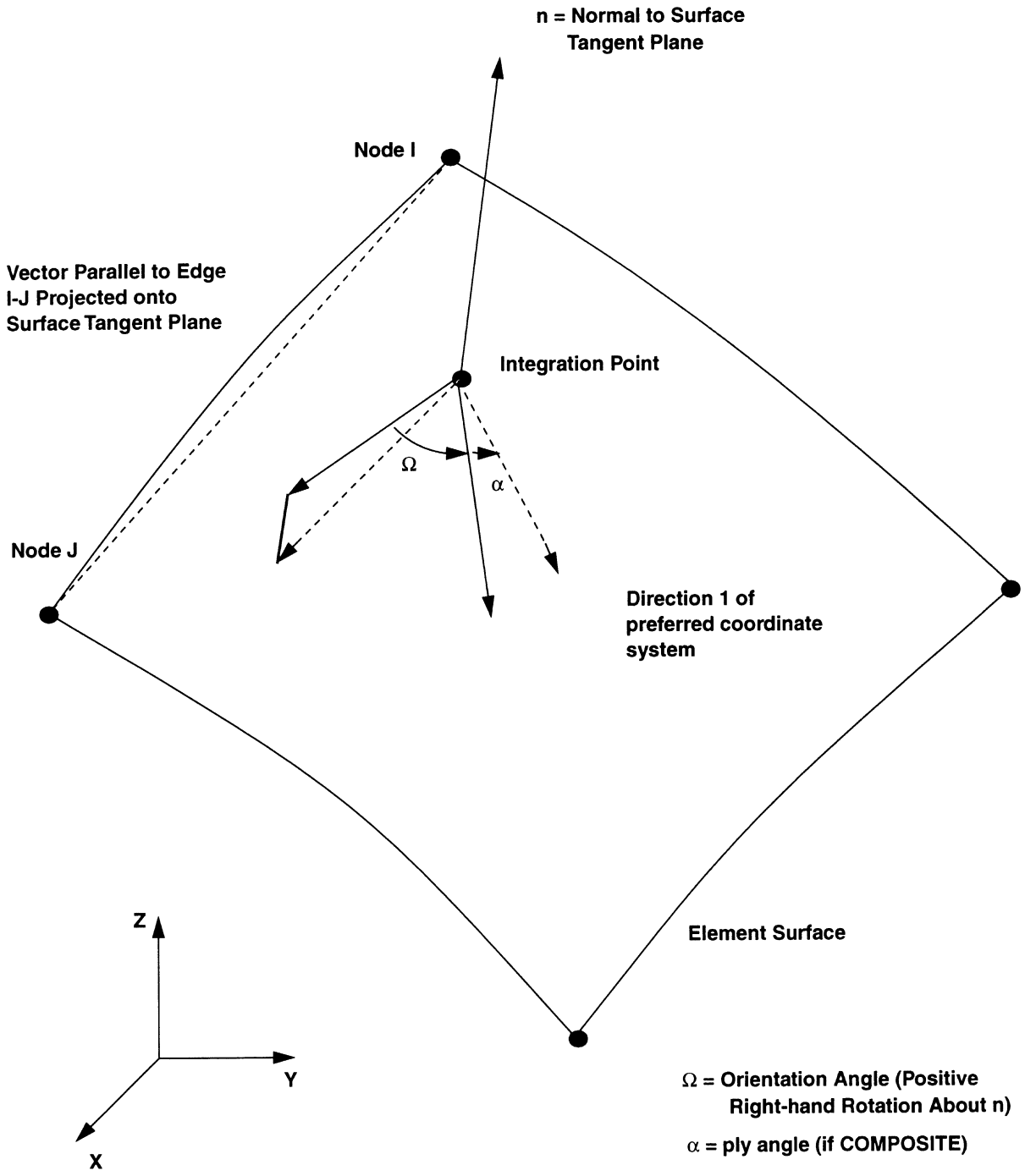


Figure A 6.2-1 Edge I-J Orientation Type

3. $$\begin{cases} \left(\frac{\varepsilon_3}{e_{3t}}\right)/F & \text{if} \quad \varepsilon_3 > 0 \\ \left(-\frac{\varepsilon_3}{e_{3c}}\right)/F & \text{if} \quad \varepsilon_3 < 0 \end{cases} \qquad \text{(A 6.2-9)}$$

4. $$\left(\left|\frac{\gamma_{12}}{g_{xy}}\right|\right)/F \qquad \text{(A 6.2-10)}$$

5. $$\left(\left|\frac{\gamma_{23}}{g_{yz}}\right|\right)/F \qquad \text{(A 6.2-11)}$$

6. $$\left(\left|\frac{\gamma_{31}}{g_{zx}}\right|\right)/F \qquad \text{(A 6.2-12)}$$

where

F	is the failure index (normally, F=1.0).
e_{1t}, e_{1c}	are the maximum allowable strains in the 1 direction in tension and compression.
e_{2t}, e_{2c}	are maximum allowable strains in the 2 direction in tension and compression.
e_{3t}, e_{3c}	are maximum allowable strains in the 3 direction in tension and compression.
g_{xy}	is the maximum allowable shear strain in the xy plane.
g_{yz}	is the maximum allowable shear strain in yz plane.
g_{zx}	is the maximum allowable shear strain in zx plane.

1. Hill Failure Criterion

Assumptions:

a. Orthotropic materials only
b. Incompressibility during plastic deformation
c. Tensile and compressive behaviors are identical

At each integration point, MARC calculates:

$$\left(\frac{\sigma_1^2}{X^2} - \frac{\sigma_1\sigma_2}{X^2} + \frac{\sigma_2^2}{Y^2} + \frac{\sigma_{12}^2}{S^2}\right)/F \qquad \text{(A 6.2-13)}$$

where

F	is the failure index
X	is the maximum allowable stress in the 1 direction
Y	is the maximum allowable stress in the 2 direction
S	is the maximum allowable shear stress

2. Hoffman Failure Criterion

NOTE

Hoffman criterion is essentially Hill criterion modified to allow unequal maximum allowable stresses in tension and compression.

At each integration point, MARC calculates:

$$\left\{\left(\frac{1}{X_t}-\frac{1}{X_c}\right)\sigma_1+\left(\frac{1}{Y_t}-\frac{1}{Y_c}\right)\sigma_2+\frac{\sigma_1^2}{X_tX_c}+\frac{\sigma_2^2}{Y_tY_c}+\frac{\sigma_{12}^2}{S^2}-\frac{\sigma_1\sigma_2}{X_tX_c}\right\}/F \qquad \text{(A 6.2-14)}$$

where: X_t, X_c, Y_t, Y_c, S, F are as before.

NOTE

For small ratios of, for example, $\frac{\sigma_1}{X_t}$, the Hoffman criteria may become *negative* due to the presence of the linear terms.

3. Tsai-Wu Failure Criterion

Tsai-Wu is a tensor polynomial failure criterion. At each integration point, MARC calculates:

$$\left\{\left(\frac{1}{X_t}-\frac{1}{X_c}\right)\sigma_1+\left(\frac{1}{Y_2}-\frac{1}{Y_c}\right)\sigma_2+\frac{\sigma_1^2}{X_tX_c}+\frac{\sigma_2^2}{Y_tY_c}+\frac{\sigma_{12}^2}{S_2}+2F_{12}\sigma_1\sigma_2\right\}/F \qquad \text{(A 6.2-15)}$$

where: F_{12} is the "normal stress interaction" parameter.

NOTE

In order for the Tsai-Wu failure surface to be closed,

$$F_{12}^2 < \frac{1}{X_tX_c}\bullet\frac{1}{Y_tY_c}$$

See Wu, R.Y. and Stachurski, 2, *Evaluation of the Normal Stress Interaction Parameter in the Tensor Polynomial Strength Theory for Anisotropic Materials*, *Journal of Composite Materials*, Vol. 18, Sept. 1984, pp. 456-463.

Simpler forms of the above strain energy function are

$$W = C_{10}(I_1 - 3) + C_{01}(I_2 - 3) \quad \text{Mooney-Rivlin}$$
$$W = C_{10}(I_1 - 3) \quad \text{Neo-Hookean} \qquad \text{(A 6.4-1)}$$

Use the MOONEY model definition option to activate the elastomeric material option in the MARC program and enter the material constants C_{10}, C_{01}, C_{11}, C_{20}, C_{30}.

The TEMPERATURE EFFECTS model definition option can be used to input the temperature dependency of the constants C_{10} and C_{01}. The user subroutine UMOONY can be used to modify all five constants C_{01}, C_{10}, C_{11}, C_{20}, and C_{30}. For viscoelastic, the additional model definition option VISCEL MOON must be included.

The original Ogden strain energy form with N terms is

$$W = \sum_{n=1}^{N} \frac{\mu_n}{\alpha_n}(\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n} - 3) \qquad \text{(A 6.4-2)}$$

This has been enhanced to allow dilatational behavior.

$$W = \sum_{n=1}^{N} \frac{\mu_n}{\alpha_n}\{J^{-\alpha_n/3}(\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n}) - 3\} + 4.5K(J^{1/3} - 1)^2 \qquad \text{(A 6.4-3)}$$

where α_i, μ_i are material constants, and K is the bulk modulus. If no bulk modulus is given, it is taken to be virtually incompressible. This model is different from the Mooney model in several respects. The MOONEY option is with respect to the invariants of the right Cauchy-Green strain tensor and implicitly assumes that the material is incompressible. The Ogden formulation is with respect to the eigenvalues of the right Cauchy-Green strain, and the presence of the bulk modulus implies some compressibility. Using a two-term series will result in identical behavior as the Mooney model. The material data is given through the OGDEN model definition option, or user subroutine UOGDEN. For viscoelastic behavior, the additional model definition option VISCELOGDEN must be included.

Additional definitions:

$$C_{ii} = \lambda_i^2$$

where C_{ii} is the right Cauchy-Green strain.

$$E_{ii} = I_{ii} + 2C_{ii}$$

where Eii is the Green-Lagrange strain

$$J = \lambda_1 \cdot \lambda_2 \cdot \lambda_3 .$$

The pressure variable of the Hermann formulation is related to the strains by:

$$p = 3K\left(J^{\frac{1}{3}} - 1 \right)$$

where

K is the bulk modulus.

Additionally, a damage model may be included with the Ogden model.

The rubber foam model which is based on Ogden formulation has a strain energy form as follows:

$$W = \sum_{n=1}^{N} \frac{\mu_n}{\alpha_n} (\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n} - 3) + \sum_{n=1}^{N} \frac{\mu_n}{\beta_n} (1 - J^{\beta_n}) \qquad \text{(A 6.4-4)}$$

where μ_n, α_n, β_n are material constants. The model reduces to incompressible Ogden model when β_n equals zero.

If the strain energy function expressed by Equation A 6.4-1, Equation A 6.4-3, or Equation A 6.4-5 is not adequate, user subroutine UENERG can be used to define the strain energy function.

The stress may be calculated from the strain energy expressed by Equation A 6.4-6. Since it is a nonlinear relation, you must perform an incremental procedure.

$$s_{ij} = \frac{\partial W}{\partial E_{ij}} \qquad \text{(A 6.4-5)}$$

MARC internally calculates the stress-strain relation

$$\dot{S}_{ij} = L_{ijkl}\dot{E}_{kl} \qquad \text{(A 6.4-6)}$$

Strain invariants in elastomers are discussed below. In the rectangular block in Figure 6.4-2, λ_1, λ_2, and λ_3 are the principal stretch ratios along the edges of the block defined by

$$\lambda_i = (l_i + u_i)/l_i \qquad \text{(A 6.4-7)}$$

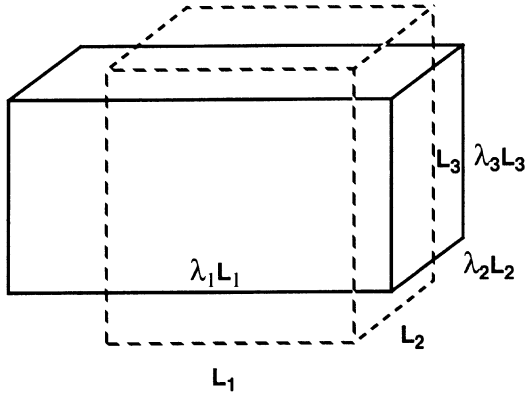


Figure A 6.4-1 Rectangular Rubber Block

In practice, the material behavior is (approximately) incompressible, leading to the constraint equation

$$\lambda_1\lambda_2\lambda_3 = 1 \qquad \text{(A 6.4-8)}$$

the strain invariants are defined as

$$I_1 = \lambda_1^2 + \lambda_2^2 + \lambda_3^2$$

$$I_2 = \lambda_1^2\lambda_2^2 + \lambda_2^2\lambda_3^2 + \lambda_3^2\lambda_1^2 \qquad \text{(A 6.4-9)}$$

$$I_3 = \lambda_1^2\lambda_2^2\lambda_3^2$$

The material constants for the Mooney-Rivlin form may be obtained from experimental data. The Mooney-Rivlin form of the strain energy density function is

$$W = C_{10}(I_1 - 3) + C_{01}(I_2 - 3) \qquad \text{(A 6.4-11)}$$

For the Mooney-Rivlin model, the force and deformation for a uniaxial test specimen may be related as

$$P = 2A_0\left(1 - \frac{1}{\lambda_1^3}\right)(\lambda C_{10} + C_{01}) \qquad \text{(A 6.4-12)}$$

where P is the force of the specimen, A_0 is the original area of the specimen, and λ_1 is the uniaxial stretch ratio. This equation provides a simple way to determine the Mooney-Rivlin constants. The Mooney-Rivlin constitutive equation is applicable if the plot of

$$P/2A_0\left(1 - \frac{1}{\lambda_1^3}\right) \qquad \text{(A 6.4-13)}$$

versus the stretch ratio results in an approximately straight line, as shown in Figure A 6.4-3.

If only the Young's modulus *E* is supplied, and full uniaxial data are not available then

$$C_{01} \cong 0.25C_{10} \qquad \text{(A 6.4-14)}$$

is a reasonable assumption. The constants then follow from the relation:

$$6(C_{10} + C_{01}) \cong E \qquad \text{(A 6.4-15)}$$

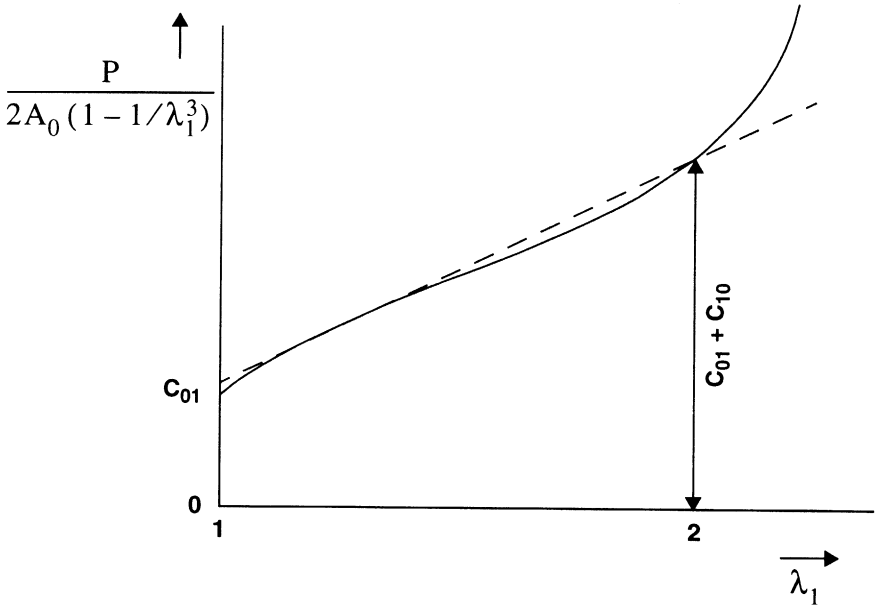
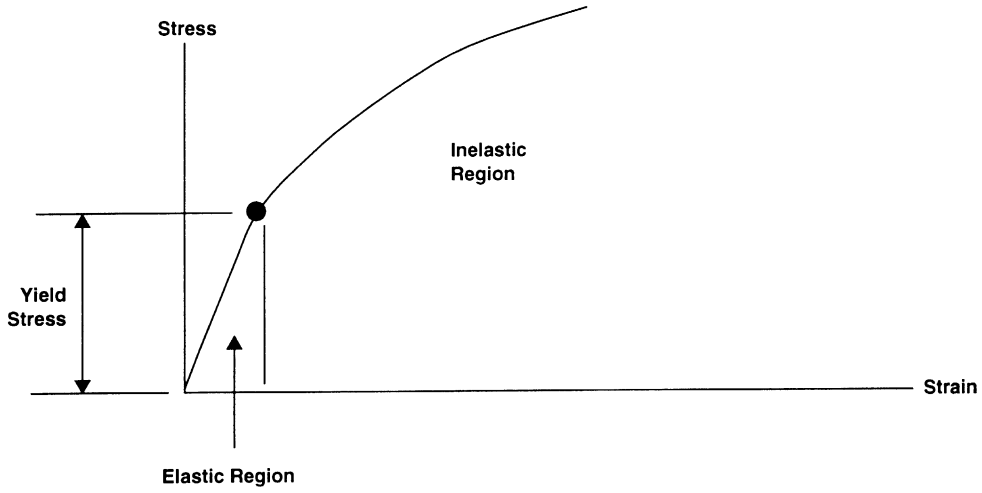


Figure A 6.4-3 An Approximate Straight Line Using the Mooney-Rivlin Constitutive Equation

A 6.5 TIME-INDEPENDENT INELASTIC BEHAVIOR

In uniaxial tension tests of most metals (and many other materials), the following phenomena can be observed. If the stress in the specimen is below the yield stress of the material, the material will behave elastically and the stress in the specimen will be proportional to the strain. If the stress in the specimen is greater than the yield stress, the material will no longer exhibit elastic behavior, and the stress-strain relationship will become nonlinear. Figure A 6.5-1 shows a typical uniaxial stress-strain curve. Both the elastic and inelastic regions are indicated.



Note: Stress and Strain are total quantities.

Figure A 6.5-1 Typical Uniaxial Stress-Strain Curve (Uniaxial Test)

Within the elastic region, the stress-strain relationship is unique. Therefore, if the stress in the specimen is increased (loading) from zero (point 0) to σ_1 (point 1), and then decreased (unloading) to zero, the strain in the specimen is also increased from zero to ε_1, and then returned to zero. The elastic strain is completely recovered upon the release of stress in the specimen. Figure A 6.5-2 illustrates this relationship.

The loading-unloading situation in the inelastic region is different from the elastic behavior. If the specimen is loaded beyond yield to point 2, where the stress in the specimen is σ_2 and the total strain is ε_2, upon release of the stress in the specimen the elastic strain, ε_2^e, is completely recovered. However, the inelastic (plastic) strain, ε_2^p, remains in the specimen. Figure A 6.5-2 illustrates this relationship. Similarly, if the specimen is loaded to point 3 and then unloaded to zero stress state, the plastic strain ε_3^p remains in the specimen. It is obvious that ε_2^p is not equal to ε_3^p. We can conclude that in the inelastic region:

- Plastic strain permanently remains in the specimen upon removal of stress.
- The amount of plastic strain remaining in the specimen is dependent upon the stress level at which the unloading starts (path-dependent behavior).

Linear Mohr-Coulomb Material

The deviatoric yield function, as shown in Figure A 6.5-6, is assumed to be a linear function of the hydrostatic stress.

$$f = \alpha J_1 + J_2^{1/2} - \frac{\bar{\sigma}}{\sqrt{3}} = 0 \qquad \text{(A 6.5-10)}$$

where

$$J_1 = \sigma_{ii} \qquad \text{(A 6.5-11)}$$

$$J_2 = \frac{1}{2}\sigma'_{ij}\sigma'_{ij}, \qquad \sigma'_{ij} = \sigma_{ij} - \frac{1}{3}\delta_{ij}\sigma_{kk} \qquad \text{(A 6.5-12)}$$

Analysis of linear Mohr-Coulomb material based on the constitutive description above is available in MARC through the ISOTROPIC model definition option. Through the ISOTROPIC option, the values of σ and α are entered. Note that, throughout the program, the convention that the tensile direct stress is positive is maintained, contrary to its use in many soil mechanics texts .

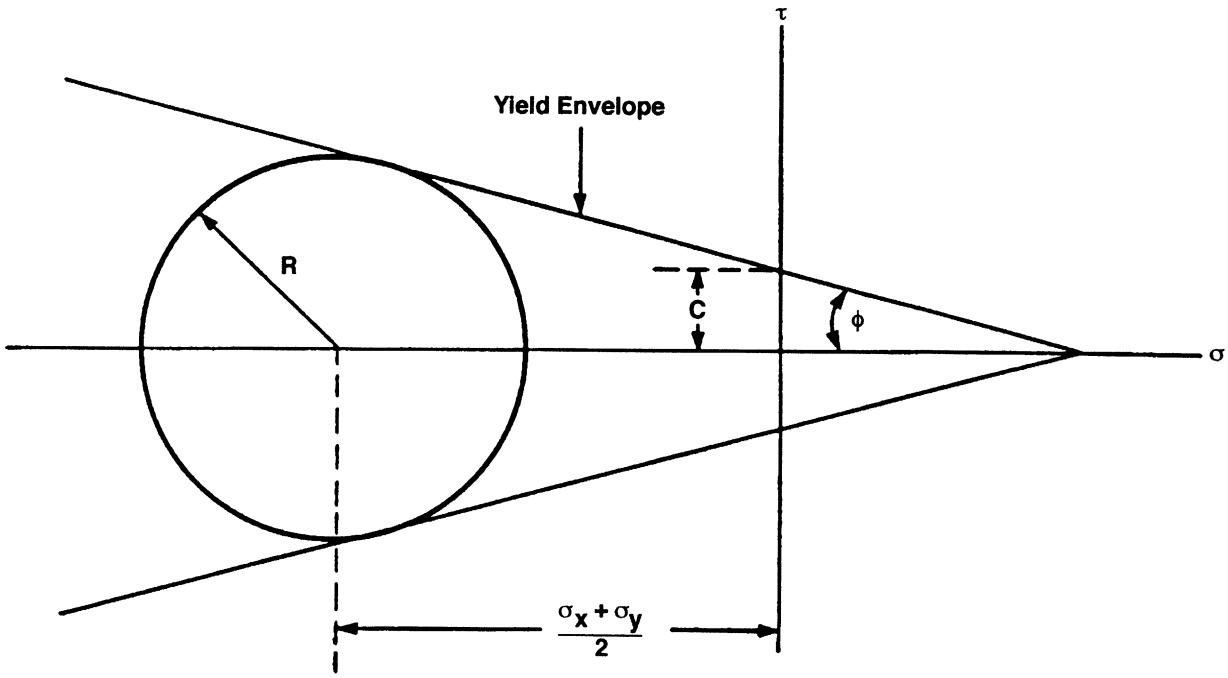


Figure A 6.5-3 Yield Envelope of Plane Strain (Linear Mohr-Coulomb Material)

The constants α and σ may be related to c and ϕ by

$$c = \frac{\bar{\sigma}}{3(1-12\alpha^2)^{1/2}}; \qquad \frac{3\alpha}{(1-3\alpha^2)^{1/2}} = \sin\phi \qquad \text{(A 6.5-13)}$$

where c is the cohesion and ϕ is the angle of friction.

Parabolic Mohr-Coulomb Material

The hydrostatic dependence is generalized to give a yield envelope which is parabolic in the case of plane strain (see Figure A 6.5-7).

$$f = (3J_2 + \sqrt{3}\beta\bar{\sigma}J_1)^{1/2} - \bar{\sigma} = 0 \qquad \text{(A 6.5-14)}$$

The parabolic yield surface is obtained in MARC through the ISOTROPIC definition option. Enter the values $\bar{\sigma}$ and β through the model definition option ISOTROPIC.

where

$$\bar{\sigma}^2 = 3\left(c^2 - \frac{\alpha^2}{3}\right) \qquad \beta = \frac{\alpha}{(3(3c^2 - \alpha^2))^{1/2}} \qquad \text{(A 6.5-15)}$$

where c is the cohesion.

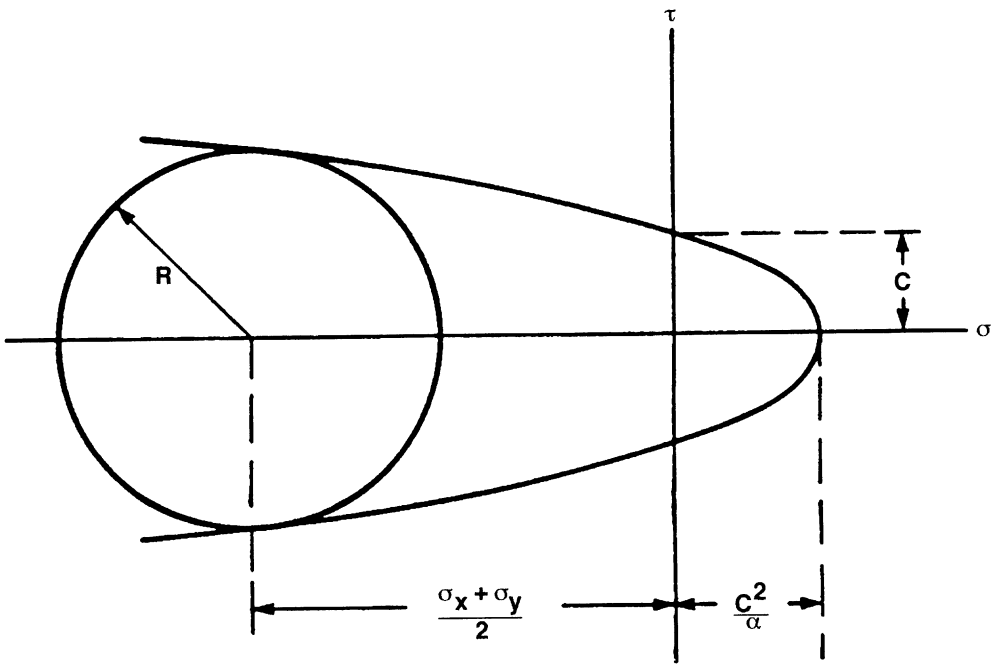


Figure A 6.5-4 Resultant Yield Condition of Plane Strain (Parabolic Mohr- Coulomb Material)

Let

$$\alpha^n(h) = 1 - \exp(-h/\lambda^n) \tag{A 6.6-40}$$

and

$$\beta^n(h) = \alpha^n(h)\,\lambda^n/h$$

The increment total stress-strain relation may then be expressed as

$$\Delta\sigma_{ij}(t_m) = \left[G^{\infty}_{ijkl} + \sum_{n=1}^{N} \beta^n(h)\, G^n_{ijkl}\right]\Delta\varepsilon_{kl} - \sum_{n=1}^{N} \alpha^n(h)\,\sigma^n_{ij}(t_m - h) \tag{A 6.6-41}$$

or the expression for each term is

$$\Delta\sigma^n_{ij}(t_m) = \beta^n(h)\, G^n_{ijkl}\Delta\varepsilon_{kl}(t_m) - \alpha^n(h)\,\sigma^n_{ij}(t_m - h) \tag{A 6.6-42}$$

In the MARC program, the incremental equation for the total stress is expressed in terms of the short term moduli (See Equation A 6.6-35).

$$\Delta\sigma_{ij}(t_m) = \left\{G^{o}_{ijkl} - \sum_{n=1}^{N} [1 - \beta^n(h)]\, G^n_{ijkl}\right\}\Delta\varepsilon_{kl}(t_m) - \sum_{n=1}^{N} \alpha^n(h)\,\sigma^n_{ij}(t_m - h) \tag{A 6.6-43}$$

In this way, the instantaneous elastic moduli can be specified through the ISOTROPIC or ORTHOTROPIC options. Moreover, since the TEMPERATURE EFFECTS option acts on the instantaneous elastic moduli, it is more straightforward to use the short term values instead of the long term ones. Note that the set of equations given by Equation A 6.6-42 and Equation A 6.6-43 can directly be used for both anisotropic and isotropic materials.

Isotropic Viscoelastic Material

For an isotropic viscoelastic material, MARC assumes that the deviatoric and volumetric behavior are fully uncoupled and that the behavior can be described by a time dependent shear and bulk modules. The bulk moduli is generally assumed to be time independent, however this is an unnecessary restriction of the general theory.

Both the shear and bulk moduli can be expressed in a Prony series

$$G(t) = G^{\infty} + \sum_{n=1}^{N} G^n \exp(-t/\lambda_D^n) \qquad \text{(a)}$$

$$K(t) = K^{\infty} + \sum_{n=1}^{N} K^n \exp(-t/\lambda_v^n) \qquad \text{(b)} \qquad \text{(A 6.6-44)}$$

with short term values given by

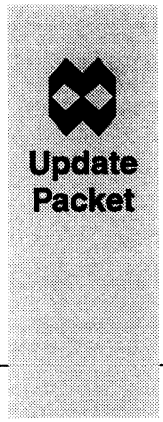
$$G^0 = G^{\infty} + \sum_{n=1}^{N} G^n \qquad \text{(a)}$$

$$K^0 = K^{\infty} + \sum_{n=1}^{N} K^n \qquad \text{(b)} \qquad \text{(A 6.6-45)}$$

Let the deviatoric and volumetric component matrices π_D and π_V be given by

$$\pi_D = \begin{bmatrix} 4/3 & -2/3 & -2/3 & 0 & 0 & 0 \\ -2/3 & 4/3 & -2/3 & 0 & 0 & 0 \\ -2/3 & -2/3 & 4/3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \qquad \text{(A 6.6-46)}$$

$$\pi_V = \begin{bmatrix} 1 & 1 & 1 & 0 & 0 & 0 \\ 1 & 1 & 1 & 0 & 0 & 0 \\ 1 & 1 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$



Volume B: Element Library K6.2 Update Packet

The following pages with a • by the page number have been either modified or added to the *Volume B, Element Library* manual to reflect changes in the K6.2 release.

