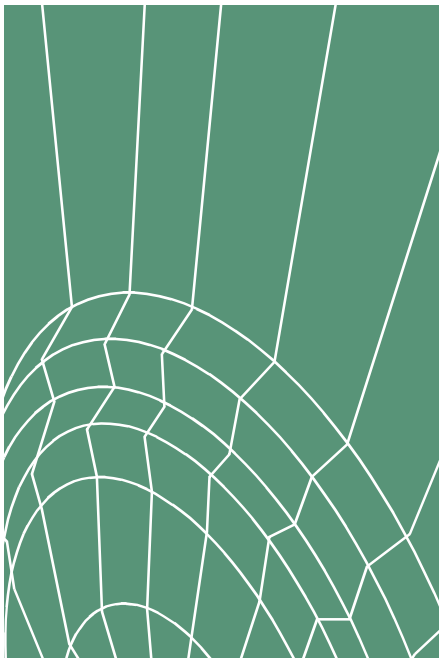
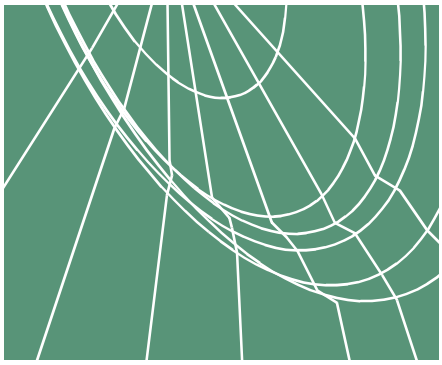
MSC SOFTWARE
SIMULATING REALITY



MSC.Marc
Volumes A - E



Additions and Corrections

Version 2001

Corporate
MSC.Software Corporation
815 Colorado Boulevard
Los Angeles, CA 90041-1777
Telephone: (323) 258-9111 or (800) 336-4858
FAX: (323) 259-3638

Europe
MSC.Software Corporation
Innsbrucker Ring 15
Postfach 80 12 40
81612 München, GERMANY
Telephone: (49) (89) 431 9870
Fax: (49) (89) 436 1716

Asia Pacific
MSC.Software Corporation
Entsuji-Gadelius Building
2-39, Akasaka 5-chome
Minato-ku, Tokyo 107, JAPAN
Telephone: (81) (03) 3505-0266
Fax: (81) (03) 3505-0914

Worldwide Web
www.mscsoftware.com

Document Title: **MSC.Marc Volumes A - D: Additions and Corrections**

Part Number:

Revision Date: February, 2001

Proprietary Notice

Trademarks

C O N T E N T S

MSC.Marc Volumes A - E: Additions and Corrections

Volume A: Theory and User Information

Volume B: Element Library

Chapter 2
MSC.Marc Element Classifications

Chapter 3
Element Library

Volume C: Program Input

Chapter 2
Parameters

Chapter 3
Model Definition Options

Volume E: Demonstration Problems, Part II

Chapter 3
Plasticity and Creep

Volume E: Demonstration Problems, Part III

Chapter 5
Heat Transfer

Chapter 6
Dynamics

Volume E: Demonstration Problems, Part IV

Chapter 7
Contact

Chapter 8
Advanced Topics

Volume A: Theory and User Information

Chapter 4 Introduction to Mesh Definition

Local Adaptivity

The adaptive mesh generation capability increases the number of elements and nodes to improve the accuracy of the solution. The capability is applicable for both linear elastic analysis and for nonlinear analysis. The capability can be used for lower-order elements, 3-node triangles, 4-node quadrilaterals, 4-node tetrahedrals, and 8-node hexahedral elements.

When used in conjunction with the ELASTIC parameter for linear analysis, the mesh is adapted and the analysis repeated until the adaptive criteria is satisfied. When used in a nonlinear analysis, an increment is performed. If necessary, this increment is followed by a mesh adjustment which is followed by the analysis of the next increment in time. While this can result in some error, as long as the mesh is not overly coarse, it should be adequate.

Number of Elements Created

The adaptive meshing procedure works by dividing an element and internally tying nodes to insure compatibility. Figure 4-20 shows the process for a single quadrilateral element.

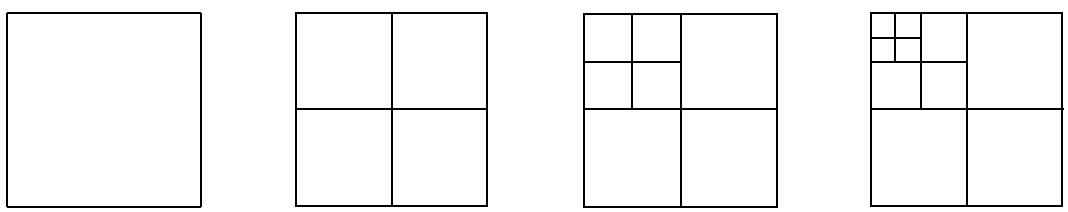


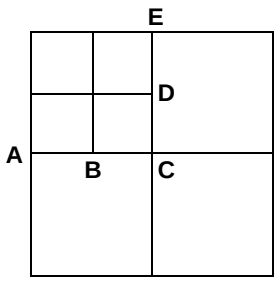
Figure 4-20 Single Quadrilateral Element Process

A similar process occurs for the triangles, tetrahedrons, and hexahedrons elements. You can observe that for quadrilaterals the number of elements expands by four with each subdivision; similarly, the number of elements increases by eight for hexahedrals. If full refinement occurs, you observe that the number of elements is $2^{(\text{level}x2)}$ for quadrilaterals and $2^{(\text{level}x3)}$ for hexahedrons elements. The number of levels also limits the amount of subdivisions that may occur.

Level	Number of Elements	
	Quadrilaterals	Hexahedrals
0	1	1
1	4	8
2	16	64
3	64	512
4	256	4096

For this reason, it is felt that the number of levels should, in general, be limited to three.

When adaptive meshing occurs, you can observe that discontinuities are created in the mesh as shown below:



To ensure compatibility node B is effectively tied to nodes A and C, and node D is effectively tied to nodes C and E. All of this occurs internally and does not conflict with other user-defined ties or contact.

Boundary Conditions

When mesh refinement occurs, boundary conditions are automatically adjusted to reflect the change in mesh. The rules listed below are followed:

1. Fixed Displacement

   For both 2-D and 3-D, if both corner nodes on an edge have identical boundary conditions, the new node created on that edge has the same boundary conditions. For 3-D, if all four nodes on a face have identical boundary conditions, the new node created in the center of the face has the same boundary conditions. Note that identical here means the same in the first degree of freedom, second degree of freedom, etc. independently of one another.

2. Point Loads

   The point loads remain unchanged on the original node number.

3. Distributed Loads

   Distributed loads are automatically placed on the new elements. Caution should be used when using user subroutine FORCEM as the element numbers can be changed due to the new mesh process.

4. Contact

   The new nodes generated on the exterior of a body are automatically treated as potential contact nodes. The elements in a deformable body are expanded to include the new elements created. After the new mesh is created, the new nodes are checked to determine if they are in contact.

CAUTION: None of the nodes of an element being subdivided should have a local coordinate system defined through the TRANSFORMATION option.

Location of New Nodes

When an element is refined, the default is that the new node on an edge is midside to the two corner nodes. As an alternative, the SURFACE and ATTACH NODE options can be used or user subroutine UCOORD can be used. The SURFACE option can be used to describe the mathematical form of the surface. If the corner nodes of an edge are attached to the surface, the new node is placed upon the actual surface.

This is illustrated in Figure 4-21 and Figure 4-22, where initially a single element is used to represent a circle. The circle is defined with the SURFACE option and the original four nodes are placed on it using the ATTACH NODE option. Notice that the new nodes are placed on the circle.

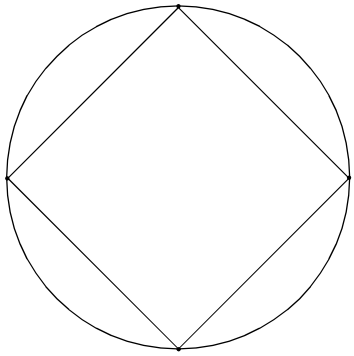
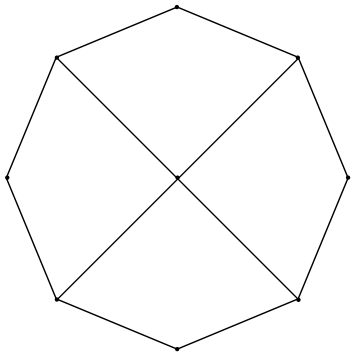
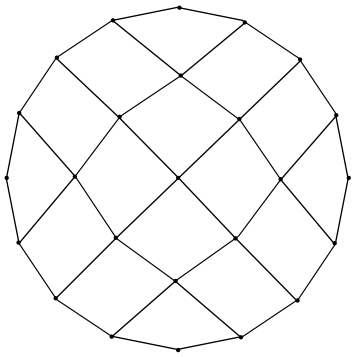


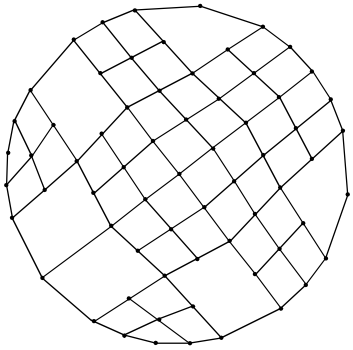
Figure 4-21 Original Mesh and Surface



Level 1 Refinement



Level 2 Refinement



Level 2 and 3 Refinement

Figure 4-22 Levels of Refinement

Adaptive Criteria

The adaptive meshing subdivision occurs when a particular adaptive criteria is satisfied. Multiple adaptive criteria can be selected using the ADAPTIVE model definition option. These include:

Mean Strain Energy Criterion

The element is refined if the strain energy of the element is greater than the average strain energy in an element times a given factor, f_1.

$$\text{element strain energy} > \frac{\text{total strain energy}}{\text{number of elements}} * f_1 \tag{4-31}$$

Zienkiewicz-Zhu Criterion

The error norm is defined as either

$$\pi^2 = \frac{\int(\sigma^* - \sigma)^2 dV}{\int\sigma^2 dV + \int(\sigma^* - \sigma)^2 dV} \quad \gamma^2 = \frac{\int(E^* - E)^2 dV}{\int E^2 dV + \int(E^* - E)^2 dV} \tag{4-32}$$

The stress error and strain energy errors are

$$X = \int(\sigma^* - \sigma)^2 dV \text{ and } Y = \int(E^* - E)^2 dV \tag{4-33}$$

where σ^* is the smoothed stress and σ is the calculated stress. Similarly, E is for energy.

An element is subdivided if

$\pi > f_1$ and

$X_{el} > f_2 * \text{X/NUMEL} + f_3 * X * f_1 / \pi / \text{NUMEL}$

or

$\gamma > f_1$ and

$Y_{el} > f_4 * \text{Y/NUMEL} + f_5 * Y * f_1 / \gamma / \text{NUMEL}$

where NUMEL is the number of elements in the mesh. If f_2, f_3, f_4, and f_5 are input as zero, then $f_2 = 1.0$.

Zienkiewicz – Zhu Plastic Strain Criterion

The plastic strain error norm is defined as $\alpha^2 = \frac{\int(\varepsilon^{p*} - \varepsilon^p)^2 dV}{\int \varepsilon^{p2} dV + \int(\varepsilon^{p*} - \varepsilon^p)^2 dV}$.

The plastic strain error is $A = \int(\varepsilon^{p*} - \varepsilon^p)^2 dV$. The allowable element plastic strain error is $AEPS = f_2 * \mathrm{A}/\mathrm{NUMEL} + f_3 * A * f_1/\alpha/\mathrm{NUMEL}$. The element will be subdivided when $\alpha > f_1$ and $A_{el} > AEPS$. NUMEL is the number of elements in the mesh.

Zienkiewicz-Zhu Creep Strain Criterion

Zienkiewicz-Zhu creep strain error norm is defined as

$\beta^2 = \frac{\int(\varepsilon^{c*} - \varepsilon^c)^2 dV}{\int \varepsilon^{c2} dV + \int(\varepsilon^{c*} - \varepsilon^c)^2 dV}$. The creep strain error is $B = \int(\varepsilon^{c*} - \varepsilon^c)^2 dV$. The allowable element creep strain error is

$AECS = f_{2*} B/\mathrm{NUMEL} + f_3 * B * f_1/\beta/\mathrm{NUMEL}$.

The element will be subdivided when $\beta > f_1$ and $B_{el} > AECS$. NUMEL is the number of elements in the mesh.

Equivalent Values Criterion

This method is based upon either relative or absolute testing using either the equivalent von Mises stress, the equivalent strain, equivalent plastic strain or equivalent creep strain. An element is subdivided if the current element value is a given fraction of the maximum (relative) or above a given absolute value.

$$\sigma_{vm} > f_1 \sigma_{vm}^{max} \text{ or } \sigma_{vm} > f_2$$

$$\varepsilon_{vm} > f_3 \sigma_{vm}^{max} \text{ or } \varepsilon_{vm} > f_4$$

Node Within A Box Criterion

An element is subdivided if it falls within the specified box. If all of the nodes of the subdivided elements move outside the box, the elements are merged back together.

Nodes In Contact Criterion

An element is subdivided if one of its nodes is associated with a new contact condition. In the case of a deformable-to-rigid contact, this implies that the node has touched a rigid surface. For deformable-to-deformable contact, the node can be either a tied or retained node. Note that if chattering occurs, there can be an excessive number of elements generated. Use the level option to reduce this problem.

Temperature Gradient Criterion

An element is subdivided if the gradient in the element is greater than a given fraction of the maximum gradient in the solution. This is the recommended method for heat transfer.

User-defined Criterion

User subroutine UADAP can be used to prescribe a user-defined adaptive criteria.

Previously Refined Mesh Criterion

Use the refined mesh from a previous analysis as the starting point to this analysis. The information from the previous adapted analysis is read in.

Global Remeshing

In the analysis of metal or rubber, the materials may be deformed from some initial (maybe simple) shape to a final, very often, complex shape. During the process, the deformation can be so large that the mesh used to model the materials may become highly distorted, and the analysis cannot go any further without using some special techniques. Remeshing/rezoning in Marc is a useful feature to overcome the difficulties.

In the release before MSC.Marc 2000, the global remeshing/rezoning is done manually. When the mesh becomes too distorted because of the large deformation to continue the analysis, the analysis is stopped. A new mesh is created based on the deformed shape of the contact body to be rezoned. A data mapping is performed to transfer necessary data from the old, deformed mesh to the new mesh. The contact tolerance is recalculated (if not specified by you) and the contact conditions are redefined, and the analysis continues.

After the release of MSC.Marc 2000, the above steps are done automatically (see Figure 4-23). Based on the different remeshing criteria you specified, the program determines when the remeshing/rezoning is required. The automatic remeshing control can be instructed through the ADAPT GLOBAL option or through automatic time stepping control. With automatic time stepping control, remeshing (if allowed) is forced when the mesh of the body is distorted during the analysis. At the point of remeshing/rezoning on a 2-D application, the program finds the outline of the body to be rezoned and repairs the outline to remove possible penetrations. Then, the program calls the mesher to create a new mesh based on the clean outline. Furthermore, the program performs data transfer from the old mesh to the new mesh, redetermines the contact conditions, and continues the analysis.

The automatic remeshing/rezoning feature can be activated using REZONING,1 parameter. Remeshing/rezoning can be carried out for one or more contact bodies at one increment. Different bodies can use different remeshing/rezoning criteria. The remeshing/rezoning criteria, the bodies to be rezoned, the element target length and other remeshing control parameters for the new meshes are specified by you or by the program via the ADAPT GLOBAL model and history definition options.

Note: Automatic remeshing/rezoning feature only works for two-dimensional cases in the current release of Marc.

Only updated Lagrangian formulation can be used for the feature.

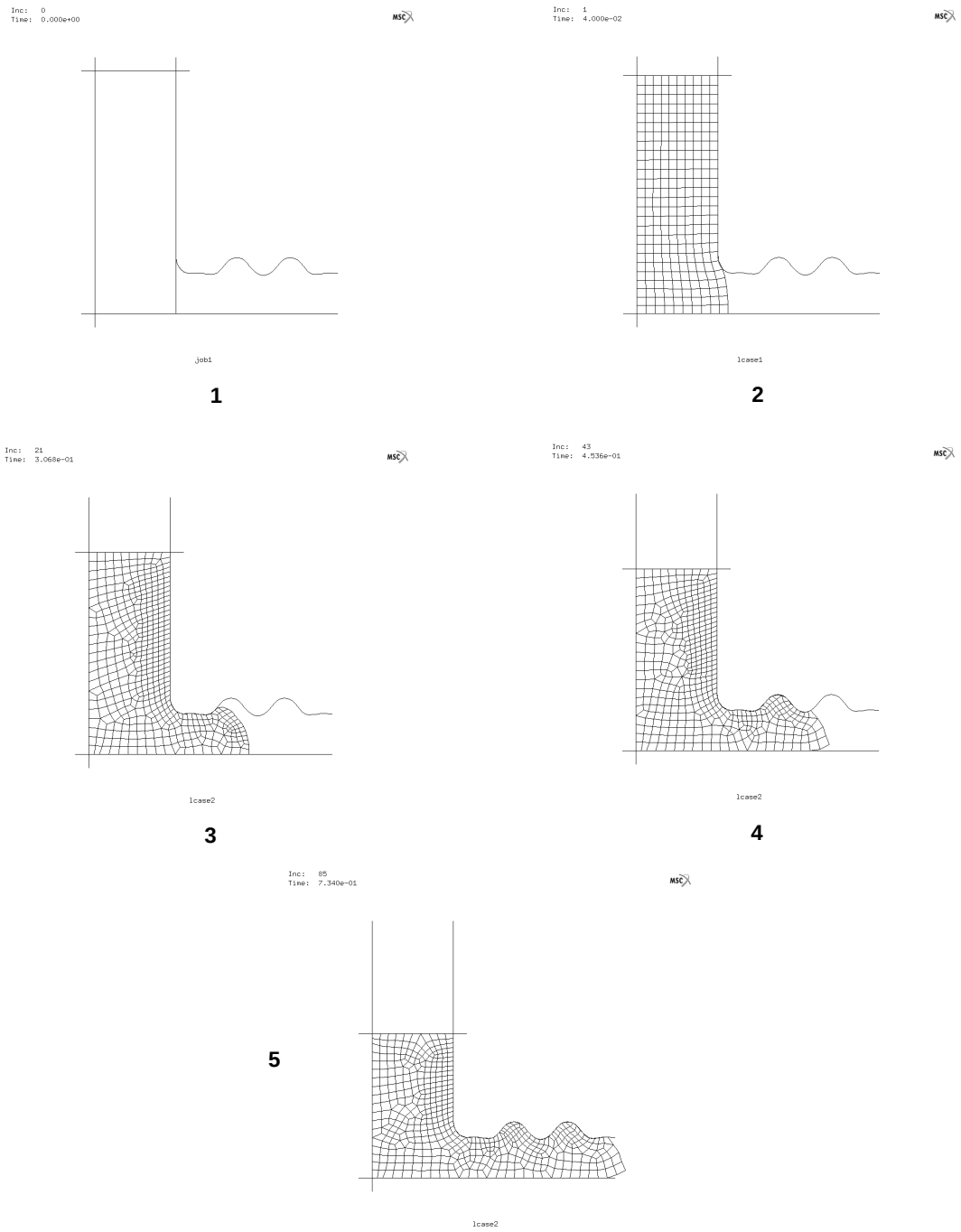


Figure 4-23 Automatic Remeshing and Rezoning of a Rubber Seal

Remeshing Criteria

It is possible to choose any combinations of the following remeshing criteria: Element Distortion, Contact Penetration, Increment, Angle Deviation or Immediate.

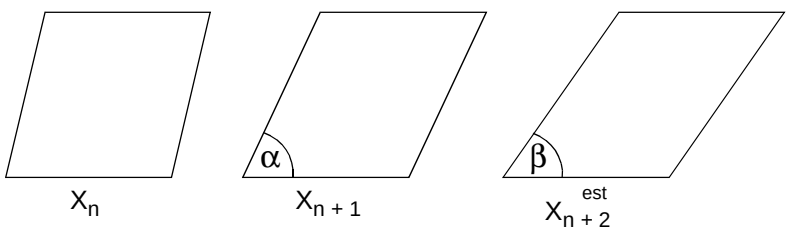
Element Distortion

The identified body is remeshed when the distortion in the elements becomes large, or it is anticipated that the distortion will become large.

The element distortion criteria is based upon examining the angles of the elements at the end of the increment and an estimate of the change in the angle in the next increment.

Given that X_n is the coordinates in the beginning of the step and that ΔU_n are the displacements in the increment, then:

$$X_{n+1} = X_n + \Delta U_n \text{ and } X_{n+2}^{est} = X_{n+1} + \Delta U_n$$



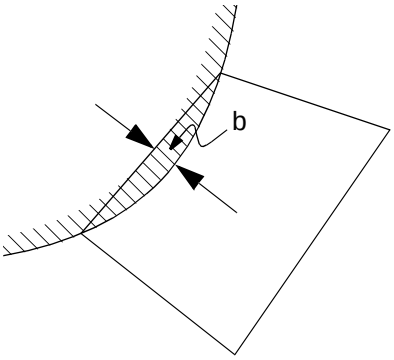
If $\cos\alpha > 0.8$ and $\cos\beta > 0.9$, remesh, or if $\cos\alpha > 0.9$ and $\cos\beta > \cos\alpha$, remesh which is equivalent to

$0 < \alpha < 36°$ and $0 < \beta < 25°$ remesh or
$144° < \alpha < 180°$ and $155° < \beta < 180°$ remesh or
$0 < \alpha < 25°$ and $\beta < \alpha$ or
$155° < \alpha < 180°$ and $\beta < \alpha$.

Contact Penetration

The identified body is remeshed when the curvature of the contact body is such that the current mesh cannot accurately detect penetration.

The penetration remeshing criteria is based upon examining the distance between the edge of the element and the representation of the other contact surface.



If $b >$ penetration limit, remeshing is required. The penetration limit can be specified by you. By default, the penetration limit is 2*(contact tolerance). The contact tolerance can be input by you or computed internally by the program.

Please note that this check does not apply to the self-contact situation.

Increment

Remeshing occurs at specified increment frequency.

Angle Deviation

The identified body remeshes when the angles in the element have a deviation from the ideal angle greater than a specified amount. The ideal angle for quadrilaterals or hexahedrals is 90°. The ideal angle for triangles and tetrahedrals is 60°. The default of 40° indicates that any angle in the range $50 \leq \alpha \leq 130$ is acceptable for quadrilaterals.

Immediate

The identified body is remeshed before performing any analysis.

Remeshing Techniques

The remeshing techniques include outline extraction and repair and the mesh generation. After the outline is extracted and repaired, the mesh generator is called to create a mesh. For 2-D remeshing, the new mesh is created either through the built-in mesh generator using the overlay method or through a standalone mesh generator. When a standalone mesh generator is called, the program, by default, pauses while waiting for the mesh generator to create the new mesh. This can be memory intensive as both program and mesher are using memory. However, the program can be stopped automatically while the mesh is being created, freeing the memory for the mesh generation with the program automatically resuming after the meshing is complete. In Marc, this is accomplished by using the AUTO RESTART option, `-autorst`, through the command line parameter. In Mentat, this is instructed through JOB->JOB PARAMETERS ->REMESHING CONTROL->STOP AND RESTART.

Mesh Generation

Overlay meshing: This is a quadrilateral mesh generator. The 2-D overlay mesher is included within Marc. It creates a quadrilateral mesh by forming a regular grid covering the center area of a body. A projection is then used to project all boundary nodes onto the real surface and form the outer layer elements. For the surface that is not in contact with other bodies, a cubic spline line is used to make the outline points smoother. This mesher also allows up to two level refinements on boundary where finer edges are needed to capture the geometry detail, and one level of coarsening in the interior where small elements are not necessary. This refinement and coarsening are performed using the tying constraints (see Figure 4-24).

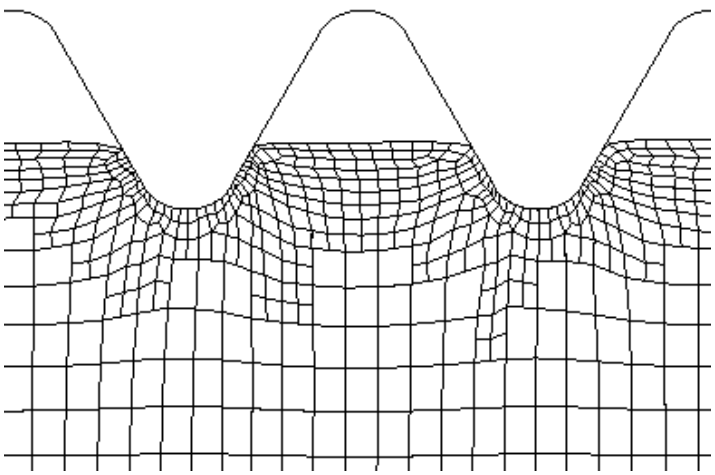


Figure 4-24 Local Refinement and Coarsening

In general, overlay meshing produces a better quality element. However, it does not take geometry with holes inside. It may not create a good mesh with geometry that has a very thin region or very irregular shape. Also, because the regular grid is created based on the global coordinate system, it may create a poorer mesh if the geometry is not aligned with the global coordinate system.

Advancing front meshing: This mesher creates either triangular, quadrilateral, or mixed triangular and quadrilateral mesh. For a given outline boundary, it starts by creating the elements along the boundary. The new boundary front is then formed when the layer of elements is created. This front advances inward until the complete region is meshed. Some smoothing technique is used to improve the quality of the elements. In general, this mesher works with any enclosed geometry and for geometry that has holes inside. The element size can be changed gradually from the boundary to the interior allowing smaller elements near the boundary with no tying constraints necessary (see Figure 4-25).

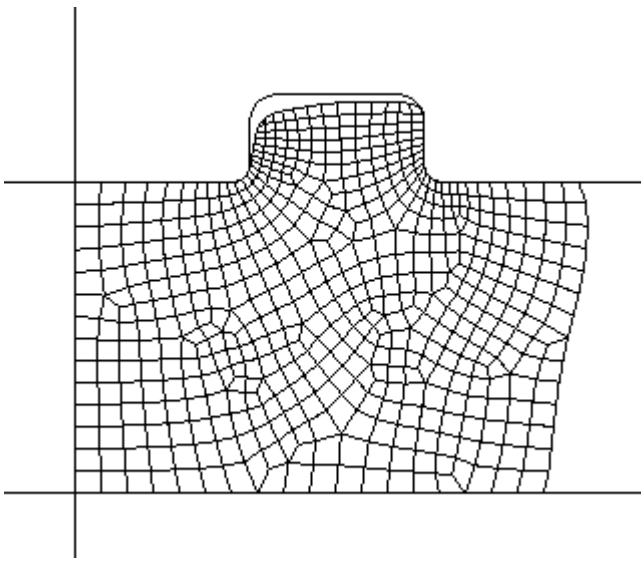


Figure 4-25 Advancing Front Meshing

Delaunay Triangulation: This mesher creates only the triangular mesh. All the triangles satisfy the Delaunay Triangulation property. It takes all the seed points on the improved outlines as initial triangulation points. The triangulation is implemented by sequential insertion of new points into the triangulation until all the triangles satisfy the local density and quality requirement. Delaunay Triangulation Algorithm assures the triangular mesh created has the best quality possible for the given set of points. The mesher also allows geometry to have holes inside the body and a variation of the elements with different sizes (Figure 4-26).