Chapter 1

Replacing Pages:	New Pages:	Moved Pages:
B 1.9-13 through B 1.9-14	B 1.9-13 through B 1.9-14	

Chapter 3

B 3.9-1 through B 3.9-2	B 3.9-1 through B 3.9-2	
B 3.11-3 through B 3.11-4	B 3.11-3 through B 3.11-4	
B 3.12-1 through B 3.12-2	B 3.12-1 through B 3.12-2	
B 3.78-3 through B 3.78-4	B 3.78-3 through B 3.78-4	
B 3.79-3 through B 3.79-4	B 3.79-3 through B 3.79-4	
B 3.115-3 through B 3.115-4	B 3.115-3 through B 3.115-4	

Volume B
Element
Library

K6.2
Updates

Table B 1.9-3 Summary of Element Properties

Element Type	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
1	2	2	1	1	3	2	Yes
2	3	3	1	1	2	2	Yes
3	4	2	1	4	2	2	Yes
4	4	2	1	9	12	14	Yes
5	2	1	1	3	3	2	No
6	3	3	1	1	2	2	Yes
7	8	3	3	8	3	3	Yes
8	3	2	1	7	9	11	Yes
9	2	1	0	1	3	3	Yes
10	4	3	1	4	2	2	Yes
11	4	3	1	4	2	2	Yes
12	4	1	2	1	2/3	2/3	No
13	2	1	0	3	8	13	No
14	2	1	1	3	6	6	Yes
15	2	2	0	3	4	5	Yes
16	2	1	0	3	4	5	Yes
17	3	2	0	3	6	5	No
18	4	2	1	4	3	3	No
19	6	3	1	4	2	2	Yes
20	4	3	3	4	3	2	Yes
21	20	3	3	27	3	3	Yes
22	8	2	3	4	6	3	Yes
23	20	1	0	20	3	3	No
24	8	2	1	28	9	11	Yes
25	2	1	1	3	7	6	Yes
26	8	2	1	9	2	2	Yes
27	8	3	1	9	2	2	Yes
28	8	3	1	9	2	2	Yes
29	10	3	1	9	2	2	Yes
30	8	2	1	9	3	3	No
31	2	2	1	2	6	3	No

Table B 1.9-3 Summary of Element Properties (Continued)

Element Type	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
32	8	3	1	9	3	2	No
33	8	3	1	9	3	2	No
34	10	3	1	9	3	2	No
35	20	3	3	27	4	3	No
36	2	1	0	1	1	3	No
37	3	2	0	1	1	2	No
38	3	2	0	1	1	2	No
39	4	2	0	4	1	2	No
40	4	2	0	4	1	2	No
41	8	2	0	9	1	2	No
42	8	2	0	9	1	2	No
43	8	3	0	8	1	3	No
44	20	3	0	27	1	3	No
45	3	1	1	2	3	2	Yes
46	8	1	0	10	2	2	No
47	10	1	0	10	2	2	No
48	8	1	0	10	2	2	No
49	6	2	1	1	3	3	Yes
50	3	3	0	1	2	3	No
51	2	1	0	1	3	3	No
52	2	1	0	3	6	6	Yes
53	8	2	1	4	2	2	Yes
54	8	3	1	4	2	2	Yes
55	8	3	1	4	2	2	Yes
56	10	3	1	4	2	2	Yes
57	20	3	3	8	3	3	Yes
58	8	3	1	4	3	2	No
59	8	3	1	4	3	2	No
60	10	3	1	4	3	2	No
61	20	3	3	8	4	3	No
62	8	3	3	9	3	2	No

Element 9

B 3.9 Three-Dimensional Truss

Element type 9 is a simple linear straight truss with constant cross-section. The strain-displacement relations are written for large strain, large displacement analysis. All constitutive relations may be used with this element. This element can be used as an actuator in mechanism analyses.

> **NOTE:** This element has no bending stiffness.

Quick Reference

Type 9

Two- or three-dimensional, two-node, straight truss. Used by itself or in conjunction with any 3-D element, this element will have three coordinates and three degrees of freedom. Otherwise, it has two coordinates and two degrees of freedom.

Connectivity

Two nodes per element (see Figure B 3.9-1).

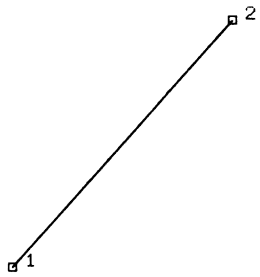


Figure B 3.9-1 Two-Noded Truss

Geometry

The cross-sectional area is input in the first data field (EGEOM1). Default area is equal to 1.0. The second and the third data fields are not used. The fourth field is used to define the length of the element when used as an actuator. The ACTUATOR history definition option, or the user subroutine UACTUAT can be used to modify the distance of the link.

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u displacement
2 = v displacement
3 = w displacement (optional)

Tractions

Distributed loads according to the value of IBODY are as follows:

Load Type	Description
0	Uniform load (force per unit length) in the direction of the global x-axis.
1	Uniform load (force per unit length) in the direction of the global y-axis.
2	Uniform load (force per unit length) in the direction of the global z-axis.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION AXIS option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION AXIS option.

Output of Strains

Uniaxial in the truss member.

Output of Stresses

Uniaxial in the truss member.

Transformation

The global degrees of freedom for any node may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Points

Only one integration point available.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output as for total Lagrangian approach. Cross section will be updated.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See B 3.36 for a description of the conventions used for entering the flux and film data for this element.

Load Type	Description
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION AXIS option.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in the x- and y-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION AXIS option.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element in global coordinates is:

1 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = γ_{rz}

Output of Stresses

Same as for **Strains**.

Transformation

Two global degrees of freedom may be transformed into local coordinates.

Tying

Use subroutine UFORMS.

Output Points

Output is available at the centroid or at the four Gaussian points shown in Figure B 3.11-1.

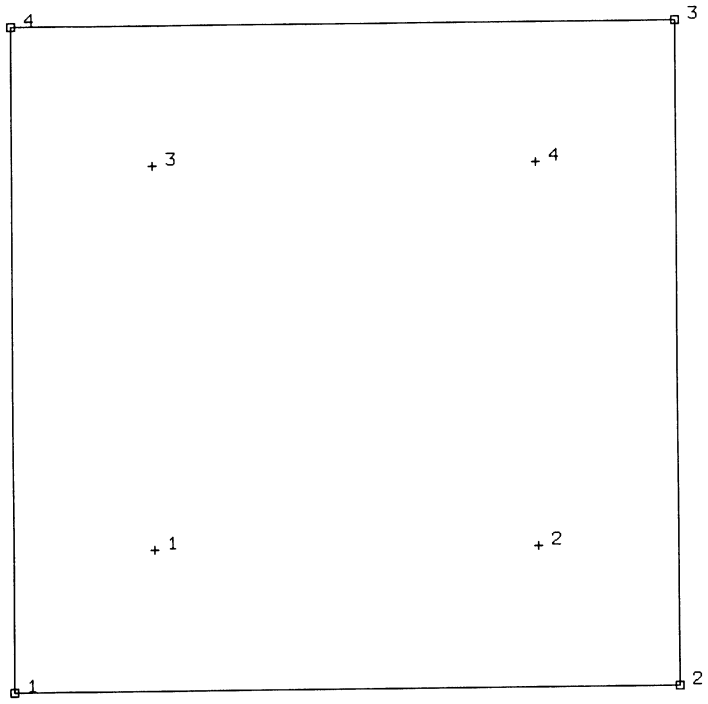


Figure B 3.11-1 Gaussian Integration Points for Element Type 11

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions. Reduced volume strain integration recommended. (See **Geometry**.)

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See B 3.39 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Assumed Strain

The assumed strain formulation is available to improve the bending characteristics of this element. Although this increases the stiffness assembly costs per element, it does improve the accuracy.

Element 12

B 3.12 Friction And Gap Link Element

This element provides frictional and gapping connection between any two nodes of a structure. Essentially, the element is based on imposition of a gap closure constraint and frictional stick or slip via Lagrange multipliers.

The element may be used with any other elements in the program by invoking suitable tying (user UFORMS), if necessary.

Three different options for the definition of the gap have been included. The default formulation is a gap in a fixed direction. This option is useful for geometrically linear analysis or geometrically nonlinear analysis if a body is not to penetrate a given flat surface.

The second formulation constrains the true distance between the two end-points of the gap to be greater or less than a specified value. This option is useful for geometrically nonlinear analysis or for analysis in which a body is not to penetrate a given circular (2D) or spherical (3D) surface. This option is activated by specification of a "1" in the seventh data field of GAP DATA model definition block.

The third formulation allows specification of closure distance and gap direction in the user subroutine GAPU. The gap direction and distance can then be updated during analysis to model sliding along a curved surface. The fixed direction gap option in GAP DATA block must be activated if GAPU is used.

Fixed Direction Gap

Description of the Fixed Direction Gap for Two-Dimensional Problems

The element is implemented in the program as a four-node element (link). The first and fourth nodes have (u, v) Cartesian displacements to couple to the rest of the structure.

Node 2 is the gap node.It has one degree of freedom, F_n, the force being carried across the link. The coordinate data for this node is used to input (n_x, n_y), the direction of $\underset{\sim}{n}$, the gap closure direction. If these data are not given (or are all zero), the program defines:

$$\underset{\sim}{n} = (\underset{\sim}{X}_4 - \underset{\sim}{X}_1) / |\underset{\sim}{X}_4 - \underset{\sim}{X}_1| ;$$

i.e., the gap closure direction is along the element in its original configuration. The user should note that, in many cases, the gap is very small (or, indeed, may be of zero length if the two surfaces are initially touching), so that inaccuracies may be introduced by taking a small difference between two large values. This is the reason for allowing separate input of (n_x, n_y).

Node 3 is the friction node. It has degrees of freedom F; the frictional force being carried across the link, and s, the net frictional slip. The coordinate data for this node may be given as (t_x, t_y), the frictional direction. If the user does not input this data, $\underset{\sim}{t}$ is defined by the program as:

$$\underset{\sim}{t} = \underset{\sim}{k} \times \underset{\sim}{n}$$

where $\underset{\sim}{n}$ is the gap direction (see above) and $\underset{\sim}{k}$ is the unit vector normal to the plane of analysis.

Description of the Fixed Direction Gap for Three-Dimensional Problems

The first and fourth nodes have (u, v, w) Cartesian displacements to couple to the rest of the structure.

Node 2 is the gap node. It has one degree of freedom, F_n; the normal force being carried across the link. The coordinate data for this node is used to input (n_x, n_y, n_z), the direction of $\underset{\sim}{n}$, the gap closure direction. If these data are not given (or are all zero), the program defines:

$$\underset{\sim}{n} = (\underset{\sim}{X}_4 - \underset{\sim}{X}_1) / \left| \underset{\sim}{X}_4 - \underset{\sim}{X}_1 \right| ;$$

i.e., the gap closure direction is along the element in its original configuration. The user should note that in many cases, the gap is very small (or, indeed, may be of zero length if the two surfaces are initially touching), so that inaccuracies may be introduced by taking a small difference between two large values. This is the reason for allowing separate input of (n_x, n_y, n_z).

Node 3 is the friction node. It has degrees of freedom (F_1, F_2); the frictional forces being carried cross the link and s, the net frictional slip.

The coordinate data for this note may be given as (t_x^1, t_y^1, t_z^1), the first frictional direction. If the user does not input these data, $\underset{\sim}{t}^1$ is defined by the program as:

$$\underset{\sim}{t}^1 = \underset{\sim}{i} \times \underset{\sim}{n}$$

where $\underset{\sim}{n}$ is the gap direction (see above) and $\underset{\sim}{i}$ is the unit vector in the global $\underset{\sim}{x}$-direction. If $\underset{\sim}{n}$ is parallel to $\underset{\sim}{i}$, the first friction direction is defined as:

$$\underset{\sim}{t}^1 = -\underset{\sim}{j}$$

where $\underset{\sim}{j}$ is the global y-direction. The second friction direction $\underset{\sim}{t}^2$ is calculated by the program as:

$$\underset{\sim}{t}^2 = \underset{\sim}{n} \times \underset{\sim}{t}^1$$

Distributed Loads

Distributed load types as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length, with magnitude and direction supplied via user subroutine FORCEM.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION AXIS option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION AXIS option.

Point Loads

Point loads and moments may be applied at the end nodes.

Output Of Strains

Generalized strains:

1 = ε_{zz} = axial stretch
2 = κ_{xx} = curvature about local x-axis of cross section.
3 = κ_{yy} = curvature about local y-axis of cross section.
4 = γ = twist per unit length.

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-moment
4 = axial torque

Stresses at integration points in the cross section are only printed if explicitly requested or if plasticity is present.

Transformation

Displacement and rotations at the nodes may be transformed to a local coordinate system.

Tying

Use tying type 100 for fully moment-carrying joints. Use tying type 103 for pin joints.

Output Points

Centroid or two Gaussian integration points. The first point is near the first node in the CONNECTIVITY description of the element. The second point is near the second node in the CONNECTIVITY description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange is available. Finite is not recommended since the cross section is assumed to remain constant.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See B 3.36 for a description of the conventions used for entering the flux and film data for this element.

Load Type	Description
4	Nonuniform load per unit length; magnitude and direction supplied via user subroutine FORCEM.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION AXIS option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION AXIS option.

Point Loads

Point loads and moments may be applied at the end nodes.

Output of Strains

Generalized strains:

1 = ε_{zz} = axial stretch
2 = κ_{xx} = curvature about local x-axis of cross section.
3 = κ_{yy} = curvature about local y-axis of cross section.
4 = η = warping.
5 = γ = twist.

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-moment
4 = bimoment
5 = axial torque

Layer stresses are only printed if explicitly requested or if plasticity is present.

Transformation

Displacement and rotations at the nodes may be transformed to a local coordinate reference.

Tying

Use tying type 100 for fully moment-carrying joints, tying type 103 for pin joints.

Output Points

Centroid or two Gaussian integration points. The first point is near the first node in the CONNECTIVITY description of the element. The second point is near the second node in the CONNECTIVITY description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange procedure is available. Finite strain is not recommended since it is assumed that the cross section does not change.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See B 3.36 for a description of the conventions used for entering the flux and film data for this element.

Load Type	Description
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION AXIS option.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x- and y-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION AXIS option.

For all nonuniform pressures, body and shear forces, magnitude is supplied through subroutine FORCEM.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element are:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz} = 0$
$4 = \gamma_{xy}$

Output of Stresses

Same as for **Strains**.

Transformation

The two global degrees of freedom at the corner nodes may be transformed into local coordinates.

Tying

Use subroutine UFORMS.

Output Points

Output is available at the centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain in global coordinate directions. This element does not lock for nearly incompressible materials.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 121. See B 3.121 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Volume C: Program Input K6.2 Update Packet

Packet Update

The following pages with a • by the page number have been either modified or added to the *Volume C, Program Input* manual to reflect changes in the K6.2 release.

Chapter 1

Replacing Pages:	New Pages:	Moved Pages:
C 1-3 through C 1-4	C 1-3 through C 1-4	

Chapter 2

C 2-iii through C 2-iv	C 2-iii through C 2-iv	
C 2-13 through C 2-16	C 2-13 through C 2-16	
C 2-27 through C 2-28	C 2-27 through C 2-28	
C 2-35 through C 2-36	C 2-35 through C 2-36	
C 2-41 through C 2-42	C 2-41 through C 2-42	
		2-75 to 3-76

Chapter 3

C 3-vii through C 3-xii	C 3-vii through C 3-xii
C 3-45 through C 3-54	C 3-45 through C 3-54
C 3-57 through C 3-58	C 3-57 through C 3-58
C 3-75 through C 3-76	C 3-75 through C 3-76
C 3-81 through C 3-82	C 3-81 through C 3-82
C 3-89 through C 3-90	C 3-89 through C 3-90
	C 3-94.1 through C 3-94.2
C 3-97 through C 3-98	C 3-97 through C 3-98
C 3-119 through C 3-122	C 3-119 through C 3-122
C 3-137 through C 3-138	C 3-137 through C 3-138
C 3-145 through C 3-146	C 3-145 through C 3-146
C 3-177 through C 3-180	C 3-177 through C 3-180
C 3-189 through C 3-190	C 3-189 through C 3-190
C 3-197 through C 3-198	C 3-197 through C 3-198
C 3-201 through C 3-202	C 3-201 through C 3-202
C 3-207 through C 3-210	C 3-207 through C 3-210
	C 3-210.1 through C 3-210.2
C 3-215 through C 3-218	C 3-215 through C 3-218

Chapter 3 (Continued)

Replacing Pages:	New Pages:	Moved Pages:
C 3-221 through C 3-230	C 3-221 through C 3-230	
C 3-245 through C 3-246	C 3-245 through C 3-246	
C 3-257 through C 3-264	C 3-257 through C 3-264	
C 3-295 through C 3-296	C 3-295 through C 3-296	
C 3-309 through C 3-310	C 3-309 through C 3-310	
C 3-335 through C 3-338	C 3-335 through C 3-338	
C 3-349 through C 3-350	C 3-349 through C 3-350	

Chapter 5

C 5-v through C 5-viii	C 5-v through C 5-viii	
C 5-11 through C 5-12	C 5-11 through C 5-12	
C 5-21 through C 5-28	C 5-21 through C 5-28	
	C 5-34.1 through C 5-34.2	
C 5-35 through C 5-38	C 5-35 through C 5-38	
C 5-41 through C 5-44	C 5-41 through C 5-44	
	C 5-52.1	
C 5-63 through C 5-64	C 5-63 through C 5-64	
	C 5-64.1 through C 5-64.2	
C 5-69 through C 5-70	C 5-69 through C 5-70	
C 5-83 through C 5-84	C 5-83 through C 5-84	
C 5-99 through C 5-100.1	C 5-99 through C 5-100.1	

Appendix A

C A-5 through C A-6	C A-5 through C A-6	

Volume C
Program
Input

K6.2
Updates

Old Format

The list of items using the OLD format can be specified in three different forms, depending on user convenience:

1. A range of items may be specified as:

   ```
   l m n
   ```

   which implies items l through m by n; if n is not specified, it is taken as 1. Note that the range can be either increasing or decreasing.

2. A list of items may be specified as:

   -n a_1 a_2 a_3 .... a_n

   which implies that n items are to be given, and they are a_1 , ... a_n .

3. A setname may be specified as:

   ```
   MYSET
   ```

   which implies that all items previously specified to be in the set MYSET are to be used. The items in a set are specified using the DEFINE option.

New Format

The set of items using the NEW format can be expressed as a combination of one or more subsets. These subsets can be specified in three different forms, depending on user convenience. The operations that can be performed between subsets are as follows:

```
AND
INTERSECT
EXCEPT
```

In forming a set, subsets are combined in binary operations going from left to right. Hence, a set may be formed as:

1. SUBLIST1 AND SUBLIST2

   which implies all items in `SUBLIST1` and `SUBLIST2`. Duplicate items will be eliminated and the resultant set will be sorted.

2. SUBLIST1 INTERSECT SUBLIST2

   which implies only those items occurring both in `SUBLIST1` and `SUBLIST2`.

3. SUBLIST1 EXCEPT SUBLIST2

   which implies all items in `SUBLIST1` except those which occur in `SUBLIST2`.

4. SUBLIST1 AND SUBLIST2 EXCEPT SUBLIST3 INTERSECT SUBLIST4

   which implies take the items in `SUBLIST1` and `SUBLIST2` and remove those items that occur in `SUBLIST3`. Then, if these items also occur in `SUBLIST4`, include them in the set.

The SUBLISTS can have the form:

1. A range of items may be specified as:

   ```
   l TO m BY n
   ```

   or

   ```
   1 THROUGH M BY n
   ```

   which implies items l through m by n; if BY n is not included, it is assumed to be by 1. Note that the range can be either increasing or decreasing.

2. A string of items can be specified as:

   a_1 a_2 a_3 ... a_n

   which implies that n items are to be included. If continuation cards are necessary then a "C" or CONTINUE should be the last item on the card.

3. A setname may be specified as:

   ```
   MYSET
   ```

   which implies that all items previously specified to be in the set MYSET are to be used. The items in a set are specified using the DEFINE option.

Sorted vs. Unsorted Lists

In the MARC program, most lists are sorted lists. That is, regardless of the order of the list items on the list card, the MARC program returns these items sorted from lowest to highest. Unsorted lists are required in several places, however. These places are:

1. List of nodes in the TYING option.
2. List of nodes in the SUPERINPUT option.
3. List of degrees of freedom in the FIXED DISP option.

When defining unsorted lists, the sublist connectors EXCEPT and INTERSECT may not be used. Setnames may be used as long as the sets themselves are unsorted. In the MARC program, degree of freedom sets are always unsorted. Unsorted node sets may be defined by using set type NDSQ (for "node sequence") rather than set type NODE (see DEFINE model definition option).

Chapter 2: Parameters

2.2 Analysis Types

ELASTIC

2.2.1 Elastic Analysis with Multi-Loads

Description

This parameter allows repeated back substitution on a series of load vectors. This option uses the decomposed stiffness matrix. Total loads must be input with the POINT LOAD, DIST LOADS, or CHANGE STATE/THERMAL LOADS blocks after END OPTION.

The decomposed stiffness matrix is only written once to the restart tape, following increment 0. When using the RESTART option with this parameter, always restart at increment 0.

When the ADAPTIVE meshing option is used in conjunction with this option, only the loads before the END OPTION (increment zero) will be considered. This load is then re-analyzed until the error criteria is satisfied.

NOTES: This card should *never* be used with any cards which flag nonlinear analysis or which change the stiffness matrix, e.g., the SCALE parameter card or the DISP CHANGE option.

If temperature dependent material properties are included, then a new assembly is performed (if temperature loading is on).

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1-10	1st	A	Enter the word ELASTIC.
11-15	2nd	I	Element storage parameter, to reduce storage in elastic analysis. Set to 1, so that creep, swelling, plastic, incremental strains, and plastic strain rates and incremental stresses are not stored. Set to 2, so that in addition to 1 above, strain energies, thermal strains, and elastic strains are not stored. Note, if you request these items on the post file and they are not stored, the information will be incorrect.

ADAPTIVE

2.2.2 Adaptive Mesh Refinement

Description

This parameter allows an adaptive meshing analysis to be performed. The error criteria used is specified through the ADAPTIVE model definition option.

In an elastic analysis, the program will iterate based upon the exitation given to satisfy an error tolerance. The ELASTIC parameter must be included.

In a nonlinear incremental analysis, the program will adapt the mesh at each increment, or user controlled frequency to improve the solution.

New elements will be created as described in *Volume A*.

This capability may be used with lower order linear, triangular, quadrilateral, tetrahedral, and brick elements.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1-8	1st	A	Enter the word ADAPTIVE.
11-15	2nd	I	Enter an upper bound to the number of elements in the mesh.
16-20	3rd	I	Enter an upper bound to the number of nodes in the mesh.
21-25	4th	I	Enter a 1 to continue to perform an incremental analysis if the number of nodes or elements created would exceed the maximums specified. The previous mesh will be used.

LINEAR

2.2.3 Matrices Saved for Linear Analysis

Description

This parameter allows additional values to be stored rather than being recalculated during subsequent increments. This means an increase in the overall size of the workspace used for the problem, but may actually result in a reduced computation time. The efficiency of this parameter is highly dependent upon the analysis data and the machine on which the problem is computed. It has proven very effective in reducing computation time for linear elastic and small displacement dynamic problems. When set to 0, the parameter has also been used effectively on nonlinear problems such as rigid plastic flow.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1-10	1st	A	Enter the word LINEAR.
11-15	2nd	I	Enter 0 (default) to save BETA matrix (Strain-Displacement). Enter 1 to save the BETA matrix and the stress-strain law.

FOLLOW FOR

2.2.13 Follower Forces: Distributed Load Application on Current Geometry - Application of Total Boundary Condition

Description

The FOLLOW FOR parameter is used for follower force (e.g., pressure) problems. When this parameter is used, all distributed loads are formed on the basis of current geometry. This parameter requires the use of the residual load correction and, therefore, forces the use of that option regardless of other parameter cards (i.e., the NO LOADCOR parameter is ignored).

Whenever FOLLOW FOR is used, the distributed load magnitude given in user subroutine FORCEM must be the total magnitude to be reached after the current increment, and not the incremental magnitude. If the loads are specified, incremental values are given as usual.

In a coupled thermal-stress analysis the fluxes will be based upon the current geometry.

Boundary conditions in structural analysis are normally entered as incremental values. To specify total values, use the third field of this option

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1-10	1st	A	Enter the words FOLLOW FOR.
11-15	2nd	I	Enter a 1 if follower force stiffness is not required (default). Enter a 2 if follower force stiffness is to be included. Enter a 3 if the follower force is based upon the displacement at the beginning of the increment, as opposed to the last iteration. Enter a -1 if the undeformed geometry is required but total values of load are to be used.
16-20	3rd	I	Enter a 1 if total values of boundary conditions are to be entered on DISP CHANGE, POINT LOADS, and DIST LOADS option as opposed to the default incremental loads.

NOTE: If the follower force stiffness is included the use of the SOLVER option may be used to specify a nonsymmetric formulation. This improves convergence, but will result in longer solver times.

Follower force stiffness is available for element types (3, 7, 10, 11, 18, 72, 75, 80, 82, 84, 114, 115, 116, 117, 118, and 119).

BUCKLE

2.2.14 Buckling Load Estimation via Eigenvalue Analysis

Description

This parameter specifies the use of elastic buckling load estimation by eigenvalue analysis, based on a perturbation of the tangent stiffness. Multiple eigenvectors are allowed for the case where the closest root to the current load set is not pertinent. Power sweep with Gram-Schmidt orthonormalization is the algorithm used. The BUCKLE history definition option or the BUCKLE INCREMENT modal definition option controls the eigenvalue extraction. The RECOVER history definition option allows for modal stress recovery or storing eigenvectors on post tape.

For additional information about this type of analysis, see *Effective Use of the Incremental Stiffness Matrices in Nonlinear Geometric Analysis* by Dupuis, Pfaffinger and Marcal in the *MARC Reference Library, Volume F.*

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1-6	1st	A	Enter the word BUCKLE.
11-15	2nd	I	Maximum number of buckling modes to be estimated at any time.
16-20	3rd	I	Number of buckling modes with positive eigenvalues to be estimated at any time. In many buckling problems, collapse modes corresponding to loads of opposite magnitude to those of interest exist. By specifying a larger number of modes (say 5) in Columns 11-15 and one or two modes in this field, the user can ensure he will get the one or two modes he is interested in. The program will stop the modal search when all these modes have been formed, or when all the modes requested in columns 11-15 have been formed, whichever occurs first. If this field is left blank, all modes asked for in columns 11-15 will be formed regardless of sign.
21-25	4th	I	Enter 1 if modal stress recovery or storing eigenvectors on post tape is to be performed in this analysis.
26-30	5th	I	Enter a 3 to perform non-axisymmetric Fourier buckling.

2.2.20 Heat Transfer (Convection) Analysis

Description

This parameter specifies a heat transfer (convection) analysis instead of displacement/stress analysis. For more information about heat transfer capabilities in the MARC program, see the *MARC Reference Library, Volume A*.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1-4	1st	A	Enter the word HEAT.
11-15	2nd	I	Temperature distribution in thickness direction of heat transfer shell elements 50, 85, 86, 87, and 88. Enter 0 for linear temperature distribution in thickness direction. Enter 1 for quadratic temperature distribution in thickness direction. Default is 0.
16-20	3rd	I	Maximum number of latent heats associated with any material type. Default is 0.
21-25	4th	I	Enter 1 to obtain gradients and fluxes at integration points. Enter 2 to obtain in addition external flux values at nodal points. If this field is left blank, only temperatures at integration points and nodal temperatures will be printed.
26-30	5th	I	Enter a 2 to include convective terms. This will automatically initiate the nonsymmetric solver.

JOULE

2.2.21 Joule Heating (Coupled Thermo-Electrical) Analysis

Description

This parameter allows the user to perform a coupled thermo-electrical (Joule heating) problem. The coupling between the electrical problem and the thermal problem is because: (1) the resistance in the electric problem is dependent on temperatures and (2) the internal heat generation in the thermal problem is a function of the electric flow. For more information about the finite element formulation of Joule heating problems see the *MARC Reference Library, Volume A*.

In the analysis of Joule heating, the model definition cards JOULE, VOLTAGE, DIST CURRENT, and POINT CURRENT must be used for the definition of electric problems. However, options for the heat transfer analysis remain unchanged.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1-5	1st	A	Enter the word JOULE.
11-15	2nd	I	Enter a 1 for conventional model Enter a 2 if electrical properties are a strong function of temperature.

2.2.26 Acoustic Analysis

Description

This parameter activates the capability to perform an acoustic analysis in a cavity with rigid reflecting surfaces. Model definition option ISOTROPIC is used to prescribe the material behavior of the fluid. Model definition options FIXED PRESSURE, DIST SOURCES, and POINT SOURCE are used to prescribe boundary conditions. The program will calculate the fundamental frequencies when the MODAL SHAPE option is encountered. A transient analysis may be performed using the DYNAMIC CHANGE option.

For more information about the acoustic analysis capability in the MARC program, see the *MARC Reference Library, Volume A*. For information about elements used in acoustic analysis, see *Volume B*.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1-10	1st	A	Enter the word ACOUSTIC.
11-15	2nd	I	Maximum number of modes to be used in the modal superposition dynamic option. If the inverse power sweep method is used for eigenvalue analysis, it is also the number of mode shapes and frequencies to be extracted.
16-20	3rd	I	Flag to indicate the Lanczos method will be used. Set to 1 to force the Lanczos method.
21-25	4th	I	Enter 1 if modal stress recovery or storing eigenvectors on post tape is to be performed in this analysis.

RADIATION

2.2.27 Radiation Analysis

Description

This parameter activates the radiation analysis capability in the heat transfer option for the analysis of thermal radiation between axisymmetric bodies or in cavities.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1-9	1st	A	Enter the word RADIATION.
11-15	2nd	I	View factor calculation flag (IRADFL). Set to 0 to calculate view factors during analysis. Set to 1 to read view factors from a file. Default is 0.
16-20	3rd	I	File number for view factors. View factors will be written if IRADFL=0, and will be read if IRADFL=1. When IRADFL=0 and IFILVF=0, the view factors will not be saved. Default is 0.
21-25	4th	I	Temperature unit flag. Set to 1 if user input is in degrees Celsius. Set to 2 if user input is in degrees Kelvin. Set to 3 if user input is in degrees Fahrenheit.

CAUTION: Do not enter temperatures in degrees Rankine.

26-35	5th	F	Enter scale factor for power and length unit conversions for Setfan-Boltzmann constant calculation, σ=5.6696E-8(W/M^2-K^4)/(SCALNT)^2.
36-40	6th	I	Number of division used inview factor calculation. Default is 3.
41-45	7th	I	Number of Gauss points used in each subdivion. Default is 3.
46-50	8th	I	Enter unit number for debug printout.
51-55	9th	I	Enter a 1 for alternative (K4) method for calculation of view factors.

Chapter 3: Model Definition Options

ADAPTIVE

3.2.16 Define Error Criteria Used in Adaptive Analysis

Description

This model definition set controls the error criteria for adaptive meshing. The ADAPTIVE parameter option must also be included.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-8	1st	A	Enter the word ADAPTIVE.
2nd card series			
1-5	1st	I	Enter the number of criteria to use.
6-10	2nd	I	Enter the unit number to read data, defaults to input.
11-15	3rd	I	Enter the frequency to perform adaptive meshing, default is every increment.
16-20	4th	I	Enter the unit number to which the adaptive mesh data will be written.

The third and fourth card series are repeated in pairs for each criteria selected.

3rd card series

1-5	1st	I	Enter the criteria type: 1: Mean Strain Energy subdivide element if: element strain energy > f_1 * total strain energy/ NUMEL f_2 to f_6 is not used 2: Zienkiewicz-Zhu Error Criteria The error norm is defined as

$$\pi = \frac{\int (\sigma^* - \sigma)^2 dV}{\int \sigma^2 dV + \int (\sigma^* - \sigma)^2 dV} \quad \text{or}$$

$$\Upsilon^2 = \frac{\int (E^* - E)^2 dV}{\int E^2 dV + \int (E - E^*)^2 dV}$$

The stress error and strain energy errors are

$$X = \int (\sigma^* - \sigma)^2 dV \text{ and } Y = \int (E^* - E)^2 dV$$

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

The allowable element stress error is
AES = f_2 * X/NUMEL + f_3 * X * f_1/π/NUMEL
The allowable element strain energy error is
AEE = f_4 * Y/NUMEL + f_5 * X * f_1/γ/NUMEL
where
NUMEL is the number of elements in the mesh.
If f_2, f_3, f_4, f_5 are input as zero f_2 = 1.0.
If stress error testing is to be performed, $f_1 \neq 0$ and f_2 and/or $f_3 \neq 0$, $f_4 = 0$, $f_5 = 0$.
The element will be subdivided when:
$\pi > f_1$ and $X_{e1} >$ AES.
If strain energy error testing is to be performed.
$f_1 \neq 0$ and $f_2 = 0$, $f_3 = 0$, $f_4 \neq 0$, and/or $f_5 \neq 0$.
The element will be subdivided when:
$\gamma > f_1$ and Y_{e1} > AEE
The default is f_2 = 1.0 if f_2, f_3, f_4, f_5 are input as 0.0.
It is advisable that $f_2 + f_3 \approx 1$ or $f_4 + f_5 \approx 1.0$.

3: Stress Discontinuity
(not yet implemented)

4: Location within Box
subdivide element if at least one of the nodes:
$f_1 < x < f_2$ and $f_3 < y < f_4$ and $f_5 < z < f_6$

5: Node in Contact
subdivide element if at least one of the nodes is in contact or belongs to a segment which is contacted
f_1 to f_6 is not used, enter 0 or blank

6: Aspect Ratio
(not yet implemented)

7: Skewness Ratio
(not yet implemented)

8: Solution Gradient (used only for heat transfer)
subdivide element if:
gradient/maximum gradient > f_1
typical value of f_1 = 0.75 (f_1 must be < 1.0)
f_2 to f_6 is not used, enter 0 or blank

9: Equivalent stress, strain
subdivide element if:
von Mises stress > f_1 * maximum von Mises stress
or
von Mises stress > f_2
or

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
			Equivalent strain > f_3 * maximum equivalent strain or Equivalent strain > f_4 Equivalent plastic strain > f_5 maximum equivalent plastic strain or Equivalent plastic strain > f_6
			10: User Subroutine UADAP subdivide element if: user/maximum user > f_1 or user > f_2 f_3 to f_6 is not used, enter 0 or blank user is defined in user subroutine uadap
			11: Previously Defined Mesh Use the refined mesh from a previous analysis as the starting point to this analysis.(see 4th card series).
6-10	2nd	I	Enter the maximum number of levels to adapt an element.
11-22	3rd	A	Enter the name of the element set that will use this criteria. Default is to apply the adaptive criteria to all elements.

4th card series (except for criteria type 11)

1-10	1st	E	First parameter f_1
11-20	2nd	E	Second parameter f_2
21-30	3rd	E	Third parameter f_3
31-40	4th	E	Fourth parameter f_4
41-50	5th	E	Fifth parameter f_5
51-60	6th	E	Sixth parameter f_6

4th card series (criteria type 11)

Include the data file written by the previous analysis (the unit number was specified on the second card series).

ATTACH NODE

3.2.17 Define the Nodes Which are Attached to Surfaces for Adaptive Mesh Capability

Description

This option allows one to attach nodes to a curve or surface. This option is used in conjunction with the SURFACE option in an adaptive mesh analysis. If two points on an edge of an element are attached to a curve or surface, and new points created by the adaptive procedure will be placed on the curve. This improves the geometric modeling.

NOTE: In the case of updated Lagrange, or if no surface is defined, the new nodes are placed midway between the previous nodes.

A node may be attached to as many as three surfaces, any additional surfaces will be ignored.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word ATTACH NODE.
2nd card series			
1-5	1st	I	Enter the number of data sets.
6-10	2nd	I	Enter the unit number to read data, defaults to input.
3rd card series			
1-5	1st	I	Enter curve/surface identifier.
4th card series			

Enter a list of nodes which are attached to this curve or surface.

3.2.18 Define the Curves and Surfaces used to Specify Coordinate Data

Description

This option allows the definition of a curve for one or two dimensional problems and a surface for three dimensional problems. These geometrical entities will be used with the ADAPTIVE option for elastic analyses. The original nodal points may be associated (ATTACH option in Mentat II) with these geometric entities. In such cases when new nodes are created during the adaptive process they will automatically be placed on the curve or surface. This improves the geometric modeling.

NOTE: If no surface is defined, the new nodes are placed midway between the master nodes.

A node may be attached to as many as three surfaces, any additional surfaces will be ignored.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-6	1st	A	Enter the word SURFACE.
2nd card series			
1-5	1st	I	Enter the number of curves to be defined.
6-10	2nd	I	Enter the unit number to read geometric information, defaults to input.
3rd card series			
1-5	1st	I	Curve/Surface identifier
6-10	2ndt	I	Enter the curve or surface type: 1: 2-D Polyline 2: 2-D Circular Arc 3: 2-D Circle 4: 2-D NURB Curve (full description) 5: 2-D NURB Curve (internally generate) 6: Plane 7: Sphere 8: Cylinder 9: NURB Surface (full description) 10: NURB Surface (internally generate)

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
A. FOR 2-D POLYLINE			
4th card series			
1-5	1st	I	Number of points on polyline.
5th card series			
Enter the coordinate points, one per line.			
1-10	1st	I	X-coordinate.
11-20	2nd	I	Y-coordinate.
B. FOR CURVE TYPE CIRCULAR ARC			
4th card series			
1-10	1st	E	X-coordinate of starting point.
11-20	2nd	E	Y-coordinate of starting point.
21-30	3rd	E	X-coordinate of ending point.
31-30	4th	E	Y-coordinate of ending point.
41-50	5th	E	X-coordinate of center.
51-60	6th	E	Y-coordinate of center.
61-70	7th	E	Radius.
C. FOR CURVE TYPE CIRCLE			
4th card series			
1-10	1st	E	X-coordinate of center.
11-20	2nd	E	Y-coordinate of center.
21-30	3rd	E	Radius.
D. FOR CURVE TYPE 2-D NURB - FULL DESCRIPTION			
4th card series			
1-5	1st	I	Number of control points (NPU).
6-10	2nd	I	Order of NURB (NOU).
5th card series			
Enter NPU homogeneous coordinates in E10 format (8 per line) use as many lines as necessary. The homogeneous coordinates are between 0 and 1.			
6th card series			
Enter (NPU plus NOU) knot vectors in E10 format (8 per line) use as many lines as necessary. The components of the knot vector are between 0 and 1.			

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

7th card series

Enter the physical coordinates of control points - 1 control point per line (2 coordinates). There should be NPU lines.

1-10	1st	E	X-coordinate of control point.
11-20	2nd	E	Y-coordinate of control point.

E. FOR CURVE TYPE 2-D NURB - INTERNALLY GENERATED

4th card series

1-5	1st	I	Number of control points (NPU). Minimum is 4.

5th card series

Enter the physical coordinates of control points - 1 control point per line (2 coordinates). There should be NPU lines.

1-10	1st	E	X-coordinate of control point.
11-20	2nd	E	Y-coordinate of control point.

F. FOR PLANE

4th card series

Enter coordinates at four points, one point per line.

1-10	1st	E	X-coordinate.
11-20	2nd	E	Y-coordinate.
21-30	3rd	E	Z-coordinate.

G. FOR CURVE TYPE SPHERE

4th card series

1-10	1st	E	X-coordinate of center.
11-20	2nd	E	Y-coordinate of center.
21-30	3rd	E	Z-coordinate of center.
31-40	4th	E	Radius.

H. FOR CURVE TYPE CYLINDER/CONE

4th card series

1-10	1st	E	X-coordinate of starting point on axis.
11-20	2nd	E	Y-coordinate of starting point on axis.
21-30	3rd	E	Z-coordinate of starting point on axis.
31-40	4th	E	Radius at starting point.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
41-50	5th	E	X-coordinate of ending point on axis.
51-60	6th	E	Y-coordinate of ending point on axis.
61-70	7th	E	Z-coordinate of ending point on axis.
71-80	8th	E	Radius at ending point.

I. FOR CURVE TYPE 3-D NURB - FULL DESCRIPTION

4th card series

1-5	1st	I	Number of control points in first direction (NPU).
6-10	2nd	I	Order of NURB in first direction (NOU).
11-15	3rd	I	Number of control points in second direction (NPV).
16-20	4th	I	Order of NURB in second direction (NOV).

5th card series

Enter NPU times NPV homogeneous coordinates in E10 format (8 per line) use as many lines as necessary. The homogeneous coordinates are between 0 and 1.

6th card series

Enter (NPU plus NOU) plus (NPV plus NOV) knot vectors in E10 format (8 per line) use as many lines as necessary. The components of the knot vector are between 0 and 1.

7th card series

Enter the physical coordinates of NPU times NPV control points 1 control point per line (3 coordinates). There should be NPU times NPV lines.

1-10	1st	E	X-coordinate of control point.
11-20	2nd	E	Y-coordinate of control point.
21-30	3rd	E	Z-coordinate of control point.

J. FOR CURVE TYPE 3-D NURB - INTERNALLY GENERATED

4th card series

1-5	1st	I	Number of control points in first direction (NPU). Minimum is 4.
6-10	2nd	I	Not used enter a 0.
11-15	3rd	I	Number of control points in second direction (NPV). Minimum is 4.

5th card series

Enter the physical coordinates of NPU times NPV control points 1 control point per line (3 coordinates). There should be NPU times NPV lines.

1-10	1st	E	X-coordinate of control point.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
11-20	2nd	E	Y-coordinate of control point.
21-30	3rd	E	Z-coordinate of control point.

GEOMETRY

3.2.19 Specify Geometrical Data

Description

The element geometry is specified in distinct sets. The information required varies from one element type to another. As a rule, the thickness of shell elements is given in the first variable (EGEOM1) to be defined. The geometry for a particular element may be defined repeatedly and only the last data is used. This feature is designed for local variations of geometric data.

The geometry block is unnecessary if the element description does not require either EGEOM1, EGEOM2, or EGEOM3. (See *Volume B*).

Note that the NODAL THICKNESS model definition block can also be used for the input of beam/shell thickness.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-8	1st	A	Enter the word GEOMETRY.
2nd card series			
1-5	1st	I	Number of distinct sets of element geometries to be input (optional).
6-10	2nd	I	Enter tape number for input of geometry defaults to input.
3rd card series			
Element geometries. The third and fourth cards are entered as pairs, one for each distinct data set.			
1-10	1st	F	EGEOM1
11-20	2nd	F	EGEOM2
21-30	3rd	F	EGEOM3
31-40	4th	F	EGEOM4
41-50	5th	F	EGEOM5
51-60	6th	F	EGEOM6
			See library element descriptions in Quick Reference of Volume B for the meaning of EGEOM1, etc. for each element type.

TRANSFORMATION

3.2.21 Define Nodal Coordinates for Transformation

Description

This option defines nodal coordinates for calculation of a direction cosine matrix, which is then used for transforming the global degrees of freedom of a specified node to a new local coordinate system.

This block may be used to set up local coordinate systems at a number of flagged nodes, for application of boundary conditions in a transformed system, or for printout purposes. Five points should be noted:

1. The displacements and loads or reactions will be output in the transformed system at such nodes.
2. The transformation will be done on all Cartesian displacements. Thus, for the shell elements, the derivative degrees of freedom become the derivative of the transformed displacements with respect to the original surface coordinate system.
3. Transformations are assumed to be orthogonal.
4. All kinematic conditions such as boundary conditions and ties at that node must be input in the transformed system.
5. All concentrated nodal loads must be applied in the transformed system.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-14	1st	A	Enter the word TRANSFORMATION.
2nd card series			
1-5	1st	I	Number of distinct sets of transformations data to be entered (optional).
6-10	2nd	I	Enter unit number for input of transformation data, defaults to input.
11-15	3rd	I	Enter 1 to suppress printout of transformation data.

The third and fourth cards are entered as pairs, one for each distinct data set.

3rd card series			
1-5	1st	I	Node number. Enter 0 to read a list of nodes. See 4th card series.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
6-15	2nd	F	Global coordinates of a first point A such that the vector from this point to the node is direction 1 of the local coordinate system. (See Figure 3-1a.)
16-25	3rd	F	
26-35	4th	F	
36-45	5th	F	Global coordinates of a second point, such that this point, the first point, and the node define the 1-2 plane of the local coordinate system.
46-55	6th	F	
56-65	7th	F	

NOTE: Direction 2 of the local coordinate system will be constructed perpendicular to direction 1 such that this second point has a positive 2 coordinate in the local 1-2 plane (see Figure 3-1b).

Direction 3 of the local coordinate system is given by a cross product of direction 1 with direction 2.

4th card series

Include only if the first field in the 3rd card series is 0.

Enter a list of nodes for which the above transformation is applied.

Note that for elements in a plane (e.g.; 1, 2, 3, 5, 6, 10, 11, 12, 15, 16, 17, 19, etc.) only the first two coordinates of the first point (columns 6-15 and 16-25) need be supplied.

See Figure 3-1 on the following page.

ERROR ESTIMATE

3.2.32 Create Error Estimation

Description

The user can request that the program give information regarding the error associated with the finite element discretization. There are two measures, the first evaluates the stress discontinuity between elements. A large value implies that the stresses gradients are not accurately represented in the finite element mesh. In a classical linear elastic solution, this could be resolved by choosing quadratic elements over linear elements or refining the mesh.

The second error measure examines geometric distortion in the model. It first examines the aspect ratios and warpage of the elements and in subsequent increments measures how much these ratios change. This measure can be used to indicate if the original mesh is good and whether at a later time rezoning is required.

The evaluation of the stress error measure is moderately expensive. The evaluation of the geometric error measure is very inexpensive.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the words ERROR ESTIMATE.
2nd card series			
1-5	1st	I	Enter a 1 if the stress measure is to be evaluated.
6-10	2nd	I	Enter a 1 if the geometric measure is to be evaluated.

USDATA

3.2.33 Invoke USDATA User Subroutine for Initialization

Description

This option invokes the call to user subroutine USDATA for the initialization of user variables. These variables (data) is stored in a common block MYDATA that may be used in other user subroutines. This option requires the definition of the amount of data in the common block in REAL*4 words. The data will be automatically saved on the restart file for use in subsequently analysis.

Format

FORMAT		DATA	
FIXED	FREE	TYPE	ENTRY
1st card series			
1-6	1st	A	Enter the word USDATA.
11-15	2nd	I	Enter the number of real*4 words to be defined in common block MYDATA via user subroutine USDATA.

3.3.2 Specify Direct or Iterative Solver

Description

This option defines the solver to be used in the analysis. You can specify either the direct or iterative solver. The choice of whether the in-core or out-of-core procedure is used is automatically determined by the program, based upon the amount of workspace required and the number given on the SIZING option. You can also select whether the symmetric or nonsymmetric solver is used. At this time, only a direct, nonsymmetric solver is available. Additionally, you can specify if the solution of a non-positive definite system is to be obtained.

When the iterative solver is chosen, additional parameters must be defined which are used to control the accuracy.

NOTE: It is not recommended to use the iterative solvers for beam or shell models, because these problems are ill-conditioned, resulting in a large-number of iterations. For a well-conditioned system, the number of iterations should be less than (and possible much less than) the square root of the total number of degrees of freedom in the system.

NOTE: The user controls the maximum number of iterations allowed. If this is a positive number, the program will stop if this is exceeded. If this is a negative number, the program will print a warning and continue to the next Newton-Raphson iteration or increment.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word SOLVER.
2nd card series			
1-5	1st	I	0 = Profile Direct Solver. 1 = EBE Iterative. 2 = Sparse Iterative. 3 = Hardware Provided. 4 = Sparse Direct Solver
6-10	2nd	I	Enter 1 if the nonsymmetric solver is to be used.
11-15	3rd	I	Enter 1 if the solution of non-positive definite system is to be obtained.
3rd card series			
Only necessary if EBE iterative solver is used.			
1-5	1st	I	Enter maximum number of groups; defaults to 30 times the number of element types.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
6-10	2nd	I	Enter 1 if the previous solution is to be used as the initial trial solution.
11-15	3rd	I	Enter maximum number of conjugate-gradient iterations. Default is 1000, see note on previous page.
16-20	4th	I	Enter the type of preconditioner: Enter 0 for no preconditioner. Enter 1 for incomplete Cholesky preconditioner.

4th card series

Only necessary if EBE iterative solver is to be used.

1-10	1st	F	Enter tolerance on conjugate gradient convergence for stress analysis. Default is 0.01.
11-20	2nd	F	Enter tolerance on conjugate gradient convergence for heat transfer analysis.
21-30	3rd	F	Condition number cut-off. If calculated condition number is less than this value, the analysis will be stopped.

3rd card series

Only necessary if the sparse iterative solver is used.

1-5	1st	I	Enter maximum number of conjugate-gradient iterations. Default is 1000, see note on previous page.
6-10	2nd	I	Enter 1 if the previous solution is to be used as the initial trial solution.
11-15	3rd	I	Enter a 3 for diagonal preconditioner. Enter a 4 for scaled-diagonal preconditioner. Enter a 5 for incomplete Cholesky preconditioner.

4th card series

Only necessary if the sparse iterative solver is to be used.

1-10	1st	F	Enter tolerance on conjugate gradient convergence for stress analysis. Default is 0.001.

Table 3-3 POST Codes (Continued)

Codes	Description	
431-436	Plastic strain in global coordinate system tensor	
441-446	Creep strain in global coordinate system tensor	
Post Codes for Heat Transfer Analysis		
9 or 180	Total temperature	
181-183	Components of temperature gradient T	
184-186	Components of flux	
Post Codes for Bearing Analysis		
190	Pressure	
191-193	Components of pressure gradient	
194-196	Mass flux vector	
Post Codes for Joule Heating Analysis		
87	Voltage	
88	Current	
89	Heat generated	
Post Codes for Acoustic Analysis		
190	Pressure	
191-193	Components of pressure gradient	
Post Codes for Electrostatic Analysis		
130	Electric potential	(V)
131-133	Components of electric field intensity	(E)
134-136	Components of electric displacement	(D)
Post Codes for Magnetostatic Analysis		
140	Magnetic potential (2-D analysis only)	(A_z)
141-143	Components of magnetic induction	(B)
144-146	Components of magnetic field intensity	(H)
Post Codes for Transient Electromagnetic Analysis		
131-133	Components of electric intensity	(E)
137-139	Components of Lorentz force	
134-136	Components of electric displacement	(D)
141-143	Components of magnetic induction	(B)
144-146	Components of magnetic field intensity	(H)

Table 3-3 POST Codes (Continued)

Codes	Description	
147-149	Components of current density	(J)
Post Codes for Harmonic Electromagnetic Analysis		
131-133	Real components of electric intensity	(E)
134-136	Real components of electric displacement	(D)
137-139	Real components of Lorentz force	
141-143	Real components of magnetic induction	(B)
144-146	Real components of magnetic field intensity	(H)
147-149	Real components of current density	(J)
151-153	Imaginary components of electric field intensity	(E)
154-156	Imaginary components of electric displacement	(D)
157-159	Imaginary components of Lorentz force	
161-163	Imaginary components of magnetic induction	(B)
164-166	Imaginary components of magnetic field intensity	(H)
167-169	Imaginary components of current density	(J)
Post Codes for Soil Analysis		
171	Porosity	
172	Void ratio	
173	Pore pressure	
174	Preconsolidation pressure	

Note that codes 1-6 are the generalized strains, and are section (not layer) quantities, so that the layer number need not be given.

Codes 11-16 give generalized stress quantities if no layer number is given for shell analysis. If a layer number is given, these are physical layer quantities.

For heat transfer, code 9 is used for all heat transfer elements.

When using shells in heat transfer, it is important to enter a code for each layer in chronological order if post file is to be correctly read by the INITIAL STATE or CHANGE STATE options.

Note that one does not need to select nodal values (i.e. displacement, velocities and accelerations, and temperature for a heat transfer run) as these are automatically written to the post file.

Eigenmodes (dynamic analysis) and eigenvectors (buckling analysis) are written to the post file only if indicated by the RECOVER or modal INCREMENT /BUCKLE INCREMENT option.

3.3.6 SDRC Interface

Description

This option allows the user to generate an Universal file which is compatible with the SDRC I-DEAS™ program. Two types of quantities may be on this file, element and nodal quantities. The element quantities stresses, strains, etc., can either be component values or invariant values or both. They are an average value within the element. The nodal quantities are values extrapolated from the integration points and a weighted average is calculated. Extreme care should be used interpreting the results using beam and/or shell elements.

Format

1st card series

1-4	1st	A	Enter the word SDRC.
11-15	2nd	I	Enter the unit number to which to write file, default is 40.
16-20	3rd	I	Frequency to write out file.

If no element date is required, skip to the 4th card series.

2nd card series

1-10	1st	A	Enter the word ELEMENT.

Repeat 3rd card series as often as required.

3d card series

1-5	1st	I	Enter a: 1 for stresses. 2 for total strains. 3 for creep strains. 4 for thermal strains. 5 for plastic strains. 6 for strain energy. 7 for stress/yield stress. 8 for failure indexes.
6-10	2nd	I	Enter a layer number if shell elements. IL If the value in the first field is a 2, no layer number is required.

If no nodal data is required, skip to the next model definition point.

4th card series

1-8	1st	A	Enter the word NODAL.

Repeat 5th card series as often as required

5th card series

1-5	1st	I	Enter a: 1 for displacements. 2 for velocities. 3 for acceleration. 4 for reaction forces. 5 for temperatures. 6 for generalized stresses. 7 for generalized strains. 8 for top/middle/bottom layer stresses. 9 for top/middle/bottom elastic strain. 10 for top/middle/bottom plastic strain. 11 for top/middle/bottom creep strain. 13 for eigenmode. 14 for harmonic displacement and reaction.

NOTE: If shell elements are not included in the model, specifying 8, 9, 10, or 11 will result in only the top or actual component.

PRINT ELEMENT

3.3.7 Specify Elements to be Included in Output

Description

This option allows the user to choose which elements, and what quantities associated with an element are to be printed. If the user does not specify NODE on the first card series, these values are at the integration points. If the user specifies the word NODE, these values are the extrapolated nodal values.

> **NOTE:** This option revokes any NO PRINT that precedes it. Therefore, NO PRINT followed by PRINT ELEM and not followed by PRINT NODE will result in the selected element printout and full nodal printout. Use PRINT NODE with a blank node list to suppress node output.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the words PRINT ELEMENT.
11-20	2nd	A	Enter the word NODE (optional).
2nd card series			
1-5	1st	I	Enter the number of sets to be given below (optional).
6-10	2nd	I	Increment between printout. Default is print every increment.
11-15	3rd	I	File unit to which output is to be written. Defaults to standard output, unit 6.

Card series 3, 4, and if necessary, 5 and 6 are given once for each data set.

3rd card series			
1-80	1st	A	Enter one or more of the following: STRAIN output total strain. STRESS output total stress. PLASTIC output plastic strain. CREEP output creep, swelling and viscoelastic strain. THERMAL output thermal strain

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY	
			ENERGY	output of strain energy densities: • total strain energy • incremental total strain energy • total elastic strain energy • incremental elastic strain energy • plastic strain energy • incremental plastic strain energy
			CRACK	output of cracking strain
			CAUCHY	output Cauchy stress.
			STATE	output state variables.
			PREFER	output stresses in preferred system.
			ELECTRIC	output electric field and electric flux
			MAGNETIC	output magnetic field and magnetic flux
			CURRENT	output current
			ALL	output of all of the above.

4th card series

Enter a list of elements to be printed.

NOTE: To suppress all element print-out, enter a blank list for the list of elements.

5th card series

Enter a list of integration points to be printed, or a list of nodes to be printed if the NODE option is specified on the 1st card. This node list is a list of internal element node numbers (ranging from 1 to the maximum number of nodes per element).

6th card series

Enter a list of layers to be printed. This is only necessary if there are either thin walled beam or shell elements in the mesh, (that is, element types 1, 4, 5, 8, 13, 14, 15, 16, 17, 22, 24, 25, 45, 49, 50, 72, 75, 76, 77, 78, 79, 89, 90, 96, 97).

3.4.1 Control Option for Stress Analysis

Description

This option allows the user to input parameters governing the convergence and the accuracy for nonlinear analysis. For heat transfer analysis, see Section 3.9.

For coupled thermal-stress analysis, card series 4 must be used in addition to either card series 3a or 3b.

For nonlinear static analysis, the controls are described in *Volume A*. They do not appear on the restart tape, and so must be re-entered on a restart run. A description of the iteration procedures and convergence tests is given in *Volume A*.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st Card Series			
1-7	1st	A	Enter the word CONTROL.
2nd Card Series			
1-5	1st	I	Maximum number of load steps/increments in this run. Default is 9999. This is a cumulative number and is usually used to stop the run when restart is being used. If ELASTIC parameter card is included, this field is ignored and all load cases are analyzed.
6-10	2nd	I	Maximum number of recycles/increments during an increment for plasticity, or other tangent modulus nonlinearities. Default is 3. This should usually be increased to 10 for rigid-plastic flow option. If a negative number is entered, then the program will do a maximum of the absolute value entered. If convergence has not been obtained, a warning is given and the program proceeds to the next increment. This is not recommended.
11-15	3rd	I	Minimum number of recycles during an increment for plasticity or other tangent modulus nonlinearities. Default is 0. Note that this data field forces this number of recycles to take place at all subsequent increments. Caution, this value is overwritten by the PROPORTIONAL INCREMENT data block.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
16-20	4th	I	Flag for convergence testing. If set to 0 or left blank, testing is done on residuals. If set to one, testing is done on displacements. If set to two, testing is done on strain energy. Note that testing on displacements or strain energy always requires at least one iteration. If nonlinear analysis is done with the CENTROID option, the residuals are not calculated and testing is always done on displacements. Note that nonlinear analysis with the CENTROID option is not recommended.
21-25	5th	I	Flag to specify relative or absolute error testing. If equal to 0, testing is done on relative error. If equal to 1, testing is done on absolute value. If set to 2, testing is done on relative error testing unless reactions or incremental displacements are below minimum value than absolute tolerances, testing is used.
26-30	6th	I	Iterative procedure flag. 1. Full Newton-Raphson (default). 2. Modified Newton-Raphson (no reassembly during iteration). 3. Newton-Raphson with strain correction modification (see Volume A). 8. Secant method.
31-35	7th	I	Nonpositive definite flag. If set to 1, solution of nonpositive definite system is forced. Note that with use of gap and Herrmann elements the matrix always is nonpositive definite, and this entry has no significance.
36-40	8th	I	*Plasticity* Radial return flag. If set to 0, MARC standard mean normal method is used in the plasticity equations. If set to 1, radial return method will be used. Implicit Creep or Visco plasticity Set to 0 for elastic tangent. Set to 1 for secant Set to 2 for radial return (IMPLICIT CREEP only).
41-45	9th	I	To print convergence control messages to log file, enter a 1.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
46-50	10th	I	Control on initial stress stiffness. 0 Normal-full contribution. 1 Mooney or Ogden use deviatoric stress only. 2 No initial stress stiffness. 3 Use stress at beginning of increment, not last iteration.
3rd card series			
1-10	1st	F	If relative residual checking: Maximum allowable value of maximum residual force divided by maximum reaction force. Default is 0.10. If relative displacement checking: Maximum allowable value of the change in displacement increment divided by the displacement increment. Default is 0.10.
11-20	2nd	F	If relative residual checking: Maximum allowable value of maximum residual moment divided by maximum reaction moment. Default is 0.0, in which case, no checking on residual moment occurs. If relative displacement checking: Maximum allowable value of the change in rotational increment divided by the rotational increment. Default is 0.0, in which case, no checking on change in rotational increment occurs.
21-30	3rd	F	If relative residual checking: Minimum reaction force, if reaction force is less than this value, checking will be bypassed or absolute testing will be performed. If relative displacement checking: Minimum displacement, if displacement increment is less than this value, checking will be bypassed or absolute testing will be performed.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
31-40	4th	F	If relative residual checking: Minimum moment, if moment is less than this value, checking will be bypassed or absolute testing will be performed. If relative displacement checking: Minimum rotation, if rotation increment is less than this value, checking will be bypassed.
41-50	5th	F	If absolute testing: Maximum value of residual force. Default is 0.0 in which case, no checking on residual force will take place.
51-60	6th	F	If absolute testing: Maximum value of residual moment. Default is 0.0 in which case, no checking on residual moments will take place.
61-70	7th	F	$\dot{\bar{\varepsilon}}_o$ initial strain rate. (Rigid-Plastic Analysis only)
71-80	8th	F	$\dot{\bar{\varepsilon}}$ cutoff strain rate. (Rigid-Plastic Analysis only)

4th card series

Only necessary for COUPLED analysis.