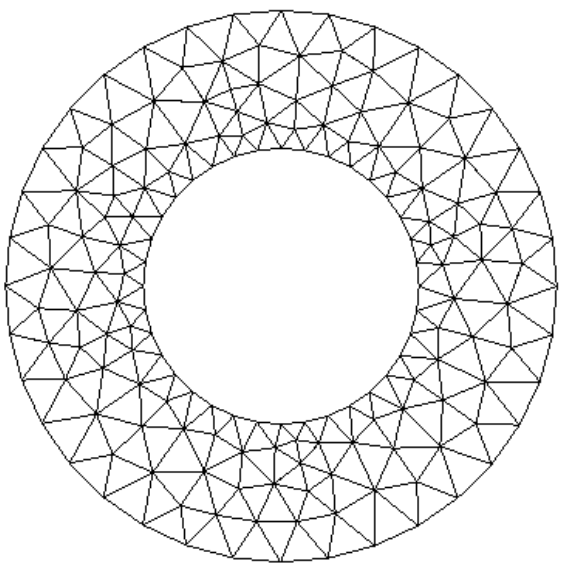


Figure 4-26 Meshing with Delaunay Triangulation

Outline Extraction and Repair

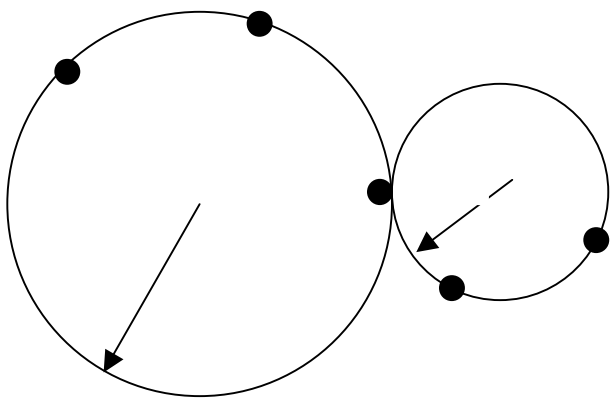
The outline consists of all the boundary edges of the contact body. Once the outline is extracted from the mesh, it is checked against other contacting bodies. The penetration from other contacting bodies is marked and the outline is corrected according to the penetration. If the outline is to be used for the advancing front or Delaunay mesher, some refinement and corrections are required to obtain better outline points. These

outline points become boundary nodes in the new mesh and cannot be altered during the meshing process. The following procedures are taken by the program to prepare the new outline for the remeshing:

Step 1: **Marking the hard points:** The hard points are those points that represent important features of the original outline. Hard points are points that mark the beginning of a contact or the last of the contact, and the points that represent a sharp corner (such as 90° angle).

Step 2: **Marking the points with target element size and minimum element size:** The outline points are placed based on the target element size. The refinements and the user-defined outline points are allowed to change this control. However, the minimum element size is used to make sure the outline segment is not too small for the mesh generation.

Step 3: **Marking the points with curvature consideration:** The curvature control is used to allow small outline segments to be used on the boundary where curvature radii are small. Three neighboring outline points are used to calculate the curvature radius. The associate curvature circle can then be formed by a number of line segments. With the same number of line segments used to approximate a circle, smaller outline segments for smaller curvature circles can be had (Figure 4-27).



$$l = \frac{2\pi R}{n}$$

Figure 4-27 Curvature Consideration

Step 4: **Marking the points with thin region consideration:** In the area where a thin region is formed, small elements are preferred. This can be done by detecting the thin region and using smaller outline segments in the area. The segment length used for the thin area is to allow at least three elements to be presented across the thin area.

Step 5: **Smoothing the outline points:** Smoothing is required on the outline so that the segment length can gradually vary.

Step 6: **Interpolations:** Interpolation is the actual process to create a new outline based on the extracted outline and the marking of the outline points. Linear interpolation is used on the contact area to prevent penetration into the other contact bodies. Cubic spline line interpolation is used for the free surfaces.

Remeshing Based on the Target Number of Elements

Instead of giving the element size, you can give the target number of elements for the remeshing. The number of elements in the new mesh can also be controlled by using a percentage tolerance to ensure that the new mesh does not have too many or too few elements. However, this tolerance control requires remeshing trials and it cannot be used with automatic stop and restart control.

Chapter 5
Structural Procedure Library

Load Incrementation

Several history definition options are available in Marc to input mechanical and thermal load increments (see Table 5-8). The choice is between a fixed and an automatic load stepping scheme. For a fixed scheme, the load step size remains constant during a load case. The fixed schemes are AUTO LOAD for static mechanical, CREEP INCREMENT for creep, DYNAMIC CHANGE for dynamic mechanical and TRANSIENT NON AUTO for thermal/thermo-mechanically coupled. For an adaptive scheme, the load step size changes from one increment to the other and also within an increment depending on convergence criteria and/or user-defined criteria. The adaptive schemes are AUTO STEP and AUTO INCREMENT for static mechanical, AUTO CREEP and AUTO STEP for creep, AUTO STEP and TRANSIENT for dynamic mechanical and AUTO STEP and TRANSIENT for thermal/thermo-mechanically coupled.

The main automatic scheme is AUTO STEP, which has been greatly enhanced in the 2001 release. Previously, one or more criteria based upon strain, stress displacement, or temperature increments for controlling the time step had to be given. These criteria are still available, but there is a new criterion for automatically controlling the load step based upon the number of recycles. This allows the option to be used directly with default settings for mechanical, thermal and thermomechanically coupled analysis problems. More details on the AUTO STEP scheme are given in the next section.

While the improved AUTO STEP option is designed as the default adaptive stepping procedure, it may still be advantageous to use one of the other available adaptive time-stepping options. This is the case for the following analysis types:

- Post buckling or snap-through analyses requires the so-called arclength method which is available through the AUTO INCREMENT option. This option can only be used in static mechanical analyses and the applied load is automatically increased or decreased in order to maintain a certain arclength. AUTO INCREMENT can also be used for general situations without instabilities, but, in general, the AUTO STEP option is preferred for these situations.
- For creep analysis, the available adaptive options are AUTO CREEP and AUTO STEP. The AUTO CREEP option supports some features which are not available through AUTO STEP (see the description in Volume C). If AUTO STEP is used for creep problems, it may be advisable to use a creep strain increment criterion and a stress increment criterion as additional user-defined criteria in combination with the default recycle criterion. AUTO STEP is usually more reliable in cases involving creep and contact.
- For the case of automatic load stepping for a thermally loaded elastic-creep/elastic-plastic-creep stress analysis, the only available scheme is AUTO THERM CREEP (see Volume C).

- For thermally driven mechanical problems, the AUTO THERM option can be used. The thermal loads derived from a thermal analysis are applied using the CHANGE STATE option in a mechanical analysis. The load step of the mechanical analysis is automatically adjusted based upon user-specified (allowed) changes in temperature from the thermal analysis per increment. For example, if there is a change of 50° in the thermal analysis in one increment and you only allowed a change of 10° per increment in the mechanical analysis, AUTO THERM splits up the thermal increment into five mechanical increment.

Table 5-8 History Definition Options for Load Incrementation

Load Type	Fixed	Adaptive
Mechanical	AUTO LOAD DYNAMIC CHANGE CREEP INCREMENT	AUTO STEP AUTO INCREMENT AUTO TIME* AUTO THERM AUTO CREEP AUTO THERM CREEP
Thermal	TRANSIENT NON AUTO	AUTO STEP TRANSIENT

* The option is obsolete, use AUTO STEP instead

Selecting Load Increment Size

Selecting a proper load step increment is an important ingredient in a nonlinear solution scheme. Large steps often lead to many recycles per increment and, if the step is too large, it can lead to inaccuracies and non convergence. On the other hand, using too small steps is inefficient.

When a fixed load stepping scheme is used, it is important to select an appropriate load step size that captures the loading history and allows for convergence within a reasonable number of recycles. For complex load histories, it is often necessary to break up the analysis into separate load cases with different step sizes. For fixed stepping, there is an option to have the load step automatically cut back in case of failure to obtain convergence. When an increment diverges, the intermediate deformations after each recycle can show large fluctuations and the final cause of program exit can be any of maximum number of recycles reached (exit 3002), elements going inside out (exit 1005 or 1009) or, in a contact analysis, nodes sliding off a rigid contact body (exit 2400). These deformations are normally not visible as post results (there is a feature to allow for the intermediate results to be available on the post file, see the POST option). If the cut-back feature is activated and one of these

failures occurs, the state of the analysis at the end of the previous increment is restored from a copy kept in memory, and the increment is subdivided into a number of sub increments. The step size is halved until convergence is obtained or the user-specified number of cut-backs has been performed. Once an increment is converged, the rest of the original increment is completed in equally sized steps unless further cut-backs are necessary. No results are written to the post file during subincrementation, and the original increment count is preserved.

Automatic Load Incrementation

In many nonlinear analyses, it is useful to have Marc figure out the appropriate load step size automatically. The AUTO INCREMENT option is a so-called arc-length method and is designed for applications like post buckling and snap-through analysis. This method is described in detail in Chapter 11, Arc-Length Methods on page 11.

AUTO STEP

The scheme appropriate for most other applications is AUTO STEP. The primary control of the load step is based upon the number of recycles needed to obtained convergence. There are a number of optional user-specified physical criteria that can be used to additionally control the load step. For the recycle based option, the user specifies a desired number of recycles. This number is used as a target value for the load stepping scheme. If the number of recycles needed to obtain convergence exceeds the desired number, the load step size is reduced, the recycle counter is reset to zero and the increment is performed again with the new load step. The factor with which the time step is cut back defaults to 1.2 and can be specified by you. The load step for the next increment is increased if the number of recycles required in the current increment is less than the desired number. The same factor that is used for decreasing the time step is used for increasing it. The load step is never increased during an increment. In addition, the same type of cut-back feature for fixed load stepping is available for this scheme as well. If the maximum number of recycles is reached (exit 3002), elements go inside out (exit 1005/1009), or nodes slide off the end of a rigid contact body (exit 2400), the load step is halved and the increment is redone. Whenever the load step is reduced (either due to number of recycles becoming larger than desired or one of the four exits), the state at the end of the previous increment is restored from a copy kept in memory. This is the principal behavior of the scheme.

There are some exceptions to the basic scheme outlined above. If an increment is consistently converging with the original load step and the number of recycles exceeds the desired number, the number of recycles is allowed to go beyond the

desired number until convergence or up to the user specified maximum number. The time step is then decreased for the next increment. An increment is determined to be converging if the convergence ratio was decreasing in three previous recycles.

Special rules also apply in a contact analysis. For quasi-static problems, the AUTO STEP option is designed to only use the automated penetration check option (see CONTACT option, 7th field of 2nd data block; option 3 is always used). Even if you flag the increment splitting penetration check option, Marc internally converts it to automated penetration check. During the recycles, the contact status can keep changing (new nodes come in contact, nodes slide to new segments, separate etc.). Whenever the contact status changes during an increment, a new set of contact constraints are incorporated into the equilibrium equations and more recycles are necessary in order to find equilibrium. These extra recycles, which are solely due to contact changes, are not counted when the comparison is made to the desired number for determining if the load step needs to be decreased within the increment. Thus, only true Newton-Raphson iterations are taken into account. For the load step of the next increment, the accumulated number of recycles during the previous increment is used. This ensures that the time step is not increased when there are many changes in contact during the previous increment.

In addition to allowing Marc to use the number of recycles for automatically controlling the step size for AUTO STEP, user-specified physical criteria can be used for controlling the step size. You can specify the maximum allowed incremental change within certain ranges for specific quantities during an increment. The quantities available are displacements, rotations, stresses, strains, strain energy, and temperature (in thermal or thermomechanically coupled analyses). These criteria can be utilized in two ways. By default, they are used as limits, which means that the load step is decreased if a criterion is violated during the current increment, but they do not influence the decision to change the load step for the next increment (that is, only the actual number of recycles versus desired number of recycles controls the load step for the next increment). The criteria can also be used as targets; in which case, they are used as the main means for controlling the time step for the current and next increments. If the calculated values of the criteria are higher than the user-specified values, the time step is scaled down. If the obtained values for a converged increment are less than the user specified, the time step is scaled up. The scale factor used is the ratio between the actual value and the target value, and this factor is limited by user-specified minimum and maximum factors (defaults to 0.1 and 10 respectively). If this type of load step control is used together with the recycle based control, the time step can be reduced due to whichever criterion that is violated. The decision to increase the step size for the next increment is based upon the physical criteria.

In many analyses, it is convenient to obtain post file results at specified time intervals. This is naturally obtained with a fixed load stepping scheme but not with an automatic scheme. Traditionally, the post output frequency is given as every n^{th} increment. With the AUTO STEP option, you can request post output to be obtained at equally spaced time intervals. In this case, the time step is temporarily modified to exactly reach the time for output. The time step is then restored in the following increment.

The AUTO STEP option also has an artificial damping feature available for mechanical statics analysis. If the time step is decreased to below the user-specified minimum time step, Marc normally stops with exit number 3015, but if the artificial damping feature is activated, the analysis is continued with a smaller time step. The solution is stabilized by adding a factored mass matrix to the stiffness matrix. This artificial stabilization is turned off once the time step increases above the minimum time step. No post file results are written while the artificial damping is active. The critical parameter for this feature is the (artificial) mass density. You need to specify an appropriate value for all materials in the model.

The defaults of the AUTO STEP scheme are carefully chosen to be adequate in a wide variety of applications. There are cases, however, when the settings may need to be modified. Assume that the default settings are used, which means that the recycle based control is active with an initial load of one per cent of the total. If the structure is weakly nonlinear, convergence can be obtained in just a few recycles even if the step is large. In certain cases this may lead to an inaccurate solution. It can also lead to problems if an initially weakly nonlinear structure suddenly exhibits stronger nonlinearities; for instance, occurrence of plasticity or parts coming into contact. Possible remedies to this problem include:

(i) decrease the time step scale factor from 1.2 to a smaller number so the step size does not grow so rapidly;

(ii) use a physical criterion like maximum increment of displacements to limit the load step;

(iii) use the maximum time step to limit large steps;

(iv) decrease the desired and maximum number of recycles to make the scheme more prone to decrease the load step if more recycles are needed.

Another situation is if the structure is highly nonlinear and convergence is slow. In this case, it may be necessary to increase the desired number and maximum number of recycles. In general, there is a close connection between the convergence tolerances used and the desired number and maximum number of recycles. In many cases, it may be beneficial to use one or more physical criteria; for example, the increment of plastic strain as targets for controlling the load step.

Data Transfer from Axisymmetric Analysis to 3-D Analysis

In many cases, it is possible to begin the numerical simulations as a two-dimensional axisymmetric problem even though the final problem is fully three-dimensional. This is advantageous because of the large computational savings. For this to be useful, the first stage of the problem should be truly axisymmetric. The second stage of the problem can be fully three-dimensional. The AXITO3D model definition option is used in the input file of the 3-D problem to transfer results from an axisymmetric analysis into a 3-D analysis. The data from the axisymmetric analysis is stored on the post file. In the 3-D analysis, the results from axisymmetric analysis are used as initial conditions.

There are three steps in performing an axisymmetric to 3-D analysis.

1. Run axisymmetric analysis.
2. Expand axisymmetric model to 3-D model and transfer data from axisymmetric model to 3-D model.
3. Run 3-D analysis.

Most quadrilateral axisymmetric elements (in 3-D case, hexahedral elements), including Herrmann elements, and most available materials, such as metal and rubber, can be used in the axisymmetric to 3-D analysis. For problems involving large deformation, either total or updated Lagrangian formulation must be used. Thermal and dynamic effects are considered.

For more detailed information on this feature, see the model definition option, AXITO3D, in *MSC.Marc Volume C: Program Input*.

Chapter 6 Nonstructural Procedure Library

Output

Marc prints out both the nodal temperatures and the temperatures at the element centroid when the CENTROID parameter is used, or at the integration points if the ALL POINTS parameter is invoked. You can also indicate on the HEAT parameter for the program to print out the temperature gradients and the resulting nodal fluxes.

To create a file of element and nodal point temperatures, use the POST model definition option. This file can be used as temperature input for performing a thermal stress analysis. This file is processed using the CHANGE STATE option in the subsequent thermal stress analysis. This post file can also interface with Mentat to plot temperature as a function of time.

Heat Transfer with Convection

Marc has the capability to perform heat transfer with convection if the velocity field is known. The numerical solutions of the convection-diffusion equation have been developed in recent years. The streamline-upwind Petro-Galerkin (SUPG) method has been implemented into the Marc heat transfer capability.

The elements which are available are described in Table 6-1.

Table 6-1 Heat Transfer Convection Elements

Element Type	Description
36, 65	2-, 3-node link
37, 39, 41, 50, 69, 85, 86	3-, 4-, 8-node planar
38, 40, 42, 70, 87, 88, 122	4-, 4-, 8-node axisymmetric
43, 44, 71, 123, 133	8-, 20-node hexahedron
135	4-node tetrahedral
133	10-node tetrahedron
131, 132	6-node triangular

To activate the convection contribution, use the HEAT parameter and set the fifth field to 2. Due to the nonsymmetric nature of the convection term, the nonsymmetric solver is used automatically. Specify the nodal velocity vectors using the VELOCITY option. To change velocity, use VELOCITY CHANGE. If nonuniform velocity vectors are required, user subroutine UVELOC is used. This capability can be used in conjunction with the Rigid-Plastic Flow on page 85 to perform a coupled analysis, in which the velocity fields are obtained.

Technical Background

The general convection-diffusion equation is:

$$\frac{\partial T}{\partial t} + v \cdot \nabla T = \nabla \cdot (\kappa \nabla T) + Q \tag{6-32}$$

The perturbation weighting functions are introduced as:

$$W = N + \alpha\left(\frac{h}{2|v|}(v \cdot \nabla N)\right) + \beta\left(\frac{h}{4|v|}\Delta t(v \cdot \nabla N)\right) \tag{6-33}$$

N is the standard interpolation function in Equation 6-1. The upwinding parameter, α, is the weighting used to eliminate artificial diffusion of the solution; while the beta term, β, is to avoid numerical dispersion. $|v|$ is the magnitude of local velocity vectors. T is the temperature, κ is the diffusion tensor. Q is the source term and Δt is the time increment.

The optimal choice for α and β are:

$$\alpha = \coth(Peclet/2) - (2/Peclet)$$

$$\beta = C/3 - (2/Peclet)*(\alpha/C)$$

where $Peclet$ is the local Peclet number in the local element and C is the local Courant number:

Peclet = density * specific heat * characteristic length * magnitude of the fluid velocity/conductivity

Peclet = ρ*c*h*$|v|/k$

and

$C = |v|*(\Delta t)/h$ where (Δt) is the time increment.

The characteristic length h is defined in [Ref 12] where $C \leq 1$ is required for numerical stability. When $C > 1$, the β is set to be zero and a large time step is recommended to avoid numerical dispersion.

Note: The interpolation function N is not the time-space functions defined in [Ref 6], so that most Marc heat transfer elements can be used. The convection contribution of heat transfer shell elements is limited due to the definitions of the perturbation weighting function and the interpolation function.

Denote the virtual potential by W; then, the variational formulation is

$$\int_V \mu^{-1}(\nabla \times W) \bullet (\nabla \times A) dV = \int_V W \bullet J dV + \int_V W \bullet (\nabla \times \mu^{-1} B_r) dV + \int_\Gamma W \bullet (H \times n) dA \tag{6-64}$$

where n is the outward normal to V at the boundary Γ.

In the three-dimensional case, the Coulomb gauge, Equation 6-63, is enforced with a penalty formulation. The resulting term added to the variational formulation, Equation 6-64 reads:

$$\int_V \mu^{-1}(\nabla \times W) \bullet (\nabla \times A) dV = \int_V W \bullet J dV + \int_V W \bullet (\nabla \times \mu^{-1} B_r) dV + \int_\Gamma W \bullet (H \times n) dA + \int_V r(\nabla \bullet W)(\nabla \bullet A) dV \tag{6-65}$$

The default value used for r is:

$$r = 10^{-4} \left| \mu^{-1} \right| \tag{6-66}$$

Chapter 7
Material Library

Table 7-1 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, 142-148 $\{\varepsilon\} = [1/E_{xx}]\{\sigma\}$
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 = 2,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 $\begin{Bmatrix}\varepsilon_{mm}\\ \varepsilon_{\theta\theta}\\ \gamma_T\end{Bmatrix} = \begin{bmatrix}1/E_{mm} & -\nu_{\theta m}/E_{\theta\theta} & 0\\ -\nu_{m\theta}/E_{mm} & 1/E_{\theta\theta} & 0\\ 0 & 0 & 1/G_{m\theta}\end{bmatrix}\begin{Bmatrix}\sigma_{mm}\\ \sigma_{\theta\theta}\\ \tau_T\end{Bmatrix}$

Table 7-1 Classes of Stress-Strain Relations (Continued)

Class 6	NDI = 3,NSHEAR = 1 Plane Strain, Axisymmetric with No Twist, Elements 151-154. $$\begin{Bmatrix}\varepsilon_{xx}\\ \varepsilon_{yy}\\ \varepsilon_{zz}\\ \gamma_{xy}\end{Bmatrix} = \begin{bmatrix} 1/E_{xx} & -\nu_{yx}/E_{yy} & -\nu_{zx}/E_{zz} & 0\\ -\nu_{xy}/E_{xx} & 1/E_{yy} & -\nu_{zy}/E_{zz} & 0\\ -\nu_{xz}/E_{xx} & -\nu_{yz}/E_{yy} & 1/E_{zz} & 0\\ 0 & 0 & 0 & 1/G_{xy}\end{bmatrix}\begin{Bmatrix}\sigma_{xx}\\ \sigma_{yy}\\ \sigma_{zz}\\ \tau_{zx}\end{Bmatrix}$$ $$\nu_{yx} = \nu_{xy}E_{yy}/E_{xx} \qquad \nu_{zy} = \nu_{yz}E_{zz}/E_{yy} \qquad \nu_{xz} = \nu_{zx}E_{xx}/E_{zz}$$
Class 7	NDI = 2,NSHEAR = 3 Thick Shell, Elements 22, 75 $$\begin{Bmatrix}\varepsilon_{xx}\\ \varepsilon_{yy}\\ \gamma_{xy}\\ \gamma_{yz}\\ \gamma_{zx}\end{Bmatrix} = \begin{bmatrix} 1/E_{xx} & -\nu_{yx}/E_{yy} & 0 & 0 & 0\\ -\nu_{xy}/E_{xx} & 1/E_{yy} & 0 & 0 & 0\\ 0 & 0 & 1/G_{xy} & 0 & 0\\ 0 & 0 & 0 & 1/G_{yz} & 0\\ 0 & 0 & 0 & 0 & 1/G_{zx}\end{bmatrix}\begin{Bmatrix}\sigma_{xx}\\ \sigma_{yy}\\ \tau_{xy}\\ \tau_{yz}\\ \tau_{xy}\end{Bmatrix}$$ $$\nu_{xy} = \nu_{xy}E_{yy}/E_{xx}$$
Class 8	NDI = 3,NSHEAR = 3 Three-Dimensional Brick Elements, Elements 149, 150 $$\begin{Bmatrix}\varepsilon_{xx}\\ \varepsilon_{yy}\\ \varepsilon_{zz}\\ \gamma_{xy}\\ \gamma_{yz}\\ \gamma_{zx}\end{Bmatrix} = \begin{bmatrix} 1/E_{xx} & -\nu_{yx}/E_{yy} & -\nu_{zx}/E_{zz} & 0 & 0 & 0\\ -\nu_{xy}/E_{xx} & 1/E_{yy} & -\nu_{zy}/E_{zz} & 0 & 0 & 0\\ -\nu_{xz}/E_{xx} & -\nu_{yz}/E_{yy} & 1./E_{zz} & 0 & 0 & 0\\ 0 & 0 & 0 & 1/G_{xy} & 0 & 0\\ 0 & 0 & 0 & 0 & 1/G_{yz} & 0\\ 0 & 0 & 0 & 0 & 0 & 1/G_{zx}\end{bmatrix}\begin{Bmatrix}\sigma_{xx}\\ \sigma_{yy}\\ \sigma_{zz}\\ \tau_{xy}\\ \tau_{yz}\\ \tau_{zx}\end{Bmatrix}$$

Mohr-Coulomb Material (Hydrostatic Yield Dependence)

Marc includes options for elastic-plastic behavior based on a yield surface that exhibits hydrostatic stress dependence. Such behavior is observed in a wide class of soil and rock-like materials. These materials are generally classified as Mohr-Coulomb materials (generalized von Mises materials). Ice is also thought to be a Mohr-Coulomb material. The generalized Mohr-Coulomb model developed by Drucker and Prager is implemented in Marc. There are two types of Mohr-Coulomb materials: linear and parabolic. Each is discussed on the following pages.

Linear Mohr-Coulomb Material

The deviatoric yield function, as shown in Figure 7-18, is assumed to be a linear function of the hydrostatic stress.

$$f = \alpha J_1 + J_2^{1/2} - \frac{\bar{\sigma}}{\sqrt{3}} = 0 \tag{7-97}$$

where

$$J_1 = \sigma_{ii} \tag{7-98}$$

$$J_2 = \frac{1}{2}\sigma'_{ij}\,\sigma'_{ij} \tag{7-99}$$

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.

The constants α and $\bar{\sigma}$ can be related to c and ϕ by

$$c = \frac{\bar{\sigma}}{[3(1-12\alpha^2)]^{1/2}}\,; \qquad \frac{3\alpha}{(1-3\alpha^2)^{1/2}} = \sin\phi \tag{7-100}$$

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

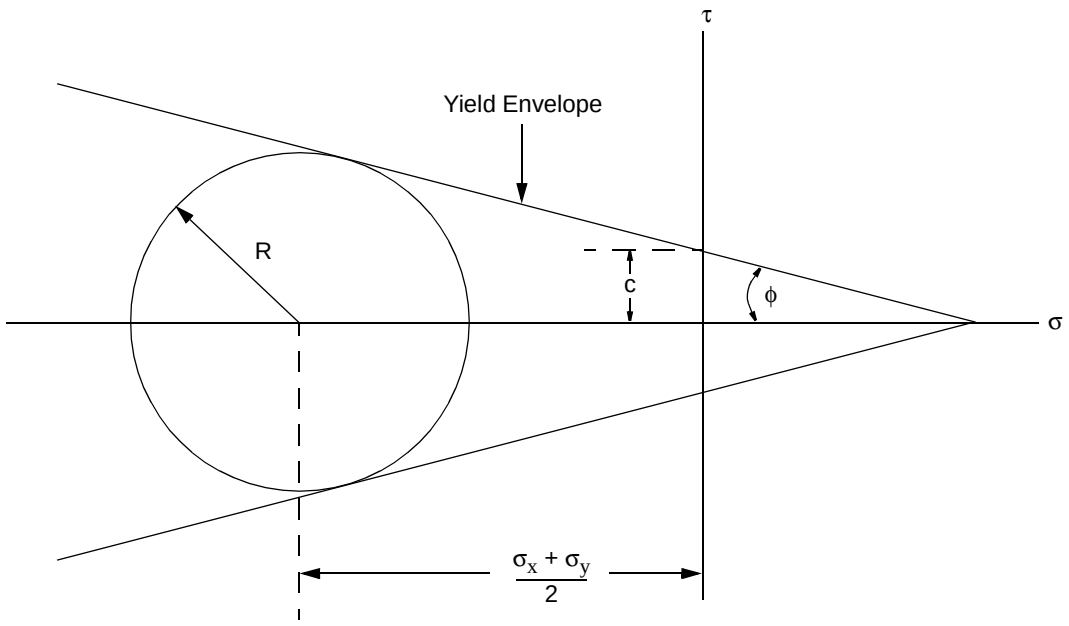


Figure 7-18 Yield Envelope of Plane Strain (Linear Mohr-Coulomb Material)

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 7-19).

$$f = (3J_2 + \sqrt{3}\beta\bar{\sigma}J_1)^{1/2} - \bar{\sigma} = 0 \tag{7-101}$$

The parabolic yield surface is obtained in Marc through the ISOTROPIC model definition option. Enter the values σ and β through the model definition option ISOTROPIC.

$$\bar{\sigma}^2 = 3\left(c^2 - \frac{\alpha^2}{3}\right) \qquad \beta = \frac{\alpha}{(3(3c^2 - \alpha^2))^{1/2}} \tag{7-102}$$

where c is the cohesion.