1-10	1st	F	Maximum nodal temperature change allowed. Used to control automatic time step scheme for heat transfer. Default value of 20.
11-20	2nd	F	Maximum nodal temperature change allowed before properties are reevaluated and matrices reassembled. Default value of 100.
21-30	3rd	F	Maximum error in temperature estimate used for property evaluation. This control provides a recycling capability to improve accuracy in highly nonlinear heat-transfer problems (e.g., latent heat, radiation boundary conditions). Default is 0, which bypasses this test. Set to maximum temperature error which is considered acceptable.

INITIAL STATE

3.4.9 Initialize State Variables

Description

This option provides various ways of initializing the state variables throughout the model. The number of state variables per integration point is defined in the STATE VARS parameter card. The default is one, with temperature always being the first state variable at an integration point. If more than one state variable per integration point has been assigned, this option may be used repeatedly to initialize all the state variables. The default value of state variables not initialized is zero.

Four ways of providing the state variable initial values are shown below:

- Read the range of elements, integration points and layers and a corresponding state variable value.
- Read the initial values through user subroutine INITSV.
- Read the initial values from a step of the binary or formatted POST output file from a previous heat transfer analysis with the MARC program. This technique is most common for thermal stress analysis to initialize temperature (the first state variable at any point). With this option the program assumes direct correspondence of the POST file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by the user.
- Read a list of elements, integration points and layers and a corresponding state variable value.

NOTE: Initial temperature values read in by this option are assumed to define the stress-free temperature field. Temperature changes which cause thermal strains are read in through the CHANGE STATE or AUTO THERM options. In the current version of the MARC program, initial stress free temperatures may also be input on the PROPERTY option - the values used by the program are those last appearing in the input data.

In a coupled analysis, the temperatures are not independent state variables, and the INITIAL TEMPERATURE option must be used.

Format

FORMAT		DATA	
FIXED	FREE	TYPE	ENTRY
1st card series			
1-13	1st	A	Enter the words INITIAL STATE.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
2nd card series			
1-5	1st	I	Enter the state variable identifier for the state variable being set (1,2,etc.). 1=temperature. If more than one state variable is being used, the STATE VARS parameter card must be included.
6-10	2nd	I	Enter a 1 to initialize the state variable via card 3 series below. See also the third field on this card. Enter a 2 to initialize the state variable via user subroutine INITSV. This subroutine will now be called in a loop over all elements in the mesh. Enter a 3 to read the file values of the state variable from the POST tape written by a previous heat transfer analysis. In this case, the fourth and fifth fields must also be defined. Enter a 4 to initialize the state variable via card series 5, 6, 7, and is given below. See also the third field on this card.
11-15	3rd	I	Only nonzero if the second field is set 1 to or 4. Then this entry gives the number of pairs of cards in series 3 and 4 or in series 5, 6, 7, and 8 used to input the state variable.
16-20	4th	I	Only used if the second field is set to 3. Then this entry defines the unit number from which the POST file information from the previous heat transfer run is to be read. Defaults to unit 24 for a formatted post tape and to unit 25 for a binary post tape.
21-25	5th	I	Only used if the second field is set to 3. In that case, this entry defines the increment number on the heat transfer run POST file to be used as the definition of the initial state variable values.
26-30	6th		Not used, enter a zero.
31-35	7th	I	Set to 1 if option 3 is used, and a formatted POST file is used.
36-40	8th	I	Only nonzero if the second field is set to 2. Set to 1 to suppress printout of state variable values that are initialized in INITSV.
41-45	9th	I	Enter the post code number to be read into this state variable, default is 9 (temperature).

INITIAL TEMP

3.4.12 Define Initial Temperatures

Description

This option provides initial temperatures at nodal points for thermal stress problems.

> **NOTE:** For shell analyses, a uniform temperature will be used through the thickness direction.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the words INITIAL TEMP.
2nd card series			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional). Enter a 1 if user subroutine USINC will be used. In this case, card sets 3 and 4 are not used.
6-10	2nd	I	Enter tape number for input of prescribed temperatures data, defaults to input.
11-15	3rd	I	Flag to indicate that initial conditions are read from previously generated POST tape. Set to 1.
16-20	4th	I	Only nonzero if the third field is set to 1. Then this entry defines the unit number from which the POST tape information will be read.
21-25	5th	I	Enter step number to be read.
26-30	6th	I	Enter a 1 if a formatted POST tape is used.
Card series 3 and 4 are given in NSET pairs, only if the flag in the third field is not equal to 1.			
3rd card series			
1-10	1st	E	Initial temperature.
4th card series			
			Enter list of nodes for which the above initial temperature is applied.

POINT TEMP

3.4.13 Define Point Temperatures

Description

This option defines temperatures at nodal points for an uncoupled thermal stress problems at the end of the increment.

NOTE: For shell analyses, a uniform temperature will be used through the thickness direction.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the words POINT TEMP.
2nd card series			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional).
6-10	2nd	I	Enter tape number for input of prescribed temperatures data, defaults to input.
11-15	3rd	I	Flag to indicate that temperatures are read from previously generated POST tape. Set to 1.
16-20	4th	I	Only nonzero if the third field is set to 1. Then this entry defines the unit number from which the POST tape information will be read.
21-25	5th	I	Enter step number to be read.
26-30	6th	I	Enter a 1 if a formatted POST tape is used.

Card series 3 and 4 are given in NSET pairs, only if the flag in the third field is not equal to 1.

3rd card series			
1-10	1st	E	Temperatures at the end of the increment.
4th card series			
			Enter list of nodes for which the above initial temperature is applied.

3.5.1 Define Two-Dimensional Contact Surface

Description

This option allows for the input of 2-D contact surface definition (rigid or deformable) in contact problems. It also allows the user to input friction type, relative sliding velocity for sticking conditions, contact tolerance, average and cut-off strain rates, location of center of rotation, initial angular position of surface, velocity of center of rotation, angular velocity, as well as friction coefficient.

NOTE: Always define deformable surfaces before rigid surfaces.

If the UMOTION option and user subroutine MOTION are used, velocity data may be skipped.

If the UFRICTION option and user subroutine UFRIC are used, friction data may be skipped, but the friction type must be identified.

If, in a coupled thermal-stress-contact problem, the UHTCOEF option and user subroutine UHTCOE are used, the film coefficient and sink temperature data of a free surface may be skipped.

If the UHTCON option and user subroutine UHTCON are used in a coupled thermal-stress-contact problem, film coefficient data between surfaces in contact may be skipped.

The following data may be changed upon restart:

Friction type	4th field	2nd card
Maximum number of separations	6th field	2nd card
Suppression of splitting	7th field	2nd card
Relative sliding velocity	1st field	3rd card
Contact distance	2nd field	3rd card
Average strain rate	3rd field	3rd card
Cutoff strain rate	4th field	3rd card
Separation force	5th field	3rd card
Bias factor	6th field	3rd card

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-7	1st	A	Enter the word CONTACT.
2nd card series			
1-5	1st	I	Number of surfaces to be defined.
6-10	2nd	I	Maximum number of entities to be created for any surface.
11-15	3rd	I	Upper bound to the number of nodes that lie on the periphery of any deformable surface.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
16-20	4th	I	Friction type 0: No Friction 1: Shear Friction 2: Coulomb Friction 3: Shear Friction for Rolling 4. Coulomb Friction for Rolling
21-25	5th	I	Enter a 1 for the calculation of Coulomb friction based on nodal force instead of nodal stress. Default is 0.
26-30	6th	I	Maximum number of separations allowed in each increment. Default is 9999.
31-35	7th	I	Enter a 1 for the suppression of the splitting of an increment in fixed time step procedure. Enter a 2 for adaptive time step procedure. Default is 0.
36-40	8th	I	Enter a 1 for the interference kinematic check. Enter a 2 to suppress bounding box checking (this may eliminate pentration, but will slow down the solution). Enter a 3 to not reset NCYCLE=0; this will speed up the solution but may result in instabilities. Enter a 4 is used for analytical surfaces only; check for separation only when convergent solution, similar to PWL approach.
41-45	9th	I	Control separations within an increment. When a 0 is entered, if the force on a node is greater than the separation force, the node will separate and an iteration will occur. When a 1 is entered, if a node which was in contact at the end of the previous increment has a force greater than the separation force, the node will not separate in this increment, but will separate at the beginning of the next increment. When a 2 is entered, if a new node comes into contact during this increment, it will not be allowed to separate during this increment (prevents chattering). When a 3 is entered, both (1) and (2) above will be in effect.
46-50	10th	I	Enter a 1 to ignore shell thickness for 3-D contact.
3rd card series			
1-10	1st	F	Relative sliding velocity between surfaces below which sticking is simulated (RVCNST). Default = 1.0.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
11-20	2nd	F	Distance below which a node is considered touching a surface (ERROR). Leave blank if you want MARC to calculate it. This number is also used to divide splines. If splines are used, this must be defined.
21-30	3rd	F	Average strain rate. Used in rigid-plastic analysis only--to start a problem. Leave blank if you want MARC to calculate it.
31-40	4th	F	Cutoff strain rate below which flow stresses drop linearly to zero. Used in rigid-plastic analysis only. Leave blank if you want MARC to calculate it. Default is 1.e-4.
41-50	5th	F	Separation force above which a node separates from a surface (FNTOL). Default is the maximum residual force.
51-60	6th	F	Contact tolerance BIAS factor. (0-1)

Card series 4, 5, 6, 7, and 8 are repeated once for each surface to be defined.

4th card series

1-5	1st	I	Surface number.
6-10	2nd	I	Number of sets of geometrical data to be input for this rigid surface (NETTY). Enter 0 if deformable surface.
11-15	3rd	I	For rigid surfaces, enter a 1 if surface is a symmetry plane. For deformable surfaces, enter a 1 if K3 style deformable-deformable contact will be used. Note that in this case results are dependent upon the order in which contact surfaces are defined.
16-20	4th	I	When a node contacts multiple rigid surfaces, it will contribute a load to a surface, the load surface being indeterminate. Setting this field to 1 will insure that all nodes contacting this surface will contribute their load to this surface. This only influences the load summary table.
21-25	5th	I	Enter a 1 if analytic form is to be used.

5th card series

For a deformable surface, only the friction coefficients are read and the other entries are ignored.

1-10	1st	F	First coordinate of initial position of center of rotation.
11-20	2nd	F	Second coordinate of initial position of center of rotation.
21-30	3rd	F	Not used; enter a zero.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
31-40	4th	F	First component of velocity of center of rotation.
41-50	5th	F	Second component of velocity of center of rotation.
51-60	6th	F	Angular velocity, about center of rotation (radians/time).
61-70	7th	F	Friction coefficient.

6th card series

6th card series only necessary for COUPLED *analysis.*

1-10	1st	F	Heat Transfer coefficient (film) to environment.
11-20	2nd	F	Environment sink temperature.
21-30	3rd	F	Contact heat transfer coefficient (film).
31-40	4th	F	Surface temperature. (Required for rigid surface only.)

A. FOR 2-D DEFORMABLE SURFACES

7a card series

1-80	1st	I	Enter a list of element of which the surface is comprised.

The 7th and 8th card series are repeated for as many geometrical data as required (NETTY).

B. FOR 2-D RIGID SURFACE (LINE-SEGMENT)

7b card series

1-5	1st	I	Enter a 1 for straight line segments (ITYPE).
6-10	2nd	I	Number of points required to define polyline (NPOINT).

The 8(b)th card series is repeated once for each point entered.

8b card series

1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.

C. FOR 2-D RIGID SURFACE (CIRCULAR ARC)

7c card series

1-5	1st	I	Enter a 2 for circular arc (ITYPE).
6-10	2nd	I	Method of describing circular arc (METHOD). See Figure 3-5.

The 8c card series is repeated four times.

8c card series

1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
11-15	3rd	I	Number of points along the second direction of surface (NPOINT2).

*The 9(f) card series is repeated (NPOINT1 * NPOINT2) times for poly-surfaces.*

9f card series

1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.
21-30	3rd	F	Third coordinate of point.

G. FOR 3-D RIGID SURFACE (NURBS)

8g card series

1-5	1st	I	Enter 9 for NURBS.
6-10	2nd	I	Number of control points along u-direction (NPTU).
11-15	3rd	I	Number of control points along v-direction (NPTV).
16-20	4th	I	Order along u-direction (NORU).
21-25	5th	I	Order along v-direction (NORV).
26-30	6th	I	Number of subdivisions along u-direction, default 50.
31-35	7th	I	Number of subdivisions along v-direction, default 50.
36-40	8th	I	Number of trimming curves.

*The 9(g) card series is repeated (nptu*nptv) for control points.*

9g card series

1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.
21-30	3rd	F	Third coordinate of point.

*The 10(g) card series is repeated (nptu*nptv) for homogeneous coordinate.*

10g card series

1-10	1st	F	Homogeneous coordinate ($0 \le h \le 1$).

The 11(g) card series is repeated (nptu+noru)+(nptv+norv) for knot vectors.

11g card series

1-10	1st	F	Knot vector($0 \le k \le 1$).

H. FOR 3-D RIGID SURFACE (CYLINDER)

8h card series

1-5	1st	I	Enter 10 for Cylinder.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
6-10	2nd	I	Number of subdivisions.
9h card series			
1-10	1st	F	First coordinate of center point on top surface.
11-20	2nd	F	Second coordinate of center point on top surface.
21-30	3rd	F	Third coordinate of center point on top surface.
31-40	4th	F	Radius of top surface
41-50	5th	F	First coordinate of center point on bottom surface.
51-60	6th	F	Second coordinate of center point on bottom surface.
61-70	7th	F	Third coordinate of center point on bottom surface radius of bottom surface.
71-80	8th	F	Radius of bottom surface.

If the radius is negative value in 4th field the normal of cylinder is outward. Default is inward.

I. FOR 3-D RIGID SURFACE (SPHERE)

8icard series			
1-5	1st	I	Enter 11 for Sphere.
6-10	2nd	I	Number of subdivisions.
9icard series			
1-10	1st	F	First coordinate of center point.
11-20	2nd	F	Second coordinate of center point.
21-30	3rd	F	Third coordinate of center point.
31-40	4th	F	Radius of sphere.

If the radius is negative value in 4th field the normal of sphere is outward. Default is inward.

CONTACT NODE

3.5.8 Define Nodes for Surface Contact

Description

This option is used to define which nodes in a body may potentially contact other surfaces. This option can be used to reduce the computational cost if a body has many exterior nodes, yet it is known for which nodes contact may occur. If this option is not used, all exterior surface nodes will be checked for contact.

NOTE: If this option is used and a node number is not explicitly listed, that node may penetrate other bodies.

NOTE: In a restart analysis, if these values are to be changed, use the REAUTO option and specify the CONTACT NODE after the END OPTION.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-12	1st	A	Enter the words CONTACT NODE.
2nd card series			
1-5	1st	I	Enter the number of bodies for which exterior nodes are defined
3rd card series			
1-5	1st	I	Body number
4th card series			
1-80	1st	I	Enter a list of nodes that are potential contact nodes

UCONTACT

3.5.9 Invoke User Subroutine for User Defined Contact Conditions

Description

This option calls the user subroutine UCONTACT to define contact conditions (see *Volume D*). In general, the four stages in the MARC Contact procedure are: checking contact, enforce contact constraint, check separation, and check penetration. The user can substitute their own procedure to replace the default MARC one.

NOTE: Use only in conjunction with the CONTACT option.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-6	1st	A	Enter the word UCONTACT.

4. Isotropic Elastic Incompressible Material Mooney-Rivlin form – This material can be represented by a certain class of strain energy functions. The form of this function is:

$$W = C_{10}(I_1 - 3) + C_{01}(I_2 - 3) + C_{11}(I_1 - 3)(I_2 - 3) + C_{20}(I_1 - 3)^2 + C_{30}(I_1 - 3)^3$$

where I_1, I_2 are the first and second invariants of the elastic strain. This strain energy function can represent the Neo-Hookean materials (C_{01}, C_{11}, C_{20}, and C_{30} are zero) or Mooney-Rivlin materials (C_{11}, C_{20} and C_{30} are zero). The material has a nonlinear relation between stress and strain; hence, an incremental procedure must be performed. Alternative energy functions can be specified via user subroutine UENERG.

The stress-strain relations can be expressed as $\sigma_{ij} = \dfrac{\partial W}{\partial \varepsilon_{ij}}$.

The constants are supplied by the user through the MOONEY option. Note that this material model can be used to represent large-strain elastic materials. It is used in conjunction with the Herrmann formulation elements (for plane strain, axisymmetric and three-dimensional problems) or plane stress elements.

5. Isotropic Elastic Incompressible Material-Ogden Formulation - This nonlinear elastic material is represented using the Ogden strain energy function. This material model can be used to represent large-strain elastic materials. It is used in conjunction with the Herrmann formulation elements (for plane strain, axisymmetric and three-dimensional problems) or plane stress elements. The strain energy function is

$$W = \sum_{n=1} \frac{\mu_n}{\alpha_n} J^{-\alpha_n/3} \left(\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n} - 3\right) + 4.5K\left(J^{1/3} - 1\right)^2$$

6. Isotropic Elastic Foam Material - This nonlinear elastic material has the characteristic that it may have both large strain deviatoric and volumetric behavior. The material model is used in conjunction with the displacement elements. The strain energy function is

$$W = \sum_{n=1} \frac{\mu_n}{\alpha_n} \left(\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n} - 3\right) + \sum_{n=1} \frac{\mu_n}{\beta_n} \left(1 - J^{\beta_n}\right)$$

7. General Anisotropic Nonlinear Elastic Material – This can be represented by the hypoelastic material model. The material has a nonlinear relation between stress and strain; hence, an incremental procedure must be performed. The stress-strain relation can be expressed as $\dot{\sigma}_{ij} = C_{ijkl}\dot{\varepsilon} + g_{ij}$

where C_{ijkl} and g_{ij} are functions of elastic strain and temperature. The material data is described by the user subroutine HYPELA. Model definition block HYPOELASTIC should be used to direct the MARC program to call user subroutine HYPELA when necessary.

B. Elastic-Plastic Behavior

Elastic-plastic material can be described using a variety of models. The differentiation between these models is due to:

a) the inclusion or exclusion of elastic effects;

b) the yield function;

c) the flow rule, and

d) the hardening rule.

8. Rigid-Plastic Material – This is the only material model which excludes the elastic strains. The capability is based on the iteration for the velocity field in an incompressible, non-Newtonian fluid. The nonlinear stress-strain relation can be expressed as $S_{ij} = G(\bar{\varepsilon})\,\varepsilon_{ij}$. Note that as the material is incompressible, a traction boundary condition must be specified; otherwise the stress field will only be known to an arbitrary hydrostatic pressure. Only the yield stress need be entered.

9. Elastic-Perfectly-Plastic Material – This is a material which behaves elastically until it reaches the yield stress. The material has no ability to support additional load (in a uniaxial sense) upon yield. The material has a nonlinear relation between the stress rate and strain rate and is path dependent. The user needs only specify the elastic constants and the yield stress. The program uses the von Mises yield function and the associated flow law.

10. Elastic-Plastic Isotropic-Hardening Material – This is a model which behaves elastically until it reaches the yield stress. After yielding the material strains or work hardens according to the isotropic model. The yield surface uniformly expands in all directions with increasing equivalent plastic strain. The additional information necessary to define the rate of growth of the yield surface is prescribed through the WORK HARD option.

11. Elastic-Plastic Kinematic-Hardening Material – This is a material whose yield surface behavior is governed by the kinematic hardening model. In this model the yield surface translates in stress space depending upon the change in plastic strain. The rate of translation is prescribed through the WORK HARD option. This option is set through the ISOTROPIC, ORTHOTROPIC, or ANISOTROPIC model definition option.

12. Elastic-Plastic Combined-Hardening Material – This is a material whose yield surface behavior is governed by a combination of both the isotropic and kinematic hardening models. That is, the yield surface both expands in size and shifts in space. This behavior is given through the WORK HARD option. This option is set through the ISOTROPIC, ORTHOTROPIC, or ANISOTROPIC model definition option.

13. Elastic-Plastic-ORNL Hardening – This is a model whose yield surface is governed by the ORNL constitutive theory. This model also allows plastic creep interaction. This option is set through the ISOTROPIC, ORTHOTROPIC, or ANISOTROPIC model definition option.

As discussed above models 6, 7, 8, 9, 10, and 11 use the von Mises yield function. It is

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
21-30	3rd	F	Mass density. Note that since the program is dimensionally independent, the mass density must be used for dynamics.
31-40	4th	F	Coefficient of thermal expansion.
41-50	5th	F	Initial (stress-free) temperature. This temperature is used in the calculation of temperature dependent properties only (see TEMPERATURE EFFECTS). It never causes thermal straining.

NOTE: INITIAL STATE option is preferred.

51-60	6th	F	Enter the equivalent (von Mises) tensile yield stress (default is 10^{20}).
61-70	7th	F	Equivalent (von Mises) 10th cycle tensile yield stress, for use with ORNL constitutive theories. (See Volume A)
71-75	8th	I	Material type identification (1,2,3...). For cross-referencing TEMPERATURE EFFECTS and WORK HARD data.

4th Card Series

This card series must be included only when the COUPLE *parameter card is included to indicate a coupled thermal-stress analysis. In such problems the thermal properties are given here.*

1-10	1st	F	Enter the reference temperature value of the isotropic thermal conductivity.
11-20	2nd	F	Enter the reference temperature value of the specific heat (per unit reference mass).
21-30	3rd	F	Enter the reference mass density.
31-40	4th	F	Not used; enter 0.
41-50	5th	F	Enter the emissivity for radiating cavities.

5th card series

Enter a list of elements to which the above properties is applied.

ISOTROPIC

3.6.2 Define Mechanical Data for Rigid-Plastic Materials

Description

This option allows the user to define material properties for a rigid-plastic material. The user must also associate these material properties with a list of element numbers.

To define the dependence of these properties on temperature, use the TEMPERATURE EFFECTS Model Definition block.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
Card Series 1			
1-10	1st	A	Enter the word ISOTROPIC.
Card Series 2			
1-5	1st	I	Enter the number of sets of rigid-plastic material data to follow (optional).
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.

Card series 3, 4, 5, and 6 are repeated as a set, once for each set of isotropic material defined.

Card Series 3			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing TEMPERATURE EFFECTS and WORK HARD data.
Card Series 4			

The data entered in the following cards should be the values at the lowest temperature expected during an analysis, not necessarily at the stress-free temperature.

1-10	1st	F	Penalty.
11-20	2nd	F	Not used; enter a zero.
21-30	3rd	F	Mass density (stress analysis).
31-40	4th	F	Coefficient of thermal expansion.
41-50	5th	F	Equivalent (von Mises) tensile yield stress.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

Card Series 5

Necessary only in a coupled thermal-stress analysis.

1-10	1st	F	Thermal conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).
31-40	4th	F	Not used; enter a zero.
41-50	5th	F	Emissivity.

NOTE: The Penalty is only used if a non plane stress analysis is performed, using conventional displacement elements, ie. not Herrmann elements.

Card Series 6

Enter a list of elements associated with this material.

ORTHOTROPIC

3.6.3 Define Mechanical Data for Orthotropic Materials

Description

This option allows the user to define material properties, a yield criterion, and a strain hardening law for an orthotropic material. The user may also associate these material properties with a list of element numbers.

To define the dependence of these properties on temperature, use the ORTHO TEMP Model Definition block.

Defaults for this option are VON MISES yield criterion, ISOTROPIC strain hardening law (with a slope of 0. if the WORK HARD Model Definition block is omitted), and an equivalent yield stress of 10^{20}. Therefore, the default is a perfectly elastic non-yielding orthotropic material.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
Card Series 1			
1-10	1st	A	Enter the word ORTHOTROPIC.
Card Series 2			
1-5	1st	I	Enter the number of sets of ortho-tropic material data to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.

Card Series 3-8 are entered as a set, once for each data set.

Card Series 3			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing ORTHO TEMP, WORK HARD data and user subroutines.
6-15	2nd	A	Enter one of the following yield criteria: VON MISES – von Mises (Default). NORM ORNL – Normal ORNL. CRMO ORNL – 2-1/4 Cr-Mo ORNL. REVP ORNL – Reversed Plasticity ORNL. ARST ORNL – Full alpha reset ORNL. GEN-PLAST – Generalized Plasicity Model. VISCO PLAS – Viscoplastic model through subroutine UVSCPL.

ISOTROPIC

3.6.3 Define Mechanical Data for Rigid-Plastic Materials

Description

This option allows the user to define material properties for a rigid-plastic material. The user must also associate these material properties with a list of element numbers.

To define the dependence of these properties on temperature, use the TEMPERATURE EFFECTS Model Definition block.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
Card Series 1			
1-10	1st	A	Enter the word ISOTROPIC.
Card Series 2			
1-5	1st	I	Enter the number of sets of rigid-plastic material data to follow (optional).
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.

Card series 3, 4, 5, and 6 are repeated as a set, once for each set of isotropic material defined.

Card Series 3			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing TEMPERATURE EFFECTS and WORK HARD data.
Card Series 4			

The data entered in the following cards should be the values at the lowest temperature expected during an analysis, not necessarily at the stress-free temperature.

1-10	1st	F	Penalty.
11-20	2nd	F	Not used; enter a zero.
21-30	3rd	F	Mass density (stress analysis).
31-40	4th	F	Coefficient of thermal expansion.
41-50	5th	F	Equivalent (von Mises) tensile yield stress.

Format

FORMAT		DATA	
FIXED	FREE	TYPE	ENTRY

Card Series 5

Necessary only in a coupled thermal-stress analysis.

1-10	1st	F	Thermal conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).
31-40	4th	F	Not used; enter a zero.
41-50	5th	F	Emissivity.

NOTE: The Penalty is only used if a non plane stress analysis is performed, using conventional displacement elements, ie. not Herrmann elements.

Card Series 6

Enter a list of elements associated with this material.

ANISOTROPIC

3.6.4 Model Definition Block for Stress or Coupled Thermal Stress Analysis

Description

A general temperature dependent orthotropic material model is available through the MARC input deck by the use of the ORTHOTROPIC and ORTHO TEMP options. If a more general model is needed, the user may supply such a model through the user subroutines ANELAS, HOOKLW, ANEXP, ANKOND, ANPLAS, or ORIENT.

Two ways to request a call to these subroutines are shown below:

- Use the flag (3rd card, sixth field) on the ORTHOTROPIC option to modify the material data entered there.
- Use the ANISOTROPIC Model Definition block to call these subroutines.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word ANISOTROPIC.
2nd card series			
1-5	1st	I	Enter the number of anisotropic material data sets to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.
Card series 3,4, and 5 are repeated as a set NSET times.			
3rd card series			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing ORTHO TEMP, WORK HARD data and user subroutines.
6-15	2nd	A	Enter one of the following yield criteria: VON MISES - von Mises (default) NORM ORNL - Normal ORNL CRMO ORNL - 2-1.3 Cr-Mo ORNL REVP ORNL - Reversed Plasticity ORNL ARST ORNL - Full alpha reset ORNL

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
16-25	3rd	A	Enter one of the following hardening rules: ISOTROPIC - Isotropic hardening (default) KINEMATIC - Kinematic hardening COMBINED - Combined hardening (isotropic/kinematic)
26-30	4th	I	Enter a 1 if user subroutines ANELAS, ANEXP, ANPLAS, and HOOKLW are to be called. Enter a 2 if the anisotropic stress strain law, etc., is to be entered in card series (4a, 4b, 4c, 4d, 4e, 4f).

4th card series

1-10	1st	E	Mass density (stress analysis)
11-20	2nd	E	Equivalent (von Mises) yield stress
21-30	3rd	E	If ORNL yielding, 10th cycle yield stress
31-40	4th	E	Mass density (heat transfer analysis)
41-50	5th	E	Specific Heat

Card series 4a, 4b, and 4c used to define anisotropic elastic stress strain relation. Card series 4a only required if the fourth field is a "2".

Card series 4a

1-10	1st	C11
11-20	2nd	C12
21-30	3rd	C13
31-40	4th	C14
41-50	5th	C15
51-60	6th	C16
61-70	7th	C22
71-80	8th	C23

Card series 4b only required if the fourth field is a "2".

Card series 4b

1-10	1st	C24
11-20	2nd	C25
21-30	3rd	C26
31-40	4th	C33
41-50	5th	C34
51-60	6th	C35
61-70	7th	C36
71-80	8th	C44

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

Card series 4c only required if the fourth field is a "2"

Card series 4c

1-10	1st	C45
11-20	2nd	C46
21-30	3rd	C55
31-40	4th	C56
41-50	5th	C66

Use only as many terms as are required for the element type chosen. All three cards must be used. For plane stress only C11, C12, C13 C22, C23, C23, C33 must be entered.

Card Series 4d is only required if the fourth field is a "2". It defines the anisotropic thermal expansion coefficients.

Card series 4d

1-10	1st	α11
11-20	2nd	α12
21-30	3rd	α13
31-40	4th	α22
41-50	5th	α23
51=6-	6th	α33

Card Series 4e is only required if the fourth field is a "2". It defines the anisotropic plasticity.

Card series 4e

1-10	1st	YRDIR1	Direct stress anisotropic yield ratios.
11-20	2nd	YRDIR2	
21-30	3rd	YRDIR3	
31-40	4th	YRSHR1	Shear stress anisotropic yield ratios.
41-50	5th	YRSHR2	
51-60	6th	YRSHR3	

Card series 4f only required if the fourth field is a "2" on a coupled analysis.