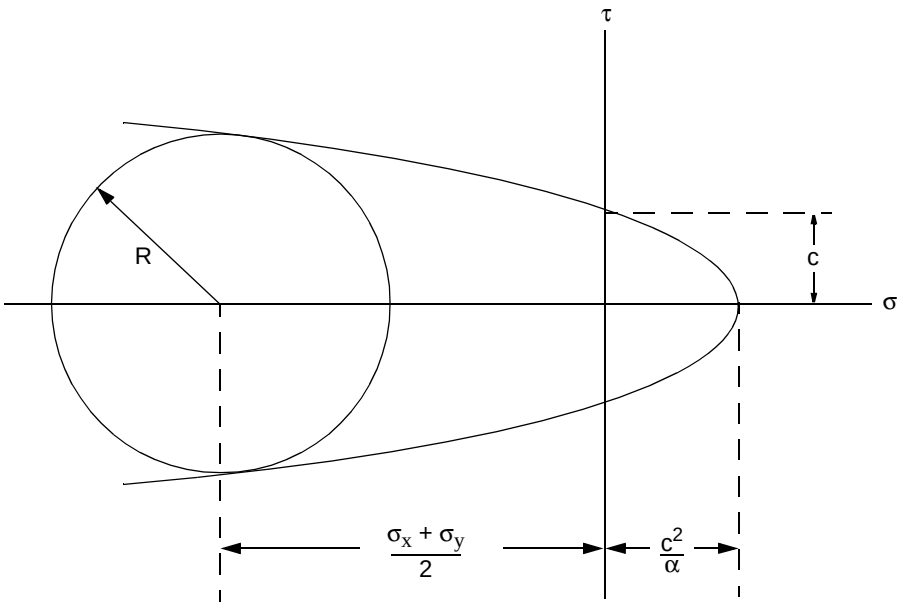


Figure 7-19 Resultant Yield Condition of Plane Strain (Parabolic Mohr-Coulomb Material)

Buyukozturk Criterion (Hydrostatic Stress Dependence)

This yield criterion [Ref. 2], which orginally has been proposed as a failure criterion, has the general form:

$$f = \beta\sqrt{3}\bar{\sigma}J_1 + \gamma J_1^2 + 3J_2 - \bar{\sigma}^2 \qquad (7\text{-}103)$$

Through the ISOTROPIC model definition option, the user has to define $\bar{\sigma}$ and the factor β, where γ has a fixed value of 0.2. The Buyukozturk criterion reduces to the parabolic Mohr-Coulomb criterion if $\gamma = 0$.

Damage Models

In many structural applications, the finite element method is used to predict failure. This is often performed by comparing the calculated solution to some failure criteria, or by using classical fracture mechanics. Previously, we discussed two models where the actual material model changed due to some failure, see Progressive Composite Failure on page 22 and the previous section on Low Tension Material on page 100. In this section, the damage models appropriate for ductile metals and elastomeric materials will be discussed.

Ductile Metals

In ductile materials given the appropriate loading conditions, voids will form in the material, grow, then coalesce, leading to crack formation and potentially, failure. Experimental studies have shown that these processes are strongly influenced by hydrostatic stress. Gurson studied microscopic voids in materials and derived a set of modified constitutive equations for elastic-plastic materials. Tvergaard and Needleman modified the model with respect to the behavior for small void volume fractions and for void coalescence.

In the modified Gurson model, the amount of damage is indicated with a scalar parameter called the void volume fraction f. The yield criterion for the macroscopic assembly of voids and matrix material is given by:

$$F = \left(\frac{\bar{\sigma}}{\sigma_y}\right)^2 + 2q_1 f^* \cosh\left(\frac{q_2 \sigma_{kk}}{2\sigma_y}\right) - [1 + (q_1 f^*)^2] = 0 \qquad (7\text{-}233)$$

as seen in Figure 7-53.

The parameter q_1 was introduced by Tvergaard to improve the Gurson model at small values of the void volume fraction. For solids with periodically spaced voids, numerical studies [10] showed that the values of $q_1 = 1.5$ and $q_2 = 1$ were quite accurate.

Chapter 8
Contact

Numbering of Contact Bodies

When defining contact bodies for a deformable-to-deformable analysis, it is important to define them in the proper order. As a general rule, a body with a finer mesh should be defined before a body with a coarser mesh.

Note: For problems involving adaptive meshing or automated remeshing, care must be taken to satisfy this rule before as well as after the mesh change.

If one has defined a body numbering which violates the general rule, or if the rule is violated upon remeshing, then a CONTACT TABLE model or history definition option can be used to modify the order in which contact will be established. This order can be directly user-defined or decided by the program. In the latter case, the order is based on the rule that if two deformable bodies might come into contact, searching is done for nodes of the body having the smallest element edge length. It should be noted that this implies single-sided contact for this body combination, as opposed to the default double-sided contact.

Automatic Penetration Checking Procedure

To detect contact between bodies whose boundaries are moving towards each other, an automatic penetration checking procedure is available. This procedure significantly increases accuracy and stability for models in which boundary nodes are displacing significantly. Typical examples include metal forming processes (sheet forming and forging), highly deformable elastomeric models (rubber boots), and snap-fit problems (inserting a key into a lock).

The automatic penetration checking procedure is available under the CONTACT option. It is automatically activated if the adaptive loading procedure is selected. It is not currently available for dynamics or the arc length control procedure. If the automatic penetration checking procedure is selected for these two options, a different procedure, as described below, is used instead.

From a computational perspective, the automatic penetration checking procedure detects penetration each time displacements are updated.

For implicit analysis, this typically happens after a matrix solution which produces a change in the displacements due to a change in applied loads and internal forces. The procedure detects nodes traversing a contact boundary due to the change in displacements. If at least one node penetrates a contact surface, a scale factor is applied to the change in displacements such that the penetrating nodes are moved back to the contact surface.

The automatic penetration checking procedure can, therefore, be considered to be a type of a line search. The procedure also looks at the magnitude of the change in displacement of nodes which already are contacting and not necessarily penetrating. Using stability considerations, the scale factor calculated above may be further modified. In addition, for nodes on a contact boundary which are not yet contacting, a similar procedure is followed to enhance stability.

Because the procedure can reduce the change in displacements, it may require more iterations to complete an increment. It is important to ensure that the maximum allowable number of iterations to complete an increment is set to a sufficiently large value. When the adaptive loading procedure is used, or when the fixed time stepping procedure is used with automatic restarting, the increment automatically restarts if the maximum allowable number of iterations is exceeded. In the case of the adaptive loading procedure, the time step is modified.

When dynamics or the arc length control method is used, the above procedure is not available. Instead, penetration is checked for when convergence is achieved, usually after multiple iterations.

Chapter 9
Boundary Conditions

Table 9-21 Rigid Link Constraint

Tying Code	Number of Retained Nodes	Purpose	Remarks
80	2	Define a rigid link between nodes	The second retained node is an unattached node which contains the rotation

Shell-to-Solid Tying

In many problems, a region exists that is modeled with both brick elements and shell elements. A particular case of this is shown is Figure 9-20 and Figure 9-21.

In the first case, an 8-node brick which has been reduced to a triangular prism is connected to a 4-node shell. In the second case, a 20-node brick is connected to an 8-node shell. An automatic constraint equation is developed between the elements. Note that the thickness of the shell must be entered as the brick thickness.

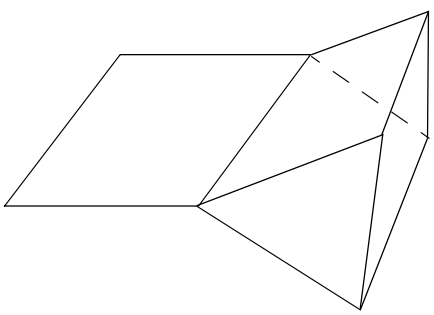


Figure 9-20 4-Node Shell-to-Solid Automatic Constraint

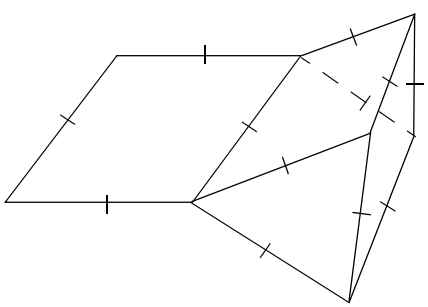


Figure 9-21 8-Node Shell-to-Solid Automatic Constraint

Cyclic Symmetry

A special set of tying constraints for continuum elements can be automatically generated by the Marc program to effectively analyze structures with a geometry and a loading varying periodically about a symmetry axis. Figure 9-22 shows an example where, on the left-hand side, the complete structure is given and, on the right-hand side, a sector to be modeled.

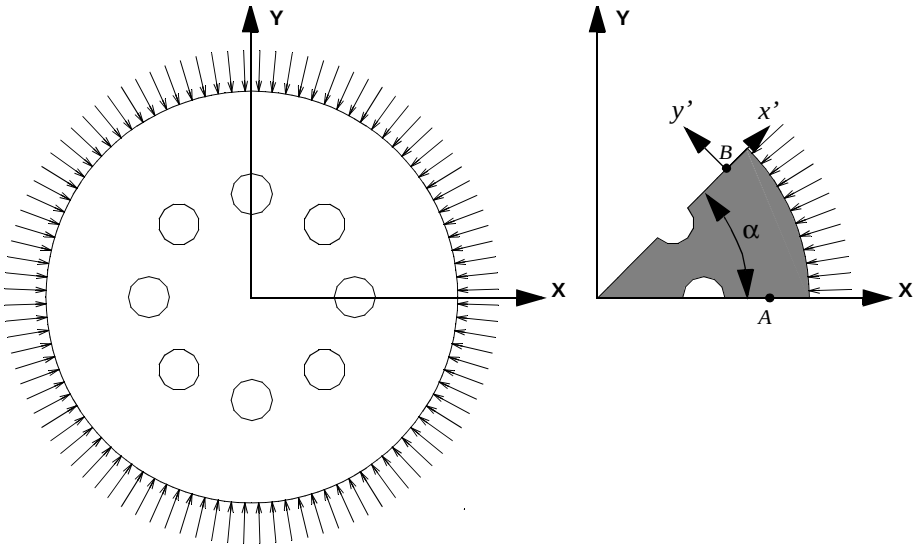


Figure 9-22 Cyclic symmetric structure: complete model (left) and modeled sector (right)

Looking at points A and B on this segment, the displacement vectors should fulfil:

$$\boldsymbol{u}'_B = \boldsymbol{u}_A \tag{9-19}$$

which can also be written as:

$$\boldsymbol{u}_B = \boldsymbol{R}\boldsymbol{u}_A \tag{9-20}$$

where the transformation matrix $\boldsymbol{R}$ depends on the symmetry axis (which, in the example above, coincides with the global Z-axis) and the sector angle α (see Figure 9-22). In Marc, the input for the option CYCLIC SYMMETRY consists of the direction vector of the symmetry axis, a point on the symmetry axis and the sector angle α. The following items should be noted:

1. The meshes do not need to line up on both sides of a sector (for example, see Figure 9-23).

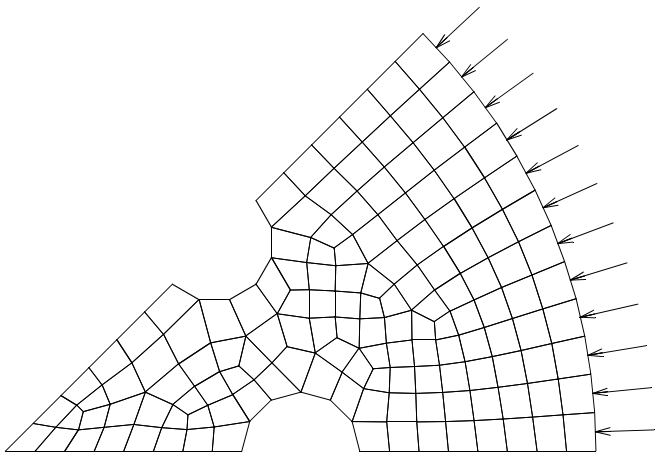


Figure 9-23 Finite element mesh for cyclic symmetric structure with different mesh densities on the sector sides

2. Any shape of the sector sides is allowed provided, that upon rotating the sector $360/\alpha$ times about the symmetry axis over the sector angle α, will result in the complete model.
3. The CYCLIC SYMMETRY option can be combined with the CONTACT option.
4. The CYCLIC SYMMETRY option can be combined with global remeshing.
5. In a coupled thermo-mechanical analysis, the temperature is forced to be cyclic symmetric ($T_A = T_B$ as in Figure 9-22).
6. A nodal point on the symmetry axis is automatically constrained in the plane perpendicular to the symmetry axis.
7. The possible rigid body motion about the symmetry axis can be automatically suppressed.

Cross Section

In various engineering applications, it is necessary to define a pre-stress in for example bolts or rivets before applying any other structural loading. Although such a pre-stressed state is often simulated using a temperature loading, it is rather difficult to arrive at a desired net force in the bolt or rivet. An easier way is to use the option CROSS-SECTION.

Generally speaking, a cross section in a model is defined by (see Figure 9-24):

1. All the nodes in the cross section;
2. The normal vector to the cross section;
3. All the elements sharing nodes in the cross section and lying on the side of the cross section corresponding to the opposite normal vector.

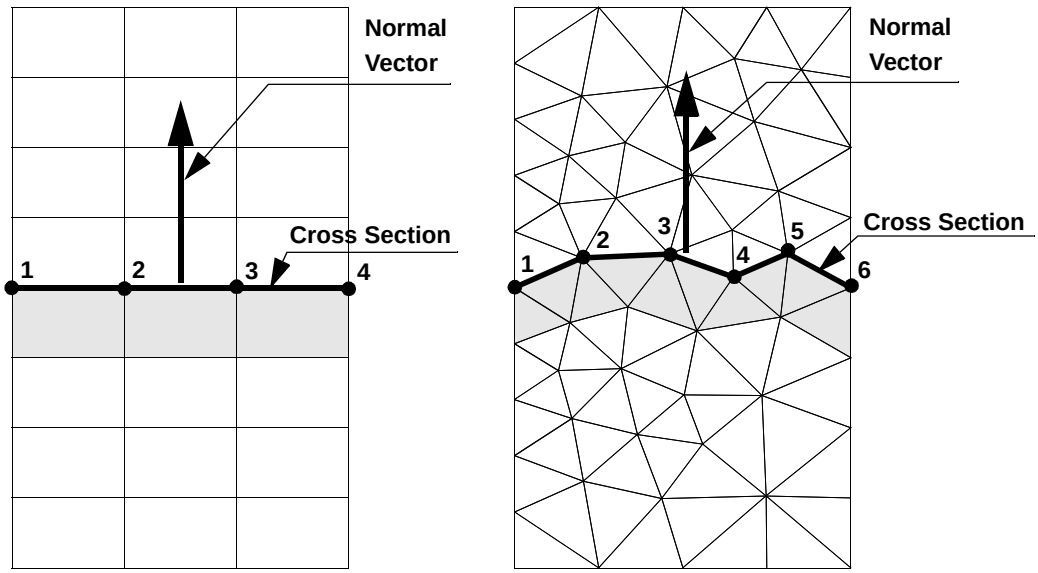


Figure 9-24 Cross section examples: regular mesh (left) and irregular mesh (right)

Notice that this option can be used for continuum as well as for beam and shell stress elements.

An additional node, called *nc*, has to be assigned to every cross section defined in a model. Such a node has only one degree of freedom, namely the displacement in the direction of the cross section normal vector, which is indicated as u_{nc}. Elements corresponding to the cross section definition now get a modified displacement field,

depending on which nodes of the elements are part of the cross section. If the third and fourth node of a 4-node element are part of the cross section, the element displacement vector is changed according to (see Figure 9-25):

$$\boldsymbol{U}_e^T = \left[\boldsymbol{u}_1^T\ ,\ \boldsymbol{u}_2^T\ ,\ \boldsymbol{u}_3^T + u_{nc}\boldsymbol{n}^T\ ,\ \boldsymbol{u}_4^T + u_{nc}\boldsymbol{n}^T\right] \tag{9-23}$$

Similarly, if only the fourth node is part of the cross section, the element displacement vector is given by:

$$\boldsymbol{U}_e^T = \left[\boldsymbol{u}_1^T\ ,\ \boldsymbol{u}_2^T\ ,\ \boldsymbol{u}_3^T\ ,\ \boldsymbol{u}_4^T + u_{nc}\boldsymbol{n}^T\right] \tag{9-24}$$

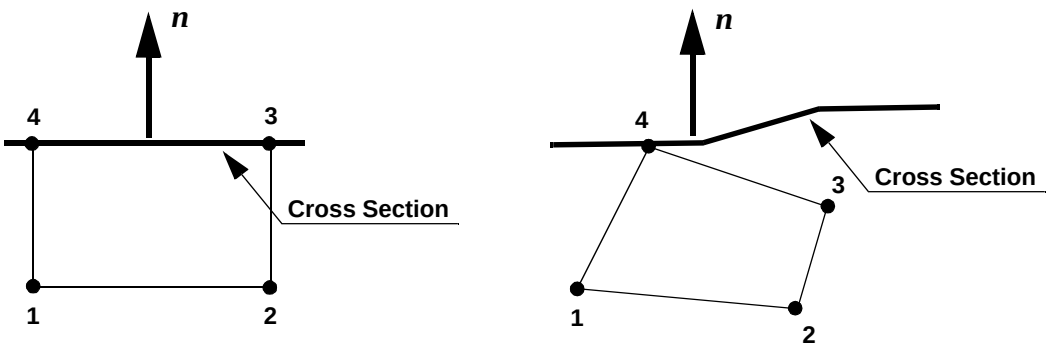


Figure 9-25 Cross section with 4-node elements: two nodes in cross section (left), one node in cross section (right)

As a result, the following is possible:

1. A force in the cross section can be defined by specifying a POINT LOAD on the first degree of freedom of node *nc*. Notice that this force works in the direction of the cross section normal vector. The resulting displacement of node *nc* can be seen as the shortening of the cross section elements.
2. A constant overclosure in the cross section can be obtained by specifying the first degree of freedom of node *nc* via the FIXED DISP or DISP CHANGE option. Then, the reaction force in node *nc* is the section force in the direction of the cross section normal vector.

During an analysis, you can switch from one of the above options to the other. So, to simulate the behavior of a pre-stressed bolt under additional loading conditions, you should first define the pre-tension force via the POINT LOAD option and subsequently use the DISP CHANGE option to "lock" the bolt by setting the incremental displacement Δu_{nc} to zero.

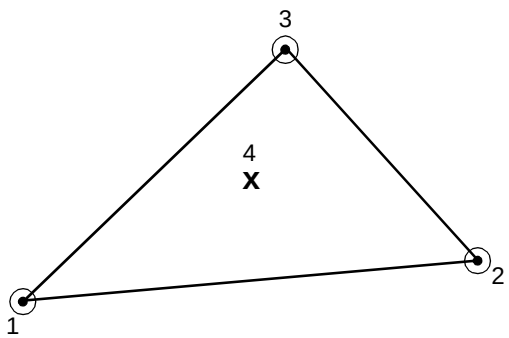
If you are only interested in the total force in the cross section, the displacement u_{nc} should be set to zero. Then, the total force in the cross section is given by the reaction force in node *nc*.

Chapter 10
Element Library

Incompressible Elements

Incompressible and nearly incompressible materials can be modeled by using a special group of elements in the program. These elements, based on modified Hermann variational principles, are capable of handling large deformation effects as well as creep and thermal strains. The incompressibility constraint is imposed by using Lagrange multipliers. Generally, the low (linear) order elements have a single additional node which contains the Lagrange multiplier, while the high-order (quadratic) elements have Lagrange multipliers at each corner node.

Elements 155 (plane strain triangle), 156 (axisymmetric triangle), and 157 (3-D tetrahedron) are low-order elements and are exceptions within the incompressible element group. They have an additional node located at the center of the elements and have Lagrange multipliers at each corner node. The shape function of the center node is a bubble function. See Figure 10-1 for an element 155 case. The degrees of freedom of the center node is condensed out on the element level before the assembly of the global matrix.



⊙ Nodes with both displacements and pressure as degrees of freedom

X Center node with only displacement degrees of freedom. The shape function for this node is a bubble function.

Figure 10-1 Element 155

Large Strain Elasticity

The incompressible elements based on Hermann formulations can be used for large strain analysis of rubber-like materials, using either total Lagrangian formulation or updated Lagrangian formulation.

Chapter 11
Solution Procedures for Nonlinear Systems

Convergence Controls

The default procedure for convergence criterion in Marc is based on the magnitude of the maximum residual load compared to the maximum reaction force. This method is appropriate since the residuals measure the out-of-equilibrium force, which should be minimized. This technique is also appropriate for Newton methods, where zero-load iterations reduce the residual load. The method has the additional benefit that convergence can be satisfied without iteration.

The basic procedures are outlined below.

1. RESIDUAL CHECKING

$$\frac{\|F_{residual}\|_\infty}{\|F_{reaction}\|_\infty} < TOL_1 \tag{11-44}$$

$$\frac{\|F_{residual}\|_\infty}{\|F_{reaction}\|_\infty} < TOL_1 \text{ and } \frac{\|M_{residual}\|_\infty}{\|M_{reaction}\|_\infty} < TOL_2 \tag{11-45}$$

$$\|F_{residual}\|_\infty < TOL_1 \tag{11-46}$$

$$\|F_{residual}\|_\infty < TOL_1 \text{ and } \|M_{residual}\|_\infty < TOL_2 \tag{11-47}$$

Where F is the force vector, and M is the moment vector. TOL_1 and TOL_2 are control tolerances. $\|F\|_\infty$ indicates the component of F with the highest absolute value. Residual checking has two drawbacks. First, if the CENTROID parameter is used, the residuals and reactions are not calculated accurately. Second, in some special problems, such as free thermal expansion, there are no reaction forces. The program uses displacement checking in either of these cases.

2. DISPLACEMENT CHECKING

$$\frac{\|\delta u\|_\infty}{\|\Delta u\|_\infty} < TOL_1 \tag{11-48}$$

$$\frac{\|\delta u\|_\infty}{\|\Delta u\|_\infty} < TOL_1 \text{ and } \frac{\|\delta \phi\|_\infty}{\|\Delta \phi\|_\infty} < TOL_2 \tag{11-49}$$

$$\|\delta u\|_\infty < TOL_1 \tag{11-50}$$

$$\|\delta u\|_\infty < TOL_1 \text{ and } \|\delta \phi\|_\infty < TOL_2 \tag{11-51}$$

where Δu is the displacement increment vector, δu is the correction to incremental displacement vector, $\Delta\phi$ is the correction to incremental rotation vector, and $\delta\phi$ is the rotation iteration vector. With this method, convergence is satisfied if the maximum displacement of the last iteration is small compared to the actual displacement change of the increment. A disadvantage of this approach is that it results in at least one iteration, regardless of the accuracy of the solution.

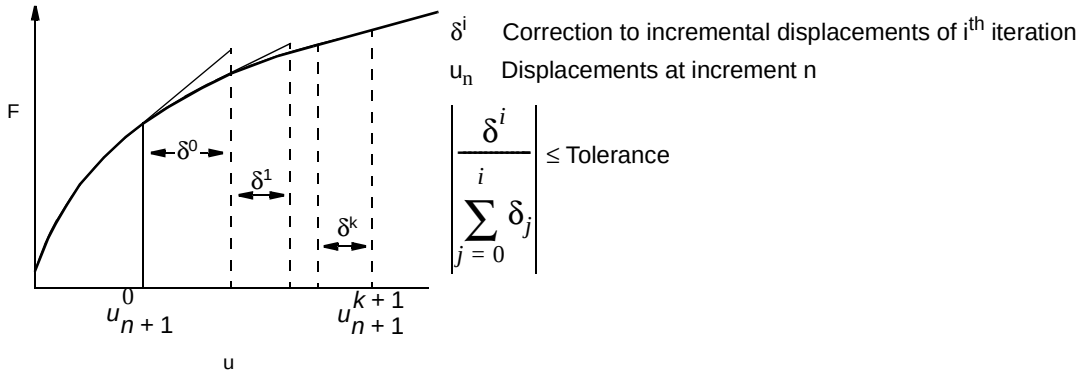


Figure 11-7 Displacement Control

3. STRAIN ENERGY CHECKING

This is similar to displacement testing where a comparison is made between the strain energy of the latest iteration and the strain energy of the increment. With this method, the entire model is checked.

$$\frac{\delta E}{\Delta E} < TOL_1 \tag{11-9}$$

where ΔE is the strain energy of the increment and δE is the correction to incremental strain energy of the iteration. These energies are the total energies, integrated over the whole volume. A disadvantage of this approach is that it results in at least one iteration, regardless of the accuracy of the solution. The advantage of this method is that it evaluates the global accuracy as opposed to the local accuracy associated with a single node.

Besides the basic procedures, Marc provides more choices for convergence control. Different problems require different schemes to detect the convergence efficiently and accurately. To do this, the following combinations of residual checking and displacement checking are available.

4. RESIDUAL OR DISPLACEMENT CHECKING

   This procedure does convergence checking on both residuals (Procedure 1) and displacements (Procedure 2). Convergence is obtained if one converges.

5. RESIDUAL AND DISPLACEMENT CHECKING

   This procedure does a convergence check on both residuals and displacements (Procedure 4). Convergence is achieved if both criteria converge simultaneously.

For problems where maximum reactions or displacements are extremely small (even close to the round-off errors of computers), the convergence check based on relative values could be meaningless if the convergence criteria chosen is based on these small values. It is necessary to check the convergence with absolute values; otherwise, the analysis is prematurely terminated due to a nonconvergent solution. Such situations are not predicable and usually happen at certain stages of an analysis. For example, problems with stress free motion (rigid body motion or free thermal expansion) and small displacements (springback or constraint thermal expansion) may need to check absolute value at some stage of the analysis. However, it is not convenient and it is also difficult to determine when to check the absolute value and how small the absolute criterion value should be used due to its unpredictability. In order to reduce human interventions and improve the robustness of an FE analysis, Marc allows you to use the AUTO SWITCH option to switch the convergence check scheme if the above mentioned situation occurs during an FE analysis. Turning ON the AUTO SWITCH option allows Marc to automatically change the convergence check scheme to Procedure 4 if small reactions or displacements are detected. This function can be deactivated by turning the AUTO SWITCH option OFF or specifying an absolute value check.