Card series 4f

1-10	1st	K11
11-20	2nd	K12
21-30	3rd	K13
31-40	4th	K22
41-50	5th	K23
51-60	6th	K33

Card series 5

Enter a list of elements associated with this material. (Do not enter composite elements which use this material in its layers.)

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
51-60	6th	F	C_{20}, higher order constants.
61-70	7th	F	C_{30}, higher order constants.

Card Series 5

(Only necessary in a coupled thermal-stress analysis)

1-10	1st	F	Conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).

NOTE: In a coupled thermal-stress analysis, the thermal material model defaults to isotropic heat transfer behavior.

Card Series 6

Enter a list of element numbers associated with this particular elastomeric material.

NOTES: This material model may only be used for elements using the Herrmann (incompressible) formulation or plane stress elements. These material identifications may not be referenced by any COMPOSITE group.

The values C_{10}, C_{01}, C_{20}, C_{20}, and C_{30} can be redefined using the subroutine UMOONY.

Although a general strain energy function may be defined by using the subroutine UENERG, it is still required to define the elements associated with the material identifier here.

OGDEN

3.6.7 Define Data for Ogden Material Model

Description

This option allows the user to define the data associated with the Ogden model for incompressible and nearly incompressible rubber material. The strain energy function for this model has the form.

$$W = \sum_{n=1}^{N} \frac{\mu_n}{\alpha_n} J^{-\alpha_n/3} \left(\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n} - 3\right) + 4.5K\left(J^{1/3} - 1\right)^2$$

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word OGDEN.
2nd card series			
1-5	1st	I	Enter the number of sets of Ogden material data to follow (optional).
6-10	2nd	I	Enter the logical unit number for input. Defaults to input deck.
Card series 3, 4, 5, 6, and 7 are repeated for each data set.			
3rd card series			
1-5	1st	I	Enter the material identification.
6-10	2nd	I	Enter the number of terms (N) that defines the strain energy function.
4th card series			
1-10	1st	F	Enter the bulk modulus (K), default is such that material is incompressible.
11-20	2nd	F	Enter the mass density.
21-30	3rd	F	Enter the coefficient of thermal expansion.
Card Series 5			
(Only necessary in a coupled thermal-stress analysis)			
1-10	1st	F	Conductivity.
11-20	2nd	F	Specific heat.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
21-30	3rd	F	Mass density (heat transfer analysis).

NOTE: In a coupled thermal-stress analysis, the thermal material model defaults to isotropic heat transfer behavior.

Card series 6 is repeated once for each term specified in the 3rd card series.

6th card series

1-10	1st	F	Enter the modulus.
11-15	2nd	F	Enter the power.

7th card series

Enter a list of element numbers associated with this particular elastomeric material.

FOAM

3.6.8 Define Data for Foam Material Model

Description

This option allows the user to define the data associated with the Foam model for highly compressible rubber material.

The strain energy function for this model has the form.

$$W = \sum_{n=1}^{N} \frac{\mu_n}{\alpha_n} (\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n} - 3) + \sum_{n=1} \frac{\mu_n}{\beta_n} (1 - J^{\beta_n})$$

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word FOAM.
2nd card series			
1-5	1st	I	Enter the number of sets of Foam material data to follow (optional).
6-10	2nd	I	Enter the logical unit number for input. Defaults to input deck.

Card series 3, 4, 5, 6, and 7 are repeated for each data set.

3rd card series			
1-5	1st	I	Enter the material identification.
6-10	2nd	I	Enter the number of terms (N) that defines the strain energy function.
4th card series			
1-10	1st	F	Not used, enter zero.
11-20	2nd	F	Enter the mass density.
21-30	3rd	F	Enter the coefficient of thermal expansion.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

Card Series 5

(Only necessary in a coupled thermal-stress analysis)

1-10	1st	F	Conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).

NOTE: In a coupled thermal-stress analysis, the thermal material model defaults to isotropic heat transfer behavior.

Card series 6 is repeated once for each term specified in the 3rd card series.

6th card series

1-10	1st	F	Enter the modulus (μ_n).
11-15	2nd	F	Enter the power for deviatoric behavior (α_n).
21-30	3rd	F	Enter the power for volumetric behavior (β_n).

7th card series

Enter a list of element numbers associated with this particular elastomeric material.

NOTE: If the bulk modulus is entered then βi = zero for all values of i.
If the bulk modulus is zero and all βi are zero, then the material is treated as an Ogden material.

STRAIN-RATE

3.6.9 Define Strain-Rate Dependent Yield Stress

Description

This data block allows the definition of a strain-rate dependent yield stress, for use in dynamic and flow (e.g. extrusion) problems. This may also be used in static analysis by entering a fictitious time using the TIME STEP option. The zero strain rate yield stress is given on the PROPERTY, ISOTROPIC, ORTHOTROPIC, and ANISOTROPIC options. This block must be repeated for each different material for which strain rate data are necessary. The yield stress variation with strain rate is given using one of two options:

A. The breakpoints and slopes for a piecewise linear approximation to the yield stress, strain rate curve are given. The strain rate breakpoints should be in ascending order, or

B. The yield stress and stain rate data points lying on the yield stress, strain rate curve are input directly. The data is entered in ascending order of strain rate. This method is flagged by entering the word DATA on the 1st card.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-11	1st	A	Enter the words STRAIN RATE.
13-80	2nd	A	Enter the word DATA to indicate that option B is being used.
2nd card series			
1-5	1st	I	For option A, enter the number of slopes of yield versus strain rate curve. For option B, enter the number of data points.
6-10	2nd	I	Material type identification (1,2,3...) for cross-reference to ISOTROPIC, ORTHOTROPIC, and ANISOTROPIC blocks.
11-15	3rd	I	Unit number for input of the set of this block. Defaults to cards.
card series 3a			
Card series 3a is used in conjunction with option A. The number of cards in this series is equal to that given in the first field of card series 2.			
1-10	1st	F	Enter the slope of the yield versus strain-rate curve.
11-20	2nd	F	Enter the strain-rate value above which the above slope becomes operational. Note, the first strain-rate breakpoint must be zero.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

card series 3b

Card series 3b is used in conjunction with option B.

1-10	1st	F	Enter the value of the yield strength.
11-20	2nd	F	Enter the associated strain rate. Note that the first strain-rate must be zero.

WORK HARD

3.6.10 Define Workhardening Data

Description

This block allows the user to specify the material stress-strain relation for elastic-plastic behavior. Further details on this block are given in *Volume A*. The workhardening data can be entered in one of three forms.

A. The breakpoints and slopes for a piecewise linear approximation to the stress-strain curves are given. The piecewise linear curve is entered in ascending order of equivalent plastic strain.

B. The stress and plastic strain data points lying on the stress-strain curve are input directly. The data is entered in ascending order of plastic strains. This method is flagged by entering the word DATA on the WORK HARD card. These data points are used to calculate slope breakpoint data.

C. By user subroutine WKSLP. This routine will be called for every integration point where elastic-plastic behavior occurs. See *Volume D* for details. Note that if this option is used, it must be used for ALL material types.

This block must be repeated for each different material for which workhardening data is necessary.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-9	1st	A	Enter the words WORK HARD.
11-80	2nd	A	Enter the word DATA to indicate that option B is being used.
2nd card series			
1-5	1st	I	For option A enter number of slopes of the work-hardening curve. For option B, enter the number of data points. For option C, enter a -1.
6-10	2nd	I	This is the same in the first field except it is for the data associated with the 10th cycle yield used in the ORNL constitutive theory.
11-15	3rd	I	Material type identification (1,2,3 etc.) for cross-referencing with the ISOTROPIC, ORTHOTROPIC, and ANISOTROPIC blocks.
16-20	4th	I	Enter unit number for input of workhardening data, defaults to input.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

Card series 3a and 4a are used in conjunction with Option A.

card series (3a)

The number of cards entered is equal to the number of slopes entered above.

Included only if the first field of 2nd card series > 0.

1-15	1st	F	Enter the slope of the workhardening curve.
16-30	2nd	F	Enter the breakpoint when the slope becomes operative. The breakpoint and slope data should be described in ascending order of the equivalent plastic strain, the first slope starting at zero plastic strain.

NOTE: The work-hardening slope should be for a uniaxial tension specimen, and is the change in stress per unit of plastic strain, not per unit of total strain. See Volume A.

card series (4a)

Included only if the first field of 2nd card series > 0. Then number of cards is equal to that number.

1-15	1st	F	Slope of 10th cycle workhardening curve (stress change per plastic strain change).
16-30	2nd	F	Breakpoint when above slope becomes operative. First breakpoint should be at zero plastic strain.

Card series 3b and 4b are used in conjunction with Option B.

card series (3b)

The number of cards entered is equal to the number of data points entered above.

1-15	1st	F	Enter the equivalent stress.
16-30	2nd	F	Enter the equivalent plastic strain. The data should be described in ascending order of equivalent plastic strain, the first data set starting at zero plastic strain.

card series (4b)

Included only if the first field of 2nd card > 0. Then, number of cards is equal to that number.

1-15	1st	Enter the equivalent stress associated with the 10th cycle workhardening curve.
16-30	2nd	Enter the equivalent plastic strain.

TEMPERATURE EFFECTS

3.6.11 Define Effects of Temperature

Description

This data block defines the variation of element properties (Young's modulus, yield stress, Poisson's ratio, and coefficient of thermal expansion) with temperature. The values read in through either the PROPERTY, ISOTROPIC, or POWDER option are those at the lowest temperature specified. Properties are not defined below the lowest temperature. The temperature dependency can be entered using one of the following two options:

- The variation of a particular property with temperature is specified as a piecewise linear curve. Breakpoints must be given in ascending order of temperature.
- The particular property value and temperature lying on the relevant curve are input directly. Data points must be given in increasing order of temperature. This option is flagged by entering the word DATA on the 1st card.

NOTE: For Mooney materials, the temperature dependence for C10 and C01 can be defined by replacing C10 for "Young's modulus" and C01 for "Poisson's ratio". The other constants can be specified by utilizing user subroutine UMOONY.

Format

FORMAT		DATA	
FIXED	FREE	TYPE	ENTRY

1st card series

1-19	1st	A	Enter the words TEMPERATURE EFFECTS.
21-80	2nd	A	Enter the word DATA to indicate that option B is used.

For option A, use card series 2a, 3a, 4a, 5a, 6a, 7a and 8a. For option B used card series 2b, 3b, 4b, 5b, 6b, 7b and 8b, below.

Option A

card series (2a)

1-5	1st	I	Number of slopes of yield stress versus temperature curve.
6-10	2nd	I	Number of slopes of Young's modulus versus temperature curve.
11-15	3rd	I	Number of slopes of Poisson's ratio versus temperature curve.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

The following Card Series are used only in a coupled thermal-stress analysis.

Card Series 18a

The number of cards in this series is the number in Card Series 3a, first field.

1-15	1st	F	Enter the slope of K_{11} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

Card Series 19a

The number of cards in this series is n, the number in Card Series 3a, second field, or 0 if n=-1.

1-15	1st	F	Enter the slope of K_{22} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

Card Series 20a

The number of cards in this series is n, the number in Card Series 3a, third field, or 0 if n=-1.

1-15	1st	F	Enter the slope of K_{33} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

Card Series 21a

The number of cards in this series is the number in Card Series 3a, fourth field.

1-15	1st	F	Enter the slope of specific heat vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

Card Series 22a

The number of cards in this series is the number in Card Series 3a, fifth field.

1-15	1st	F	Enter latent heat value.
16-30	2nd	F	Solidus temperature (lower phase change limit).
31-45	3rd	F	Liquidus temperature (upper phase change limit).

Option B

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
Card Series 2b			
1-5	1st	I	Number of data points of yield.
6-10	2nd	I	Number of data points of E_{11} vs temperature curve.
11-15	3rd	I	Number of data points of E_{22} vs temperature curve. Enter -1 to have (E_{22} vs. temp.) ≡ (E_{11} vs. temp.).
16-20	4th	I	Number of data points of E_{33}. Enter -1 to have (E_{33} vs. temp.) ≡ (E_{11} vs. temp.).
21-25	5th	I	Number of data points of ν_{12}.
26-30	6th	I	Number of data points of ν_{23}. Enter -1 to have (ν_{23} vs. temp.) ≡ (ν_{12} vs. temp.).
31-35	7th	I	Number of data points of ν_{31} Enter -1 to have (ν_{31} vs. temp.) ≡ (ν_{12} vs. temp.).
36-40	8th	I	Number of data points of G_{12}.
41-45	9th	I	Number of data points of G_{23}. Enter a -1 to have (G_{23} vs. temp.) ≡ (G_{12} vs. temp.).
46-50	10th	I	Number of data points of G_{31}. Enter a -1 to have (G_{31} vs. temp.) ≡ (G_{12} vs. temp.).
51-55	1th1	I	Number of data points of α_{11}.
56-60	12th	I	Number of data points of α_{22}. Enter a -1 to have (α_{21} vs. temp.) ≡ (α_{11} vs. temp.).
61-65	13th	I	Number of data points of a_{33}. Enter a -1 to have (α_{33} vs. temp.) ≡ (α_{11} vs. temp.).
66-70	14th	I	Number of data points of the workhardening vs. temperature curve.
71-75	15th	I	Enter the material identification for this data set.
76-80	16th	I	Enter the unit number for input of this data. Defaults to input deck.
Card Series 3b			
Include this card only in a coupled thermal-stress analysis.			
1-5	1st	I	Number of data points of K_{11}
6-10	2nd	I	Number of data points of K_{22}. Enter a -1 to have (K_{22} vs. temp.) ≡ (K_{11} vs. temp.).

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
5th card series			
1-10	1st	F	γ_{xy} – Maximum absolute value of shear strain in xy-plane.
11-20	2nd	F	γ_{yz} – Maximum absolute value of shear strain in yz-plane.
21-30	3rd	F	γ_{zx} – Maximum absolute value of shear strain in zx-plane.
For HOFFMAN or HILL:			
4th card series			
1-10	1st	A	Enter either HOFFMAN or HILL.
11-20	2nd	F	X – Maximum tensile stress in x-direction.
21-30	3rd	F	X_c – Maximum absolute value of compressive stress in x-direction.
31-40	4th	F	Y – Maximum tensile stress in y-direction.
41-50	5th	F	Y_c – Maximum absolute value of compressive stress in y-direction.
51-60	6th	F	Z – Maximum tensile stress in z-direction.
61-70	7th	F	Z_c – Maximum absolute value of compressive stress in z-direction.

NOTE: X_c, Y_c and Z_c default to the values of X, Y, and Z, respectively, if left undefined for HOFFMAN. For HILL, X_c, Y_c and Z_c are assumed to be equal to X, Y, and Z respectively and are not used.

5th card series			
1-10	1st	F	S_{xy} – Maximum absolute value of shear stress in xy-plane.
11-20	2nd	F	S_{yz} – Maximum absolute value of shear stress in yz-plane.
21-30	3rd	F	S_{zx} – Maximum absolute value of shear stress in zx-plane.
31-40	4th	F	Failure index. Default is 1.0.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

For TSAI-WU:

4th card series

1-10	1st	A	Enter the word TSAI-WU
11-20	2nd	F	X – Maximum tensile stress in x-direction.
21-30	3rd	F	X_C – Maximum absolute value of compressive stress in x-direction.
31-40	4th	F	Y – Maximum tensile stress in y-direction.
41-50	5th	F	Y_C – Maximum absolute value of compressive stress in y-direction.
51-60	4th	F	Z – Maximum tensile stress in z-direction.
61-70	5th	F	Z_C – Maximum absolute value of compressive stress in z-direction.

NOTE: X_C, Y_C and Z_C default to the values of X, Y, and Z, respectively, if left undefined

5th card series

1-10	1st	F	S_{XY} – Maximum absolute value of shear stress in xy-plane.
11-20	2nd	F	S_{YZ} – Maximum absolute value of shear stress in yz-plane.
21-30	3rd	F	S_{ZX} – Maximum absolute value of shear stress in zx-plane.
31-40	4th	F	Failure index. Default is 1.0.
41-50	5th	F	F_{XY} – Interactive strength tensor constant for the xy-plane.
61-60	6th	F	F_{YZ} – Interactive strength tensor constant for the yz-plane.
61-70	7th	F	F_{ZX} – Interactive strength tensor constant for the zx-plane.

NOTE: F_{xy} should be such that $F_{xy}^2 < \frac{1}{XX_c} \cdot \frac{1}{YY_c}$, etc

For UFAIL:

FORMAT		DATA	
FIXED	FREE	TYPE	ENTRY

For UFAIL type failure criteria, enter only the word UFAIL in the 4th card series and leave all other fields blank. The 5th card series is not used for UFAIL.

4th card series

1-10	1st	A	Enter the word UFAIL.

DAMAGE

3.6.17 Define Properties for Damage Materials

Description

This option allows the user to define a set of data for a specific material which includes a damage model. The specific model and associated data can also be specified with this option. For elastic-plastic materials, the damage model is based on a Gurson model for the yield surface definition for materials with voids. Void nucleation and void growth are based on a model by Tvergaard and Needleman. For elastomeric materials, the model is based on a modified Ogden strain energy function.

The normal data for a specific material are defined with the ISOTROPIC, WORK HARD, and OGDEN options. Cross-reference to this material is made with the material number. Additional data for the initial void volume fraction can be defined with user subroutine UVOID. Other nucleation models are allowed via the user subroutine UVOIDN.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the words DAMAGE.
2nd card series			
1-5	1st	I	Enter the number of distinct sets of material properties to be input (optional).
6-10	2nd	I	Enter the logical unit number for reading damage data. Defaults to input.

The 3rd and 4th card series are entered as pairs, once for each distinct data set.

3rd card series			
1-5	1st	I	Material type identification (1, 2, 3, etc.) for cross-referencing to ISOTROPIC or ODGEN option.
6-10	2nd	I	Method of void nucleation. Enter 0 for no nucleation. Enter 1 for plastic-strain controlled nucleation. Enter 2 for stress controlled nucleation. Enter 3 fo nucleation controlled by user subroutine UVOIDN Enter 4 for elastomeric damage model Enter 5 for elastomeric damage model controlled by user subroutine UELDAM

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

4a card series

Use only if the method of void nucleation (shown above) is 0, 1, or 2.

1-10	1st	F	First yield surface multiplier q_1 (recommended is $q_1 = 1.5$).
11-20	2nd	F	Second yield surface multiplier q_2 (recommended is $q_2 = 1$).
21-30	3rd	F	Initial void volume fraction.
31-40	4th	F	Critical void volume fraction. This value represents the value at which coalesence of voids start (f_c).
41-50	5th	F	Failure void volume fraction (f_F). This is the value of the void volume fractron at which the stiffness of the material has reduced to zero.
51-60	6th	F	If strain controlled, enter the mean strain for nucleation. If stress controlled, enter the mean stress for nucleation.
61-70	7th	F	Standard deviation in nucleation relation.
71-80	8th	F	Volume fraction of void nucleating particles.

NOTE: The presence of these cards in the model definition deck automatically overwrite the yield criterion specified for a specific material on the ISOTROPIC card. Currently, the model can only be used for isotropic hardening materials.

4b card series

Use only for elastomeric damage model.

1-10	1st	F	Deviatoric damage rate.
11-20	2nd	F	Maximum deviatoric damage factor.
21-30	3rd	F	Volumetric damage rate.
31-40	4th	F	Maximum volumetric damage factor.

GAP DATA

3.6.18 Define Data for Gap Elements

Description

This option allows the user to specify all of the data associated with gap elements. These data include gap closure distance, gap elastic stiffness, contact coefficient of friction, and momentum ratio.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st Card Series			
1-10	1st	A	Enter the words GAP DATA.
2nd Card Series			
1-5	1st	I	Number of sets of gap data to be input.
6-10	2nd	I	Unit number for reading data. Defaults to input.

The 3rd and 4th card series are entered as pairs, once for each set of gap data.

3rd Card Series			
1-10	1st	F	For a fixed direction gap, enter the gap closure distance U_{c1}. For a true distance gap, enter the minimum distance d between end points.

NOTE: If d>0, the two end points will never be closer than a distance |d| apart. If d<0, the two end points will never be farther apart than |d|.

11-20	2nd	F	μ, the contact coefficient of friction.
21-30	3rd	F	K_{GAP}, the elastic stiffness of the closed gap in the contact direction. Default: Gap is rigid when closed.
31-40	4th	F	$K_{FRICTION}$, the elastic stiffness of the closed gap in the friction direction. Default: Gap is rigid when closed.
41-50	5th	F	User supplied momentum ratio for first gap node. Default: The MARC program will calculate this ratio internally.
51-60	6th	F	User supplied momentum ratio for fourth gap node. Default: The MARC program will calculate this ratio internally.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
61-65	7th	I	Enter 0 for fixed direction gap. Enter 1 for true distance gap. Default is 0.
66-70	8th	I	Enter 0 if gap is open during increment 0. Enter 1 if gap is closed during increment 0. Default is 0.

4th card series

Enter a list of gap elements to be associated with this set of gap data.

COMPOSITE

3.6.19 Define Properties for Laminated Composite Materials

Description

This option allows the user to define the layer-by-layer material identifications, layer thicknesses, and orientation angles for a laminated composite material and to associate this information with an element number. Property data for each material identification is entered using the ISOTROPIC, ORTHOTROPIC, ANISOTROPIC, or HYPOELASTIC options.

To specify a user-defined orientation, use the ORIENTATION option. Note that an input error will result if the COMPOSITE option is specified for non-layered elements.

This option is available for shell elements or beams in a plane (type 16). It is not available for open and closed section beam elements.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word COMPOSITE.
2nd card series			
1-5	1st	I	Enter the number of composite group data sets to follow.
6-10	2nd	I	Unit number for input. Defaults to standard input (unit 5).
3rd card series			
1-5	1st	I	Composite group number.
6-10	2nd	I	Number of layers in this group.
11-15	3rd	I	Enter 0 to input actual layer thicknesses in the second field of Card Series 4. (Default is 0; i.e., the sum of the layer thicknesses will override thickness data entered in the GEOMETRY block. Enter 1 to input percentage of total thickness in the second field of Card Series 4. In this case, element thickness is entered using the GEOMETRY data block or the NODAL THICKNESS data block. If you are using the variable thickness capability for those elements which have such an option, you must enter a 1 here and then enter percentages of total thickness in the second field of Card Series 4 below.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
36-50	6th	F	If the entry in the tenth field is 0, enter tolerance on the creep strain increment relative to the elastic strain. Default = 0.50. Note: a higher value is likely to cause stability problems. If the entry in the tenth field is 1, enter the maximum allowable creep strain increment. Default is .01.

NOTE: Use of AUTO CREEP option to input this value is preferred.

51-65	7th	F	If the entry in the tenth field is 0, enter the tolerance on the stress change per stress during creep. Default is 0.10. If the entry in the tenth field is 1, enter the maximum stress increment. Default is 100. This control is included primarily for accuracy purposes. Default value is adequate for creep laws of the type $\varepsilon = a\sigma^n$ where $3 < n < 6$. For lower values of n, tolerance can be increased; for higher values, it should be decreased.

NOTE: Use of AUTO CREEP option to input this value is preferred.

66-70	8th	I	Enter the unit number for input of creep data, defaults to input.
71-75	9th	I	Material id number.
76-80	10th	I	Enter 1 if absolute rather than relative testing is to be performed.

card series 3a

Slope and breakpoint data for equivalent creep strain rate versus temperature curve. The number entered in the first field of the second card series defines the number of cards required in card series 3.

1-15	1st	F	Enter the slope of the curve or the exponent of temperature in the exponential creep law.
16-30	2nd	F	Enter the temperature above which the slope (above) becomes operative. This entry is left blank for exponential creep law.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY

card series 4a

Enter the slope and breakpoint data for equivalent creep strain rate versus equivalent total stress curve. The number entered in the second field of the second card series defines the number of cards required in card series 4.

1-15	1st	F	Enter the slope of the curve or the exponent of stress in the exponential creep law.
16-30	2nd	F	Enter the equivalent total stress above which the slope becomes operative. This entry is left blank for exponential creep law.

card series 5a

Slope and breakpoint data for equivalent creep strain rate versus total equivalent creep strain curve. The number entered in the third field of the second card series defines the number of cards required in card series 5.

1-15	1st	F	Enter the slope of the curve or the exponent of total equivalent creep strain in the exponential creep law.
16-30	2nd	F	Enter the equivalent total creep strain above which the slope becomes operative. This entry is left blank for exponential creep law.

card series 6a

Slope and breakpoint data for total equivalent creep strain versus time curve. The number entered in the fourth field of the second card series defines the number of cards required in card series 6.

1-15	1st	F	Enter the slope of the curve or the exponent of time in the exponential creep law.
16-30	2nd	F	Enter the total time above which the slope becomes operative. This entry is left blank for exponential creep law.

card series 3b

Data points for the equivalent creep strain rate versus temperature curve. The number entered in the first field of the second card series defines the number of cards required in card series 3.

1-15	1st	F	Enter the equivalent creep strain rate or the exponent of temperature in the exponential creep law.
16-30	2nd	F	Enter the associated temperature. This entry is left blank for exponential creep law.

DAMPING

3.8.1 Define Damping Factors

Description

This option allows the input of damping factors for use with the dynamic analysis options. Two damping inputs are available depending on the user's choice of dynamic option. For modal superposition analysis, the user gives the fraction of critical damping associated with each mode of the solution. For direct integration options, the user inputs the factors weighting the mass and stiffness matrices to form the damping matrix. In both cases, the damping matrix is assumed to be formed as a linear combination of the mass and stiffness matrices of the system, see *Volume A*.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-7	1st	A	Enter the word DAMPING. For modal analysis, the second card series is used. For direct integration analysis, damping is given on a element basis, card series 3, 4, and 5 are used.
2nd card series			
(For modal analysis only.)			
1-10	1st	F	Fraction of critical damping for 1st mode.
11-20	2nd	F	Fraction of critical damping for 2nd mode.
Etc.	Etc.	Etc.	Etc.
3th card series			
For direct integration (Newmark-Beta, Houbolt or Central Difference).			
1-5	1st	I	Number of damping sets (NDMPST) to be read in. Card series 4 and 5 are given in pairs NDMPST times.
6-10	2nd	I	Enter the unit number for input of the damping data, defaults to input.
4th card series			
1-10	1st	F	Multiplier (α) for mass matrix contribution to damping matrix.
11-20	2nd	F	Multiplier (β) for stiffness matrix contribution to damping matrix.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
21-30	3rd	F	Multiplier (γ) for numerical damping.

5th card series

1-5	1st	I	First element to have these damping values.
6-10	2nd	I	Last element to have these damping values.

In a dynamic analysis, the damping matrix is evaluated as

$$C = \alpha M + (\beta + \frac{\gamma \Delta t}{\pi}) K$$

In a harmonic analysis, the damping matrix is evaluated as

$$C = \alpha M + (\beta + \frac{2\gamma}{\omega}) K$$

ANISOTROPIC

3.9.9 Model Definition Block for Heat Transfer Analysis

Description

This option specifies thermal properties defined by a call to user subroutines ANKOND and ORIENT. The subroutine ANKOND must be used for the input of constant or temperature dependent anisotropic thermal conductivities (K_{11}, K_{22}, K_{33}) and/or resistives (R_{11}, R_{22}, R_{33}) defined in the user coordinate (1,2,3) system. The TEMPERATURE EFFECTS model definition block can be used for the input of variations of specific heat with temperatures. Note that the data entered in this option should be the values at the lowest temperature expected during an analysis, not necessarily at the initial temperature.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word ANISOTROPIC.
2nd card series			
1-5	1st	I	Enter the number of anisotropic data sets to follow (optional)
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.
Card series 3,4, and 5 are repeated as a set NSET times.			
3rd card series			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing ANKOND and ORIENT, as well as to the TEMPERATURE EFFECTS option.
6-10	2nd	I	Enter a 1 if the user subroutine ANKOND is to be called. Enter a 2 if the anisotropic conductivity is to be entered in card series 4a.
4th card series			
1-10	1st	E	Mass density.
11-20	2nd	E	Specific heat per unit mass.
21-30	3rd	E	Emissivity.
Card series 4a only required if the second fields 6-10 is a "2"			
4a card series			
1-10	1st		K11
11-20	2nd		K12

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
21-30	3rd		K13
31-40	4th		K22
41-50	5th		K23
51-60	6th		K33

5th card series

Enter a list of elements associated with this material. (Do not enter composite elements which use this material in its layers.)

3.9.10 Define Variation of Element Properties in Heat Transfer Analysis

Description

This option defines the variation of element properties (conductivity, specific heat and electrical resistance) with temperature. The values read in through the PROPERTY or ISOTROPIC options are those at the lowest temperature specified. Properties are not defined below the lowest temperature. The temperature dependency can be entered using one of the following two options:

A. The variation of a particular property with temperature is specified as a piecewise linear curve. Breakpoints must be given in ascending order of temperature.

B. The particular property value and temperature lying on the relevant curve are input directly. Data points must be given in ascending order of temperature. This option is flagged by entering the work "DATA" on the 1st card.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
Card series 1			
1-19	1st	A	Enter the word TEMPERATURE EFFECTS.
21-80	2nd	A	Enter the word DATA to indicate that Option B is used.
Option A			
Card series 2a			
For option A use card series 2a, 3a, 4a, 5a and 6a. For option B use cards series 2b, 3b, 4b, 5b and 6b.			
1-5	1st	I	Number of slopes of conductivity versus temperature curve.
6-10	2nd	I	Number of slopes of specific heat versus temperature curve.
11-15	3rd	I	Number of latent heats to be entered.
16-20	4th	I	Number of slopes of resistivity versus temperature curve for Joule heating problem.
21-25	5th	I	Number of slopes for emissivity versus temperature curve for radiating cavities.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
26-30	6th	I	Not used; enter a zero.
31-35	7th	I	Material type identification (1,2,3...) for cross-reference to PROPERTY or ISOTROPIC option.
36-40	8th	I	Logical unit number for input of this set of data. Defaults to cards.

Card series 3a

Conductivity variation. Number of cards as given on card 2, first field.

1-15	1st	F	Slope of conductivity versus temperature curve.
16-30	2nd	F	Temperature above which above slope becomes operative.

Card series 4a

Specific heat variation. Number of cards as given on card 2, second field.

1-15	1st	F	Slope of specific heat versus temperature curve.
16-30	2nd	F	Temperature above which above slope becomes operative.

Card series 5a

Latent heat. Number of cards given on card 2, third field.

1-15	1st	F	Latent heat.
16-30	2nd	F	Solidus temperature (lower phase change limit).
31-45	3rd	F	Liquidus temperature (upper phase change limit).

Card series 6a

Resistivity variation for Joule heating problem. Number of cards given on card 2, fourth field.

1-15	1st	F	Slope of resistivity versus temperature curve.
16-30	2nd	F	Temperature above which above slope becomes operative.

Card series 7a

Emissivity variation for radiating cavity problems. Number of cards given on card 2, fifth field.

1-15	1st	F	Slope of emissivity versus temperature curve.
16-30	2nd	F	Temperature above which the above slope becomes operative.

CONRAD GAP

3.9.14 Define Convection/Radiation Gap

Description

This option allows the user to input emissivity, Stefan-Boltzmann constant, absolute temperature conversion factor, film coefficient, and gap closure temperature for convection/radiation gap option.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word CONRAD GAP.
2nd card series			
1-5	1st	I	Number of sets of data used to input CONRAD GAP.
6-10	2nd	I	Unit number for input of CONRAD GAP data, defaults to input.
3rd card series			
1-5	1st	I	Face identification – see Volume B "Library Element Descriptions". Note that these identifiers are different from those used for DIST FLUX.
6-15	2nd	F	Emissivity
16-25	3rd	F	Stefan-Boltzmann constant (default is 0.1714×10^{-8} BTU/HR-FT^2 -R^4).
26-35	4th	F	Absolute temperature conversion factor (i.e., degrees Rankine = 459.7 + degrees Fahrenheit; or, degrees Kelvin = 273.15 + degrees centigrade; default is 459.7).
36-45	5th	F	Film coefficient.
46-55	6th	F	GAP closure temperature.
4th card series			
			Enter a list of elements to which the above CONRAD GAP data is applied.

CHANNEL

3.9.15 Define Fluid Channel Input

Description

This option allows the user to input inlet temperature, fluxes, and film coefficient for a fluid channel in a heat transfer analysis.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word CHANNEL.
2nd card series			
1-5	1st	I	Number of sets of data used to input fluid channels.
6-10	2nd	I	Unit number for input of fluid channels data, defaults to input.
3rd card series			
1-5	1st	I	Face identification – see Volume B "Library Element Descriptions". Note that these identifiers are different from those used for DIST FLUX.
6-10	2nd	I	First (inlet) element number in the channel.
11-20	3rd	F	Inlet temperature.
21-30	4th	F	Mass flow rate.
31-40	5th	F	Film coefficient.
4th card series			
			Enter a list of elements to which the above fluid channel data is applied.

Chapter 5: History Definition Options

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
			PRES output pressure (bearing analysis) COOR output coordinates (only for rezoning) ALL output all relevant quantities
			4th card series
			Enter a list of nodes to be printed. To suppress all nodal printout, enter a blank list for the list of nodes.

NO PRINT

5.1.7 Suppress Printing

Description

This option suppresses element and nodal output. This option is revoked by using either the PRINT CHOICE, PRINT ELEMENT, or PRINT NODE option.

Format

FORMAT		DATA	
FIXED	FREE	TYPE	ENTRY

1st card series

1-10	1st	A	Enter the words NO PRINT.

PRINT VMASS

5.1.14 Print Element Volumes and Masses

Description

This option allows the user to choose which elements and associated volumes and masses to be printed. Options are provided for the user to print: (1) total volumes and masses of groups of elements and volumes and masses of each element in the group or, (2) total volumes and masses of groups of elements only. In order to have correct mass computations, mass density for each element must be entered through the ISOTROPIC, ORTHOTROPIC, etc. option. Note that volumes and masses for some special elements (e.g. gap element, semi-infinite element, etc.) will not be computed. The volumes and masses can be written on either standard output file unit 6, or user specified unit.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-11	1st	A	Enter the words PRINT VMASS.
2nd card series			
1-5	1st	I	Enter the number of sets to be given below.
6-10	2nd	I	Enter 1 for option to print only total volumes and masses for groups of elements. Default is 0.
11-15	3rd	I	File unit to which output is to be written; default to standard output, unit 6.
Card series 3 is repeated for each set.			
3rd card series			
			Enter a list of elements to be printed.

CONTROL

5.1.15 Define History Controls

This option allows the user to input parameters governing the convergence and solution accuracy for nonlinear stress analysis.

For coupled thermal-stress analysis card series 4 must be used in addition to either card series 3a or 3b.

For nonlinear static analysis, the controls are described in *Volume A*. They do not appear on the restart file, and so must be re-entered on a restart run. A description of the iteration procedures and convergence tests is given in *Volume A*.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st Card Series			
1-7	1st	A	Enter the word CONTROL.
2nd Card Series			
1-5	1st	I	Maximum number of load steps in this run. Default is 9999. This is a cumulative number and is usually used to stop the run when restart is being used. If ELASTIC parameter card is included, this field is ignored and all load cases are analyzed.
6-10	2nd	I	Maximum number of recycles during an increment for plasticity, or other tangent modulus nonlinearities. Default is 3. This should usually be increased to 10 for rigid-plastic flow option. If a negative number is entered, then the program will do a maximum of the absolute value entered. If convergence has not been obtained, a warning is given and the program proceeds to the next increment. This is not recommended.
11-15	3rd	I	Minimum number of recycles during an increment for plasticity or other tangent modulus nonlinearities. Default is 0. Note that this data field forces this number of recycles to take place at all subsequent increments.

CAUTION: This value is overwritten by the PROPORTIONAL INCREMENT data block.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
16-20	4th	I	Flag for convergence testing. If set to 0 or left blank, testing is done on residuals. If set to 1, testing is done on displacements. If set to 2, testing is done on strain energy. Note that testing on displacements or strain energy always requires at least one iteration. If nonlinear analysis is done with the CENTROID option, the residuals are not calculated and testing is always done on displacements. Note that nonlinear analysis with the CENTROID option is not recommended.
21-25	5th	I	Flag to specify relative or absolute error testing. If equal to 0, testing is done on relative error. If equal to 1, testing is done on absolute value. If set to 2, testing is done on relative error testing unless reactions or incremental displacements are below minimum value than absolute tolerances, testing is used.
26-30	6th	I	Iterative procedure flag. 1. Full Newton-Raphson. (Default) 2. Modified Newton-Raphson (no reassembly during iteration). 3. Newton-Raphson with strain correction modification (see Volume A). 8. Secant method.
31-35	7th	I	Nonpositive definite flag. If set to 1, solution of nonpositive definite system is forced. Note that with use of gap and Herrmann elements the matrix always is nonpositive definite, and this entry has no significance. *Plasticity*
36-40	8th	I	Radial return flag. If set to 0, MARC standard mean normal method is used in the plasticity equations. If set to 1, radial return method will be used. *Implicit Creep or Viscoplasticity* Enter 0 for elastic tangent. Enter 1 for secant. Enter 2 for radial return (Implicit Creep only)
41-45	9th	I	To print convergence control messages to log file, enter 1.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
46-50	10th	I	Control on initial stress stiffness 0 Normal-full contribution. 1 Mooney or Ogden use deviatoric stress only. 2 No initial stress stiffness. 3 Use stress at beginning of increment, not last iteration.
3rd card series			
1-10	1st	F	If relative residual checking: Maximum allowable value of maximum residual force divided by maximum reaction force. Default is 0.10. If relative displacement checking: Maximum allowable value of the change in displacement increment divided by the displacement increment. Default is 0.10.
11-20	2nd	F	If relative residual checking: Maximum allowable value of maximum residual moment divided by maximum reaction moment. Default is 0.0, in which case, no checking on residual moment occurs. If relative displacement checking: Maximum allowable value of the change in rotational increment divided by the rotational increment. Default is 0.0, in which case, no checking on change in rotational increment occurs.
21-30	3rd		If relative residual checking: Minimum reaction force, if reaction force is less than this value, checking will be bypassed or absolute testing will be performed. If relative displacement checking: Minimum displacement, if displacement increment is less than this value, checking will be bypassed or absolute testing will be performed.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
31-40	4th		If relative residual checking: Minimum moment, if moment is less than this value, checking will be bypassed or absolute testing will be performed. If relative displacement checking: Minimum rotation, if rotation increment is less than this value, checking will be bypassed or absolute testing will be performed.
41-50	5th	F	If absolute testing: Maximum value of residual force. Default is 0.0 in which case, no checking on residual force will take place.
51-60	6th	F	If absolute testing: Maximum value of residual moment. Default is 0.0 in which case, no checking on residual moments will take place.
61-70	7th	F	$\dot{\varepsilon}_\circ$ initial strain rate. (Rigid-Plastic Analysis only)
71-80	8th	F	$\dot{\bar{\varepsilon}}$ cutoff strain rate. (Rigid-Plastic Analysis only)

4th card series

Only necessary for COUPLED analysis.

1-10	1st	F	Maximum nodal temperature change allowed. Used to control automatic time step scheme for heat transfer. Default value of 20.
11-20	2nd	F	Maximum nodal temperature change allowed before properties are re-evaluated and matrices reassembled. Default value of 100.
21-30	3rd	F	Maximum error in temperature estimate used for property evaluation. This control provides a recycling capability to improve accuracy in highly nonlinear heat-transfer problems (e.g., latent heat, radiation boundary conditions). Default is 0, which bypasses this test. Set to maximum temperature error which is considered acceptable.

SOLVER

5.1.16 Specify Direct or Iterative Solver

Description

This option defines the solver to be used in the analysis. You can specify either the direct or iterative solver. The choice of whether the in-core or out-of-core procedure is used is automatically determined by the program, based upon the amount of workspace required and the number given on the SIZING option. You can also select whether the symmetric or nonsymmetric solver is used. At this time, only a direct, nonsymmetric solver is available. Additionally, you can specify if the solution of a non-positive definite system is to be obtained.

When the iterative solver is chosen, additional parameters must be defined which are used to control the accuracy.

NOTE: It is not recommended to use the iterative solvers for beam or shell models, because these problems are ill-conditioned, resulting in a large-number of iterations. For a well-conditioned system, the number of iterations should be less than (and possible much less than) the square root of the total number of degrees of freedom in the system.

NOTE: The user controls the maximum number of iterations allowed. If this is a positive number, the program will stop if this is exceeded. If this is a negative number, the program will print a warning and continue to the next Newton-Raphson iteration or increment.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word SOLVER.
2nd card series			
1-5	1st	I	0 = Profile Direct Solver. 1 = EBE Iterative. 2 = Sparse Iterative. 3 = Hardware Provided. 4 = Sparse Direct Solver
6-10	2nd	I	Enter 1 if the nonsymmetric solver is to be used.
11-15	3rd	I	Enter 1 if the solution of nonpositive definite system is to be obtained.
3rd card series			
Only necessary if EBE iterative solver is used.			
1-5	1st	I	Enter maximum number of groups; defaults to 30 times the number of element types.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
6-10	2nd	I	Enter 1 if the previous solution is to be used as the initial trial solution.
11-15	3rd	I	Enter maximum number of conjugate-gradient iterations. Default is 1000, see note on previous page.
21-25	4th	I	Enter the type of preconditioner: Enter 0 for no preconditioner. Enter 1 for Cholesky preconditioner.

4th card series

Only necessary if EBE iterative solver is to be used.

1-10	1st	F	Enter tolerance on conjugate gradient convergence for stress analysis. Default is 0.01.
11-20	2nd	F	Enter tolerance on conjugate gradient convergence for heat transfer analysis.
21-30	3rd	F	Condition number cut-off. If calculated condition number is less than this value, the analysis will be stopped.

3rd card series

Only necessary if the sparse iterative solver is used.

1-5	1st	I	Enter maximum number of conjugate-gradient iterations. Default is 1000, see note on previous page.
6-10	2nd	I	Enter 1 if the previous solution is to be used as the initial trial solution.
11-15	3rd	I	Enter a 3 for diagonal preconditioner. Enter a 4 for scaled-diagonal preconditioner.

4th card series

Only necessary if the sparse iterative solver is to be used.

1-10	1st	F	Enter tolerance on conjugate gradient convergence for stress analysis. Default is 0.001.

POST INCREMENT

5.1.17 Define Increments between Writing on Post File

Description

This option allows the user to alter the increments at which data is written to the post file. This option has the same effect as the data in the ninth field of the POST Model Definition Option.

Format

FORMAT		DATA	
FIXED	FREE	TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the words POST INCREMENT.
2nd card series			
1-5	1st	I	Enter the number of increments between writing of post data. Defaults to write post file very increment. Enter a -1 to turn off all writing of post data until the next POST INCREMENT option.

NOTES: Post data is automatically written to the post file at the increment in which the POST INCREMENT option occurs.

This value is not saved upon restart; it must be reset through the POST model definition option or POST INCREMENT option.

Example:

POST INCREMENT
2

will write every other increment to the post file beginning with the current increment.

GAP CHANGE

5.2.2 Redefine Data for Gap Elements

Description

This option allows the user to modify the data associated with gap elements. This data includes gap closure distance, gap elastic stiffness, contact coefficient of friction, and momentum ratio.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st Card Series			
1-10	1st	A	Enter the words GAP CHANGE.
2nd Card Series			
1-5	1st	I	Number of sets of gap data to be input.
6-10	2nd	I	Unit number for reading data. Defaults to input.
The 3rd and 4th card series are entered as pairs, once for each set of gap data.			
3rd Card Series			
1-10	1st	F	For a fixed direction gap, enter the gap closure distance U_{cl}. For a true distance gap, enter the minimum distance d between end points.

NOTE: If d>0, the two end points will never be closer than a distance |d| apart. If d<0, the two end points will never be farther apart than |d|.

11-20	2nd	F	μ, the contact coefficient of friction.
21-30	3rd	F	K_{GAP}, the elastic stiffness of the closed gap in the contact direction. Default: Gap is rigid when closed.
31-40	4th	F	$K_{FRICTION}$, the elastic stiffness of the closed gap in the friction direction. Default: Gap is rigid when closed.
41-50	5th	F	User supplied momentum ratio for first gap node. Default: The MARC program will calculate this ratio internally.
51-60	6th	F	User supplied momentum ratio for fourth gap node. Default: The MARC program will calculate this ratio internally.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
61-65	7th	I	Enter 0 for fixed direction gap. Enter 1 for true distance gap. Default is 0.
66-70	8th	I	Enter 0 if gap is open during increment 0. Enter 1 if gap is closed during increment 0. Default is 0.
4th card series			
			Enter a list of gap elements to be associated with this set of gap data.

TYING CHANGE

5.2.3 Define Tying Constraints

Description

This allows the number of tying constraints to be modified or a totally new series of tying constraints to be introduced.

NOTES: The use of TYING CHANGE may increase the bandwidth beyond that calculated for the original space allocation, and therefore, the program will recalculate the nodal bandwidth and the storage allocation for the assembly and solution part of the program.

To completely remove a set of tying constraints set column 5 to 1 and column 10 to 0.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-12	1st	A	Enter the words TYING CHANGE.
2nd card series			
1-5	1st	I	Set equal to 1 to reduce the number of tying constraints at this point of the analysis. Set equal to 2 to read in an entire new set of tying constraints. If column 5 is set to 1 the new number of tying constraints has to be less than the originally specified number of ties. The tying constraints are deleted from the end of the list to the desired number of remaining ties. The list is in the same sequence as the list of ties in the input file.
6-10	2nd	I	New number of tying constraints. The cards required by the tying option are read in next if column 5 of this card is set to 2, except for the key word card.

DIST LOADS

5.2.4 Define Distributed Loads

Description

This option allows pressure (surface and volumetric) loads to be specified.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	DIST LOADS.
11-15	2nd	I	Enter a 1 if distributed load is to be applied as excitation loads in a harmonic analysis.
2nd card series			
1-5	1st	I	Enter the number of sets to distributed loads to be entered (optional).
6-10	2nd	I	Enter logical unit number for input of distributed load data, defaults to input.

The 3rd and 4th card series are entered as pairs, one for each data set.

3rd card series

(3a)

Use if not harmonic analysis.

1-5	1st	I	Parameter identifying the type of load. See library element description in Volume B.
6-15	2nd	F	Enter the magnitude of this type of distributed load. For load type 102, enter the magnitude of force per unit mass in first coordinate direction.
16-25	3rd	F	For load type 102, enter the magnitude of force per unit mass on second coordinate direction.
26-35	4th	F	For load type 102, enter the magnitude of force per unit mass in third coordinate direction.
36-40	5th	I	Distributed load index. (Distributed load index is to be used in subroutine FORCEM).

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
card series (3b)			
Use if harmonic analysis.			
1-5	1st	I	Parameter identifying the type of load. See library element description in Volume B.
6-15	2nd	F	Enter the magnitude of this type of distributed load (real component).
16-25	3rd	F	Enter the imaginary magnitude of distributed load if harmonic subincrement.
26-35	4th	F	For load type 102, enter the real component of force per unit mass in second coordinate direction.
36-45	5th	F	For load type 102, enter the imaginary component of force per unit mass in second coordinate direction.
46-55	6th	F	For load type 102, enter the real component of force per unit mass in third coordinate direction.
56-65	7th	F	For load type 102, enter the imaginary component of force per unit mass in second coordinate direction.
4th card series			
			Enter a list of elements to which the above distributed loads are applied.

POINT LOAD

5.2.5 Define Point Loads

Description

This block of data allows nodal point loads to be specified.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	POINT LOAD.
11-15	2nd	I	Enter a 1 to enter real harmonic load. Enter a 2 to enter imaginary harmonic load.
2nd card series			
1-5	1st	I	Enter number of sets of point loads to be entered (optional).
6-10	2nd	I	Enter logical unit number for input of point load data; defaults to input.

The 3rd and 4th card series are entered as pairs, once for each data set.

3rd card series			
1-10	1st	F	Nodal load associated with first degree of freedom.
11-20	2nd	F	Nodal load associated with second degree of freedom.
21-30	3rd	F	Nodal load associated with third degree of freedom.
31-40	4th	F	Nodal load associated with fourth degree of freedom.
41-50	5th	F	Nodal load associated with fifth degree of freedom.
51-60	6th	F	Nodal load associated with sixth degree of freedom.
61-70	7th	F	Nodal load associated with seventh degree of freedom.
71-80	8th	F	Nodal load associated with eighth degree of freedom.

Continuation card is necessary; must be in 6E10.3 format. Continuation cards are needed if there are more than eight degrees of freedom per node in the analysis.

PROPORTIONAL INCREMENT

5.2.6 Define Proportional Increments

Description

Using this option, the current load increment can be scaled up or down for use in the next load increment. This is most frequently used in elastic-plastic analysis where the first load increment is scaled up to the values that cause first yield. This option governs mechanical loads only; temperature changes are independent of this proportioning.

The option can precede or follow all the other options in this optional series. If it precedes a DIST LOADS, POINT LOAD, or DISP option, these options reset the proportionality factor to 1.0. If it follows either of these options, it will also scale any nonzero load or displacement increments given in these options.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-22	1st	A	Enter the words PROPORTIONAL INCREMENT.
2nd card series			
1-5	1st	I	Minimum number of cycles for each step of one increment; normally 1 (forces recycling n times). Every recycle may cause an assembly and triangularization of the stiffness matrix. The program will automatically recycle if convergence to tolerance is not achieved. The default value of recycles is 0. If this value is set above 1, more cycles are allowed but each increment is forced to cycle at least n times before solution. Use caution that no unnecessary recycling is being forced (e.g. in AUTO LOAD). Recycling is usually forced for the first few critical load steps to get convergence and then resume normal condition.
6-20	2nd	F	Ratio of the next increment of load to the present increment. Only mechanical loads and kinematic boundary conditions will be scaled.

It is permissible to follow this by a DIST LOADS, POINT LOAD, or CHANGE STATE option to change some or all of the load and temperature vectors.

Note that if the SCALE parameter card is used, the load increment that is applied in the first increment is the scaled load multiplied by the value given in the second field.

AUTO INCREMENT

5.2.7 Define Automatic Load Stepping

Description

This option allows automatic load stepping in a quasi-static analysis and is very useful for both geometric (LARGE DISP) and material (elastic-plastic) nonlinear problems. The option is capable of handling elastic/plastic snap- through phenomena and hence the post-buckling behavior of structures can be analyzed.

The user has to specify in the DIST LOADS, POINT LOAD and/or DISP CHANGE options the total loading for a sequence of load steps, and the program will automatically generate the magnitude of each load step based on an initial load step and the amount of nonlinearity occurring during the loading.

The length of the incremental displacement vector is based on the number of recycles in the previous increment. The size of the load increment is controlled by the length of the incremental displacement vector. The analysis is stopped when the total load is reached or when the maximum allowed number of increments is reached. In case of a snap-through problem, the loading can initially increase, decrease after the buckle load has been reached, and increase if the stiffness increases in the post-buckled state. Within the history definition deck, the AUTO INCREMENT option can be used as often as desired.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-14	1st	A	Enter the words AUTO INCREMENT.
2nd card series			
1-10	1st	F	Fraction of the total load increment that should be applied in the first cycle of the first increment of this AUTO INCREMENT session.
11-15	2nd	I	Maximum number of increments during this AUTO INCREMENT session.
16-20	3rd	I	Desired number of recycles per increment. Used to increase or decrease load steps during AUTO INCREMENT session. Default is 5. Please allow for more recycles via CONTROL model definition data.
21-30	4th	F	Maximum fraction of the total load that may be applied in any increment of this AUTO INCREMENT session. Default is 1 if no contact is present. Default is 0.05 if contact is present.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
31-40	5th	F	Maximum multiplier of applied arc length in norm of displacement vector to initial arc length. Defaults to maximum fraction of load divided by initial fraction of load.
41-50	6th	F	Total time period to be covered, to be used in conjunction with CONTACT analyses. Default is 1.0.

NOTES: Upon restart, before reading history definition data, this AUTO INCREMENT session is finished. The maximum number of increments allowed, the desired number of recycles, and the maximum step size for this session can be changed upon restart using the REAUTO model definition card.

The option cannot be used for thermal loading; use AUTO THERM instead.

If this option will be used for post-buckling analysis, the nonpositive definite flag in the SOLVER model definition option has to be used. This card may be added upon restart.

THERMAL LOADS

5.2.8 Define Thermal Loads

Description

This option allows input of temperature and other state variables (see STATE VARS parameter card). Used here, the loads are incremental; *i.e.*, they are in addition to any loads previously applied. The loads are total loads only if the ELASTIC parameter card is used.

One can specify either a uniform or nonuniform change in temperature (or other state variables). If a nonuniform change is desired, then the change of every state variable at every layer of every integration point of every element must be specified. In this case, the program calls user subroutine CREDE for every element in the mesh. (See Model Definition option THERMAL LOADS for more information.)

If the Fourier decomposition method is being used to analyze an arbitrarily loaded axisymmetric structure, then the THERMAL LOADS option must be invoked separately for each Fourier series term that has temperatures (state variables) associated with it. If there is no variation of these variables in the circumferential direction, then only the zeroth term of the series should be specified.

Note that on a restart run any THERMAL LOADS option before END OPTION will read data.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-13	1st	A	Enter the words THERMAL LOADS.
2nd card series			
1-5	1st	I	Set to 1 if uniform increment temperature (state variable) increment is applied to all elements. Set to 2 if non-uniform incremental total temperature (state variable) will be read via user subroutine CREDE. Set to 3 if non-uniform total temperature (state variable) will be read via user subroutine CREDE.
3rd card series			
1-80	1st	F	Include only if the first field of card 2 is 1; enter the uniform increments in temperature and any additional state variables in (8E10.3) format; will be applied to all elements.

POINT TEMP

5.2.12 Define Point Temperatures

Description

This option defines temperatures at nodal points for an uncoupled thermal stress problems at the end of the increment.

> **NOTE:** For shell analyses, a uniform temperature will be used through the thickness direction.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the words POINT TEMP.
2nd card series			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional).
6-10	2nd	I	Enter tape number for input of prescribed temperatures data, defaults to input.
11-15	3rd	I	Flag to indicate that temperatures are read from previously generated POST tape. Set to 1.
16-20	4th	I	Only nonzero if the third field is set to 1. Then this entry defines the unit number from which the POST tape information will be read.
21-25	5th	I	Enter step number to be read.
26-30	6th	I	Enter a 1 if a formatted POST tape is used.

Card series 3 and 4 are given in NSET pairs, only if the flag in the third field is not equal to 1.

3rd card series			
1-10	1st	E	Temperatures at the end of the increment.
4th card series			
			Enter list of nodes for which the above initial temperature is applied.

MOTION CHANGE

5.2.20 Define Motion of Rigid Surfaces

Description

This option is useful for prescribing the motion of rigid bodies when the CONTACT option is used. This option can be used to call user subroutine MOTION to change the rigid body speed or coefficient of friction.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word MOTION CHANGE.
2nd card series			
1-5	1st	I	Enter the number of sets of rigid bodies to be input.

The following cards (3 and 4) are entered as pairs, once for each data set.

2-D contact problems

3rd card series			
1-5	1st	I	Enter the rigid body number.
4th cards series			
1-10	1st	F	First component of velocity of center of rotation.
11-20	2nd	F	Second component of velocity of center of rotation.
21-30	3rd	F	Angular velocity (in radian/unit time) about center of rotation.
31-40	4th	F	Friction coefficient.

3-D contact problems

3rd cards series			
1-5	1st	I	Enter the rigid body number.
4th card series			
1-10	1st	F	First component of velocity of center of rotation.
11-20	2nd	F	Second component of velocity of center of rotation.
21-30	3rd	F	Third component of velocity of center of rotation.
31-40	4th	F	Angular velocity (in radian/unit time) about center of rotation.
41-50	5th	F	Friction coefficient.

RELEASE

5.2.21 Define Release Data

Description

This option is useful for the analysis of spring-back after bodies contact one another. The body number is entered and then all of the nodes which contact that body will be released at the beginning of the increment. The contact force may either be immediately removed or gradually reduced. In addition, the body must be moving away to avoid nodes recontacting during the same increment.

Format

FORMAT		DATA	
FIXED	FREE	TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word RELEASE.
11-15	2nd	I	Enter a 0 if the contact forces are to be immediately removed (default). Enter a 1 if the contact forces are to be reduced to zero over the number of increments specified in this load period.
2nd card series			
			Enter a list of bodies for which nodes currently in contact will be released.

RELEASE NODE

5.2.22 Define Nodes for Which the Boundary Condition is Gradually Released

Description

This option will remove a boundary condition constraint from a node in a gradual manner. This option is similar to changing boundary conditions, but it allows the reaction force to be brought to a zero value over a series of increments. The load will be reduced in equal steps if used in conjunction with the AUTO LOAD or DYNAMIC CHANGE option. The load will be proportionally reduced to zero if used with the AUTO TIME or AUTO INCREMENT option. If the RELEASE NODE and DISP CHANGE are given in the same load incrementation section, the DISP CHANGE option should be given first.

NOTE: This option should not be applied to nodes in contact with rigid surfaces.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word RELEASE NODE.
2nd card series			
1-5	1st	I	Enter the number of sets of data; this must be given.
6-10	2nd	I	Enter unit number for input of release data, defaults to input.
3rd card series			
			Enter a list of degrees of freedom to be released.
4th card series			
			Enter a list of nodes to be released.

ACTUATOR

5.2.23 Define the Length of the Actuator Link

Description

This option may be used in conjunction with the truss element, type 9 to simulate an actuator. This is often used in mechanism analyses to allow the prescription of the relative distance between two points. This option should be used with the LARGE DISP option whenever large rotations of the actuator or large displacements are anticipated.

The original length of the actuator is given in the fourth field of the GEOMETRY option. The actuator is treated as an elastic link.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word ACTUATOR.
2nd card series			
1-5	1st	I	Enter the number of actuators (optional).
6-10	2nd	I	Enter unit number for input of actuator data, defaults to input.
3rd card series			
(Repeat for each actuator to be modified.)			
1-5	1st	I	Enter the element number
6-15	2nd	F	Enter the new length of the actuator.

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
46-50	6th	I	Not used; enter 0.
51-60	7th	F	Enter stable time step limit, if known. The MARC program uses stresses and strain change tolerances if this is not used. Stable time step limit is needed for viscoplasticity.
3rd card series			
1-10	1st	F	If the fifth field is a 0, enter tolerance on the creep strain increment relative to the elastic strain. Default is 0.50. Note that a higher value is likely to cause stability problems. If the fifth field is a 1, enter the maximum creep strain increment allowed. Default is .01.
11-20	2nd	F	If the fifth field is a 0, enter the tolerance on the stress change per stress during creep. Default is 0.10. If the fifth field is a 0, enter the maximum stress increment. Default is 100. This control is included primarily for accuracy purposes. The default value is adequate for creep laws of the type $\varepsilon = A\sigma^n$, where $3<n<6$. For lower values of n, the tolerance can be increased; for higher values, it should be decreased.
21-30	3rd	F	Tolerance on low stress point cut-off. Points with a stress lower than this ratio relative to the maximum stress in the structure will not be used in the creep tolerance checking. Default is 0.05.
31-35	4th	I	Number of the element in which the stress change will be checked. Leave blank to check all elements for stress change. If a number of elements (but not all elements) are to be checked, enter the number of elements as a negative number, with 14 as the maximum. In this case, the actual elements are entered on the next card.
36-40	5th	I	Enter a 1 if absolute rather than relative testing is to be performed.
4th card series			
This series is only required if the entry in the seventh field of the previous card is negative.			
1-70	1st	I	Enter the elements to be checked in (14I5) format.

ACCUMULATE

5.3.3 Specify Accumulation Option

Description

This flags the start of accumulation of strains and displacements for use with the extrapolate option. If a new accumulation period is to be started immediately after an extrapolation in the same increment, the ACCUMULATE option must be preceded by the EXTRAPOLATE option.

Format

FORMAT		DATA	
FIXED	FREE	TYPE	ENTRY
1-10	1st	A	Enter the word ACCUMULATE.

5.4.3 Define Integration in Time

Description

This card series specifies the parameters required for integration in time. It may be used for either the modal or the direct integration procedure. See *Volume A* on Dynamic Options and the DYNAMIC, ACOUSTIC, or EL-MA parameter card (Chapter 2 of this Volume).