Solution of Linear Equations

The finite element formulation leads to a set of linear equations. The solution is obtained through numerically inverting the system. Because of the wide range of problems encountered with Marc, there are several solution procedures available.

Most analyses result in a system which is real, symmetric, and positive definite. While this is true for linear structural problems, assuming adequate boundary conditions, it is not true for all analyses. Marc has two main modes of solvers – direct and iterative. Each of these modes has two families of solvers, based upon the storage procedure. While all of these solvers can be used if there is adequate memory, only a subset uses spill logic for an out-of-core solution. Finally, there are classifications based upon nonsymmetric and complex systems. This is summarized below:

	Direct Profile	Direct Sparse	Iterative Sparse	Multifrontal Sparse	Vendor Provided Sparse
Solver Option	0	4	2	8	6
Real Symmetric	Yes	Yes	Yes	Yes	Yes
Real Nonsymmetric	Yes	No	No	Yes	No
Complex Symmetric	Yes	No	No	Yes	No
Complex nonsymmetric	No	No	No	Yes	No
Out-of-core	Yes	Yes	No	Yes	SGI only
Possible problem with poorly conditioned systems	No	No	Yes	No	No

The choice of the solution procedure is made through the SOLVER option.

Direct Methods

Traditionally, the solution of a system of linear equations was accomplished using direct solution procedures, such as Cholesky decomposition and the Crout reduction method. These methods are usually reliable, in that they give accurate results for virtually all problems at a predictable cost. For positive definite systems, there are no computational difficulties. For poorly conditioned systems, however, the results can degenerate but the cost remains the same. The problem with these direct methods is that a large amount of memory (or disk space) is required, and the computational costs become very large.

Nonsymmetric Systems

The following analyses types result in nonsymmetric systems of equations:

Inclusion of convective terms in heat transfer analysis
Coriolis effects in transient dynamic analysis
Fluid mechanics
Soil analysis
Follower force stiffness
Frictional contact

The first three always result in a nonsymmetric system. The last three can be solved either fully using the nonsymmetric solver, or (approximately) using a symmetric solver. The nonsymmetric problem uses twice as much memory for storing the stiffness matrix.

Complex Systems

Marc utilizes a complex operator matrix for dynamic harmonic analyses when the damping matrix is present or for transient electromagnetic analyses. The matrix is always symmetric.

Iterative Solvers

In Marc, an iterative sparse solver is available using a *sparse matrix technique*. This method is advantageous for different classes of problems.

There exist certain types of analyses for which the sparse iterative solver is not appropriate. These types include:

elastic analysis
explicit creep analysis
complex harmonic analysis
substructures
central difference techniques
eigenvalue analysis
use of gap elements

Elastic or explicit creep analysis involves repeated solutions using different load vectors. When a direct solver is used, this is performed very efficiently using back substitution. However, when an iterative solver is used, the stiffness matrix is never inverted, and the solution associated with a new load vector requires a complete re-solution.

The sparse iterative solver can exhibit poor convergence when Herrmann incompressible elements are present.

Chapter 12
Output Results

Status File

The status file contains summarized information of the analysis. Each line within the main content of this file includes the number of load cases and increments; the total cycles, separations from contact surfaces, and cutbacks numbers occurring within each increment; the accumulated total cycles, separations, cutbacks, and remeshing times numbers in the analysis as well as the time step size of each increment and the overall time achieved by the analysis. Marc reports all this information on one line upon completion of each increment. If the increment is partially completed (for example, the completion of one part of a split increment or one part of the whole increment due to cutback), Marc also reports one line for each part of the increment. Therefore, if the increment has increment splittings or cutbacks, it is reported in several lines.

Volume B: Element Library

Chapter 2 MSC.Marc Element Classifications

Special Elements

Elements 4, 8, 12, 22, 23, 24, 31, 45, 46, 47, 48, 49, 68, 72, 75, 76, 77, 78, 79, 85, 86, 87,88, 89, 90, 91, 92, 93, 94, 95, 96, 97, 98, 99, 100, 101, 102, 103, 104, 105, 106, 107, 108, 109, 110, 136, 137, 138, 139, 140, 141, 142, 143, 144, 145, 146, 147, 148, 149, 150, 151, 152, 153, 154, 155, 156, and 157

These special elements do not belong to any of the 18 element classes described in this chapter. These elements include Beam and Shell Elements (4, 8, 22, 24, 31, 49, 72, 75-79, 85-90, 98, and 138-140), Friction and Gap Elements (12 and 97), Rebar Elements (23, 46-48, and 142-148), Elastic Shear Panel (68), Semi-infinite Elements (91-94 and 101-108), Axisymmetric Elements with Bending (95 and 96), Composite Elements (149-154), Lower Triangular and Tetrahedral Elements with a cubic bubble function (155-157), and Magnetostatic Elements (109 and 110). See Chapter 3 for detailed information on these elements.

Elements types 99, 100, 136, 137, and 141 are not available.

Chapter 3
Element Library

Element 22

Quadratic Thick Shell Element

Element type 22 is an eight-node thick shell element with global displacements and rotations as degrees of freedom. Second-order interpolation is used for coordinates, displacements and rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The transverse shear strains are calculated at ten special points and interpolated to the integration points. In this way, this element behaves correctly in the limiting case of thin shells. The element can be degenerated to a triangle by collapsing one of the sides. Tying type 22, which connects shell and solid, is available for this element.

Lower-order elements, such as type 75, are preferred in contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

All constitutive relations can be used with this element.

Geometric Basis

The element is defined geometrically by the (x, y, z) coordinates of the four corner nodes and four midside nodes. The element thickness is specified in the GEOMETRY option. The stress output is given with respect to local orthogonal surface directions, V_1, V_2, and V_3 for which each integration point is defined in the following way (see Figure 3-8).

At each of the integration points, the vectors tangent to the curves with constant isoparametric coordinates are normalized:

$$\underset{\sim}{t}_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \eta}\right|, \quad \underset{\sim}{t}_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now, a new basis is being defined as follows:

$$s = t_1 + t_2, d = t_1 - t_2$$

After normalizing these vectors by $\bar{s} = s/\sqrt{2}|s|$ $\bar{d} = d/\sqrt{2}|d|$, the local orthogonal directions are then obtained as follows:

$$V_1 = \bar{s} + \bar{d}, \quad V_2 = \bar{s} - \bar{d}, \quad \text{and} \quad V_3 = V_1 \times V_2$$

In this way, the vectors $\frac{\partial x}{\partial \tilde{\xi}}$, $\frac{\partial x}{\partial \tilde{\eta}}$ and V_1, V_2 have the same bisecting plane.

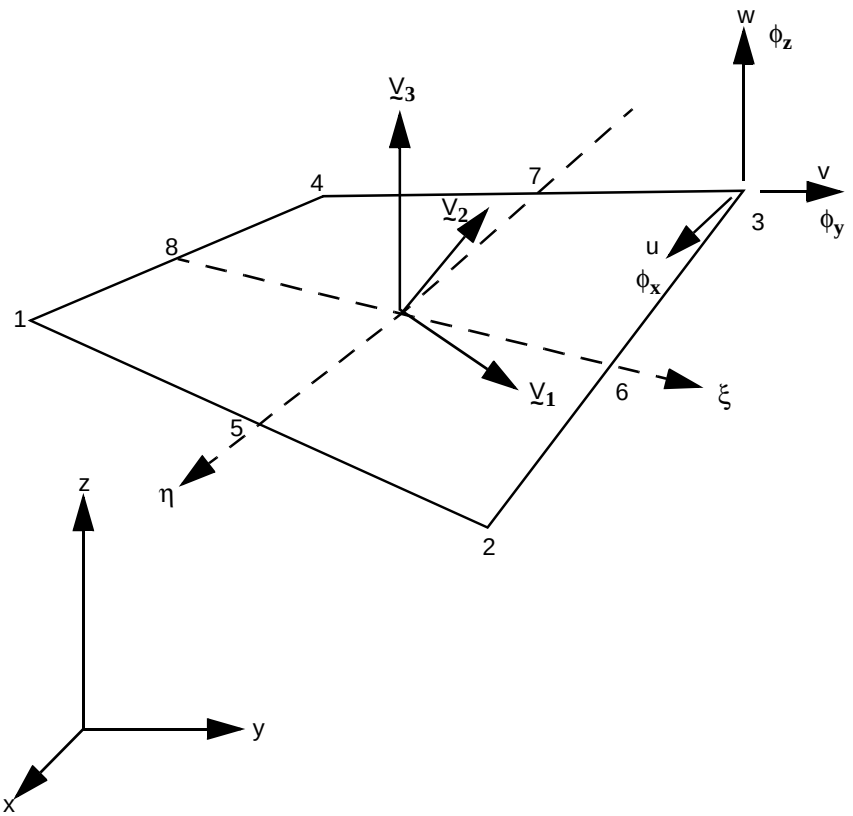


Figure 3-8 Form of Element 22

Displacements

The six nodal displacement variables are as follows:

u, v, w	Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z	Rotation components about global x-, y-, and z-axis, respectively.

Quick Reference

Type 22

Bilinear, eight-node shell element including transverse shear effects.

Connectivity

Eight nodes per element. The element can be collapsed to a triangle.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third, and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2 = EGEOM3 = EGEOM4 = 0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

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

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_x = rotation about global x-axis
5 = ϕ_y = rotation about global y-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

Distributed load types follow below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.
3	Nonuniform gravity load per surface area in -z-direction; magnitude given in the FORCEM user subroutine.
4	Nonuniform pressure with positive magnitude in -V_3-direction; magnitude given in the FORCEM user subroutine.

Load Type	Description
5	Nonuniform load per surface area in arbitrary direction; magnitude given in the FORCEM user subroutine.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to this edge.
12	Nonuniform edge load; magnitude given in the FORCEM user subroutine in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load; magnitude and direction given in the FORCEM user subroutine on 1-2 edge.
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to this edge.
22	Nonuniform edge load; magnitude given in the FORCEM user subroutine in the plane of the surface on 2-3 edge.
23	Nonuniform edge load; magnitude and direction given in the FORCEM user subroutine on 2-3 edge.
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to this edge.
32	Nonuniform edge load; magnitude given in the FORCEM user subroutine in the plane of the surface on 3-4 edge.
33	Nonuniform edge load; magnitude and direction given in the FORCEM user subroutine on 3-4 edge.
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to this edge.
42	Nonuniform edge load; magnitude given in the FORCEM user subroutine in the plane of the surface on 4-1 edge.
43	Nonuniform edge load; magnitude and direction given in the FORCEM user subroutine on 4-1 edge.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in the ROTATION A option.
102	Gravity loading in global direction. Enter 3 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 the ROTATION A option.

Point loads and moments can also be applied at the nodes.

Output of Strains

Generalized strain components are as follows:

Middle surface stretches:ε_{11} ε_{22} ε_{12}
Middle surface curvatures:κ_{11} κ_{22} κ_{12}
Transverse shear strains:γ_{23} γ_{31}

in local (V_1, V_2, V_3) system.

Output of Stresses

σ_{11}, σ_{22}, σ_{12}, σ_{23}, σ_{31} in local (V_1, V_2, V_3) system given at equally spaced layers though thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement and rotation at each node can be transformed to local direction.

Tying

Use the UFORMS user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, you have to select your loadsteps such that the rotation remains small within a load step. Thickness is only updated if the FINITE parameter is specified.

Section Stress – Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 85. See Element 85 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.

Design Variables

The thickness can be considered as a design variable.

Element 23

Three-dimensional 20-node Rebar Element

This element is an isoparametric, three-dimensional, 20-node empty brick in which you can place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 20-node brick continuum element (e.g., elements 21, 35, 57, or 61) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories can be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or via the REBAR user subroutine.

Integration

It is assumed that several “layers” of rebars are presented. The number of such “layers” is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element faces (although the rebar direction is arbitrary), so that the “thickness” direction is from one of the element faces to its opposite one. For instance (see Figure 3-29), if the layer is similar to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the “thickness” direction is from the 1, 2, 3, 4 face to 5, 6, 7, 8 face of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points (see Figure 3-29). At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross section), and orientation of the rebars. See *MSC.Marc Volume C: Program Input* for REBAR option or *MSC.Marc Volume D: User Subroutines and Special Routines* for the REBAR user subroutine.

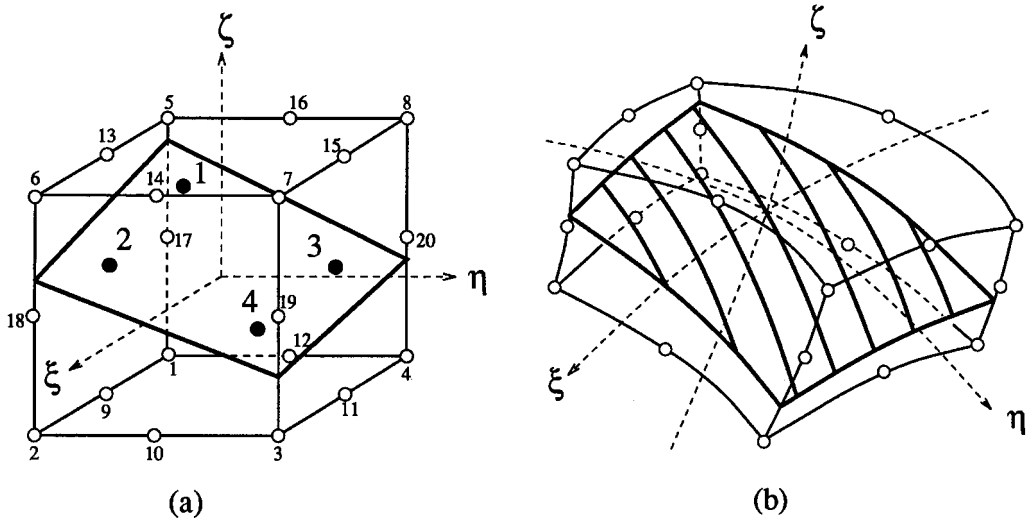


Figure 3-29 Twenty-node Rebar Element

Quick Reference

Type 23

Twenty-node, isoparametric rebar element to be used with 20-node brick continuum element.

Connectivity

Twenty nodes per element. Node numbering of the element is same as that for elements 21, 35, 57, or 61.

Geometry

If the position, equivalent thickness, and orientation of the rebars are input via the REBAR user subroutine, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, set to either 1, 2, or 3:

1. Rebar layers are similar to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the “thickness” direction is from the 1, 2, 3, 4 face to 5, 6, 7, 8 face of the element (see Figure 3-29).
2. Rebar layers are similar to the 1, 4, 8, 5 and 2, 3, 7, 6 faces of the element, the “thickness” direction is from the 1, 4, 8, 5 face to 2, 3, 7, 6 face of the element.

3. Rebar layers are similar to the 2, 1, 5, 6 and 3, 4, 8, 7 faces of the element, the “thickness” direction is from the 2, 1, 5, 6 face to 3, 4, 8, 7, face of the element.

The number of rebar layers and the isoparametric direction of the layers can also be defined using the REBAR model definition option.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 – u
2 – v
3 – w

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 20-node brick elements (e.g., element types 21, 35, 57, or 61).

Output of Strains and Stresses

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u,v,w) can be used at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 36

Three-dimensional Link (Heat Transfer Element)

This element is a simple, linear, straight link with constant cross-sectional area. It is the heat transfer equivalent of element type 9.

This element can be used as a convection-radiation link for the simulation of convective and/or radiative boundary conditions (known ambient temperatures) or, for the situation of cross convection and/or cross radiation. In order to use this element as a convection-radiation link, additional input data must be entered using the GEOMETRY option.

Quick Reference

Type 36

Three-dimensional, two-node, heat transfer link.

Connectivity

Two nodes per element (see Figure 3-52).

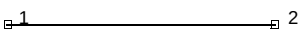


Figure 3-52 Three-dimensional Heat Link

Geometry

The cross-sectional area is input in the first data field (EGEOM1); the other fields are not used. If not specified, the cross-sectional area defaults to 1.0.

If the element is used as a convection-radiation link, the following data must be entered:

EGEOM2 = emissivity

EGEOM5 = constant film coefficient

The Stefan-Boltzmann constant and the conversion to absolute temperatures is entered through the PARAMETER model definition option.

Coordinates

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

Degrees of Freedom

One degree of freedom per node:

1 = temperature

Fluxes

Distributed fluxes according to value of IBODY are as follows:

Flux Type	Description
0	Uniform flux on first node (per cross-sectional area).
1	Volumetric flux on entire element (per volume).
2	Nonuniform flux given in the FLUX user subroutine on first node (per cross-sectional area).
3	Nonuniform volumetric flux given in the FLUX user subroutine on entire element (per volume).

Films

Same specification as **Fluxes**.

Tying

Use the UFORMS user subroutine.

Joule Heating

Capability is available.

Current

Same specification as **Fluxes**.

Output Points

A single value at the centroid is given.

Element 46

Eight-node Plane Strain Rebar Element

This element is isoparametric, plane strain, 8-node hollow quadrilateral in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 8-node plane strain continuum element (for example, element 27 or 32) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories can be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or via the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-64), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points. At each such integration point on each layer, you must input, via either the REBAR option or the REBAR user subroutine, the position, equivalent thickness (or, alternatively, spacing and area of cross section), and orientation of the rebars. See *MSC.Marc Volume C: Program Input* for REBAR option or *MSC.Marc Volume D: User Subroutines and Special Routines* for the REBAR user subroutine.

Quick Reference

Type 46

Eight-node, isoparametric rebar element to be used with 8-node plane strain continuum element.

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element 27 or 32.

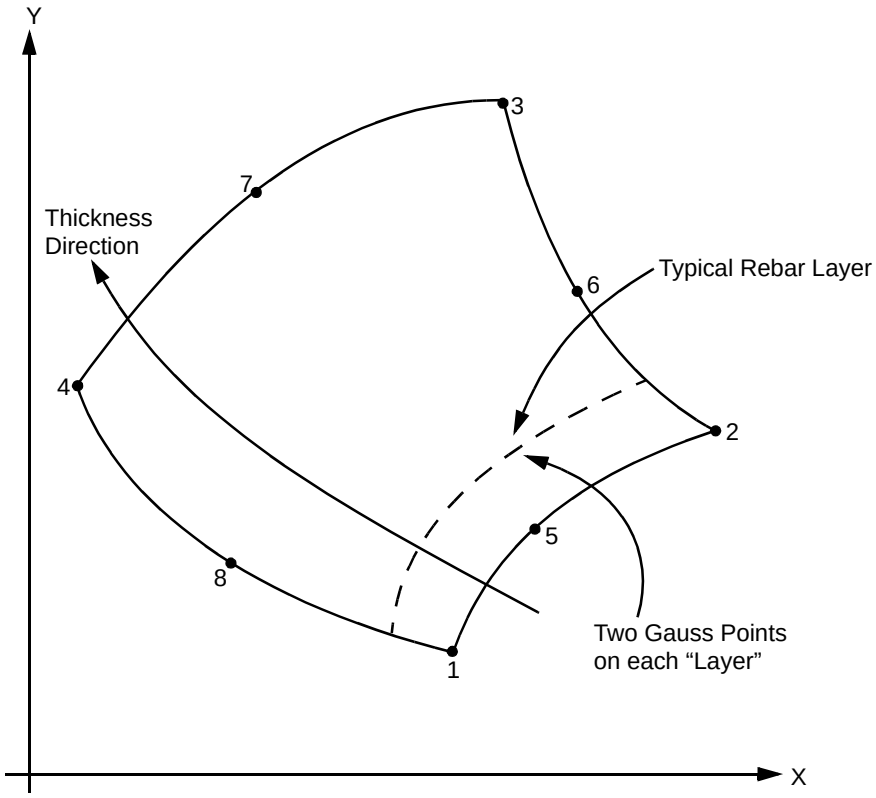


Figure 3-64 Eight-node Rebar Element Conventions

Geometry

Element thickness (in z-direction) in first field. Default thickness is unity. Note, this should not be confused with the “thickness” concept associated with rebar layers. If the position, equivalent thickness, and orientation of the rebars are input via the REBAR user subroutine, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, set to either 1 or 2:

1. Rebar layers are similar to the 1, 2 and 3, 4 edges of the element, the “thickness” direction is from the 1, 2 edge to 3, 4 edge of the element (see Figure 3-64).
2. Rebar layers are similar to the 1, 4 and 2, 3 edges of the element, the “thickness” direction is from the 1, 4 edge to 2, 3 edge of the element.

The number of rebar layers and the isoparametric direction of the layers can also be defined using the REBAR model definition option.

Coordinates

Two global coordinates x- and y-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - u
2 - v

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 8-node plane strain elements (for example, element types 27 or 32).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u,v) can be used at any node.

Special Consideration

Either REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 86

Eight-node Curved Shell (Heat Transfer Element)

This is a eight-node heat transfer shell element with temperatures as nodal degrees of freedom. Quadratic interpolation is used for the temperatures in the plane of the shell and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see the HEAT parameter). A nine-point Gaussian integration is chosen for the element in the plane of the shell and Simpson's rule is used in the thickness direction. This element can be used in conjunction with three-dimensional heat transfer brick elements through tying for heat transfer analysis. Note that only centroid or four Gaussian points are used for the output of element temperatures.

Geometric Basis

Similar to element type 22, the element is defined geometrically by the (x, y, z) coordinates of the four corner nodes and four midside nodes. The element thickness is specified in the GEOMETRY option. Local orthogonal surface directions (V_1, V_2, and V_3) for each integration point are defined below (see Figure 3-65):

At each of the integration points, the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s/(\sqrt{2}|s|) \qquad \bar{d} = d/(\sqrt{2}|d|),$$

the local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}, \quad V_2 = \bar{s} - \bar{d}, \quad \text{and} \qquad V_3 = V_1 \times V_2$$

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

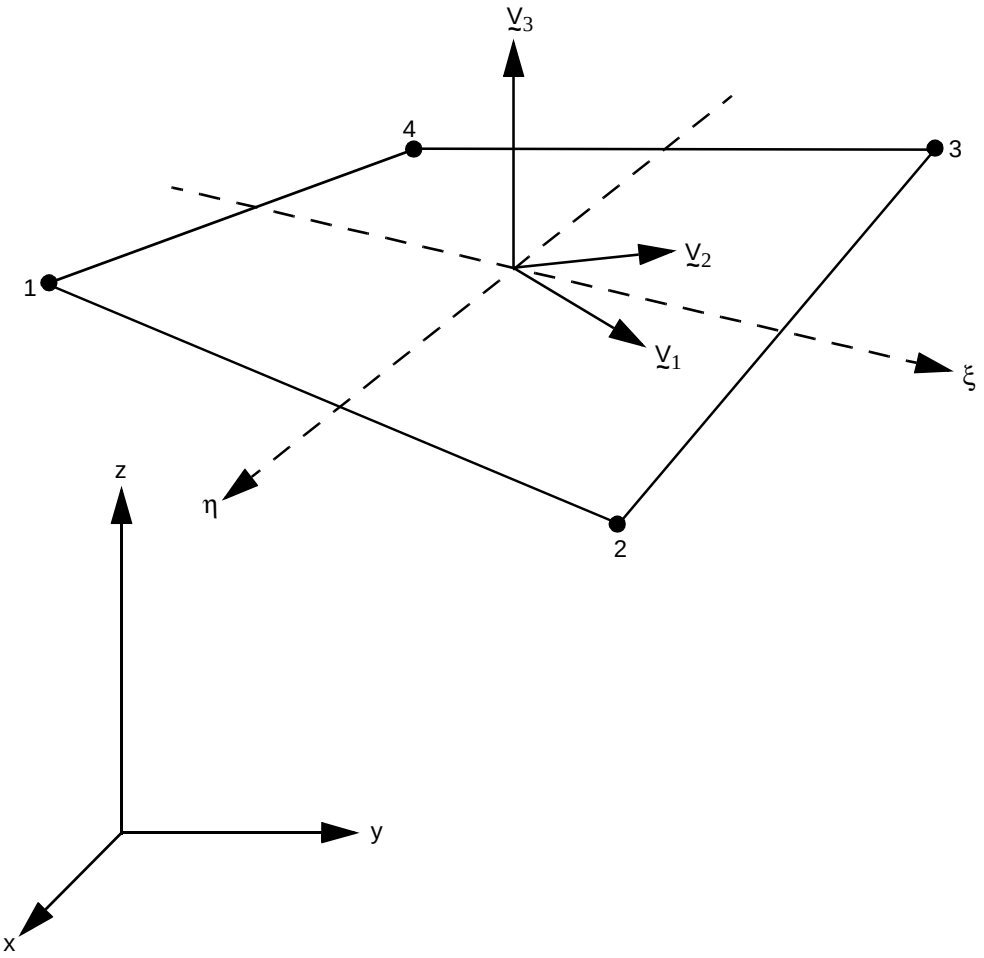


Figure 3-65 Form of Element 86

Quick Reference

Type 86

Eight-node curved heat transfer shell element.

Connectivity

Eight nodes per element.

The connectivity is specified as follows: nodes 1, 2, 3, 4 form the corners of the element, then node 5 lies at the middle of the 1-2 edge; node 6 at the middle of the 2-3 edge, etc. See Figure 3-65.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third, and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2=EGEOM3=EGEOM4=0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

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

Degrees of Freedom

N degrees of freedom per node – temperatures

N = 2 – Linear Distribution through thickness

1 = Top Surface Temperature
2 = Bottom Surface Temperature

N = 3 – Quadratic distribution through thickness

1 = Top Surface Temperature
2 = Bottom Surface Temperature
3 = Mid Surface Temperature

The first field of the HEAT parameter is used to specify whether a linear or quadratic variation will be used through the thickness.

Fluxes

Two types of distributed fluxes:

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude of flux is defined in the FLUX user subroutine.

Surface Fluxes

Flux Type	Description
5	Uniform flux per unit surface area on top surface.
6	Nonuniform flux per unit surface area on top surface, magnitude of flux is in the FLUX user subroutine.

Flux Type	Description
2	Uniform flux per unit surface area on bottom surface.
4	Nonuniform flux per unit surface area on bottom surface, magnitude of flux is defined in the FLUX user subroutine.

Point fluxes can also be applied at nodal degrees of freedom.

Films

Same specification as **Fluxes**.

Tying

Standard types 85 and 86 with three-dimensional heat transfer brick elements.

Shell Sect – Integration Through The Shell Thickness Direction

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Three points are enough for linear response. The default is 11 points.

Output Points

Temperatures are printed out at the integration points through the thickness of the shell at the centroid or Gaussian integration points in the plane of the shell. The first point in the thickness direction is on the surface of the positive normal.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as **Fluxes**.

Element 109

Eight-node 3-D Magnetostatic Element

This is an eight-node 3-D magnetostatic element with linear interpolation functions. It is similar to element type 43. The coefficient matrix is numerically integrated using eight (2 x 2 x 2) integration points.

Quick Reference

Type 109

Eight-node 3-D magnetostatic element.

Connectivity

Eight nodes per element.