In the case of explicit analysis, IDYN=5, the time step will be adjusted each increment to insure that the stability limit is not violated.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-14	1st	A	Enter the words DYNAMIC CHANGE.
2nd card series			
1-15	1st	F	Time step size.
16-30	2nd	F	Period of time for this set of boundary conditions.
31-35	3rd	I	Number of time steps in this set of boundary conditions.
36-40	4th	I	This field is not used.
41-45	5th	I	Reassembly interval for mass and stiffness matrices; for linear problems, set equal to value in the third field.
46-50	6th	I	This field is not used, enter a 0.
51-60	7th	F	Enter γ for Newmark operator, default is γ=0.5 or what was used in previous DYNAMIC CHANGE option.
61-70	8th	F	Enter β for Newmark operator, default is β=0.25 or what was used in previous DYNAMIC CHANGE option.

AUTO TIME

5.4.4 Automatic Time Stepping

Description

This option allows automatic time-stepping in dynamic analysis or in coupled thermal-stress analysis (either quasi-static or dynamic). This option is able to handle either linear or nonlinear analysis. When used for dynamic analysis, the Newmark beta operator must be used (as this is the only one which allows variable time steps).

The user has to specify (in the POINT LOAD, DIST LOADS, and/or FIXED DISP options), the total loading for a sequence of load steps, and the program will automatically generate the magnitude of the load step such that equilibrium is satisfied with a minimum number of iterations.

The size of the load vector is controlled by the error in equilibrium of the previous increment/ iteration. The analysis is stopped when the total load is reached or when the maximum allowed number of increments is reached.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-9	1st	A	Enter the words AUTO TIME.
2nd card series			
1-10	1st	F	Initial fraction of load/time.
11-20	2nd	F	Maximum fraction of load/time to apply in an increment, AUTMAX. Default is 1.0.
21-30	3rd	F	Initial time step.
31-40	4th	F	Total time period to be covered (TOTINC).
41-50	5th	F	Smallest reduction factor, FCSML. Default is .5.
51-60	6th	F	Largest increase factor, FCLRG. Default is 2.0.
61-65	7th	I	Maximum number of increments, IOTNUM.
66-70	8th	I	Desired number of recycles per increment.
71-80	9th	F	Minimum time step allowed.

NOTE: The increment of time in a step n+1 is always in the range:

$$FCSML*\Delta t^{n} \leq \Delta t^{n+1} \leq FCLRG*\Delta t^{n}$$

and

$$\Delta t^{n+1} \leq AUTMAX*TOTINC$$

5.5.7 Define Film Coefficients and Sink Temperatures

Description

This block allows film coefficients and associated sink temperatures to be input. Nonuniform films or sink temperatures may be obtained via user subroutine FILM, see *Volume D*.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-5	1st	A	Enter the word FILMS.
2nd card series			
1-5	1st	I	Number of sets of data used to input film (optional).
6-10	2nd	I	Unit number for input of film data, defaults to input.
The 3rd and 4th card series are entered as pairs, once for each data set.			
3rd card series			
1-5	1st	I	Face identification. Same as for FLUX – see Volume B, "Library Element Descriptions".
6-15	2nd	F	Reference value of film coefficient.
16-25	3rd	F	Reference value of sink temperature (reference values may be modified by subroutine FILM).
26-30	4th	I	Film coefficient index (optional).
31-35	5th	I	Sink temperature index (optional). (Film coefficient and sink temperature indices are to be used in subroutine FILM).
4th card series			
			Enter a list of elements to which the above film data is applied.

VELOCITY CHANGE

5.5.8 Modify Nodal Velocity Components

Description

This option allows the specification of the nodal velocity components in a heat transfer analysis, where the convective terms are to be included. The convective option is specified by placing a 2 in the fifth field of the HEAT parameter option. The nodal velocity components are defined by specifying the velocity magnitude of a series of components for sets of nodes. This data may be input from cards or from an auxiliary input device. Moreover, the velocity values may be respecified, or initialized if no previous data was entered via user subroutine UVELOC. See *Volume D*, Section 1.

A summary of nodal velocities will appear in the printout. This can be suppressed by specifying a nonzero value for the print-suppress parameter.

Format

FORMAT FIXED	FORMAT FREE	DATA TYPE	ENTRY
1st card series			
1-10	1st	A	Enter the word VELOCITY CHANGE.
2nd card series			
1-5	1st	I	Number of sets of cards used to input nodal velocity components. If a negative value is entered, user subroutine UVELOC will be called for every node.
6-10	2nd	I	Enter the unit number for input of the velocity field. Default to unit 5, unless the INPUT TAPE parameter card has been used.
11-15	3rd	I	Set to 1 to suppress printout of the summary of nodal velocity components.

Card series 3, 4, and 5 should be repeated for each data set.

3rd card series			
1-10	1st	F	Enter the magnitude of the velocity in the first coordinate-direction for which the velocity will be given. Additional velocity components in other coordinate directions must be specified on the same card in F10 format. The number of components must equal the number of directions.

FORMAT		DATA	
FIXED	FREE	TYPE	ENTRY

4th card series

Enter a list of coordinate directions in which the velocity will be specified.

NOTE: List verbs EXCEPT and INTERSECT are illegal here.

5th card series

Enter a list of nodes for which the velocity vector as defined in card series 3 and 4 applied

Exit Numbers 2001-3000

Exit Number	Explanation
2004	The determinant of the stiffness matrix becomes zero or negative when the indicated node has been reached during Gaussian elimination during direct solution. This means the stiffness matrix is nonpositive definite. This condition is usually caused by failure to remove all rigid body modes, or by incorrect material properties (e.g., Poisson's ratio > 0.5: Note these situations may arise through temperature dependence of properties), or by giving clockwise numbering to plane elements. In nonlinear cases, the structure may have buckled or reached a plastic limit load. If desired, the program may be forced to continue by setting IDEV=3 (PRINT card) or setting columns 31-35 on the CONTROL card, but this is only recommended in combination with the AUTO INCREMENT method.
2006	Maximum half-bandwidth exceeded. This error is detected just before out-of-core solution. MARC System error; consult analyst.
2007	System error in sparse conjugate gradient solver; contact MARC.
2008	Maximum connectivity has been exceeded during application of tying constraints. Check the TYING option data. MARC System error; consult analyst.
2009	Normal exit caused by asking for intermediate restart (before tying).
2010	Normal exit caused by asking for intermediate restart (after tying).
2011	Errors encountered during application of TYING equations; printout indicates specific problem.
2012	One of the surface directions in Tying Type 22 (intersecting shells of type 22) has zero length. This is caused by bad surface normal coordinate at one of the nodes being tied.
2014	Search vector for eigen extraction is zero. Caused by inadequate guess vector, or asking for more eigenvalues than the system contains. In last case, ask for fewer eigenvalues. In first case, consult analyst.
2015	MARC system error – not enough work space to do fluid/solid interface calculations; consult analyst.
2016	MARC system error – not enough work space to do fluid/solid interface calculations; consult analyst.
2017	MARC system error – not enough work space to do fluid/solid interface calculations; consult analyst.
2018	MARC system error – not enough work space to do fluid/solid interface calculations; consult analyst.

Exit Number	Explanation
2020	Conjugate gradient iterative solver fails to converge in required number of iterations.
2100	The partial element stiffness generation has ended as requested by the EDIT/REGEN option.
2109	Mode with zero energy is found during modal response. Probable cause is failure to ask for MODAL SHAPE option before DYNAMIC CHANGE is used.
2201	Attempt to do a recover substructure output, but have not yet solved for external nodes.
2300	Attempt to do application of boundary conditions in complex analysis using APPBC option. This is currently not available.
2400	A boundary area node tried to slide out of surface definition.
2401	A node that was considered in contact with a segment, slipped away. Check your die geometry definition. Consult analyst.

Exit Numbers 3001-4000

Exit Number	Explanation
3001	The maximum number of increments on the CONTROL option, card 2, columns 1-5 has been reached.
3002	One of several error conditions has been detected and the run aborted. Any restart written during the current increment is probably unusable. A message occurring during the current increment indicates the cause of the error. The following are the more common messages:

3002(a)

"NO CONVERGENCE TO TOLERANCE IN SPECIFIED CYCLES"

This means the stiffness approach fails to find a convergent displacement increment.

The tolerance is set in columns 1-10, card 3, CONTROL option; and the number of cycles in columns 1-5, card 2, PROPORTIONAL INCREMENT option or columns 16-20, card 2, CONTROL option. This condition is most usually caused by too tight a tolerance, too large a load step, or the solution reaching buckling load or limit load.

Volume D: User Subroutine Special Routines K6.2 Update Packet

Update Packet

The following pages with a • by the page number have been either modified or added to the *Volume D, User Subroutines Special Routines* manual to reflect changes in the K6.2 release.

Index of Subroutines

Replacing Pages:	New Pages:
xiii through xiv	xiii through xiv

Chapter 1: Introduction

D 1-3 through D 1-4	D 1-3 through D 1-4

Chapter 1: Section 2

D 1-89 through D 1-90	D 1-89 through D 1-90
D 1-103 through D 1-104	D 1-103 through D 1-104
D 1-131 through D 1-132	D 1-131 through D 1-132
D 1-139 through D 1-144	D 1-139 through D 1-144
D 1-157 through D 1-160	D 1-157 through D 1-160
D 1-163 through D 1-164	D 1-163 through D 1-164
D 1-171 through D 1-172	D 1-171 through D 1-172

Chapter 1: Section 5

D 1-205 through D 1-208	D 1-205 through D 1-208
	D 1-216.1 through D 1-216.2

Chapter 2:

D 2-13 through D 2-14	D 2-13 through D 2-14

Volume D
User Subroutines & Special Routines

K6.2 Updates

Page

R

S

T

U

Page

V

W

Y

Z

Note on Double Precision

In the K5 and subsequent versions, the MARC program is written completely in double precision. Hence, on all machines (excluding the CRAY), an IMPLICIT REAL *8 (A-H, O-Z) statement is required in the user subroutines. This is to ensure that variables passed between the MARC program and the user routine are compatible and to ensure that any common blocks included will be correct.

Note on Parallel Processing

In the K5 version, the user subroutines are not called from within loops executing in parallel, and hence the use of common blocks is generally acceptable. In subsequent versions, this will not be true, and variables associated with a particular element will not be available through the common blocks used in the past. The analysis portion of MARC has already been restructured to handle this.

Format

The following quantities are available in all user subroutines:

```
TIME AT BEGINNING OF INCREMENT:    CPTIM
TIME INCREMENT:                    TIMINC
AVAILABLE THROUGH
       include 'path/common creeps'

INCREMENT NUMBER:             INC
SUBINCREMENT NUMBER:          INCSUB
AVAILABLE THROUGH
       include 'path/common/concom'

NUMBER OF ELEMENTS IN MESH:        NUMEL
NUMBER OF NODES IN MESH:           NUMNP
MAXIMUM NUMBER OF DEGREES OF FREEDOM PER NODE:     NDEG
MAXIMUM NUMBER OF COORDINATE DIRECTIONS:           NCRD
AVAILABLE THROUGH
       include 'path/common/dimen'
```

In a coupled analysis, reference variable **IPASS** to determine if the current iteration is a stress or heat transfer iteration:

```
IPASS = 1          STRESS
IPASS = 2          HEAT TRANSFER
AVAILABLE THROUGH
       include 'path/common/concom'
```

The following quantities are available in user subroutines which are in an element loop:

```
ELEMENT NUMBER:     M
AVAILABLE THROUGH
       include 'path/common/far'

ELSTO ELEMENT NUMBER:         N
```

```
INTEGRATION POINT NUMBER:     NN
LAYER NUMBER:                 KC
AVAILABLE IN
       include 'path/common/lass'

NUMBER OF NODES IN ELEMENT:  NNODE
AVAILABLE IN
       include 'path/common/elmcom'

NUMBER OF DIRECT COMPONENTS OF STRESS:  NDI
NUMBER OF SHEAR COMPONENTS OF STRESS:   NSHEAR
SIZE OF STRESS STRAIN LAW:              NGENS
ELEMENT TYPE:      JTYPE
AVAILABLE THROUGH
       include 'path/common/elmcom'

MAXIMUM NUMBER OF LAYERS PER ELEMENT:    NEQST
MAXIMUM NUMBER OF INT. PTS PER ELEMENT:  NSTRES
AVAILABLE THROUGH
       include 'path/common/nzro1'
```

To determine the coordinates of integration point **NN** of element **M** and to place these coordinates in array **CCINT**, use the following procedure:

```
include 'path/common/lass'
include 'path/common/dimen'
include 'path/common/space'
include 'path/common/heat'
include 'path/common/array4'
DIMENSION CCINT(12)
LA1 = (N-1)* NELSTR + ICRXPT + (NN-1)*NCRD
D0 1 II = 1, NCRD
CCINT(II) = VARS(LA1)
1 LA1 = LA1 + 1
```

NOTE: This is only available after the first stiffness matrix assembly.

To obtain the array of internal node numbers of an element, use variable **LM**:

```
include 'path/common/blnk'
```

The first NNODE numbers of LM are the internal node numbers.

To determine the internal node number **LINT** of user (external) node number **LEXT** use the following procedure:

```
include 'path/common/dimen'
include 'path/common/arrays'
include 'path/common/develp'
include 'path/common/space'
LINT = IBSRCH (LEXT, INTS (INOIDS), NUMNP,1)
```

Table 1.2.1-1: Allowable Anisotropy (Continued)

Library Element Number	Allowable Transformations to Preferred Operation	Size of R. Matrix (IRDIM) for IRDIM=1 No Anisotropy Possible	Number of Direct Stresses (NDI)	Number of Shear Stresses (NSHEAR)
61	Orthogonal in (x,y,z) space	6	3	3
62	Orthogonal in z-r plane	6	3	3
63	Orthogonal in z-r plane	6	3	3
64	None	1	1	0
65	None	1	0	0
66	Orthogonal in z-r plane	6	3	3
67	Orthogonal in z-r plane	6	3	3
68	None	1	0	1
69	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
70	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
71	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
72	Orthogonal in $\underline{V}^1$ -$\underline{V}^2$	3	2	1
73	Orthogonal in z-r plane	6	3	3
74	Orthogonal in z-r plane	6	3	3
75	Orthogonal in $\underline{V}^1$ -$\underline{V}^2$	5	2	3
76	None	1	1	1
77	None	1	1	0
78	None	1	1	1
79	None	1	1	0
80	Orthogonal in x-y plane	4	3	1
81	Orthogonal in x-y plane	4	3	1
82	Orthogonal in z-r plane	4	3	1
83	Orthogonal in z-r plane	4	3	1
84	Orthogonal in (x,y,z) space	6	3	3

Table 1.2.1-1: Allowable Anisotropy (Continued)

Library Element Number	Allowable Transformations to Preferred Operation	Size of R. Matrix (IRDIM) for IRDIM=1 No Anisotropy Possible	Number of Direct Stresses (NDI)	Number of Shear Stresses (NSHEAR)
85	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
86	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
87	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
88	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
89	None	1	2	1
90	None	1	2	3
91	Orthogonal in x-y plane	4	3	2
92	Orthogonal in z-r plane	4	3	1
93	Orthogonal in x-y plane	4	3	1
94	Orthogonal in z-r plane	4	3	1
95	Orthogonal in z-r plane	6	3	3
96	Orthogonal in z-r plane	6	3	3
97	None	1	0	0
98	None	1	1	2
99	None	-	-	-
100	None	-	-	-
101	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
102	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
103	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
104	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-
105	Use subroutine ANKOND to supply anisotropic conductivity	-	-	-

ANKOND

1.2.7 Input of Anisotropic Thermal Conductivity Matrix

Description

For anisotropic heat transfer analysis, this subroutine allows the user to define an anisotropic conductivity matrix at each integration point in each element. The anisotropic conductivity matrix is defined with respect to the preferred orientation specified in **ORIENTATION** option. This routine is also used for anisotropic electrical resistance in a Joule heating analysis.

Format

Subroutine **ANKOND** is written with the following header cards:

```
 SUBROUTINE ANKOND (COND,CANISO,N,NN,KC,MATNO,ID,T,DT,TIME,DELTME,
* JOULHT)
 IMPLICIT REAL *8 (A-H, O-Z)
 DIMENSION COND(ID,ID),CANISO(3)

         user coding

 RETURN
 END
```

where

COND is the conductivity matrix, k_{ij}:

$$q_i = k_{ij}\frac{\partial T}{\partial x_j}$$

This is to be re-defined as necessary by the user.

This matrix is passed in as set-up for anisotropic conductivity, so that if the user does not re-define it, it remains anisotropic according to $k_{ji}(T)$ given on the **ISOTROPIC/ORTHOTROPIC** and **TEMPERATURE EFFECTS** data cards.

CANISO are the anisotropic conductivities k_{ji} (T) established by the user via data cards.

N is the element number.

NN is the integration point number.

KC is the layer number.

MATNO is the material identifier.

ID	is the size of the **COND** matrix, i.e., the number of derivatives. $\frac{\partial T}{\partial x_j}$
T	is the temperature at the beginning of the time increment.
DT	is the estimated temperature increment.
TIME	is the transient time at the beginning of the increment.
DELTME	is the increment of time.
JOULHT	=0 return thermal conductivity.
	=1 return electrical conductivity.

UVOIDN

1.2.20 Definition of the Void Nucleation Rate

Description

This subroutine allows the definition of the void nucleation rate in a material using the Gurson model. This subroutine is called if the void nucleation method under the DAMAGE model definition option is set to 3.

In this model, the yield surface is given as:

$$F = \frac{\sigma_e^2}{\sigma_m^2} + 2q_1 f\cosh\left(\frac{q_2\sigma_{KK}}{2\sigma_m}\right) - (1+q_1 f)^2 = 0$$

where:

σ_e	is the effective stress.
σ_m	is the equivalent tensile stress.
f	is the void ratio.

Format

Subroutine **UVOIDN** is written with the following header cards:

```
  SUBROUTINE UVOIDN(A,B,M,NN,KC,MATS,EPL,EPLAS,S,NDI,NSHEAR,DT,
 +DTDL)
  IMPLICIT REAL *8 (A-H, O-Z)
  DIMENSION DT(1),DTDL(1),EPL(1)

        user coding

  RETURN
  END
```

where:

A	is the multiplier as shown below.
B	is the multiplier as shown below.
M(1)	is the user element number.
M(1)	is the internal element number.
NN	is the integration point number.
KC	is the layer number.

MATS	is the material id.
EPL	is the plastic strain components.
EPLAS	is the equivalent plastic strain.
S	is the stress array.
NDI	is the number of direct components.
NSHEAR	is the number of shear components.
DT	is the array of state variables, temperature first.
DTDL	is the array of increment of state variables.

In this subroutine, the full stress matrix is used, so NDI = 3, and NSHEAR = 3.
In the classical Gurson model, $q_1 = 1$ and $q_2 = 1$.
In the Tvergaard model, $q_1 = 1.5$ and $q_2 = 1$.

In this subroutine, the following type of stress controlled nucleation rate can be specified:

$$\dot{f} = \lambda \dot{\sigma}_m + D \dot{\sigma}_{kk}$$

with σ_m, the equivalent von Mises stress and σ_{kk}, the hydrostatic stress.

UENERG

1.2.24 Strain Energy Function

Description

This subroutine allows the user to define his own elastic strain energy function for incompressible materials. Normally the five constant second order model is entered using the **MOONEY** model definition option. This option must still be used to invoke this user subroutine.

Format

Subroutine **UENERG** is written with the following header cards:

```
 SUBROUTINE UENERG (W,W1,W2,W11,W12,W22,WI1,WI2,C10,C01,C11,C20,
+C30,N,NN)
 IMPLICIT REAL *8 (A-H, O-Z)
 DIMENSION N(2)

       user coding

 RETURN
 END
```

where

W is the strain energy density.

W1 is $\partial W/\partial I1$

W2 is $\partial W/\partial I2$

W11 is $\partial^2 W/\partial I1^2$

W12 is $\partial^2 W/\partial I1 \partial I2$

W22 is $\partial^2 W/\partial I2^2$

WI1 is the first strain invariant of the left Cauchy-Green tensor - 3.0.

WI2 is the second strain invariant of the left Cauchy-Green tensor - 3.0.

C10, C01, C11, C20, C30
are the five material parameters of the Mooney formulation.

N(1) is the user element number.

N(2) is the internal element number.

NN is the integration point number.

UOGDEN

1.2.25 Definition of Ogden Material Parameters

Description

This subroutine allows the definition of the Ogden material parameters. Additionally, any temperature dependence of these properties may be entered here. The OGDEN option must be used to indicate that the element uses this material law, and the number of terms in the series must be entered through the model definition option.

The strain energy function for this material is written as :

$$W = \sum_{i=1}^{n} \frac{\mu_i}{\alpha_i} (\lambda_1^{\alpha_i} + \lambda_2^{\alpha_i} + \lambda_3^{\alpha_i} - 3) + 4.5K(J^{1/3} - 1)^2$$

Format

Subroutine **UOGDEN** is written with the following header cards:

```
 SUBROUTINE UOGDEN(MATS,NSER,M,NN,KC,INC,CPTIM,TIMINC,XMTDAT,BULK,
+DT,DTDT)
 IMPLICIT REAL *8 (A-H, O-Z)
 DIMENSION XMTDAT(2,NSER),M(2),DT(1),DTDL(1)

       user coding

 RETURN
 END
```

where

MATS	is the material id.
NSER	is the number of terms in the series.
M(1)	is the user element number.
M(2)	is the internal element number.
NN	is the integration point number.
KC	is the layer number.
INC	is the increment number.
CPTIM	is the time at the beginning of the increment.
TIMINC	is the time step.

XMTDAT(1,i)	is the value of μ_i.
XMTDAT(2,i)	is the value of α_i.
BULK	is the bulk modulus.
DT	is the array of state variables, temperature first.
DTDT	is the array of increments of state variables.

UELDAM

1.2.26 Definition of Damage Parameters in Ogden Model

Description

This subroutine allows the user to define the damage parameters for the Ogden model. There are two types of damage: one is associated with the deviatoric (shear) behavior, and one is associated with the dilatational (volumetric) behavior (additional details may be found in Volume A). This routine is therefore called twice per integration point, once for deviatoric behavior and once for volumetric behavior. This routine is called only if the damage type is set to 5 through the DAMAGE model definition option.

Format

Subroutine **UELDAM** is written with the following header cards:

```
 SUBROUTINE UELDAM(M,NN,KC,INC,ITYPE,MATS,CPTIM,TIMINC,TOTEND,
+TOTENV,SURFDM,SURFVM,DT,DTDT,DAMD,DAMB,DDAMD,DDAMV)
 IMPLICIT REAL *8 (A-H, O-Z)
 DIMENSION DT(1),DTDL(1),M(2)

       user coding

 RETURN
 END
```

where

M(1)	is the user element number.
M(2)	is the internal element number.
NN	is the integration point number.
KC	is the layer number.
INC	is the increment number.
ITYPE	indicates which damage data required. = 1 enter deviatoric behavior. = 2 enter volumetric behavior.
MATS	is the material id.
CPTIM	is the time at the beginning of the increment.
TIMINC	is the time step.
TOTEND	is the instantaneous deviatoric energy.
TOTENV	is the instantaneous volumetric energy.

SURFDM	is the current radius of the damage surface in deviatoric strain space.
SURFVM	is the current radius of the damage surface in volumetric strain space.
DT	is the array of state variables, temperature first.
DTDT	is the array of increments of state variables.
DAMD	is the value of the deviatoric damage parameter.
DAMV	is the value of the volumetric damage parameter.
DDAMD	is the derivative of the damage parameter with respect to the deviatoric strain.
DDAMV	is the derivative of the damage parameter with respect to the volumetric strain.

If ITYPE = 1, then **DAMD** and **DDAMD** should be defined.

If ITYPE = 2, then **DAMV** and **DDAMV** should be defined.

USELEM

1.2.33 User-Defined Element

Description

This subroutine allows the user to calculate their own finite element stiffness or mass matrix. This may also be used as interface with other numerical techniques. In general, in the finite element calculation, several matrices are required, hence, for a particular element, this routine is called a multiple number of times. The calls and the user requirements are defined as follows:

`IFLAG=1` Return the equivalent nodal loads (`F`) given distributed surface or body loads. If the **ELASTIC, AUTO TIME, AUTO INCREMENT** or **FOLLOW FORCE** options are used, these are total loads or else incremental loads. In a heat transfer analysis, this is the total flux vector.

`IFLAG=2` Return the element tangent stiffness matrix (`K`). For an elastic analysis, this is the usual stiffness. For a heat transfer matrix analysis, this is the conductivity matrix. Also calculate the total internal forces (`R`). This is not necessary in a linear elastic analysis if the **LOAD COR** option has been turned off.

`IFLAG=3` Return the mass matrix (`M`) for a dynamic analysis or specific heat matrix for a heat transfer problem.

`IFLAG=4` Calculate the incremental strains (`DE`), generalized stresses (`GSIGS`) and the internal force (`R`). For a linear elastic solution, if only displacements are required, the user does not need to return any values.

`IFLAG=5` Output element results if so desired.

To use this option, parameter card **USER** must be included to define the size of the element stiffness matrix and other critical dimensions and the element type given on the connectivity must be a negative number.

Format

Subroutine **USELEM** calls for the following header cards:

```
      SUBROUTINE USELEM(M,XK,XM,NNODE,NDEG,F,R,
     * JTYPE,DISPT,DISP,NDI,NSHEAR,IPASS,NSTATS,NGENS,
     * NSTRES,COORD,NCRD,IFLAG,IDSS,T,DT,ETOTA,GSIGS,DE,
     * GEOM1,GEOM2,GEOM3,GEOM4,GEOM5,GEOM6, SIGXX, NSTRMX)
      IMPLICIT REAL *8 (A-H, O-Z)

      DIMENSION XK(IDSS,IDSS),XM(IDSS,IDSS),DISPT(NDEG,1),DISP(NDEG,1)
      DIMENSION T(NSTATS,1),DT(NSTATS,1),COORD(NCRD,1)
      DIMENSION ETOTA(NGENS,1),GSIGS(NGENS,1),DE(NGENS,1)
      DIMENSION F(NDEG,1),R(NDEG,1)

           user coding

      RETURN
      END
```

where

M	is the user element number
XK	is the stiffness matrix
XM	is the mass matrix
NNODE	is the number of nodes per element
NDEG	is the number of degrees of freedom per node
F	is the externally applied equivalent nodal loads array
R	is the internal forces array
JTYPE	is the element type
DISPT	is the total nodal displacements array of this element
DISP	is the incremental nodal displacements of this element
NDI	is the number of direct components of stress
NSHEAR	is the number of shear components of stress
IPASS	Flag to indicate which pass for coupled analysis
IPASS=0	during an uncoupled analysis
IPASS=1	during a heat transfer pass
IPASS=2	during a stress analysis pass
NSTATS	is the number of state variables
NGENS	is the number of generalized strains
NSTRESS	is the number of integration points

COORD	is the original nodal coordinates array
NCRD	is the number of coordinates per node
IFLAG	indicates what is to be returned by the user
IFLAG=1	Called by OPRESS during formation of load vector User returns F.
IFLAG=2	Called by OASEMB during formation of stiffness matrix User returns XK,R.
IFLAG=3	Called by OASMAS during formation of mass matrix User returns XM.
IFLAG=4	Called by OGETST during stress recovery User returns R,GSIGS,DE,ETOTA,SIGXX
IFLAG=5	Called by SCIMP during output phase User prints results
IDSS	is the size of element stiffness matrix
T	is the state variables
DT	is the increment of state variables
ETOTA	is the total strain array
GSIGS	is the generalized stress array
DE	is the increment of strain array
GEOM1	is the first geometric parameter
GEOM2	is the second geometric parameter
GEOM3	is the third geometric parameter
GEOM4	is the fourth geometric parameter
GEOM5	is the fifth geometric parameter
GEOM6	is the sixth geometric parameter.
SIGXX	is layer stresses for shell elements and is equal to GSIGS for continuum element.
NSTRMX	maximum number of stresses per integration points and is equal to NGENS for continuum element.

Note that the stiffness matrix is normally symmetric. If a nonsymmetric formulation is used, the SOLVER option should be used to indicate this.

URPFLO

1.2.35 User Routine for Rigid-Plastic Flow

Description

This subroutine allows the user to define the current yield stress as a function of the equivalent strain rate, equivalent strain, temperature, and user defined state variables. This subroutine is used in conjunction with the transient R-P FLOW option.

Format

Subroutine **URPFLO** is written with the following header cards:

```
 SUBROUTINE URPFLO(MDUM,NN,LAYERS,MATS,INC,NDI,NGENS,NCRD,NSTAT,
+CPTIM,TIMINC,EBAR,ERATE,DT,DTDL,STATS,DSTATS,COORD,YD)
 IMPLICIT REAL *8 (A-H, O-Z)
 DIMENSION MDUM(2), STATS(NSTAT), DSTATS(NSTAT), COORD(NCRD)

       user coding

 RETURN
 END
```

Where

passed into routine:

MDUM	element number
NN	integration point number
LAYERS	layer number
MATS	material id
INC	increment number
NDI	number of direct components
NGENS	total number of components
NCRD	number of coordinates
NSTAT	number of state variables excluding temperature
CPTIM	time at beginning of increment
TIMINC	incremental time
DT	temperature at beginning of increment
DTDL	incremental temperature
EBAR	total equivalent strain at beginning of increment

STATS	values of state variables excluding temperature at beginning of increment
ERATE	equivalent strain rate
COORD	integration point coordinates

to be passed back:

YD	equivalent stress; if not calculated here, program will find the value of yd from the input data
DSTATS	incremental state variables (excluding temperature)

1.3.2 Input of Special Viscoplastic Strain Rate Law

Description

Subroutine **CRPLAW** may also be used for calculating the viscoplastic behavior. All the creep quantities are treated as viscoplastic strain quantities when the appropriate flag is set on the **CREEP** parameter card.

The basic information on the use of this subroutine can be found in Section 1.2.12 of this manual. Additionally, one can use common block **VISCPL**.