See Figure 3-170 for numbering. Nodes 1-4 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 5-8 are the nodes on the other face, with node 5 opposite node 1, and so on.

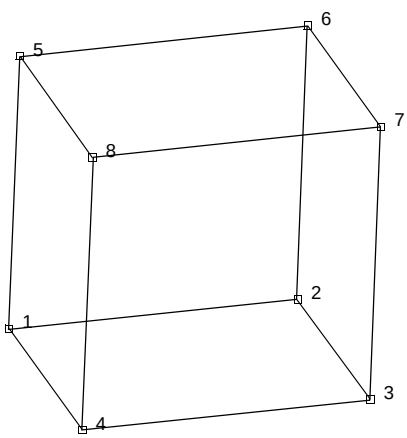


Figure 3-170 Eight-node 3-D Magnetostatic Element

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

The value of the penalty factor is given in the second field EGEOM2 (default is 0.0001).

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

Distributed Currents

Distributed currents are listed in the table below:

Current Type	Description
0	Uniform current on 1-2-3-4 face.
1	Nonuniform current on 1-2-3-4 face; magnitude is supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g. centrifugal force); magnitude and direction is supplied through the FORCEM user subroutine.
4	Uniform current on 6-5-8-7 face.
5	Nonuniform current on 6-5-8-7 face.
6	Uniform current on 2-1-5-6 face.
7	Nonuniform current on 2-1-5-6 face.
8	Uniform current on 3-2-6-7 face.
9	Nonuniform current on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform current on 4-3-7-8 face.
12	Uniform current on 1-4-8-5 face.
13	Nonuniform current on 1-4-8-5 face.
20	Uniform current on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face.
22	Uniform body force per unit volume in -z-direction.
23	Nonuniform body force per unit volume (for example, centrifugal force); magnitude and direction is supplied through the FORCEM user subroutine.
24	Uniform current on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face.
26	Uniform current on 2-1-5-6 face.

Current Type	Description
27	Nonuniform load on 2-1-5-6 face.
28	Uniform current on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face.
32	Uniform current on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face.
40	Uniform shear 1-2-3-4 face in 1⇒2 direction.
41	Nonuniform shear 1-2-3-4 face in 1⇒2 direction.
42	Uniform shear 1-2-3-4 face in 1⇒2 direction.
43	Nonuniform shear 1-2-3-4 face in 2⇒3 direction.
48	Uniform shear 6-5-8-7 face in 5⇒6 direction.
49	Nonuniform shear 6-5-8-7 face in 5⇒6 direction.
50	Uniform shear 6-5-8-7 face in 6⇒7 direction.
51	Nonuniform shear 6-5-8-7 face in 6⇒7 direction.
52	Uniform shear 2-1-5-6 face in 1⇒2 direction.
53	Nonuniform shear 2-1-5-6 face in 1⇒2 direction.
54	Uniform shear 2-1-5-6 face in 1⇒5 direction.
55	Nonuniform shear 2-1-5-6 face in 1⇒5 direction.
56	Uniform shear 3-2-6-7 face in 2⇒3 direction.
57	Nonuniform shear 3-2-6-7 face in 2⇒3 direction.
58	Uniform shear 3-2-6-7 face in 2⇒6 direction.
59	Nonuniform shear 3-2-6-7 face in 2⇒6 direction.
60	Uniform shear 4-3-7-8 face in 3⇒4 direction.
61	Nonuniform shear 4-3-7-8 face in 3⇒4 direction.
62	Uniform shear 4-3-7-8 face in 3⇒7 direction.
63	Nonuniform shear 4-3-7-8 face in 3⇒7 direction.
64	Uniform shear 1-4-8-5 face in 4⇒1 direction.
65	Nonuniform shear 1-4-8-5 face in 4⇒1 direction.
66	Uniform shear 1-4-8-5 face in 1⇒5 direction.
67	Nonuniform shear 1-4-8-5 face in 1⇒5 direction.

For all nonuniform currents, body forces per unit volume and loads, the magnitude and direction is supplied via the FORCEM user subroutine.

Currents are positive into element face.

Joule Heating

Capability is not available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is available.

Output Points

Centroid or eight Gaussian integration points.

Tying

Use the UFORMS user subroutine.

Element 110

Twelve-node 3-D Semi-infinite Magnetostatic Element

This is a 12-node 3-D semi-infinite magnetostatic element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-7-3 direction. Mappings are such that the element expands to infinity. The coefficient matrix is numerically integrated using 12 (2 x 3 x 2) integration points.

Quick Reference

Type 110

Twelve-node 3-D semi-infinite magnetostatic element.

Connectivity

Twelve nodes per element. See Figure 3-171 for numbering. The 1-2-6-5 face should be connected to a standard eight-node 3-D stress element and the 2-3-7-6, 5-6-7-8, 1-4-8-5 and 1-2-3-4 faces should be either connected to another 12-node 3-D semi-infinite stress element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

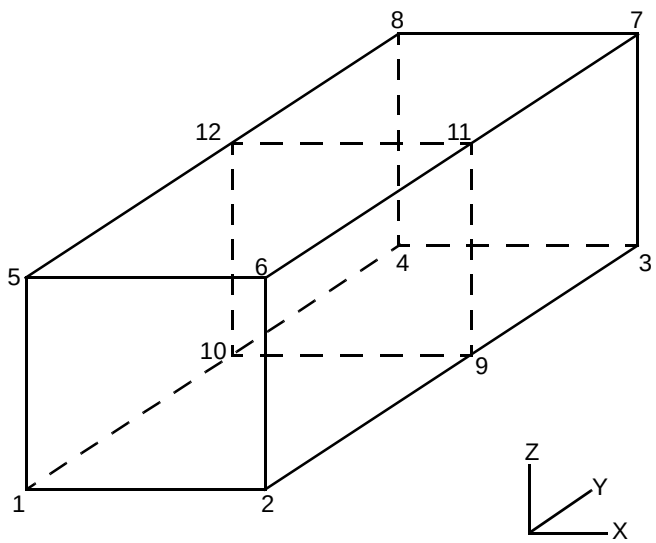


Figure 3-171 Twelve-node 3-D Semi-infinite Magnetostatic Element

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

The value of the penalty factor is given in the second field EGEOM2 (default is 0.0001).

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

Distributed Currents

Distributed currents are listed in the table below:

Current Type	Description
0	Uniform current per unit area on 1-2-3-4 face of the element.
1	Nonuniform current per unit area on 1-2-3-4 face of the element; magnitude is given in the FORCEM. user subroutine
2	Uniform current per unit volume on whole element.
3	Nonuniform current per unit volume on whole element; magnitude is given in the FORCEM user subroutine.
4	Uniform current per unit area on 5-6-7-8 face of the element.
5	Nonuniform current per unit area on 5-6-7-8 face of the element; magnitude is given in the FORCEM user subroutine.
6	Uniform current per unit area on 1-2-6-5 face of the element.
7	Nonuniform current per unit area on 1-2-6-5 face of the element; magnitude is given in the FORCEM user subroutine.
8	Uniform current per unit area on 2-3-7-6 face of the element.
9	Nonuniform current per unit area on 2-3-7-6 face of the element; magnitude is given in the FORCEM user subroutine.
12	Uniform current per unit area on 1-4-8-5 face of the element.
13	Nonuniform current per unit area on 1-4-8-5 face of the element; magnitude is given in the FORCEM user subroutine.

Point currents can be applied at nodes.

Output Points

Center or 12 Gaussian integration points (see Figure 3-172).

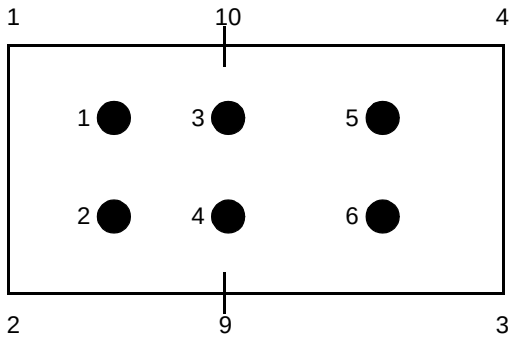


Figure 3-172 1-2 Integration Point Locations for Element 110

Joule Heating

Capability is not available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is available.

Tying

Use the UFORMS user subroutine.

Element 142

Eight-node Axisymmetric Rebar Element with Twist

This element is similar to element 48, but is written for axisymmetric applications with torsional strains. It is a hollow, isoparametric 8-node quadrilateral in which you can place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 8-node axisymmetric continuum element with twist (for example, element 66 or 67) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several “layers” of rebars are presented. The number of such “layers” is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the “thickness” direction is from one of the element edges to its opposite one. For instance (see Figure 3-213), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the “thickness” direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross section), and orientation of the rebars. See *MSC.Marc Volume C: Program Input* for the REBAR option or *MSC.Marc Volume D: User Subroutines and Special Routines* for the REBAR user subroutine.

Quick Reference

Type 142

Eight-node, isoparametric rebar element with torsional strains.

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element 66 or 67.

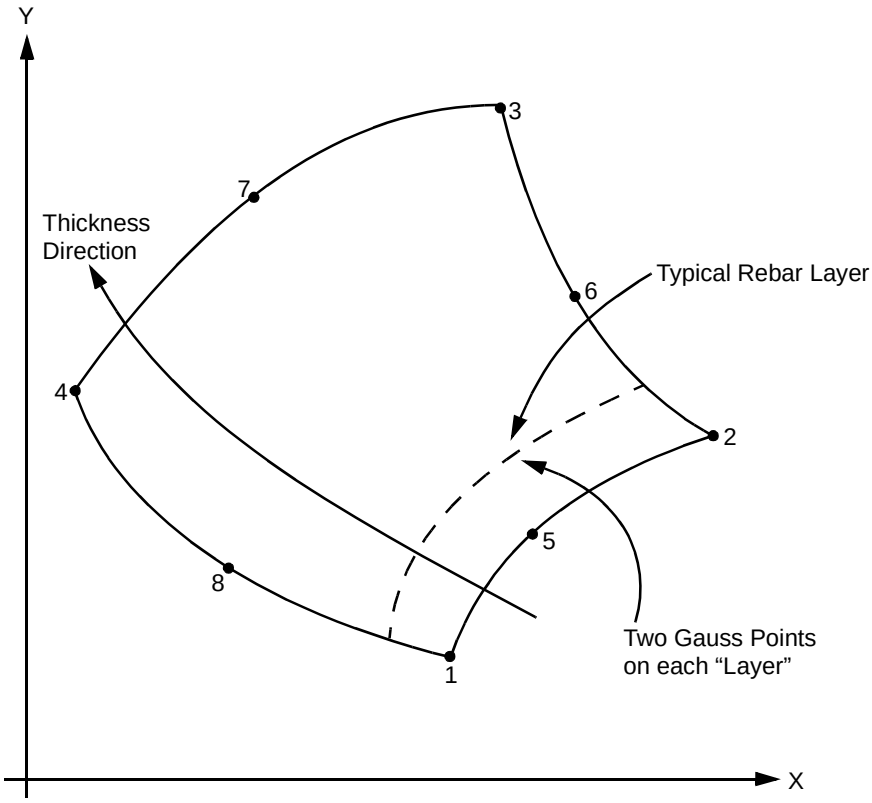


Figure 3-213 Eight-node Rebar Element Conventions

Geometry

If the position, equivalent thickness, and orientation of the rebars are input via the REBAR user subroutine, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, enter either 1 or 2:

1. Rebar layers are similar to the 1, 2 and 3, 4 edges of the element, the “thickness” direction is from the 1, 2 edge to 3, 4 edge of the element (see Figure 3-213).
2. Rebar layers are similar to the 1, 4 and 2, 3 edges of the element, the “thickness” direction is from the 1, 4 edge to 2, 3 edge of the element.

The number of rebar layers and the isoparametric direction of the layers can also be defined using the REBAR model definition option.

Coordinates

Two global coordinates in z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - z
2 - r
3 - θ

Tractions

Point loads can be applied at the nodes but no distributed loads are available. Distributed loads are applied only to corresponding 8-node axisymmetric element with twist (for example, element types 66 or 67).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformations

Any local set (u,v) can be used in the (z-r) plane at any node.

Special Considerations

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 155

Plane Strain, Low-order, Triangular Element, Herrmann Formulations

This is a isoparametric, plane strain, 3 + 1-node low-order, triangular element with an additional pressure degree of freedom at each of the three corner nodes (see Figure 3-226). It is written for incompressible or nearly incompressible plane strain applications. The shape function for the center node is a bubble function. Therefore, the displacements and the coordinates for the element are linearly distributed along the element boundaries. The stiffness of this element is formed using three Gaussian integration points. The degrees of freedom of the center node are condensed out on the element level before the assembly of the global matrix.

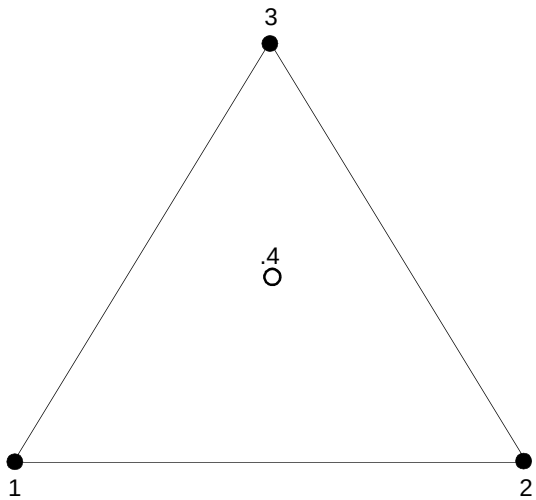


Figure 3-226 Form of Element 155

This element can be used for incompressible elasticity via total Lagrangian formulations or for rubber elasticity and elasto-plasticity via updated Lagrangian (F^eF^p) formulations. To activate large strain analysis via updated Lagrangian formulations, use either the ELASTICITY, 2 or PLASTICITY, 5 parameter (see *MSC.Marc Volume A: Theory and User Information* and *MSC.Marc Volume C: Program Input* for more information).

Integration

Three integration points are used to correctly interpolate the cubic shape function.

Quick Reference

Type 155

3 + 1-node, isoparametric, plane strain, triangular element using Herrmann formulation. Written for incompressible or nearly incompressible applications.

Connectivity

Four nodes per element (see Figure 3-226). Node numbering for the first three nodes is the same as for element type 6; that is, counterclockwise at three corners. The fourth node is located at the element center.

Geometry

Thickness of the element stored in the first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates in the x- and y-directions. Marc automatically calculates the coordinates of the fourth (center) node of the element.

Degrees of Freedom

Displacement output in global components is:

1 - u
2 - v

3 - p

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.

Load Type	Description
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in the ROTATION A 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 the ROTATION A option.

Output of Strain and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1 = global xx strain
2 = global yy strain
3 = global zz strain
4 = global xy strain

If only element type 155, 156, or 157 is used in an analysis, the post file contains only one integration point for each element. The element stresses and strains in the point are the averaged results over three integration points of the element. This is to reduce the size of the post file.

Transformation

Any local set (u, v) can be used at any node.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available with the ELASTICITY, 2 or PLASTICITY, 5 parameter.

Element 156

Axisymmetric, Low-order, Triangular Element, Herrmann Formulations

This is an isoparametric, axisymmetric, 3 + 1-node, low-order, triangular element with an additional pressure degree of freedom at each of the three corner nodes (see Figure 3-227). It is written for incompressible or nearly incompressible axisymmetric applications. The shape function for the center node is a bubble function. Therefore, the displacements and the coordinates for the element are linearly distributed along the element boundaries. The stiffness of this element is formed using three Gaussian integration points. The degrees of freedom of the center node are condensed out on the element level before the assembly of the global matrix.

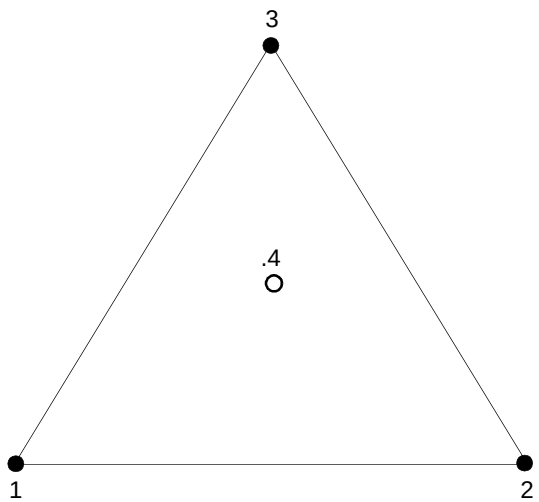


Figure 3-227 Form of Element 156

This element can be used for incompressible elasticity via total Lagrangian formulations or for rubber elasticity and elasto-plasticity via updated Lagrangian (F^eF^p) formulations.To activate large strain analysis via updated Lagrangian formulations, use either the ELASTICITY, 2 or PLASTICITY, 5 parameter (see *MSC.Marc Volume A: Theory and User Information* and *MSC.Marc Volume C: Program Input* for more information).

Integration

Three integration points are used to correctly interpolate the cubic shape function.

Quick Reference

Type 156

3 + 1-node, isoparametric, axisymmetric, triangular element using Herrmann formulation. Written for incompressible or nearly incompressible applications.

Connectivity

Four nodes per element (see Figure 3-227). Node numbering for the first three nodes is the same as for element type 2; that is, counterclockwise at three corners. The fourth node is located at the element center.

Coordinates

Two global coordinates in the z- and r-directions. Marc automatically calculates the coordinates of the fourth (center) node of the element.

Degrees of Freedom

1 - u
2 - v
3 - p

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.

Load Type	Description
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in the ROTATION A 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 the ROTATION A option.

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

Concentrated loads applied at the nodes must be the value of the load integrated around the circumference.

Output of Strain and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1 = global zz strain
2 = global rr strain
3 = global $\theta\theta$ strain
4 = global zr strain

If only element type 155, 156, or 157 is used in an analysis, the post file contains only one integration point for each element. The element stresses and strains in the point are the averaged results over three integration points of the element. This is to reduce the size of the post file.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available with the ELASTICITY, 2 or PLASITICITY, 5 parameter.

Element 157

Three-dimensional, Low-order, Tetrahedron, Herrmann Formulations

This element is a three-dimensional, isoparametric,4 + 1-node, low-order, tetrahedron with an additional pressure degree of freedom at each of the four corner nodes (see Figure 3-228). It is written for incompressible or nearly incompressible three-dimensional applications. The shape function for the center node is a bubble function. Therefore, the displacements and the coordinates for the element are linearly distributed along the element boundaries. The stiffness of this element is formed using four Gaussian integration points. The degrees of freedom of the center node are condensed out on the element level before the assembly of the global matrix.

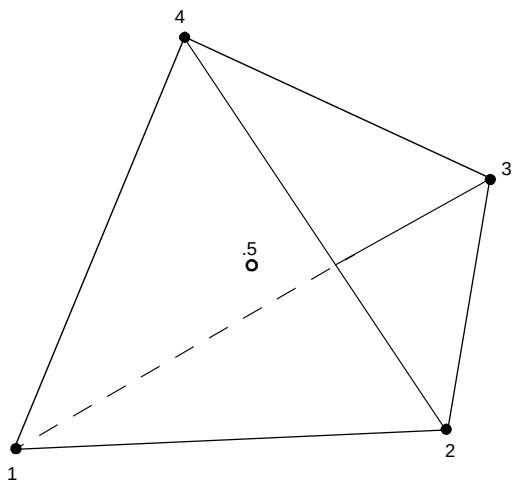


Figure 3-228 Form of Element 157

This element can be used for incompressible elasticity via total Lagrangian formulations or for rubber elasticity and elasto-plasticity via updated Lagrangian (F^eF^p) formulations.To activate large strain analysis via updated Lagrangian formulations use either the ELASTICITY, 2 or PLASTICITY, 5 parameter (see *MSC.Marc Volume A: Theory and User Information* and *MSC.Marc Volume C: Program Input* for more information).

Integration

Four integration points are used to correctly interpolate the cubic shape function.

Quick Reference

Type 157

4 + 1-node, isoparametric, three-dimensional, tetrahedron using Herrmann formulation. Written for incompressible or nearly incompressible applications.

Connectivity

Five nodes per element (see Figure 3-228). Node numbering for the first four nodes is the same as for element type 134; that is, nodes 1, 2, 3, being the corners of the first face in counterclockwise order when viewed from inside the element and node 4 on the opposing vertex. The fifth node is located at the element center.

Coordinates

Three global coordinates in the x-, y- and z-directions. Marc automatically calculates the coordinates of the fifth (center) node of the element.

Degrees of Freedom

1 - u
2 - v
3 - p

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x direction.
9	Nonuniform body force per unit volume in x direction.

Load Type	Description
10	Uniform body force per unit volume in y direction.
11	Nonuniform body force per unit volume in y direction.
12	Uniform body force per unit volume in z direction.
13	Nonuniform body force per unit volume in z direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in the ROTATION A 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 the ROTATION A option.

Output of Strain and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1 = global xx strain
2 = global yy strain
3 = global zz strain
4 = global xy strain
5 = global yz strain
6 = global zx strain

If only element type 155, 156, or 157 is used in an analysis, the post file contains only one integration point for each element. The element stresses and strains in the point are the averaged results over three integration points of the element. This is to reduce the size of the post file.

Transformation

Any local set (u, v, w) can be used at any node.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available with parameter ELASTICITY, 2 or parameter PLASTICITY, 5.

Volume C: Program Input

Chapter 2
Parameters

SUPER

Super Element Input

Description

This parameter allows for the input of stiffness and load terms generated by the SUBSTRUCTURE commands in a previous analysis. It can be used in a multilevel substructure definition in combination with the SUBSTRUC parameter or on the main level of a substructure analysis. If this parameter is used, no other elements are necessary; in which case, the ELEMENTS parameter is not used.

Format

Format		Data	
Fixed	Free	Type	Entry
1-5	1st	A	Enter the word SUPER.
11-15	2nd	I	Enter Fortran unit number for the data base containing the super element information.
16-20	3rd	I	Enter the maximum number of nodes in any super element.
21-25	4th	I	Enter the maximum number of degrees of freedom per node number in any super element.
26-30	5th	I	Enter the number of super elements in this analysis.
31-35	6th	I	= 0 Marc superelement created using SUBSTRUCTURE in previous Marc job. = 1 Super element comes from an external program and all relevant data, except the connectivity specified in SUPERINPUT model definition data is created using the USSUBS user subroutine. In this case, the second entry should be left blank.

NEW

Use New Format

Description

This parameter allows the input of data in the new Version K style format. Input is interpreted to be in this format until an OLD parameter is encountered. This parameter must not appear embedded inside a model definition or history definition option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word NEW.
11-15	2nd	I	Enter 1 if the default width of the data fields is used for input. This will override the EXTENDED parameter. Enter 2 if the double width of the data fields is used for input.

OLD

Use Old Format

Description

This parameter allows the input of data in the old (Rev. G, H, J) style format. Input is interpreted to be in this format until a NEW parameter is encountered. This parameter must not appear embedded inside a model definition or history definition option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word OLD.

PRINT

Debug Printout

Description

This parameter allows printout of various items for debugging; however, the amount of output is increased accordingly. Default is no check printout. Multiple print flags can be set using columns 11 to 80.

Format

Format		Data	
Fixed	Free	Type	Entry
1-5	1st	A	Enter the word PRINT.
11-80	2nd	14I5	Enter as many print codes as required. Enter 1 for output of element stiffness matrices (this also prints out the shell surface metric for doubly curved shells 4, 8 and 24), consistent mass matrix and equivalent nodal loads. **CAUTION:** This produces significant output. Enter 2 for output of the matrices used in tying. (See TYING, SERVO LINK, UFORMS.) Enter 3 to force the solution of a nonpositive definite matrix. This is only recommended for the AUTO INCREMENT option to pass collapse points in the collapse analysis. This can be entered on the CONTROL option. Enter 5 to obtain additional information concerning gap convergence. In coupled analysis, set to 5 to print internal heat generated. In contact analysis, set to 5 to obtain information concerning nodes touching or separating from surfaces and also to print out the maximum residual and reaction forces. Enter 6 to obtain output of nodal value array during rezoning.

Format Fixed	Format Free	Data Type	Entry
			Enter 7 to obtain tying information in CONRAD GAP option and fluid element numbers in CHANNEL option which is used to define fluid channel input data in heat transfer analysis. Enter 8 to obtain incremental displacements in local system in contact problems. Enter 9 to obtain latent heat output. Enter 10 to obtain the stress-strain relation in the local coordinate system. Enter 11 to obtain additional information on the interlaminar stress calculation. Enter 12 to output the right hand side and solution vector. **CAUTION:** This produces significant output. Enter 14 to obtain information regarding the mesh adapting process.

FILMS

Film Coefficients and Sink Temperatures

Description

This parameter allows for the input of the maximum number of elements that have films. This parameter is not needed if the number of elements with films is not increased in the history definition section.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the FILMS.
11-15	2nd	I	Maximum number of elements that have films.

Chapter 3
Model Definition Options

NEW

Use New Format

Description

This option allows the input of data in the K style format. Input is interpreted to be in this format until an OLD option is encountered. This option must not appear embedded inside a model definition or history definition option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word NEW.
11-15	2nd	I	Enter 1 if the default width of the data fields is used for input. This will override the EXTENDED parameter. Enter 2 if the double width of the data fields is used for input.

OLD

Use Old Format

Description

This option allows the input of data in the old (G, H, or J) style format. Input is interpreted to be in this format until a NEW option is encountered. This option must not appear embedded inside a model definition or history definition option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word OLD.

WRITE

Write Connectivity and Coordinates

Description

This option allows you to write the final connectivity and coordinates to an auxiliary file. The values written are those after all internal mesh generations (MESH2D, FXORD or incremental generators) and all external (UFXORD, UFCONN) transformations have been performed. All node numbers are in the user system; that is, nonoptimized.

Note: The connectivity and coordinates data are written to the auxiliary file in the format (default or extended) based on the last related valid option (see the EXTENDED parameter as well as the NEW (model definition and history definition option) and OLD (model definition and history definition option) before the END OPTION line.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word WRITE.
11-15	2nd	I	Enter the unit number to write to. Default is unit 1.

ADAPT GLOBAL

Define Meshing Parameters Used in Global Remeshing

Description

This model definition option provides parameters used for the global remeshing.

This feature is available only for 2-D problems.

Remeshing Criteria

It is possible to choose one, two, three, or four remeshing criterias simultaneously.

Note: In general, very frequent remeshing should be avoided for effective and computationally efficient analysis. Also, since every remeshing and subsequent rezoning step involves interpolation and extrapolation of element variables, a possibility of error accumulation exists as the analysis progresses when remeshing occurs too frequently.

Increment

Remeshing occurs at specified increment frequency.

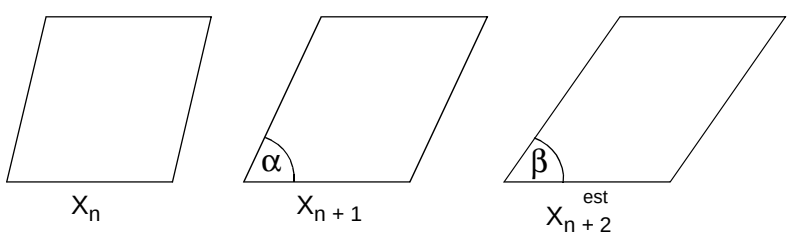
Element Distortion

The identified body is remeshed when the distortion in the elements becomes large, or it is anticipated that the distortion will become large.

The element distortion criteria is based upon examining the angles of the elements at the end of the increment and an estimate of the change in the angle in the next increment.

Given that X_n are the coordinates in the beginning of the step and that ΔU_n are the displacements in the increment, then:

$$X_{n+1} = X_n + \Delta U_n \text{ and } X_{n+2}^{est} = X_{n+1} + \Delta U_n$$



If $\cos \alpha > 0.8$ and $\cos \beta > 0.9$, remesh, or if $\cos \alpha > 0.9$ and $\cos \beta > \cos \alpha$, remesh

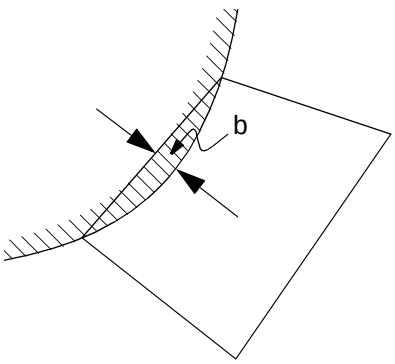
which is equivalent to

$0 < \alpha < 36°$ and $0 < \beta < 25°$ remesh or
$144° < \alpha < 180°$ and $155° < \beta < 180°$ remesh or
$0 < \alpha < 25°$ and $\beta < \alpha$ or
$155° < \alpha < 180°$ and $\beta > \alpha$

Contact Penetration

The identified body is remeshed when the curvature of the contact body is such that the current mesh cannot accurately detect penetration.

The penetration remeshing criteria is based upon examining the distance between the edge of the element and the representation of the other contact surface.



If b> penetration limit, remeshing is required. By default, penetration is 2 * (contact tolerance). The contact tolerance can be input by you or computed internally by the program.

Please note that this check does not apply to the self-contact situation.

Angle Deviation

The identified body remeshes when the angles in the element have a deviation from the ideal angle greater than a specified amount. The ideal angle for quadrilaterals or hexahedrals is 90°. The ideal angle for triangles and tetrahedrals is 60°. The default of 40° indicates that any angle in the range $50 \leq \alpha \leq 130$ is acceptable for quadrilaterals.

Immediate

The identified body is remeshed before performing any analysis.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-8	1st	A	Enter the words ADAPT GLOBAL.
2nd data block			
1-5	1st	I	Enter the number of bodies to be remeshed.
6-10	2nd	I	Enter 0 for default. Enter the unit number to read data; defaults to input.
11-15	3rd	I	Not used; enter 0.

The third, fourth, and fifth data blocks are repeated based on the number of remeshing bodies selected in the 2nd data block.

3rd data block			
1-5	1st	I	Enter 2 for Advancing Front 2-D quadrilateral or triangular mesher. Enter 3 for Delaunay 2-D triangular mesher Enter 4 for 2-D Overlay quadrilateral mesher. A valid mesher type (2-4) must be used.
6-10	2nd	I	Not used; enter 0.
11-15	3rd	I	Enter the contact body to be remeshed (default = 1) where the contact body number is defined under the CONTACT option.
4th data block			
1-5	1st	I	Enter 1 if increment frequency used as remesh criterion.
6-10	2nd	I	Enter 1 if element distortion used as remesh criterion.
11-15	3rd	I	Enter 1 if penetration is used as remesh criterion.
16-20	4th	I	If the first field is 1, enter the frequency in increments.
21-30	5th	E	Not used.
31-35	6th	I	Enter 1 if angle criterion is used.

Format Fixed	Format Free	Data Type	Entry
36-45	7th	E	If the 6th field = 1, enter maximum change in angle from the reference angle (90°) for quadrilaterals.
46-55	8th	E	If the 6th field = 1, enter the maximum change in angle from reference angle (60°) for triangles.
56-60	9th	I	Enter 1 to force remeshing before any analysis if no other data in this load case, or at the end of this increment if incremental load data given.
61-70	10th	E	Penetration limit; default = 2 * (contact tolerance).
5th data block (Two-dimensional Marc Advancing Front All Quadrilateral or Triangular Mesher) Mesher type = 2			
1-5	1st	I	Enter 0 for all quadrilateral mesh. Enter 2 for all triangular mesh.
6-15	2nd	E	Enter the target element size. Default means the target number of elements in the 4th field are used.
16-25	3rd	E	Not used; enter 0.
26-30	4th	I	Target number of elements after remeshing; default means no such control. If both the 2nd and 4th fields are default, the number of elements in the previous mesh are used.
31-35	5th	I	Curvature outline control. Enter number of divisions of line segments to fit a curvature circle; default = 36. Enter -1 to obtain uniform outline points.
36-45	6th	E	Outline smoothing ration range 0 - 1.0; default = 0.8.
46-55	7th	E	Minimum target element size; default = 1/3 * target element size.
56-65	8th	E	Percentage of change allowed for the new number of elements created. Default means no such control. Total of 5 remeshing trials will be used to create the mesh to meet the requirement. Not to be used for the remeshing with the automatic stop-and-restart option.

Format Fixed	Format Free	Data Type	Entry
5th data block (Two-dimensional Delaunay Triangular Mesher) Mesher type = 3			
1-5	1st	I	Not used; enter 0.
6-15	2nd	E	Enter the target element size.
16-25	3rd	E	Not used; enter 0.
26-30	4th	I	Target number of elements after remeshing; default means no such control.
31-35	5th	I	Curvature outline control. Enter number of divisions of line segments to fit a curvature circle. Default = 36. Enter -1 to obtain uniform outline points.
36-45	6th	E	Outline smoothing ratio (range 0-1.0); default = 0.8.
46-55	7th	E	Minimum target element size; default = 1/3 * target element size.
56-65	8th	E	Percentage of change allowed for the new number of elements created. Default means no such control. Totally 5 remeshing trials will be used to create the mesh to meet the requirement. Not to be used for the remeshing with the automatic stop-and-restart option.
5th data block (Two-dimensional Overlay Quadrilateral Mesher) Mesher type = 4			
1-10	1st	E	Enter the element target length.
11-15	2nd	I	Enter 1 if elements on the boundary in contact are to be refined one level if necessary. Enter 2 if elements on the boundary in contact are to be refined two levels if necessary.
16-20	3rd	I	Enter 1 if elements in the interior can be merged together. Four elements at a time will be merged.
21-25	4th	I	Target number of elements after remeshing; default means no such control.
26-30	5th	I	Not used; enter 0.
31-40	6th	E	Not used; enter 0.
41-50	7th	E	Not used, enter 0.
51-60	8th	E	Percentage of change allowed for the new number of elements created. Default means no such control. Totally 5 remeshing trials will be used to create the mesh to meet the requirement. Not to be used for the remeshing with the automatic stop-and-restart option.

GEOMETRY

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 defined variable (EGEOM1). The geometry for a particular element can be defined repeatedly and only the last data is used. This feature is designed for local variations of geometric data.

The GEOMETRY option is unnecessary if the element description does not require either EGEOM1, EGEOM2, or EGEOM3. (See *MSC.Marc Volume B: Element Library*).

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

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-8	1st	A	Enter the word GEOMETRY.
2nd data block			
1-5	1st	I	Number of distinct sets of element geometries to be input (optional).
6-10	2nd	I	Enter unit number for input of geometry defaults to input.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Element geometries. The 3rd and 4th data blocks 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 *MSC.Marc Volume B: Element Library* for the meaning of EGEOM1, etc. for each element type.
4th data block			
			Enter a list of elements to which the above geometry is applied.

Note: For elements 7, 10, 11, and 19, enter 1 in the EGEOM2 field to activate the constant dilatation option. This improves the behavior of the element for nearly incompressible analysis. See *MSC.Marc Volume B: Element Library* for further details. This is an alternative to the CONSTANT DILATATION parameter.

For elements 3, 7, and 11, enter 1 in the EGEOM3 field to activate the assumed strain formulation. This improves the element bending behavior. This is an alternative to the ASSUMED STRAIN parameter.

For elements 109 and 110, the penalty factor used to add the constraint for the vector potential (*MSC.Marc Volume A: Theory and User Information*) to the set of equations for magnetostatic calculations can be set in the EGEOM2 field.

CYCLIC SYMMETRY

Enter Data for a Cyclic Symmetric Structure

Description

This option is used to define data for a structure possessing cyclic symmetry, which means that the geometry and the loading vary periodically around a symmetry axis. This type of structure can be effectively analyzed by modeling only one section and applying the proper multipoint constraint equations to account for the cyclic symmetry. By defining the symmetry axis and the sector angle, the Marc program sets up the constraint equations automatically. Additionally, the rigid body rotation around the symmetry axis can be automatically suppressed.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words CYCLIC SYMMETRY.
2nd data block			
1-10	1st	E	First component of the direction cosine of symmetry axis.
11-20	2nd	E	Second component of the direction cosine of symmetry axis.
21-30	3rd	E	Third component of the direction cosine of symmetry axis.
3rd data block			
1-10	1st	E	X-coordinate of point on symmetry axis.
11-20	2nd	E	Y-coordinate of point on symmetry axis.
21-30	3rd	E	Z-coordinate of point on symmetry axis.
4th data block			
1-10	1st	E	Cyclic angle.

Format Fixed	Format Free	Data Type	Entry
5th data block			
1-5	1st	I	Enter: -1 To automatically suppress rigid body mode. 0 To have no suppression. >0 To supress at node number given.

CROSS-SECTION

Enter Data to Define Cross Sections

Description

This is option allows the definition of cross sections which can be used to simulate pre-stressed bolts or rivets. For each cross section, the option requires:

a. a special node associated with the cross section,

b. the normal to the cross section,

c. the nodes lying on the cross section and the elements connected to these nodes and lying on the side of the cross section corresponding to the negative normal direction.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word CROSS-SECTION.
2nd data block			
1-5	1st	I	Number of cross sections to be defined (optional).
6-10	2nd	I	Unit number for input of cross sections. Defaults to input.
3rd data block			
1-5	1st	I	Cross section number.
5-10	2nd	I	Additional cross section node.
11-15	3rd	I	Enter a 2 to enter the list of nodes and elements given below.
4th data block			
			No data required.

Format Fixed	Format Free	Data Type	Entry
5th data block			
1-10	1st	F	First component of normal to cross section.
11-20	2nd	F	Second component of normal to cross section.
21-30	3rd	F	Third component of normal to cross section.
6th data block			
			Enter the list of nodes in the cross section.
7th data block			
			Enter the list of elements.

SPRINGS

Input Simple Linear Spring (Dashpot)

Description

This data set is used to input any simple linear springs. For dynamic analysis, a dashpot capability is offered as well.

The force in the spring/dashpot is:

$$F = K(u_2 - u_1) + C(\dot{u}_2 - \dot{u}_1)$$

where K is the spring stiffness, C is the damping coefficient, u_2 is the displacement of the degree of freedom at the second end of the spring (third and fourth fields), and u_1 is the displacement of the degree of freedom at the first end of the spring (first and second fields).

For a nonlinear spring/dashpot, the force can be specified with the user subroutine USPRNG with the general relation:

$$F = F(u_2 - u_1, \dot{u}_2 - \dot{u}_1)$$

See *MSC.Marc Volume D: User Subroutines and Special Routines* for details.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-7	1st	A	Enter the word SPRINGS.
2nd data block			
Enter one data block per spring/dashpot.			
1-5	1st	I	Node to which first end of spring/dashpot will be attached.
6-10	2nd	I	Degree of freedom at above node to which spring/dashpot will be attached. Enter 0 for a spring/dashpot acting in the direction from the first to the second node (this also requires a 0 on the 4th field).

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Node to which other end of spring/dashpot will be attached. Enter 0 if the spring goes from the first end to the ground.
16-20	4th	I	Degree of freedom at above node to which spring will be attached. Enter 0 if the spring goes from the first end to the ground. Enter 0 for a spring/dashpot acting in the direction from the first to the second node (this also requires a 0 on the 2nd field).
21-30	5th	F	Stiffness of spring.
31-40	6th	F	Damping coefficient of dashpot (for dynamic analysis only).
41-50	7th	F	Initial force in spring.

TYING

Define Tying Constraints

Description

This option is used to define homogeneous constraints. Constraints are defined by specifying a tied node and one or more associated retained nodes. Further details are provided in *MSC.Marc Volume A: User Information*. Special types of tying can be obtained using user subroutine UFORMS (see *MSC.Marc Volume D: User Subroutines and Special Routines*).

A rigid link for either small deformation or large deformation can be implemented by using tying type 80.

To obtain tying constraint based on updated current coordinates, add 1000 to tying type code. For tying type associated with user derived tying (UFORMS), subtract 1000 to tying type code.

In a coupled thermal-mechanical analysis during the heat transfer subincrements, tying type 1 is used for all tying types except 31, 32, 33, and 34.

If TYING CHANGE is subsequently used to increase the number of constraints, the TIE parameter must also be included.

A tying constraint always consists of a tied node (removed from the system) and one or more retained nodes (which remain in the system). Each tying constraint is specified by a series of two data blocks (data blocks 3 and 3a).

If a sequence of similar tying types must be specified, a list of nodes for tied nodes (3b) and corresponding retained nodes (3c - 3d) must be given.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-5	1st	A	Enter the word TYING.
2nd data block			
1-5	1st	I	Number of constraint equations to be read (optional).
6-10	2nd	I	Unit number for input of tying data. Defaults to input.
11-15	3rd	I	Enter 1 to suppress printout of tying data.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Data blocks 3 and 3a or 3 and 3b, 3c, 3d, are given once for each constraint equation set.			
1-5	1st	I	Enter the code for tying type. SeeTable 3-2 and *MSC.Marc Volume A: User Information* for definition of default types and user-defined routines.
6-10	2nd	I	Enter 0 to indicate a list of nodes to be tied will be defined in data block 3b. Enter a node number to indicate an individual node to be tied.
11-15	3rd	I	Number of retained nodes for this tying type. If a standard Marc tying type is used this does not need to be entered.
(3a) data block			
If the number of a node is entered in the second field of the 3rd data block (above), use data block 3a to list the retained nodes.			
1-80	1st	I	Sequence of retained nodes for constraint in (16I5) format, etc.
(3b) data block			
If no tied node is entered in the second field of the 3rd data block (that is, 0 is entered), use data blocks 3b, 3c, and 3d to enter a list of nodes to be tied.			
			Enter an unsorted list of nodes to be tied.
(3c) data block			
			Enter an unsorted list of nodes which will be the first retained nodes associated with tied nodes given in data block 3b.
(3d) data block			
			Same as 3c except second retained nodes, etc. **Note:** List verbs EXCEPT, INTERSECT and sorted node sets are illegal in these lists.

Table 3-2 Summary of Standard Tying Types

Tying Code	Number of Retained Nodes	Purpose	Remarks
I≤NDEG	1	Tie the Ith degree of freedom at the tied node to the Ith degree of freedom at the retained node	
100	1	Tie all degrees of freedom at the tied node to the corresponding degrees of freedom at the retained node	
23	1	Tie axisymmetric solid node to axisymmetric-shell (element type 1) node	Both tied and retained nodes must be transformed to local system. TRANSFORMATION option must be invoked. (See *MSC.Marc Volume A: User Information*, Table 9-17)
15	Number of retained nodes is 1 less than the number of shell nodes in the z-r plane of the section	Special tying types for pipe bend element 17 to remove rigid body modes (see *Volume B: Element Library*)	
16	Number of shell nodes in the z-r plane of the section	Special tying types for pipe bend element 17 to remove rigid body modes (see *Volume B: Element Library*)	
17	2	Special tying types for pipe bend element 17 to couple bend section into pipe line (see *Volume B: Element Library*)	
18	2	Joining together the boundaries of intersecting shell, element type 4, 8 or 24. Fully moment carrying joint.	Tied node is also second retained node. Neither node can be transformed (see *MSC.Marc Volume A: User Information*, Table 9-15)
28	2	Joining intersecting shells, element type 4, 8, or 24. Pinned joint.	Tied node is also second retained node (see *MSC.Marc Volume A: User Information*, Table 9-15)
19	2	Use beam element 13 as a stiffener on shell elements 4 or 8. Tied node is beam node: First retained node is shell node, second is beam node again. Beam node should be on, or close to, the normal to the shell at the shell node.	

Table 3-2 Summary of Standard Tying Types (Continued)

Tying Code	Number of Retained Nodes	Purpose	Remarks
20	3	Create an extra node in a shell type 8 element tied to the interpolation function of the shell. Use in conjunction with tying type 21 to tie a beam element 13 or a stiffener across a shell element.	Always use after tying type 21.
21	2	Same as type 19, but tying beam to an interpolated shell node not as a vertex of an element – element type 8 only. Must be followed by type 20 to tie the interpolated shell node into the shell mesh.	Must be followed by a tying type 20.
24	2	Join intersecting shells or beams, element type 15-17.	Tied node is also second retained node. Neither node can be transformed. Tying is necessary only when there is a large angle between the two plates.
25	2	Join solid mesh to shell or beam (type 15 or 16).	Similar to 23, but no transformation needed. Tied node is also second retained node.
26	2	Join solid mesh to axisymmetric shell (type 1 or 89).	Similar to 5. Tied node is also second to retained node.
27	2	Join Fourier solid to Fourier shell (type 90).	Tied node is also second retained node.
31	2	Refine mesh of first order (linear displacement) elements in 2-D.	Tie interior nodes on refined side to corner nodes on coarse side.
32	2	Refine mesh of second order (quadratic displacement) elements in 2-D.	Tie interior nodes on refined side to the edge of an element on the coarse side.
33	4	Refine mesh of 8-node bricks	Tie interior node on the refined side to the 4 corner nodes of an element face on the coarse side.
34	8	Refine mesh of 20-node bricks	Tie interior nodes on refined side to the 8 (4 corner, 4 midside) nodes of an element on the coarse side.
52	1	Pin joint for beam types 14, 25, or 52.	
53	1	Fully moment carrying joint for beam types 14, 25, or 52.	
13	2	Joining two elements type 13 under an arbitrary angle. Fully moment carrying joint.	Tied node is also the second retained node.

Table 3-2 Summary of Standard Tying Types (Continued)

Tying Code	Number of Retained Nodes	Purpose	Remarks
>100	1	Generate several tyings of type ≤ NDEG.	Tying code is first d.o.f. multiplied by 100 added to last d.o.f.; that is, 209 means tie 2nd to 9th d.o.f. at tied node to resp. 2nd and 9th d.o.f. at retained node.
WARNING: TRANSFORMATION MUST NOT BE USED AT NODES INVOLVED IN TYING TYPES 13, 18, 19, 20, 21, 22, 24, 25, 49, 50, 51, 52, OR 53.			
80	2	Form a rigid link between tied node and retained node. This works for either small or large deformation. If a rigid region is to be modeled, use multiple ties of type 80, with the tied node of each link being a different node, and use the same common node as the retained node.	The second tied node is an extra node not connected to the structure which contains the rigid body rotation.
85	2	Tying of temperatures between shell and solid elements in heat transfer analysis (linear/quadratic/new composite temperature distribution in the thickness direction of shell elements).	Tied node is the shell node and two retained nodes are nodes of the solid element. Order of the retained nodes follows the shell node degrees of freedom. The assumption here is that the shell and brick have the same thickness.
86	3	Tying of temperatures between shell and solid element in heat transfer analysis (quadratic/new composite temperature distribution in the thickness direction of shell element).	Tied node is the shell node and three retained nodes are nodes of the solid element. Order of the retained nodes follows shell node degrees of freedom. Tied node should not have linear temperature distribution. The assumption here is that the shell and brick have the same thickness.
87	1	Tying of temperatures between two shell elements in a heat transfer analysis (linear/quadratic/new composite temperature distribution).	Tied and retained nodes are shell nodes. The tied node should have more or equal number of degree of freedom than the retained node. The assumption here is that the tied shell and retained shell have equal thickness.

SOLVER

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 Marc, based upon the amount of workspace required and the number given on the SIZING parameter. You can also select whether a symmetric or nonsymmetric solver is used. Additionally, you can specify if the solution of a nonpositive 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.

You control the maximum number of iterations allowed. If this is a positive number, Marc stops if this is exceeded. If this is a negative number, Marc prints a warning and continues to the next Newton-Raphson iteration or increment.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word SOLVER.
2nd data block			
1-5	1st	I	0 = Profile Direct Solver. 2 = Sparse Iterative. 4 = Sparse Direct Solver 6 = Hardware Provided Direct Sparse Solver 8 = Multifrontal Direct Sparse Solver.
6-10	2nd	I	Enter 1 for solving a nonsymmetric system. Only available for solvers type 0 and 8.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Enter 1 if the solution of nonpositive definite system is to be obtained.
31-35	4th - 7th	I	Not used.
36-40	8th	I	Enter, in millions, the number of four-byte words to be used by solver type 6 or 8 before going out-of-core. Default is the same behavior as for other solvers. For solver type 6, this option is only available on SGI. For solver type 8, it is available on all platforms.

3rd data block

Only necessary if the sparse iterative solver is used.

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

4th data block

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

Create File for Postprocessing

Description

This option creates a postprocessor file for time-history or variable versus variable plots using Mentat or your own postprocessing. In the latter case, the file is accessed via the utility PLDUMP given in *MSC.Marc Volume D: User Subroutines and Special Routines*.

You have two possibilities for the post file in association with restarted runs:

A. If the POST option follows the RESTART option, Marc first copies the previous post file onto the new post file (up to the restart increment), thus providing a continuous post file from the beginning of the analysis. The old post file is closed after it has been read. It is important that the POST option of the restart job requests the same post variables to be written to the post file as requested in the previous data file. Otherwise, loss of data or I/O errors can occur.

B. If the POST option precedes the RESTART option, the new post file contains only those increments analyzed in the current run.

One or the other options should be chosen – if (B) is used, a continuous post file is not created, so that (A) cannot subsequently be used for this analysis unless you combines the files with your own program.

Note: In a modal or buckling analysis in addition to POST option, the history definition option RECOVER must be used for storing eigenvectors on post file.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-4	1st	A	Enter the word POST.
2nd data block			
1-5	1st	I	Number of element variables to be written on the file (at each integration point if the ALL POINTS parameter is present; otherwise, at the centroid of each element) at each increment (optional).

Format Fixed	Format Free	Data Type	Entry
6-10	2nd	I	Unit number on which to open the new binary post file (`jid.t16`). Defaults to unit 16 if left blank. See Table B-1 in Appendix B.
11-15	3rd	I	Unit number on which to open the previous binary post file (`rid.t16`), for a restarted run. Defaults to unit 17 if left blank. Note that all data from this file (up to the restart increment) is copied to the new file upon restart, so that the post file is continuous from the start of the analysis, provided the POST option follows the RESTART option.
16-20	4th	I	Set to 0 for binary post file. Set to 1 for formatted post file. Set to 2 for both binary and formatted post file.
21-25	5th	I	**For Cray Only:** Set to 0 for internal binary. Set to 1 for IEEE binary compatible file.
26-30	6th	I	Unit number on which to open the new formatted post file (`jid.t19`). Defaults to unit 19. See Tablle B-1in Appendix B.
31-35	7th	I	Unit number on which to open the previous formatted post file, for a restart run. Defaults to unit 20. See Table B-1 in Appendix B.
36-40	8th	I	Set to 1 to convert restart file to post file with no analysis. Increments to be converted are given in the third and ninth field of RESTART model definition section.
41-45	9th	I	Number of increments between writing of post data. Defaults to write post file every increment.
46-50	10th	I	Number of user-defined post vectors. Vector is defined in user subroutine UPOSTV. This field is only used for 7- and 8-style post files.

Format Fixed	Format Free	Data Type	Entry
51-55	11th	I	Enter 1 to generate Marc K2 style post file. Enter 3 to generate Marc K3 style post file. Enter 4 to generate Marc K4 style post file. Enter 5 to generate Marc K5 style post file. Enter 6 to generate Marc K6 style post file. Enter 7 to generate Marc K7 style post file. Enter 8 to generate Marc 8 (not released) style post file. Enter 9 to generate Marc 2000 style post file. Enter 10 to generate Marc 2001 style post file (default).
56-60	12th	I	This field can be used for debugging purposes in a stress analysis: Enter 0 if no iterative data is needed. Enter 1 to get the iterative displacements. Enter 2 to get the iterative displacements and reaction/residual forces. Enter 3 to get the iterative displacements, reaction/residual forces and the touched bodies in a contact analysis. The iterative data is written as subincremental data. **Note:** The use of this option can generate a huge post file since the post data is written for every iteration.
61-65	13th	I	Only for Marc 2000 and higher style post file. Enter the total number of nodal post codes (including user-defined nodal post codes). Enter 0 for a default set of nodal data written on the post file, depending on the analysis type. Enter -1 for no nodal data written. **Note:** Displacements are not automatically written in the custom post file unless explicitly chosen. Besides the chosen quantities, if the deformation also needs to be visualized then the displacements also need to be chosen as nodal quantities.
66-70	14th	I	Not used.

Format Fixed	Format Free	Data Type	Entry

3rd data block

This data block is used for input of variables to be written on the post file. For 8- and lower style post files, only element data can be selected and the nodal data is written by default. For 9- and higher style post files, both element and nodal data can be selected. This data block is repeated for all selected element variables, and, for 9- and higher style post files, all selected nodal variables.

For Element Data

1-5	1st	I	Enter an element post code. The code numbers are described in Table 3-3.
6-10	2nd	I	Enter the layer number for shell elements or continuum composite elements. For post codes 471 and 481, enter the global identification number of the rebar layer
11-35	3rd	A	Enter a 24-character label associated with this post code for use in postprocessing.

For Nodal Data (only 9- and higher style post file):

1-5	1st	A	Enter the word NODAL.
6-10	2nd	I	Enter a nodal post code. The code numbers are described in Table 3-4.
11-59	3rd	A	Enter a 48-character label associated with this post code for use in postprocessing.