The variables in common block **VISCPL** are:

`YD`	is the equivalent stress at first yield
`YD1`	is the equivalent yield stress including current workhardening and temperature effects
`YD2`	is the equivalent stress for **ORNL** tenth cycle yield
`YD21`	is the equivalent stress including current workhardening and temperature effects for **ORNL** tenth cycle yield
`YDZER`	is the equivalent yield stress including Mohr-Coulomb terms (defaults to `YD1`).

Example

The following is a simple viscoplastic strain rate law that depends on the differences between the current stress state and the static yield stress, raised to the nth power. (Note that `T(1)`, the current equivalent stress also includes Mohr-Coulomb terms when the option is flagged.)

$$\dot{\bar{\varepsilon}} = c\,(\bar{\sigma} - \bar{\sigma}_y)^n$$

Where:

$\bar{\sigma}$	is the current total equivalent stress
$\bar{\sigma}_y$	is the current equivalent yield stress including workhardening, temperature effects and Mohr-Coulomb terms
n	is the index of the power law
c	is the constant that depends on the index n. Here the strain rate equation is made dimensionless in stress by setting $c = 0.01.../\bar{\sigma}^n_{yo}$ where σ_{yo} is the equivalent stress at first yield.

This is programmed as follows; for n=2

```
 SUBROUTINE CRPLAW(EQCP,EQCPNC,STR,CRPE,T,DT,TIMINC,CPTIM,M,NN,KC,
+MAT,NDI,NSHEAR)
 IMPLICIT REAL *8 (A-H, O-Z)
 DIMENSION T(1),DT(1),STR(1),CRPE(1)
 C = 0.01/(YD*YD)
 S = T(1) - YDZER
 EQCPNC = 0.0
 IF(S.LT.0.0)RETURN
 S=S*S
 EQCPNC=C*S*TIMINC
 RETURN
 END
```

UCOORD

1.5.7 Relocate Nodes Created During Adaptive Meshing

Description

The user subroutine **UCOORD** may be used to define the location of a new node created due to adaptive meshing. The default if this routine is not used is to put the node geometrically half way between the old nodes. This routine will be called for each new node created.

Format

Subroutine **UCOORD** is written with the following header cards:

```
SUBROUTINE UCOORD(XORD, NCRD, INOD, LM, NNODE)
IMPLICIT REAL *8 (A-H, O-Z)
DIMENSION XORD(NCRD,1),LM(1)

       user coding

RETURN
END
```

where

xord(j,i)	current coordinate j of node i
ncrdmx	number of coordinates per node
inod	node number of new node
lm(i)	nodes on which inod is depending
nnode	number of nodes on which inod is depending
nnode=2	middle of edge between lm(1) and lm(2)
nnode=3	center of triangle lm(1), lm(2),lm(3)
nnodes=4	center of plane lm(1),lm(2),lm(3),lm(4)
nnodes=4	center of tet4 lm(1),lm(2),lm(3),lm(4)
nnodes=8	center of brick lm(1),lm(2),lm(3),lm(4),lm(5),lm(6),lm(7),lm(8)

Update xord(j,inod) if desired.

UADAP

1.5.8 User Defined Error Criteria

Description

The user subroutine **UADAP** may be used to define an error criteria for adaptive meshing. The value of user must be returned. It is a measure of the quality of this element. If the value of user is greater than f1 * user_max or greater than f2, the element will be refined.

Note that the f1 and f2 must be specified on the ADAPTIVE model definition option. User_max is the largest value of user over all of the elements.

Format

Subroutine **UADAP** is written with the following header cards:

```
SUBROUTINE UADAP(MM,XORD,DSXT,NCRDMX,NDEGMX,LM,NNODE,USER)
IMPLICIT REAL *8 (A-H, O-Z)
DIMENSION XORD(NCRDMX,*),DSXT(NDEGMX,*),LM(*)

       user coding

RETURN
END
```

where

MM	user element number.
XORD	original coordinates.
DSXT	total displacements.
NCRDMX	maximum number of coordinates per node.
NDEGMX	maximum number of degrees of freedom per node.
LM	node numbers of this element.
NNODE	number of nodes per element.
USER	user error criteria.

UACTUAT

1.5.13 User Subroutine to Prescribe the Length of an ACTUATOR

Description

Subroutine **UACTUAT** allows the user to control the length of an actuator in an incremental analysis. This is often useful in mechanism analyses, where the kinematics are prescribed. This is used with the truss element, type 9, when an initial length is given in the fourth field of the geometry option.

Format

Subroutine **UACTUAT** is called with the following header cards:

```
SUBROUTINE UATUAT (M,INC,CPTIM,TIMINC,XLNGTH,OLNGTH)
IMPLICIT REAL *8 (A-H, O-Z)
DIMENSION m(2)

     user coding

RETURN
END
```

where:

M(1)	user element number.
M(2)	internal element number.
INC	increment number.
CPTIM	time.
TIMINC	time increment.
XLNGTH	length of actuator to be set by user.
OLNGTH	current length of actuator.

Block 12 - Integer Increment verification data
Block 11 is REQUIRED.
Number of records for binary : 1 Words/record: 6
Number of lines for formatted: 1 6 I13 integers

Record	Line	Format	Variable	Description
1	1	6I13	NEWCC	Switch to flag the existance of data for blocks 2 to 10 following block 11 on the incremental posttape. NEWCC = 0 info not given NEWCC = 1 rezone NEWCC = 2 substructure
			INC	Increment number
			SUBINC	Subincrement number
			JANTYP	= IANTYP + 100 if element quantities appear in this increment. IANTYP = See Table 1, Block 2. Note: This IANTYP is not necessarily equal to IANTYP in Block 2!
			KNOD	Number of nodal postvalues in this incr. = JNODE*NDEG (See Table 1, Block 1)
			IDMY1	reserved for future expansion

Block 13 - Real increment verification data
Block 12 is REQUIRED.
Number of records for binary : 1 Words/record: 6
Number of lines for formatted: 1 6 E13.6 reals

Record	Line	Format	Variable	Description
1	1	6E13.6	TIME	Transient time
			FREQ	Frequency
			GMAS	Generalized mass
			DMY2	reserved for future expansion
			DMY3	reserved for future expansion
			DMY4	reserved for future expansion

Block 14 - New non-incremental data
Block 13 is OPTIONAL. Omitted if NEWCC = 0 (Block 11)

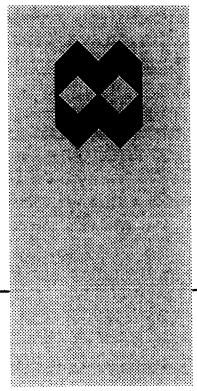
If NEWCC is non-zero, repeat Blocks 2-10 here.
(Read in new-non-incremental data.)

Block 15 - Magnitude of distributed loads
Block 14 is OPTIONAL. Omitted if NDISTL = 0 (Block 2 or 13)
Number of records for binary : 1 Words/record: NDISTL

```
            Number of lines for formatted: (NDISTL-1)/6+1

 Record Line Format Variable  Description
    1       6E13.6  DLOAD(I)  I=1,NDISTL  Distributed load magnitude

-------------------------------------------------------------------------
 Block 16 - Magnitudes of spring forces
           Block 15 is OPTIONAL.  Omitted if NSPRNG = 0 (Block 2 or 13)
            Number of records for binary : NSPRNG Words/record: 2
            Number of lines for formatted: NSPRNG

 Record Line Format Variable  Description
    J     J  2E13.6 REALF     Real spring force
                    CMPLXF    Complex spring force
   J=1,NSPRNG                 = 0.0 if ICOMPL = 0 (Block 2 or 13)

-------------------------------------------------------------------------
 Block 17 - Magnitudes of die forces
            Block 16 is OPTIONAL.  Omitted if NDIE = 0 (Block 2 or 13)
            Number of records for binary : NDIE  Words/record: 18
            Number of lines for formatted: 3*NDIE
            Items pre line for formatted : 3 lines per die,
                                           6 E13.6 reals per line
            Number of lines per die      : 3

 Record Line Format Variable  Description
    J   6E13.6,6E13.6,6E13.6
                    XDIE(I)   I=1,18  Die variables for die J

                              I =  1  X position
                                =  2  Y position
                                =  3  Z position
                                =  4  ----------------------
                                =  5  ----------------------
                               =  6  Angular position about axis of rotation
                                =  7  X velocity
                                =  8  Y velocity
                                =  9  Z velocity
                                = 10  ----------------------
                                = 11  ----------------------
                               = 12  Angular velocity about axis of rotation
                                = 13  X force
                                = 14  Y force
                                = 15  Z force
                                = 16  Moment about x
                                = 17  Moment about y
                                = 18  Moment about z

  J=1,NDIE
```

Material Curve Program

K6.2 Updates



Instruction for Use of Curve Program

1. Perform Laboratory Experiment on Specimen.
2. Collect Data.
3. Begin the Curve Program.

   Type `curve`

   The program will type:

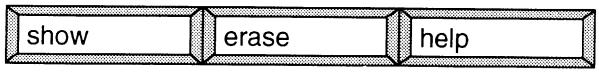
   ```
   Enter display name or <cr> for default:
   ```

   This is to define your x-terminal device,

   `enter a return.`

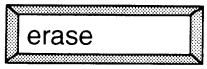
Main Menu

The main menu will come up first. On the left lower side, you will see three buttons.



These buttons will always be available.

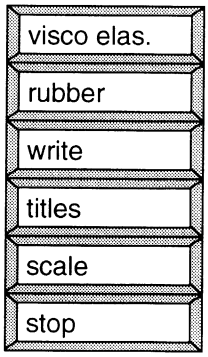
Displays the latest information.



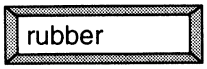
Clears the screen.

Provides on-line help.

On the right hand side are the menus that control the input of experimental data and the creation of material coefficients.



Controls the generation of a Prony series for the rate dependent viscoelastic behavior.



Controls the generation of rate independent rubber materials. This includes the following models:

Mooney-Rivlin
Signorini
3rd Order Deformation
4th Order Deformation
Yeoh Model
Ogden Model
Foam Model

Writes out the material parameters generated by the program in the format to be merged with MARC input.

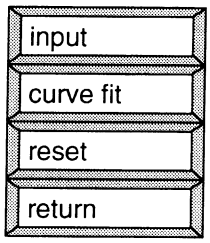
Controls the titles that will be displayed on the screen

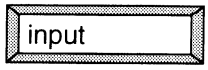
Controls the axis of the display of the curves.

Terminates the program.

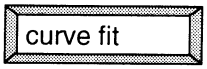
Viscoelastic Data

4. To generate viscoelastic data, pick on visco elas. The following menu appears.



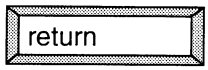


Controls the input of the experimental data as a function of time.



Selects what type of data is to be curve fit.

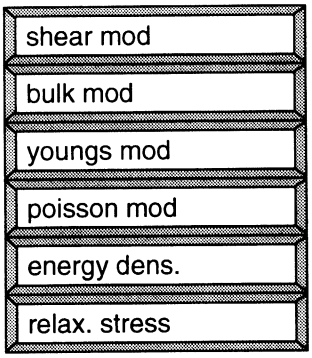
Clears out the data entered and the material coefficients calculated.



Returns to previous menu.

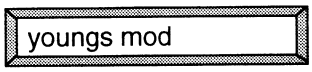
Input

Upon selecting input, one is given a choice as to what data is to be input.

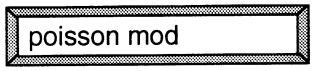


Enter the shear modulus versus time.

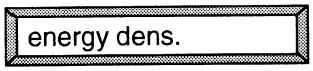
Enter the bulk modulus versus time.



Enter the Young's modulus versus time.



Enter the Poisson's ratio versus time.

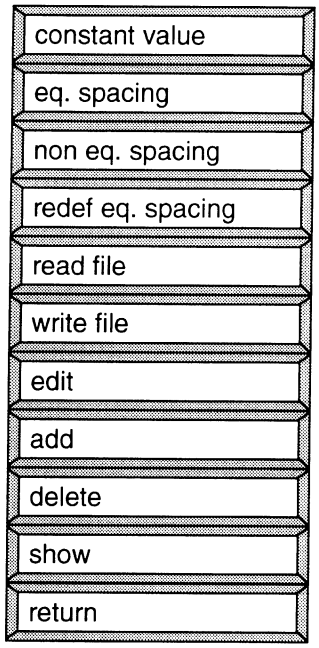


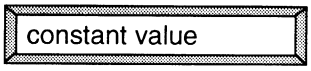
Enter the strain energy density versus time.

Enter the relaxation stress versus time.

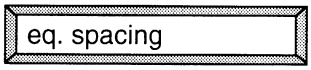
For viscoelastic curve fitting, one needs to have experimental data of either modeli, strain energy, or stress as a function of time. There is a limit of 200 data points. Additionally, one needs to define the long term value.

Upon choosing any type of material parameter, the following menu will appear.

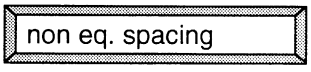




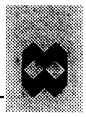
Enter a constant Shear modulus, Bulk modulus, Young's modulus, or Poisson's ratio, depending upon the previous selection.



Enter the number of points, the initial time, and the increment in time. Then for each point in time, enter the experimental value. Finally, enter the long term value.



Enter the number of points, and for each point enter the pair time and experimental value. Finally, enter the long term value.



redef eq. spacing

This interpolates between the defined experimental points, such that there are equal intervals in time. This is required.

read file

Read a file of experimental data.

After reading data, if points are not at equal intervals of time, one must redefine equal spacing.

For viscoelastic curve fitting, the format of the file should be

$time^1$, $value^1$
$time^2$, $value^2$
⋮ ⋮
$time^n$, $value^n$
0., long term value.

write file

Writes out to an file, the experimental data.

edit

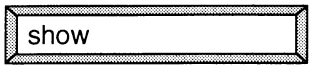
Modifies a data point. Enter the data point number to be modified, followed by the time and the new value.

add

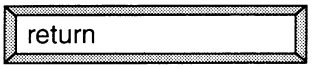
Adds a new data point, enter the time and the experimental value.

delete

Deletes an existing data point, select which point.



Redisplays Curve.

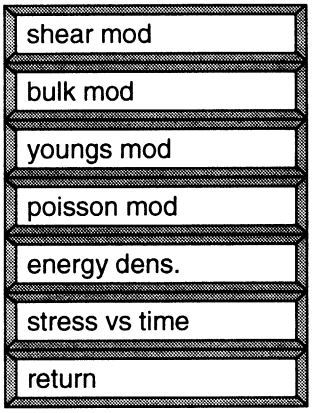


Returns to previous menu.

Curve Fit

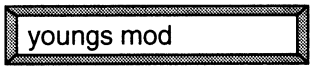
Upon selecting curve fit, the following menu appears. This controls curve fitting, the experimental data with a Prony series. You will be asked for the number of terms.

The number of terms cannot be greater than the number of data points minus two. Also, the number of terms cannot be greater than 10.



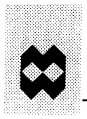
Curve fits experimental shear modulus data.

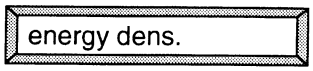
Curve fits experimental bulk modulus data.



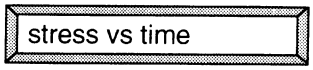
Curve fits experimental Young's modulus data.

Curve fits experimental Poisson's ratio data.

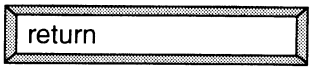




Curve fits experimental Strain Energy density data.



Curve fits experimental stress relaxation data.

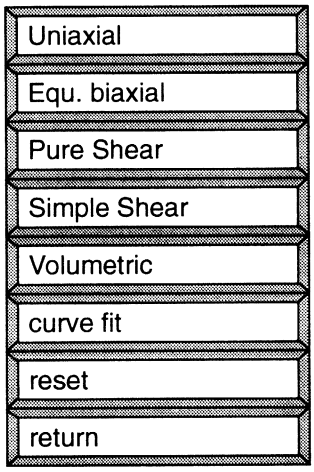


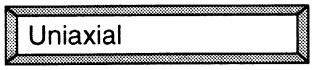
Return to previous menu.

Rubber Data

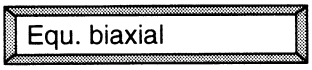
5. To generate the material coefficients for the rubber strain energy function, pick rubber.

Upon selecting the **rubber** button, the following will appear that is used to control the type of experimental data that is to be read in and the calculation of the material constants. Note that when entering material data, one needs to enter the engineering strain, the area ratio, and the engineering stress. If the area ratio (current area/initial area) is zero, it is assumed that the material is incompressible. It is possible to input the data from multiple experiments. The material constants generated will be calculate to best obtainable to fit all of the experimental data.

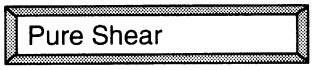




Input Uniaxial data.

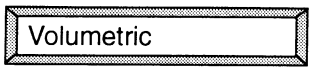


Input Equal Biaxial data.



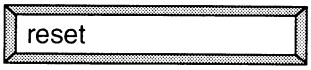
Input Pure Shear data.

Input Simple Shear data.



Input Volumetric data. The volumetric data is only used for the Ogden or Foam models.

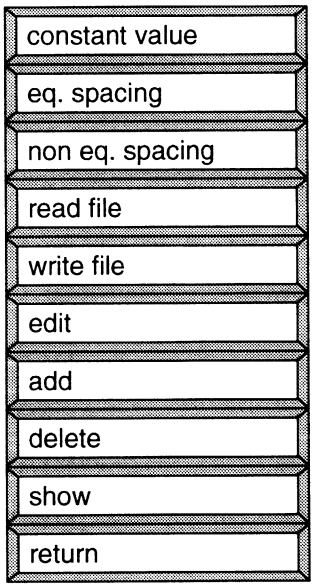
Perform curve fit to calculate constants.

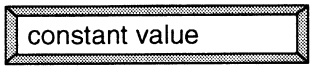


Reset.

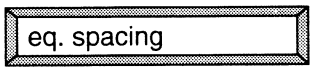
Return to previous menu.

Upon selecting Uniaxial, Equ. biaxial, Pure Shear, Simple Shear, or Volumetric experimental data to be entered, the following menu appears. For rubber curve fitting, one needs to have experimental data of strain, stress, and the area ratios. If the area ratios are not entered, the material is assumed to be incompressible. There is a limit of 200 data points.

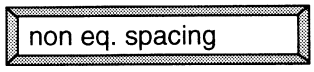




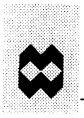
Enter either the Young's modulus *(if uniaxial data)* and a stress, or the Shear modulus and a stress *(if shear data)*, or the Bulk modulus and a stress if volumetric data.



Enter the number of points; the first strain, the increment of strain, and then for each data point, the stress and the area ratio.



Enter the number of points; the first strain, the increment of strain, and then for each data point, the stress and the area ratio.



read file

Read a file containing the experimental data. For rubber curve fitting, the format of the file should be

$$\begin{matrix} \varepsilon^1 & A_r^1 & \sigma^1 \\ \vdots & \vdots & \vdots \\ \varepsilon^n & A_r^n & \sigma^n \end{matrix}$$

where ε^i is the engineering strain

σ^i is the engineering stress

A_r^i is the relative area = A/A_0

For incompressible materials set $A_r^i = 0.0$.

write file

Write a file containing the experimental data.

edit

Enter the data point number to be modified followed by the strain, stress, and the area ratio.

add

Adds a new data point; enter the strain, stress, and the area ratio.

delete

Deletes an existing data point, select which point.

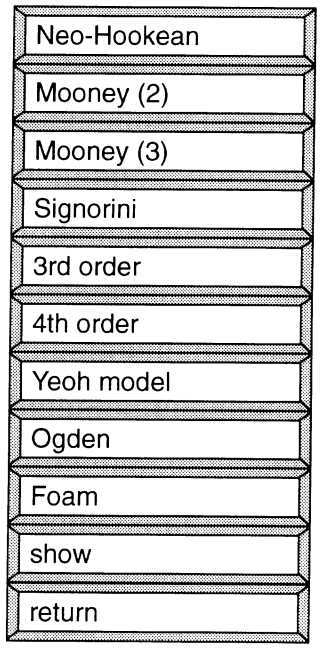
show

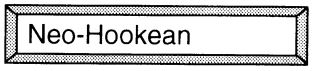
Redisplays Curve.

Returns to previous menu.

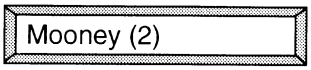
Curve Fit under Rubber Menu

Upon choosing curve fit under the rubber menu, the following menu appear. One should select a particular model. One can iterate between the different models until a satisfactory fit is obtained. The original experimental data and the curve fit will be displayed.

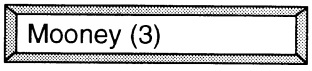




Fits data with Neo-Hookean, one constant (C10) model.

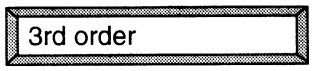


Fits data with Mooney-Rivlin, two constant (C10 and C01) model.



Fits data with Mooney-Rivlin, three constant (C10, C01, C11) model.

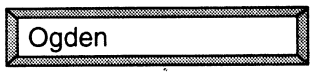
Fits data with Signorini, three constant (C10, C01, C20) model.



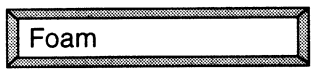
Fits data with 3rd order deformation (C10, C01, C11, C20) model.

Fits data with 4th order deformation (C10, C01, C11, C20, C30) model.

Fits data with Yeoh (C10, C20, C30) model.

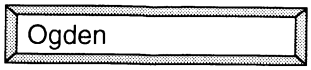


Fits data with the Ogden series model.

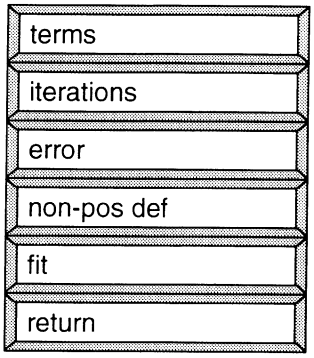


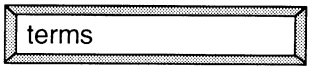
Fits data with the Foam series model.

Ogden



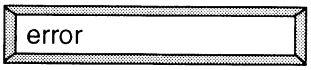
Upon selecting **Ogden** curve fit, the following menu appears.





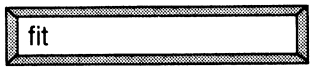
Enter the number of terms in the series. Note that the more terms requested, the more accurate the fit, but the longer the time required to fit the data. It is suggested that the number of terms should never exceed 3. The default number of terms is 1. The maximum number of terms is 5.

Enter the number of iterations used to perform the curve fitting. The larger the number of iterations, the less numerical error in the iterative process. The default is 7, the maximum is 50.

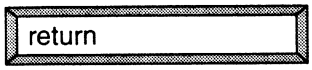


Controls the error in the iterative process, the default is 0.001.

Turns on the nonpositive definite flag. The curve fitting program uses the Hill strain energy criteria $\Delta\varepsilon \cdot \Delta\sigma > 0$ to determine stability. Turning on the nonpositive definite flag suppresses this test.

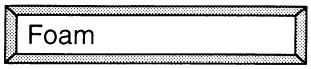


Fits the data, program will calculate μ, α for each term in the series along with the bulk modulus.

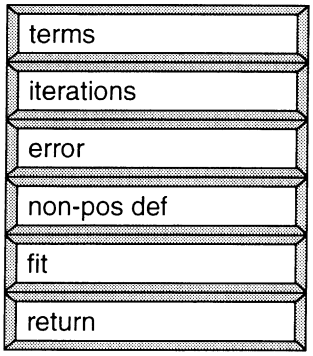


Returns to previous menu.

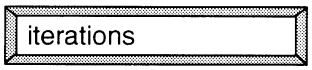
Foam



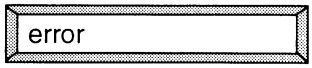
Upon selecting **Foam** curve fit, the following menu appears.



Enter the number of terms in the series. Note that the more terms requested, the more accurate the fit, but the longer the time required to fit the data. It is suggested that the number of terms should never exceed 3. The default number of terms is 1. The maximum number of terms allowed is 5.



Enter the number of iterations used to perform the curve fitting. The larger the number of iterations, the less numerical error in the iterative process. The default is 7, the maximum is 50.



Controls the error in the iterative process, the default is 0.01.

non-pos def

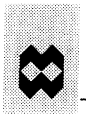
Turns on the nonpositive definite flag. The curve fitting program uses the Hill strain energy criteria $\Delta\varepsilon \cdot \Delta\sigma > 0$ to determine stability. Turning on the nonpositive definite flag suppresses this test.

fit

Fits the data. You will need to enter the effect Poisson's ratio. The curve fitting program will calculate μ, α, β for each term in the series.

return

Returns to previous menu.



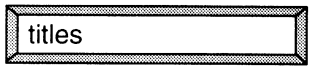
Notes

After the data is fitted, the curve program uses the material constants to verify the correlation with the experimental data. The strain data point, experimental stress, predicted stress based upon the evaluated strain energy function, and the ratio is provided in the window area.

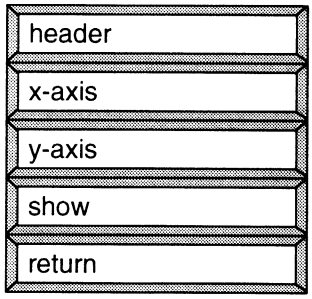
For the invariant based models *(i.e. not Ogden or Foam)* the user may also modify the material coefficients calculated by the program. You will be queried as to whether you want to do any manual changes. Simply type `yes` or `no`. If you enter new values, the new strain energy function will be used to define the stress-strain behavior.

Occasionally, when using the invariant based models, the curve fitting procedure will return negative material coefficients. While, the experimental data has been fit with the best least squares approximation, these coefficients are not physically meaningful. One can request that an alternate curve fit be performed, such that positive coefficients will be obtained. This will not produce as good a fit in the least squared sense.

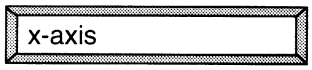
Titles



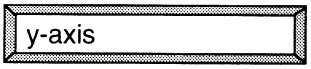
This controls the labels that appear on the graphics image. Upon selecting **titles**, the following menu appears. The graphic image is updated when the return is selected.



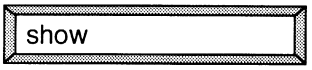
Modify main title of plot.



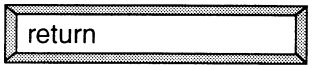
Enter the title along the x-axis, default is either "Engineering strain" or "time".



Enter the title along the y-axis, default is either "Engineering stress" or "modulus".

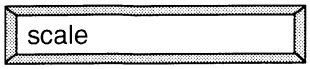


Shows the current values of the titles.

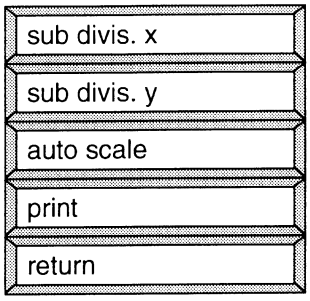


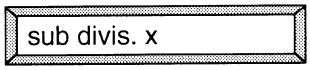
Returns to previous menu and updates graphical image.

Scale

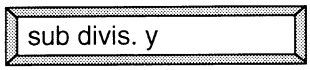


The scale menu controls the graphic image. Upon selecting **scale**, the following menu appears.

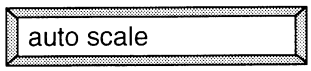




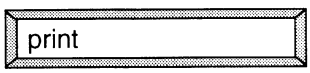
Selects the number of hash marks along the x-axis *(abscissa)*.



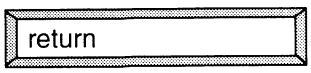
Selects the number of hash marks along the y-axis *(ordinate)*.



The graphics has two modes; auto scale on or off. In auto scale on, the experimental data and curve fit results will all fit in the viewing area. In auto scale off, the data within the user controlled limits will be displayed. Enter xmin, xmax, ymin, ymax to define these limits.



Displays the current settings of the scale option.



Returns to previous menu and updates graphical image.

MARC Corporate Headquarters
260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306, USA
Tel: (415) 329-6800
Fax: (415) 323-5892
Email: support@marc.com

MARC Analysis Research Corporation
Bredewater 26
2715 CA Zoetermeer
The Netherlands
Tel: 31-(0)79-3510-411
Fax: 31-(0)79-3517-560
Email: support@marc.nl

Nippon MARC Co.,Ltd.
P.O.Box 5056
Shinjuku Daiichi Seimei Bldg.
2-7-1 Nishi-Shinjuku
Shinjuku-ku, Tokyo 163, Japan
Tel: 81-(0)3-3345-0181
Fax: 81-(0)3-3345-1529
Email: system@marc.co.jp

MARC Software Deutschland GmbH
Ismaningerstrasse 9
85609 Aschheim
Germany
Tel: 49-(0)89-9045033
Fax: 49-(0)89-9030676
Email: support@marc.de

Nippon MARC Co.,Ltd.
Dai 2 Kimi Bldg.
2-11 Toyotsu-cho
Suita-city,Osaka 564, Japan
Tel: 81-(0)6-385-1101
Fax: 81-(0)6-385-4343

MARC Software Deutschland GmbH
Alte Dohrener Str. 66
D-30173 Hannover, Germany
Tel: 49-(0)511-800211
Fax: 49-(0)511-801042

MARC China
1704, Yanshan Hotel
138A Haidian Road
Beijing 100-086
PRC
Tel: 86-10-256-4375
Fax: 86-10-256-4860

MARC UK Ltd.
35, Shenley Pavilions, Chalkdell Drive
Shenley Wood
Milton Keynes, MK5 6LB, UK
Tel: 44-1908-506505
Fax: 44-1908-506522

Espri-MARC srl
Viale Brigata Bisagno 2/10
I-16129 Genova
Italy
Tel: 39-(0)10-585949
Fax: 39-(0)10-585949

MARC Overseas, Inc.
Podolska 50
147 00 Praha 4
Czech Republic
Tel: 42-2-6121-4123
42-2-6121-4111 x252
Fax: 42-2-6121-4123

Document Title: **MARC K6.2 Documentation Updates**
Part Number: UP-3500-01
Revision Date: September, 1995