Table 3-3 Element Post Codes

Codes	Description
1-6	Components of strain. For rigid-perfectly plastic flow problems, components of strain rate
7	Equivalent plastic strain (integral of equivalent plastic strain rate). For rigid-perfectly plastic flow problems, equivalent plastic strain rate
8	Equivalent creep strain (integral of equivalent creep strain rate)
9	Total temperature
10	Increment of temperature
11-16	Components of stress

Table 3-3 Element Post Codes (Continued)

Codes	Description
17	Equivalent Mises stress
18	Mean normal stress (tensile positive) for Mohr-Coulomb
19	User-defined variable via user subroutine PLOTV. See *MSC.Marc Volume D: User Subroutines and Special Routines*.
20	Thickness of element
21-26	Components of total plastic strain
27	Equivalent plastic strain. $\bar{\varepsilon}^{p} = \sqrt{\frac{2}{3}\Sigma\Delta\varepsilon_{ij}^{p}\Sigma\Delta\varepsilon_{ij}^{p}}$
28	Plastic strain rate
29	Total value of second state variable
31-36	Components of creep strain
37	Equivalent creep strain. $\bar{\varepsilon}^{c} = \sqrt{\frac{2}{3}\Sigma\Delta\varepsilon_{ij}^{c}\Sigma\Delta\varepsilon_{ij}^{c}}$
38	Total swelling strain (from user subroutine VSWELL)
39	Total value of third state variable
41-46	Components of Cauchy stress
47	Equivalent Cauchy stress
48	Strain energy density
49	Thickness strain for plane stress: Mooney or Ogden material
51-56	Real components of harmonic stress
57	Equivalent real harmonic stress
58	Elastic strain energy density
59	Equivalent stress/yield stress
60	Equivalent stress/yield stress (at current temperatures)
61-66	Imaginary components of harmonic stress
67	Equivalent imaginary harmonic stress
68	Plastic strain energy density
69	Current volume
71-76	Components of thermal strain

Table 3-3 Element Post Codes (Continued)

Codes	Description
78	Original volume
79	Grain size
81-86	Components of cracking strain (only for stress analysis)
91-103	Failure indices
108-109	Interlaminar shear for thick composite elements (TSHEAR parameter must be present)
111-116	Components of stress in preferred coordinate system defined by ORIENTATION option
121-126	Elastic strain
127	Equivalent elastic strain
175	Equivalent viscoplastic strain rate (powder material)
176	Relative density (powder material)
177	Void volume fraction (damage model)
<0	User-defined variable via user subroutine PLOTV. See *MSC.Marc Volume D: User Subroutines and Special Routines*.
241	Gasket Pressure.
242	Gasket Closure.
243	Plastic Gasket Closure.
251	Global components of Interlaminar normal stress; layer n is between n and n+1
254	Global components of Interlaminar shear stress; layer n is between n and n+1
261	Beam axis (required if beam moment plots are created with Mentat)
264	Axial Force
265	Moment M_{xx}
266	Moment M_{yy}
267	Shear Force V_{xz}
268	Shear Force V_{yz}
269	Torque
270	Bimoment
301	Total strains tensor
311	Stress tensor

Table 3-3 Element Post Codes (Continued)

Codes	Description
321	Plastic strain tensor
331	Creep strain tensor
341	Cauchy stress tensor
351	Real harmonic stress tensor
361	Imaginary harmonic stress tensor
371	Thermal strain tensor
381	Cracking strain tensor
391	Stresses in preferred direction tensor
401	Elastic strain tensor
411	Stress in global coordinate system tensor
421	Elastic strain in global coordinate system tensor
431	Plastic strain in global coordinate system tensor
441	Creep strain in global coordinate system tensor
451	Velocity strains (for fluids)
461	Elastic strain in preferred direction tensor
471	Global components of the rebar stresses in the undeformed configuration (Second Piola-Kirchhoff). See *MSC.Marc Volume B: Element Library* for details.
481	Global components of the rebar stress in the deformed configuration (Cauchy). See *MSC.Marc Volume B: Element Library* for details.
501	Interlaminar normal stress; layer n is between n and n+1. See *MSC.Marc Volume B: Element Library* for details.
511	Interlaminar shear stress; layer n is between n and n+1. See *MSC.Marc Volume B: Element Library* for details.
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

Table 3-3 Element Post Codes (Continued)

Codes	Description	
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)
134-136	Components of electric displacement	(D)
137-139	Components of Lorentz force	
141-143	Components of magnetic induction	(B)
144-146	Components of magnetic field intensity	(H)
147-149	Components of current density	(J)
Post Codes for Harmonic Electromagnetic Analysis		
131-133	Real components of electric field 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)

Table 3-3 Element Post Codes (Continued)

Codes	Description
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

Notes: 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 you do not need to select nodal values (that is, 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.

For post codes 411, 421, 431, and 441 global quantities for shell elements are reported for as many layers and use the same layer numbering system as regular shell quantities. Layer 1 is the top surface; layer 2 is the next surface, etc. This convention is followed from Marc 2000 on.

Table 3-4 Nodal Post Codes

Code	Description
1	Displacement
2	Rotation
3	External Force
4	External Moment
5	Reaction Force
6	Reaction Moment
7	Fluid Velocity
8	Fluid Pressure
9	External Fluid Force
10	Reaction Fluid Force
11	Sound Pressure
12	External Sound Source
13	Reaction Sound Source
14	Temperature
15	External Heat Flux
16	Reaction Heat Flux
17	Electric Potential
18	External Electric Charge
19	Reaction Electric Charge
20	Magnetic Potential
21	External Electric Current
22	Reaction Electric Current
23	Pore Pressure
24	External Mass Flux
25	Reaction Mass Flux
26	Bearing Pressure
27	Bearing Force
28	Velocity
29	Rotational Velocity

Table 3-4 Nodal Post Codes

Code	Description
30	Acceleration
31	Rotational Acceleration
32	Modal Mass
33	Rotational Modal Mass
34	Contact Normal Stress
35	Contact Normal Force
36	Contact Friction Stress
37	Contact Friction Force
38	Contact Status
39	Contact Touched Body
40	Herrmann Variable
<0	User-defined nodal quantity via user subroutine UPSTNO.

INITIAL PLASTIC STRAIN

Define Initial Plastic Strain

Description

This option provides various ways of initializing the equivalent plastic strain throughout the model. Occasionally, in metal forming analysis, it is required to define the previous amount of equivalent plastic strain. This history dependent variable represents the amount of plastic deformation that the model was subjected to, and is used in the work (strain) hardening model.

Four ways of specifying the initial equivalent plastic strain values are shown below:

- Read the range of elements, integration points and layers and a corresponding value.
- Read the initial values through user subroutine INITPL.
- Read the initial values from a step of the post output file from a previous analysis with Marc. With this option, Marc assumes direct correspondence of the post file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by you.
- Read a list of elements, integration points and layers and a corresponding value.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-13	1st	A	Enter the words INITIAL PLASTIC STRAIN.

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-5	1st	I	Not used, enter 0.
6-10	2nd	I	Enter 1 to initialize the equivalent plastic strain via the 3rd and 4th data blocks below. In this case, the third field must also be defined. Enter 2 to initialize the equivalent plastic strain via user subroutine INITPL. This subroutine is now called in a loop over all elements in the mesh. Enter 3 to read the initial values of the equivalent plastic strain from the post file written by a previous analysis. In this case, the fourth and fifth field must also be defined. Enter 4 to initialize the equivalent plastic strain via the 5th, 6th, 7th, and 8th data blocks shown below. See also the third field on this block.
11-15	3rd	I	Only nonzero if the second field is set to 1 or 4. Then this entry gives the number of pairs of data blocks in series 3 and 4 or in series 5, 6, 7 8 used to input the equivalent plastic strain. Defaults to 1.
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 run is to be read.
21-25	5th	I	Only used if the second field is set to 3. In that case, this entry defines the step number of the previous analysis.
26-30	6th		Not used, enter 0.
31-35	7th	I	Set to 1 if option 3 is used, and a formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file
36-40	8th	I	Only nonzero if the second field is set to 2. Set to 1 to suppress printout of equivalent plastic strains values that are initialized in INITPL.

Format Fixed	Format Free	Data Type	Entry

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of sets is equal to the number given in the third field above.

3rd data block

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with this value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value.
21-25	5th	I	First layer of cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value can only be bigger than 1 for beam or shell elements.

4th data block

1-10	1st	F	Initial value of the equivalent plastic strain for the above range of points.

Data blocks 5, 6, 7, and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	Initial value of the equivalent plastic strain at zeroth increment.

6th data block

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

7th data block

This data block is not necessary if the CENTROID parameter is used.

Enter a list of integration points to which the above value is applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above value is applied.

INITIAL STATE

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. 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 can 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 Marc. This technique is most common for thermal stress analysis to initialize temperature (the first state variable at any point). With this option Marc assumes direct correspondence of the post file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by you.
- 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 a coupled analysis, the temperatures are not independent state variables and the INITIAL TEMP option must be used.

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words INITIAL STATE.
2nd data block			
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 must be included.
6-10	2nd	I	Enter 1 to initialize the state variable via the 3rd and 4th data block below. See also the third field on this data block. Enter 2 to initialize the state variable via user subroutine INITSV. This subroutine is now called in a loop over all elements in the mesh. Enter 3 to read the file values of the state variable from the post file written by a previous heat transfer analysis. In this case, the fourth and fifth fields must also be defined. Enter 4 to initialize the state variable via the 5th, 6th, 7th, and 8th data blocks as given below. Also, see the third field on this data block.
11-15	3rd	I	Only nonzero if the second field is set to 1 or 4. Then this entry gives the number of pairs of data blocks 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 file and to unit 25 for a binary post file.
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 0.

Format Fixed	Format Free	Data Type	Entry
31-35	7th	I	Set to 1 if option 3 is used, and a formatted post file is used.
			For Cray Only:
			Set to 2 for IEEE binary file
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).

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of sets is equal to the number given in the third field above.

3rd data block

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with the value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value.
21-25	5th	I	First layer of cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value can only be bigger than 1 for beam or shell elements.

4th data block

1-10	1st	F	Initial value of this state variable for the above range of points.

Data blocks 5, 6, 7 and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	New total value of this state variable for the points given below at the start of the zeroth increment.

6th data block

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

Format		Data	
Fixed	Free	Type	Entry

7th data block

This data block is not necessary if the CENTROID parameter is used.

Enter a list of integration points to which the above state variable is applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above state variable is applied.

CONTROL

Control Option for Stress Analysis

Description

This option allows you to input parameters governing the convergence and the accuracy for nonlinear analysis. For heat transfer analysis, see Heat Transfer Analysis on page 439.

For coupled thermal-stress analysis, the 4th data block must be used in addition to the 3rd data block.

For nonlinear static analysis, the controls are described in MSC.Marc Volume A: Theory and User Information. They do not appear on the restart file, and so must be re-entered on a restart run.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
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 an ELASTIC parameter 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, Marc does a maximum of the absolute value entered. If convergence has not been obtained, a warning is given and Marc proceeds to the next increment. This is not recommended.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Minimum number of recycles during an increment for plasticity or other tangent modulus nonlinearities. Default is 0. **Note:** This data field forces this number of recycles to take place at all subsequent increments. **CAUTION:** This value is overwritten by the PROPORTIONAL INCREMENT option.
16-20	4th	I	Flag for convergence testing. 0 or left blank Convergence is achieved when residuals satisfy the criteria. 1 Convergence is achieved when displacements satisfy the criteria. 2 Convergence is achieved when strain energy satisfies the criteria. 4 Convergence is achieved when either residual or displacement satisfies the criterion. 5 Convergence is achieved when both residual and displacement satisfies the criterion. **Notes:** Testing on relative displacements or strain energy always requires at least one iteration. If nonlinear analysis is done with the CENTROID parameter, the residuals are not calculated and testing is always done on displacements. Nonlinear analysis with the CENTROID parameter is not recommended. If the fields are set as 0, 1, or 2, only the 3rd data block is needed. If the fields are set as 4 or 5, the 3a data block is also needed. In this case, the 3rd data block is set for residual testing and 3a data block is set for displacements check only.
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 in which case absolute tolerances testing is used.

Format Fixed	Format Free	Data Type	Entry
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 *MSC.Marc Volume A: Theory and User Information*). 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	No longer used; enter 0.
41-45	9th	I	To print convergence control messages to log file, enter 1.
46-50	10th	I	Control on initial stress stiffness. 0 Normal-full contribution. 1 For Mooney material, reduce contribution of hydrostatic pressure on initial stress stiffness according to: $\sigma^{initial} = \sigma - f_r \cdot p \cdot I$ where $\sigma^{initial}$ is the stress tensor used in the initial stress stiffness matrix, σ is the current stress tensor, f_r is entered through the PARAMETERS option, p is the hydrostatic pressure and I is a unit tensor. 2 No initial stress stiffness. 3 Use stress at beginning of increment, not last iteration. 4 Results in the inclusion of only the positive stresses in the initial stress stiffness during the equilibrium iteration. Besides faster convergence, this leads to a stable analysis of very thin shell structures.

Format Fixed	Format Free	Data Type	Entry
51-55	11th	I	Control parameter: 0 Do not allow switching of convergence testing between residuals and displacements. 1 Allow switching of convergence testing between residual and displacements if reaction forces or displacements become extremely small. **Note:** Set this parameter as 0 if any kind of absolute value testing has been set as convergence tolerance.
3rd data block			
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 is bypassed or absolute testing is performed. If relative displacement checking: Minimum displacement, if displacement increment is less than this value, checking is bypassed or absolute testing is 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 is bypassed or absolute testing is performed. If relative displacement checking: Minimum rotation: if rotation increment is less than this value, checking is bypassed.
41-50	5th	F	If absolute residual testing: Maximum value of residual force. Default is 0.0 in which case, no checking on residual force takes place. If absolute displacement tasking Maximum value of displacement increment. Default is 0.0; in which case, no checking on displacements takes place.
51-60	6th	F	If absolute residual testing: Maximum value of residual moment. Default is 0.0 in which case, no checking on residual moments takes place. If absolute displacement tasking: Maximum value of rotation increment. Default is 0.0; in which case, no checking on rotations takes place.

3a data block

Necessary only if the 4th field of 2nd data block is set to 4 or 5.

1-10	1	F	Maximum allowable value of the change in displacement increment divided by the displacement increment. Default is 0.10.
11-20	2nd	F	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	Minimum displacement, if displacement increment is less than this value, checking is bypassed or absolute testing is performed.
31-40	4th	F	Minimum rotation: if rotation increment is less than this value, checking is bypassed.

Format Fixed	Format Free	Data Type	Entry
41-50	5th	F	Maximum value of displacement increment. Default is 0.0; in which case, no checking on displacements takes place.
51-60	6th	F	Maximum value of rotation increment. Default is 0.0; in which case, no checking on rotations takes place.

4th data block

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 (for example, latent heat, radiation boundary conditions). Default is 0, which bypasses this test. Set to maximum temperature error which is considered acceptable.

PARAMETERS

Definition of Parameters used in Numerical Analysis

Description

There are many parameters that are used in the finite element calculations. These parameters can be customized for your particular application. Some of these constants can be entered in other input blocks as well. The last nonzero value is used for the calculation.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word PARAMETERS.
2nd data block			
1-10	1st	E	Enter the scale factor which, when multiplied with the incremental strain, is used to predict the incremental strain in the next increment. Default is 1.0.
11-20	2nd	E	Enter the multiplier used to calculate the penalty used to impose boundary conditions. Default is 1.e9. The penalty used is, hence, 1.e9 times the maximum diagonal value of the stiffness matrix. If the APPBC parameter is used, this option is not used.
21-30	3rd	E	Enter the penalty factor used to satisfy incompressibility in rigid plastic analysis for plane strain, axisymmetric, or solid analysis when displacement elements are used. Default is 100.
31-40	4th	E	Enter the penalty factor used to satisfy incompressibility in fluid analysis when displacement elements are used. Default is 1.e6.

Format Fixed	Format Free	Data Type	Entry
41-50	5th	E	Beta parameter used in transient dynamic analysis using Newmark-beta procedure Default is 0.25.
51-60	6th	E	Gamma parameter used in transient dynamic analysis using Newmark-beta procedure. Default is 0.50.
61-70	7th	E	Gamma1 parameter used in transient dynamic analysis using Single Step Houbolt procedure. Default is 1.5.
71-80	8th	E	Gamma parameter used in transient dynamic analysis using Single Step Houbolt procedure. Default is -0.5.
3rd data block			
1-10	1st	E	Enter the angle at which a node separates from a convex corner or becomes stuck in a concave corner for two-dimensional contact. Default is 8.625°.
11-20	2nd	E	Enter the angle at which a node separates from a convex corner or becomes stuck in a concave corner for three-dimensional contact. Default is 20.0°.
21-30	3rd	E	Enter the initial strain rate for rigid plastic analysis. Default is 1.e-4.
31-40	4th	E	Enter the cutoff strain rate for rigid plastic analysis. Default is 1.e-12.
41-50	5th	E	Enter the fraction of the mean stress that is subtracted from the stress tensor in the initial stress calculation. See the tenth field of the CONTROL option. Default is 0.0

Format Fixed	Format Free	Data Type	Entry
51-60	6th	E	Enter the factor used to calculate the drilling mode for shell elements type 22, 75, 138, 139, and 140. Default is 0.0001.
61-70	7th	E	Enter the scale factor to the incremental displacements for the increment after the rezoning increment. A value of 1 improves friction convergence, but may result in an inside-out element. Default is 1.0.

4th data block (Optional)

1-10	1st	E	Universal gas constant (R). Default is 8.314 J $mol^{-1}K^{-1}$.
11-20	2nd	E	Offset temperature between user units and absolute temperature. Default is 273.15°; that is, user input in Centigrade. If user temperature is in Kelvin (K) on Rankine (R), enter a negative value. The offset temperature is then set to zero.
21-30	3rd	E	Thermal Properties Evaluation Weight. Default is 0.5
31-40	4th	E	Surface projection factor for single step Houbolt. Default is 0.0
41-50	5th	E	Stefan Boltzman Constant. Default is 5.67051e-8W/m^2K^4.

AXITO3D

Transfer Data from Axisymmetric Analysis to 3-D Analysis

Description

In many cases, it is possible to begin the numerical simulations as a two-dimensional axisymmetric problem even through the final problem is fully three-dimensional. This is advantageous because of the large computational savings. For this to be useful, the first stage of the problem should be truly axisymmetric. The second stage of the problem can be fully three-dimensional. The AXITO3D option is used in the input file of the 3-D problem to import axisymmetric results from one analysis into a three-dimensional analysis. The data from the axisymmetric analysis is stored on the post file; hence, the appropriate data must have been previously written on the post file. This option can be used for most quadrilateral axisymmetric continuum elements. The following table shows the correlation between element types.

Axisymmetric	3-D Solid
10	7
20	7
82	84
83	84
28	21
67	21
33	35
66	35
55	57
59	61
48	23
142	23
144	146
145	146

This option cannot be used for beams, composites, and triangular elements.

There are three steps required to perform these types of analysis using Mentat.

1. Perform axisymmetric analysis:
   a. Create axisymmetric model using Mentat.
   b. Save the created model; for example, into `stage_axi.mud`.
   c. Run axisymmetric model and keep post file (`stage_axi_job1.t16` or `stage_axi_job1.t19`).

Note: Make certain that the post file contains the data to be transferred. For example, for metal plasticity using PLASTICITY,3, this would include at least stress (post code 311) and equivalent plastic strain (post code 7). For rubber elasticity using ELASTICITY,2, strain (post code 301) and stress are needed. For analysis involving temperature, post code 9 is required. Creep strain and equivalent creep strain, velocity and acceleration are needed for creep and dynamic analysis, respectively.

2. Create 3-D model using Mentat.
   a. Open axisymmetric model `stage_axi.mud`.
   b. Expand axisymmetric mesh to 3-D model via:

      MESH GENERATION⇒EXPAND⇒AXSYMMETRIC MODEL TO 3D

      and move the rigid surfaces, if any, to the deformed configuration.
   c. Define data transfer parameters via:

      INITIAL CONDITIONS⇒MECHANICAL⇒AXSYMMETRIC-3D
   d. Add additional model data such as contact surfaces if necessary.
   e. Save the created model with a different name to avoid overwriting the existing `stage_axi.mud`; for example, into `stage_3d.mud`.
3. Run 3-D model.

Note: (1) If a 3-D analysis is run outside of Mentat, use the `-pid` command line option to read in a post file of axisymmetric analysis. For example, to submit a 3-D analysis, use the command:

```
marc -j stage_3d_job1 -pid stage_axi_job1
```

(2) If remeshing/rezoning steps are performed in the axisymmetric analysis, the original mesh in the model is replaced by a new mesh. The 3-D analysis starts based on the deformed configuration.

To replace step 2-a above, do the following:

i. Open post file `stage_axi.t16` (or `stage_axi.t19`)
ii. Position to the desired increment.
iii. Rezone.
iv. Save the model to `stage_3d.mud`.
v. Close post file.
vi. Open `stage_3d.mud`.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word AXITO3D.
2nd data block			
1-5	1st	I	Not used.
6-10	2nd	I	Not used.
11-15	3rd	I	Enter the number of repetitions in θ direction.
16-20	4th	I	Increment to read off post file. -1 The last increment on the post file of an axisymmetric job (default). -2 Input the time to read the post file at 3rd data block.
21-25	5th	I	Unit number from which the post file is read. Defaults to unit 24 for a formatted post file and to unit 25 for a binary post file.
26-30	6th	I	Enter 0 for binary post file. Enter 1 for formatted post file.
31-35	7th	I	Enter 1 to move stress tensor (post code 311).
36-40	8th	I	Enter 1 to move equivalent plastic strain (post code 7).
41-45	9th	I	Enter 1 to move temperatures (post code 9 or 180).
46-50	10th	I	Enter 1 to move strain tensor (post code 301).
51-55	11th	I	Enter 1 to move plastic strain tensor (post code 321).
56-60	12th	I	Enter 1 to move thermal strain tensor (post code 371).
61-65	13th	I	Enter 1 to move creep strain tensor (post code 331).
66-70	14th	I	Enter 1 to move equivalent creep strain (post code 37).
71-75	15th	I	Enter 1 to move displacements (default). Set to 1 if total Lagrangian formulation is used. For dynamic analysis, velocity v and acceleration a are also moved if 1 is entered. **Note:** If a 0 is entered in an updated Lagrange analysis, the new mesh in 3-D has to include the prior displacements, that is the updated mesh.

Format Fixed	Format Free	Data Type	Entry
The 3rd data block is needed only if the 4th field of the 2nd data block is -2.			
3rd data block			
1-10	1st	F	Enter the time to read the post file. Default is the last increment in the post file.

CHANGE STATE

Redefine State Variables

Description

This option provides various ways of changing the state variables throughout the model. State variables are initialized in the INITIAL STATE model definition set. The number of state variables per point is defined in the STATE VARS parameter. The default is one, with temperature always being the first state variable at a point. If more than one state variable per point has been assigned, this option can be used repeatedly to change the values of all state variables. The default value is no change if this option is not used. In this option, the values of the state variable at the end of the current increment are read in. When the temperature is being defined, the following points should be noted:

- For “history following analysis”, the thermal strains are based on temperature change during this step.
- For elastic re-analysis (ELASTIC parameter) the thermal strains are always based on temperature change between the initial, stress free temperature field and the values read in here.
- The AUTO LOAD option is available for specifying a time-varying history of state variables. The value of the total state variable at the end of each increment is specified.
- The AUTO THERM option is available for automatic control of a nonlinear (elastic-plastic) temperature loaded stress problem, to be used in conjunction with this option.
- The THERMAL LOADS option can be used as an alternate to input the change of temperature. Either incremental or total temperatures can be specified using this option.
- The AUTO THERM CREEP option is available for automatic control of a thermally loaded elastic-plastic-creep problem, to be used in conjunction with this option.

Four ways of changing any state variable through CHANGE STATE are possible:

- Read a range of elements, integration points and layers and a corresponding state variable value for the end of the current step.
- Read the state variable values for the end of the current step through user subroutine NEWSV.

- Read the state variable values for the end of the current step from a named step of the post file output from a previous heat transfer analysis with Marc. With this option, Marc 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. For AUTO LOAD, a one-to-one correspondence between the thermal increments on the post file and the mechanical increments is assumed between the user-defined starting and ending post increments. For AUTO THERM or AUTO THERM CREEP, based on the user-defined allowable temperature change, the thermal increments on the post file can be subdivided into many mechanical increments. Providing state variables through the thermal post file is not currently supported for other adaptive stepping procedures.
- Read a list of elements, integration points, and layers and a corresponding state variable value.

It should be noted that the end of the current step is interpreted as the end of the current increment for fixed stepping procedures (AUTO LOAD, DYNAMIC CHANGE, CREEP INCREMENT) and as the end of the loadcase for adaptive stepping procedures (AUTO STEP, AUTO THERM, AUTO TIME, AUTO INCREMENT, AUTO CREEP).

Note: Using this option, total state variable values are input. From MSC.Marc 2001 onwards, the incremental change in the state variables is reset to 0 bevore each new increment for AUTO LOAD.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-12	1st	A	Enter the words CHANGE STATE.
2nd data block			
1-5	1st	I	Enter the state variable identifier for the state variable being changed (1,2,3,etc.) 1 = temperature. If more than one state variable is being used, the STATE VARS parameter must be included.

Format Fixed	Format Free	Data Type	Entry
6-10	2nd	I	Enter 1 to change the state variable via the 3rd and 4th data blocks below. In this case, the third field must also be defined. Enter 2 to change the state variable via user subroutine NEWSV. This subroutine is now called in a loop on all the elements in the mesh. Enter 3 to read the new values of the state variable from a post file written by a previous heat transfer analysis. In this case, the fourth and fifth field must be defined. Enter 4 to change the state variable via data blocks 5, 6, 7, and 8 below.
11-15	3rd	I	Only nonzero if the second field is set to 1 or 4. In that case, this entry gives the number of data blocks set in data blocks 3 and 4 used to input the new value of the state variable (optional).
16-20	4th	I	Only nonzero 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 read. Defaults to unit 24 for a formatted post file, and to unit 25 for a binary post file.
21-25	5th	I	Only nonzero if the second field is set to 3. In that case, this entry defines the step number on the heat transfer run post file to be read as the definition of the new value of the state variable at the end of the current step. This is currently only supported for AUTO LOAD and AUTO THERM.
26-30	6th	I	Only used if the AUTO LOAD or AUTO THERM options are in use. Give the number of sets of input to be read to define the temperature history.
31-35	7th	I	Enter 1 if formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file.

Format Fixed	Format Free	Data Type	Entry
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 defined in NEWSV.
41-45	9th	I	Enter the post code number to be read into this state variable, default is 9 (temperature).

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of sets is equal to the number given in the third field above.

3rd data block

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with this value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value.
21-25	5th	I	First layer or cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value can only be bigger than 1 for beam or shell elements.

4th data block

1-10	1st	F	New value of this state variable for the above range of points at the end of the current step.

Data blocks 5, 6, 7, and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	New total value of this state variable for the points given below at the start of the zeroth increment.

6th data block

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

Format		Data	
Fixed	Free	Type	Entry

7th data block

This data block is not necessary if the CENTROID parameter is used.

Enter a list of integration points to which the above state variable is applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above state variable is applied.

POINT TEMP

Define Point Temperatures

Description

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

Note: For shell analyses, a uniform temperature is used through the thickness direction.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words POINT TEMP.
2nd data block			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional). Enter -1 if USINC user subtroutine is used. In this case, the 3rd and 4th data blocks are not used.
6-10	2nd	I	Enter file 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 file. 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 file information is read.
21-25	5th	I	Enter step number to be read.
26-30	6th	I	Enter 1 if a formatted post file is used.
			For Cray Only:
			Set to 2 for IEEE binary file.

Format		Data	
Fixed	Free	Type	Entry

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

3rd data block

1-10	1st	E	Temperatures at the end of the increment.

4th data block

			Enter list of nodes for which the above initial temperature is applied.

CONTACT

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 you 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. In acoustic-solid analysis, it also allows for the input of reactive boundary conditions.

Notes: Always define deformable surfaces before rigid surfaces.

Always defines acoustic bodies before structural bodies.

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

If the UFRICTION option and user subroutine UFRIC are used, friction data can 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 can 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 can be skipped.

The following data can be changed upon restart:

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

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-7	1st	A	Enter the word CONTACT.
2nd data block			
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.
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 5: Stick-slip Coulomb Friction
21-25	5th	I	Enter 0 for the calculation of Coulomb friction based on nodal stress. Enter 1 for the calculation of Coulomb friction based on nodal force. Default is 0.
26-30	6th	I	Maximum number of separations allowed in each increment. Default is 9999.
31-35	7th	I	Enter 0 (default) to use the increment splitting procedure for the fixed time step procedures (AUTO LOAD, DYNAMIC CHANGE, TRANSIENT NON AUTO). Enter 1 for the suppression of the splitting of an increment in fixed time step procedure. Enter 2 for adaptive time step procedure. Default is 0. Enter 3 to use contact procedure which does not require increment splitting (automatic penetration checking procedure). **Note:** The automatic penetration checking procedure is not available for dynamics and for the arc length control method.

Format Fixed	Format Free	Data Type	Entry
36-40	8th	I	Enter 1 for the interference kinematic check. Enter 3 to not reset NCYCLE=0 when separation occurs; this speeds up the solution but might result in instabilities.
41-45	9th	I	Control separations within an increment. When 0 is entered, if the force on a node is greater than the separation force, the node separates and an iteration occurs. When 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 does not separate in this increment, but separates at the beginning of the next increment. When 2 is entered, if a new node comes into contact during this increment, it is not allowed to separate during this increment (prevents chattering). When 3 is entered, both (1) and (2) above are in effect.
46-50	10th	I	Parameter governing normal direction/thickness contribution of shell (ISH). Enter 0 – Check Node Contact with top and bottom surface Enter 1 – Nodes only come into contact with bottom layer Enter 2 – Nodes only come into contact with bottom layer and ignore shell thickness Enter -1 – Nodes only come into contact with top layer Enter -2 – Nodes only come into contact with top layer and ignore shell thickness
51-55	11th	I	Enter 1 to reduce printout of surface definition.
56-60	12th	I	Enter 1 to have separation based upon stresses not forces.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
1-10	1st	F	For friction types 1, 2, 3, or 4 enter the relative sliding velocity between bodies below which sticking is simulated (RVCNST). Default = 1.0. For friction type 5, enter the slip-to-stick transition region (β); Default is 1.e-6.
11-20	2nd	F	Distance below which a node is considered touching a body (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	Not used; enter 0.
31-40	4th	F	Not used; enter 0.
41-50	5th	F	Separation force above which a node separates from a body (FNTOL). Default is the maximum residual force. If the 12th field of the second line is 1, enter the separation stress.
51-60	6th	F	Contact tolerance BIAS factor. (0-1)
61-70	7th	F	For stick-slip model, enter the friction coefficient multiplier (α). Defaults to 1.05
71-80	8th	F	For stick-slip model, enter the friction force tolerance (e). Defaults to 0.05.

Format Fixed	Format Free	Data Type	Entry

Data blocks 4, 5, 6, 7, 8, and 9 are repeated once for each body to be defined.

Numbering of Contact Bodies: When defining contact bodies for a deformable-to-deformable analysis, it is important to define them in the proper order. As a general rule, a body with a finer mesh should be defined before a body with a coarser mesh.

Note: For problems involving adaptive meshing or automated remeshing, care must be taken to satisfy this rule before as well as after the mesh change.

4th data block

1-5	1st	I	Body number.
6-10	2nd	I	Number of sets of geometrical data to be input for this rigid body (NETTY). Enter 0 if deformable body.
11-15	3rd	I	For rigid surfaces, enter 1 if surface is a symmetry plane. For deformable bodies, enter 1 if single-sided deformable-deformable contact is used. Note that, in this case, results are dependent upon the order in which contact bodies are defined.
16-20	4th	I	Not used; enter 0.
21-25	5th	I	Enter 1 if analytic form is to be used.
26-30	6th	I	Enter -1 if body is position controlled. Enter 0 (default) if body is velocity controlled. Enter a positive number if load controlled. The number entered is the node number which has the displacement degrees of freedom of the body. The position of this node is at the center of rotation given in the 5th data block.
31-35	7th	I	Enter a positive number if load controlled and rotations are allowed. The number is the node number which has the rotation(s) of the body as degrees of freedom. The position of this node is at the center of rotation given in the 5th data block.

The first node of a load-controlled body may have the normal TRANSFORMATION option to allow for movement in user-defined directions.

The load controlled body node(s) may have FIXED DISP/POINT LOAD or connections with environment/other structures using the SPRINGS option.

Node 1 has x- and y-displacement as degree of freedom 1 and 2
Node 2 has z-rotation as degree of freedom 1

Format Fixed	Format Free	Data Type	Entry
36-40	8th	I	Contact body type (optional): 1: rigid body; 2: deformable body; 3: symmetry body; 4: heat-rigid body; 5: workpiece; 6: acoustic body.
41-64	9th	A	Contact body name (optional)

5th data block

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 0.
31-40	4th	F	First component of velocity or target position of center of rotation.
41-50	5th	F	Second component of velocity or target position of center of rotation.
51-60	6th	F	Angular velocity or angular position, about center of rotation (radians/time).
61-70	7th	F	Friction coefficient.

6th data block

The 6th data block is 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	Body temperature. (Required for rigid body only.)

Format Fixed	Format Free	Data Type	Entry

7th data block

The 7th data block is only necessary for harmonic acoustic analysis.

1-10	1st	F	$\frac{1}{k_1}$ reactive boundary coefficient.
11-20	2nd	F	$\frac{1}{c_1}$ reactive boundary coefficient.

A. For 2-D Deformable Bodies

8a data block

1-80	1st	I	Enter a list of elements of which the body is comprised.

The 8th and 9th data blocks are repeated for as many geometrical data as required (NETTY).

B. For 2-D Rigid Body (Line-Segment)

8b data block

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

The 9th data block is repeated once for each point entered.

9th data block

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

C. For 2-D Rigid Body (Circular Arc)

8c data block

1-5	1st	I	Enter 2 for circular arc (ITYPE).
6-10	2nd	I	Method of describing circular arc (METHOD). See Figure 3-3 and Figure 3-4.

Format Fixed	Format Free	Data Type	Entry

The 9c data block is repeated four times.

9c data block

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

D. For 2-D Rigid Body (Spline)

8d data block

1-5	1st	I	Enter 3 for spline (ITYPE).
6-10	2nd	I	Number of points required to define spline (NPOINT).

The 9d data block is repeated for each point to be entered.

9d data block

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

E. For 2-D Rigid Body (NURBS)

8e data block

1-5	1st	I	Enter 9 for NURBS.
6-10	2nd	I	Number of control points (NPTU).
11-15	3rd	I	Order (NORU).
16-20	4th	I	Number of subdivisions; default 50.

The 9e data block is repeated NPTU times for control points.

9e data block

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

Format

Format Fixed	Format Free	Data Type	Entry

If interpolation scheme is used the following two data blocks are ignored.

The 10e data block is repeated NPTU times for homogeneous coordinate.

10e data block

1-10	1st	F	Homogeneous coordinate between 0 and 1.

The 11e data block is repeated NPTU+ NORU times for knot vectors.

11e data block

1-10	1st	F	Component of knot vector between 0 and 1.

CONTACT

Define Three-dimensional Contact Surface

Description

This option allows for the input of 3-D contact surface definition (rigid or deformable) in contact problems. It also allows you to input friction type, relative sliding velocity for sticking condition, 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. In acoustic-solid analysis it also allows for the input of reactive boundary conditions.

Notes: Always define deformable surfaces before rigid surfaces.

Always define acoustic bodies before structural bodies.

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

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

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

If the 4-node patch option for the definition of rigid surface and the user subroutine DIGEOM are used, the coordinates of the patches can be skipped.

The following data can be changed upon restart:

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

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-7	1st	A	Enter the word CONTACT.
2nd data block			
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.
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 5: Stick-slip Coulomb Friction
21-25	5th	I	Enter 0 for the calculation of Coulomb friction based on nodal stress. Enter 1 for the calculation of Coulomb friction based on nodal force instead of nodal stress. Default is 0.
26-30	6th	I	Maximum of separations allowed in each increment. Default is 9999.
31-35	7th	I	Enter 0 (default) to use the increment splitting procedure for the fixed time step procedures (AUTO LOAD, DYNAMIC CHANGE, TRANSIENT NON AUTO). Enter 1 for the suppression of the splitting of an increment in fixed time procedure. Enter 2 for adaptive time stepping procedure. Default is 0. Enter 3 to use contact procedure which does not require increment splitting (automatic penetration checking procedure). **Note:** The automatic penetration checking procedure is not available for dynamics and for the arc length control method.

Format Fixed	Format Free	Data Type	Entry
36-40	8th	I	Enter 1 for the interference kinematic check. Enter 2 to suppress bounding box checking (this might eliminate penetration, but slows down the solution). Enter 3 to not reset NCYCLE = 0; this speeds up the solution but might result in instabilities (not relevant for iterative penetration check).
41-45	9th	I	Control separations within an increment. When 0 is entered, if the force on a node is greater than the separation force, the node separates and an iteration occurs. When 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 does not separate in this increment, but separates at the beginning of the next increment. When 2 is entered, if a new node comes into contact during this increment, it is not allowed to separate during this increment (prevents chattering). When 3 is entered, both (1) and (2) above is in effect.
46-50	10th	I	Parameter governing normal direction/thickness contribution of shell (ISH). Enter 0 – Check Node Contact with top and bottom surface Enter 1 – Nodes only come into contact with bottom layer Enter 2 – Nodes only come into contact with bottom layer and ignore shell thickness Enter -1 – Nodes only come into contact with top layer Enter -2 – Nodes only come into contact with top layer and ignore shell thickness

Format Fixed	Format Free	Data Type	Entry
51-55	11th	I	Enter 1 to reduce printout of surface definition.
56-60	12th	I	Enter 1 to have separation based upon stresses not forces.
61-65	13th	I	Enter 1 to activate beam-beam contact.
3rd data block			
1-10	1st	F	For friction types 1, 2, 3, or 4 enter the relative sliding velocity between bodies below which sticking is simulated (RVCNST). Default is 1.0. For friction type 5, enter the slip-to-stick transition region (β); Default is 1.e-6.
11-20	2nd	F	Distance below which a node is considered touching a body (ERROR). Leave it blank if you want Marc to calculate it.
21-30	3rd	F	Not used.
31-40	4th	F	Not used.
41-50	5th	F	Separation force, above which a node separates from a body (FNTOL). Default is the maximum residual force. If the 12th field of the second line is 1, enter the separation stress.
51-60	6th	F	Contact tolerance BIAS factor. (0-1)
61-70	7th	F	For stick-slip model, enter the friction coefficient multiplier (α). Defaults to 1.05
71-80	8th	F	For stick-slip model, enter the friction force tolerance (e). Defaults to 0.05.

Format Fixed	Format Free	Data Type	Entry

Data blocks 4, 5, 6, 7, 8, 9, and 10 are repeated once for each body to be defined.

Numbering of Contact Bodies: When defining contact bodies for a deformable-to-deformable analysis, it is important to define them in the proper order. As a general rule, a body with a finer mesh should be defined before a body with a coarser mesh.

Note: For problems involving adaptive meshing or automated remeshing, care must be taken to satisfy this rule before as well as after the mesh change.

4th data block

1-5	1st	I	Body number.
6-10	2nd	I	Number of sets of body entities, NSURGN, to be input for this rigid body. Enter 0 if deformable body.
11-15	3rd	I	For rigid bodies, enter 1 if body is a symmetry plane. For deformable bodies, enter 1 if single-sided deformable-deformable contact is used. Note that in this case, results are dependent upon the order in which contact bodies are defined.
16-20	4th	I	Not used; enter 0.
21-25	5th	I	Enter 1 if analytic form is to be used.
26-30	6th	I	Enter -1 if body is position controlled. Enter 0 (default) if body is velocity controlled. Enter a positive number if load controlled. The number entered is the node number which has the displacement degrees of freedom of the body. The position of this node is at the center of rotation given in the 5th data block.
31-35	7th	I	Enter a positive number if load controlled and rotations are allowed. The number is the node number which has the rotation(s) of the body as degrees of freedom. The position of this node is at the center of rotation given in the 5th data block.

The first node of a load-controlled body may have the normal TRANSFORMATION option to allow for movement in the user-defined directions.

The load controlled body node(s) may have FIXED DISP/POINT LOAD or connections with environment/other structures using the SPRINGS option.

Node 1 has x-, y-, and z-displacement as degrees of freedom 1, 2 and 3
Node 2 has x-,y-, and z-rotation as degrees of freedom 1, 2 and 3
The rotation is defined as: Rx.Ry.Rz

Format Fixed	Format Free	Data Type	Entry
36-40	8th	I	Contact body type (optional): 1: rigid body; 2: deformable body; 3: symmetry body; 4: heat-rigid body; 5: workpiece; 6: acoustic body.
41-64	9th	A	Contact body name (optional)
5th data block			
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	Third coordinate of initial position of center of rotation.
31-40	4th	F	First component of velocity or target position of center of rotation.
41-50	5th	F	Second component of velocity or target position of center of rotation.
51-60	6th	F	Third component of velocity or target position of center of rotation.
6th data block			
1-10	1st	F	Angular velocity or angular position about local axis through center of rotation.
11-20	2nd	F	Not used; enter 0.
21-30	3rd	F	Not used; enter 0.
31-40	4th	F	First component direction cosine of local axis.
41-50	5th	F	Second component direction cosine of local axis.
51-60	6th	F	Third component direction cosine of local axis.
61-70	7th	F	Friction coefficient.

Format Fixed	Format Free	Data Type	Entry

The 7th data block is only necessary for coupled analysis.

7th data block

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
31-40	4th	F	Body temperature. (Required for rigid body only.)

8th data block

The 8th data block is only necessary for harmonic acoustic analysis.

1-10	1st	F	$\frac{1}{k_1}$ reactive boundary coefficient.
11-20	2nd	F	$\frac{1}{c_1}$ reactive boundary coefficient.

A. For 3-D Deformable Body

8a data block

1-80	1st	I	Enter a list of elements of which the body is comprised.

The 8th, 9th, and 10th data blocks are repeated for each set of body entities (NSURGN).

B. For 3-D Rigid Body (Ruled Surface)

8b data block

1-5	1st	I	Enter 4 for ruled surface (ITYPE).
6-10	2nd	I	Entity type of the first surface generator (child) of the surface, JTYPE1.
11-15	3rd	I	If JTYPE1 = 1, 3, 4, 5 enter number of point required to describe first surface generator (NPOINT1). If JTYPE1 = 2 method to describe the circular arc (METH).
16-20	4th	I	Entity type of the second surface generator (child) of the surface (JPOINT2).

Format Fixed	Format Free	Data Type	Entry
21-25	5th	I	If JTYPE1 = 1, 3, 4, 5 enter number of point required to describe second surface generator (NPOINT2). If JTYPE2 = 2 method to describe the circular arc (METH).
26-30	6th	I	Number of subdivisions along first direction (NDIV1), (direction along first and second surface generator).
31-35	7th	I	Number of subdivisions along second direction (NDIV2), (direction from first surface generator to second surface generator).

The 9b data block is repeated (NPOINT1 * NPOINT2) times for ruled surface.

9b data block

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

C. For 3-D Rigid Body (Surface of Revolution)

8th data block

1-5	1st	I	Enter 5 for surface of revolution (ITYPE).
6-10	2nd	I	Entity type of the surface generator.
11-15	3rd	I	If JTYPE1 = 1, 3, 4, 5 enter number of points required to describe the generator (NPOINT). If JTYPE = 2 Method to describe the circular arc (METH).
16-20	4th	I	Number of subdivisions along the first (surface generator) direction (NDIV1).
21-25	5th	I	Number of subdivisions along the second (circumference) direction (NDIV2).

Format Fixed	Format Free	Data Type	Entry

The 9c data block is repeated NPOINT times for surface of revolution.

9c data block

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

10c data block

1-10	1st	F	First coordinate of 1st point on the axis of revolution.
11-20	2nd	F	Second coordinate of 1st point on the axis of revolution.
21-30	3rd	F	Third coordinate of 1st point on the axis of revolution.
31-40	4th	F	First coordinate of 2nd point on the axis of revolution.
41-50	5th	F	Second coordinate of 2nd point on the axis of revolution.
51-60	6th	F	Third coordinate of 2nd point on the axis of revolution.
61-70	7th	F	Total angle (degree) of rotation (Initial position of the surface generator is given on the 9th data block.)

D. For 3-D Rigid Surface (Bezier Surface)

8d data block

1-5	1st	I	Enter 6 for Bezier surface (ITYPE).
6-10	2nd	I	Number of points along the first direction of surface (NPONT1).
11-15	3rd		Number of points along the second direction of surface (NPOINT2).
16-20	4th	I	Number of subdivisions along first direction (NDIV1).
21-25	5th	I	Number of subdivisions along second direction (NDIV2).

Format Fixed	Format Free	Data Type	Entry

The 9d data block is repeated (NPOINT1 * NPOINT2) times for Bezier surface.

9d data block

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

E. For 3-D Rigid Surface (4-Node Patch)

8e data block

1-5	1st	I	Enter 7 for a surface consisting of 4-node patches (ITYPE).
6-10	2nd	I	Number of 4-node patches to be read (which makes the entire surface) NSEG.
11-15	3rd	I	Number of points to be read (NPOINT).
16-20	4th	I	Unit number. Defaults to input. Set KUNIT = -1 if data entered via user subroutine DIGEOM.
21-25	5th	I	Set to 1 if patch data is to be printed. Default: no printing.

The 9e data block is repeated NSEG times for patches not entered by means of user subroutines (ITYPE = 7 and KUNIT not -1)

9e data block

1-5	1st	I	Patch number (not necessary, can be left blank).
6-10	2nd	I	Not used.
11-15	3rd	I	First point number of this patch.
16-20	4th	I	Second point number of this patch.
21-25	5th	I	Third point number of this patch.
26-30	6th	I	Fourth point number of this patch.

Format Fixed	Format Free	Data Type	Entry

The 10e data block is repeated NPOINT times for patches not entered by means number of user subroutine (ITYPE = 7 and KUNIT not -1).

10e data block

1-5	1st	I	Point number.
6-15	2nd	F	First coordinate of this node.
16-25	3rd	F	Second coordinate of this node.
26-35	4th	F	Third coordinate of this node.

F. For 3-D Rigid Surface (Poly-Surface)

8f data block

1-5	1st	I	Enter 8 for polysurface (ITYPE).
6-10	2nd	I	Number of points along the first direction of surface (NPOINT1).
11-15	3rd	I	Number of points along the second direction of surface (NPOINT2).

The 9f data block is repeated (NPOINT1 ∗ NPOINT2) times for poly-surfaces.

9f data block

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 data block

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).

Format Fixed	Format Free	Data Type	Entry
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 9g data block is repeated (NPTU * NPTV) for control points.

9g data block

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 10g data block is repeated (NPTU * NPTV) for homogeneous coordinate.

10g data block

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

The 11g data block is repeated (NPTU + NORU) + (NPTV + NORV) for knot vectors.

11g data block

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

For each trimming curve, data blocks 12g, 13g, 14g, and 15g.

12g data block

1-5	1st	I	Enter 9 for NURBS.
6-10	2nd	I	Number of control points (NPTU).
11-15	3rd	I	Order (NORU).
16-20	4th	I	Number of subdivisions; default 50.

The 13g data block is repeated NPTU times for control points.

13g data block

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

Format Fixed	Format Free	Data Type	Entry

The 14g data block is repeated NPTU times for homogeneous coordinate.

14g data block

1-10	1st	F	Homogeneous coordinate between 0 and 1.

The 15g data block is repeated NPTU+ NORU times for knot vectors.

15g data block

1-10	1st	F	Component of knot vector between 0 and 1.

H. For 3-D Rigid Surface (Cylinder)

8h data block

1-5	1st	I	Enter 10 for Cylinder.
6-10	2nd	I	Number of subdivisions.

9h data block

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.

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

Format Fixed	Format Free	Data Type	Entry

I. For 3-D Rigid Surface (Sphere)

8i data block

1-5	1st	I	Enter 11 for Sphere.
6-10	2nd	I	Number of subdivisions.

9i data block

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.

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

INITIAL TEMP

Define Initial Temperatures

Description

This option provides initial temperatures at nodal points for thermal stress problems. For heat transfer analyses, see INITIAL TEMP for heat transfer.

Note: For shell analyses, a uniform temperature is used through the thickness direction.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words INITIAL TEMP.
2nd data block			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional). Enter -1 if user subroutine USINC is used. In this case, data blocks 3 and 4 are not used.
6-10	2nd	I	Enter file 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 file. Set to 1.
16-20	4th	I	Only nonzero 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 read. Defaults to unit 24 for a formatted post file, and to unit 25 for a binary post file.
21-25	5th	I	Enter step number to be read. If -1 is entered, the last step of the post file is used.
26-30	6th	I	Enter 1 if a formatted post file is used.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3 and 4 are given in NSET pairs, only if the flag in the third field is not equal to 1.			
3rd data block			
1-10	1st	E	Initial temperature.
4th data block			
			Enter list of nodes for which the above initial temperature is applied.

CONTACT TABLE

Define Contact Table

Description

This option is useful for deactivating or activating bodies when the CONTACT option is used. To avoid unnecessary detection of contact between bodies, you can control the detection of contact. The default for contact analysis is that every body detects the possibility of contact relative to all other bodies and itself if it is a flexible body. When the CONTACT TABLE option is entered, the default of detection for every body is overridden. Instead, you specify the relationship of detection between bodies for contact. The touching body does not contact itself unless you request it. Whenever the touched body is a flexible one, by default, the capability of double-sided contact is applied between the contacting bodies. This can be switched off by selecting single-sided contact on the CONTACT option or by setting the searching order in the CONTACT TABLE option. A positive value of the interference closure implies that there is an overlap between the bodies; a negative value implies that a gap exists.

The following control variables of contact between bodies can be modified throughout the table: contact tolerance, separation force, friction coefficient, interference closure and contact heat transfer coefficient (for coupled thermal-stress-contact analysis). In addition, you can invoke the glue option. The previous values of those control variables is not overridden unless nonzero values are entered here. For an acoustic-solid analysis, you can also modify the reactive boundary coefficients.

In the glue option, when a node contacts a rigid body, the relative tangential displacement is zero. When a node contacts a deformable body, all the translational degrees of freedom are tied.

Notes: This option should be placed after the CONTACT option.

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

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word CONTACT TABLE.
2nd data block			
1-5	1st	I	Enter the number of sets of bodies to be input.

The 3rd, 4th, and 5th data blocks are entered once for each set of bodies to be input.

3rd data block			
1-5	1st	I	Enter the touching body number.
6-15	2nd	F	Enter the contact tolerance (ERROR).
16-25	3rd	F	Enter the contact separation force (FNTOL).
26-35	4th	F	Enter the friction coefficient.
36-45	5th	F	Enter the interference closure amount, normal to the contact surface.
46-55	6th	F	Enter the contact heat transfer coefficient (coupled analysis only).
56-65	7th	F	Enter 1.0 to invoke the condition that, when a node comes into contact with this body, there is no relative normal or tangential displacement. Enter 2.0 to invoke the condition that a node is initially in contact with this body (there can be a small gap or overclosure) and there is no relative normal or tangential displacement. The node is not projected onto the surface.

Format Fixed	Format Free	Data Type	Entry
66-70	8th	I	Enter 1 to indicate that the searching order for deformable contact bodies is from the touching body to the touched bodies on the 5th data block. This might change the default order for deformable bodies, which is from bodies with a lower number to bodies with a higher number. Enter 2 to let the program decide which searching order is optimal for deformable bodies. This order is set up such that searching is done starting with the body having the smallest element edge. This option forces single-sided contact between the touching and touched bodies: searching is done only from one body to another and not the other way around.
71-75	9th	I	Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active.
4th data block (Only necessary for harmonic acoustic analysis)			
1-10	1st	F	Enter the $\frac{1}{k_1}$ reactive boundary coefficient.
11-20	2nd	F	Enter the $\frac{1}{c_1}$ reactive boundary coefficient.
5th data block			
			Enter a list of bodies for which the touching body detects contact with the parameters above.

HYPOELASTIC

Define Data for Hypoelastic Materials

Description

This option allows you to input data associated with Marc's hypoelastic material model. You must define the material stress/strain law through user subroutine HYPELA or HYPELA2 (or, for element type 52 or 98, user subroutine UBEAM).

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

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word HYPOELASTIC.
2nd data block			
1-5	1st	I	Enter the number of hypoelastic material data sets to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input file.
Data block 3, 4, 5 are repeated as a set, once for each data set.			
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing TEMPERATURE EFFECTS data and user subroutines.
6-10	2nd	I	Enter 1 to call user subroutines ANEXP and ORIENT.
11-15	3rd	I	Flag to use user subroutine HYPELA2. Enter 1 to pass in deformation gradient (F) and rotation (R). Enter 2 to pass in deformation gradient (F) and stretch ratios (λ). Enter 3 to pass in F, R, and λ.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-10	1st	F	ρ mass density (stress analysis).
11-20	2nd	F	α coefficient of thermal expansion.
21-30	3rd	F	K thermal conductivity.
31-40	4th	F	Specific heat.
41-50	5th	F	Resistivity.
51-60	6th	F	ρ mass density (heat transfer analysis).
61-70	7th	F	Emissivity.
5th data block			
			Enter a list of elements using this material model. (Do not enter composite elements using this material in a layer.)

MOONEY

Define Data for Mooney-Rivlin Materials

Description

This option allows you to enter all the material data for a Mooney-Rivlin rubber material. User subroutine UMOONY can be used to enter temperature dependent coefficients. User subroutine UENERG can be used to enter a general strain energy function. If rate effects are also present, the VISCELMOON model definition option can also be required. A list of elements can also be associated with this material.

Notes: For a Neo-Hookean material model, only C_{10} is needed.

For a Mooney/Rivlin material model, only C_{10} and C_{01} are needed.

For the full 3rd-order invariant model of Jamus, Green and Simpson, use all C_{10}, C_{01}, C_{11}, C_{20}, C_{30}.

The procedure used for the Mooney formulation is defined on the ELASTICITY parameter. If the Total Lagrange formulation is invoked, the elements in this case must be of the Herrmann formulation except for plane stress. If the Updated Lagrange formulation is invoked, the elements must be conventional displacement formulation. (Near-incompressibility is imposed using mixed approach and condensing out pressure degrees of freedom.) For plane stress, displacement elements are always used.

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

These material identifications cannot be referenced by any composite group.

The values C_{10}, C_{01}, C_{11}, C_{20}, and C_{30} can be redefined using the user subroutine UMOONY.

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

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word MOONEY.
2nd data block			
1-5	1st	I	Enter the number of data sets to follow.
6-10	2nd	I	Unit number for data input. Defaults to input file.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3, 4, and 5 are repeated as a set, once for each data set.			
3rd data block			
1-5	1st	I	Material identification number.
4th data block			
1-10	1st	F	C_{10} – Mooney-Rivlin constant.
11-20	2nd	F	C_{01} – Mooney-Rivlin constant.
21-30	3rd	F	ρ – mass density (stress analysis).
31-40	4th	F	α – coefficient of thermal expansion.
41-50	5th	F	C_{11} – higher order constants
51-60	6th	F	C_{20} – higher order constants.
61-70	7th	F	C_{30} – higher order constants.
71-80	8th	F	K – bulk modulus.
5th data block			
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).
31-40	4th	F	Electrical resistance.
41-50	5th	F	Emissivity.
6th data block			
			Enter a list of element numbers associated with this particular elastomeric material.

OGDEN

Define Data for Ogden or Principal Stretch Based Material Model

Description

This option allows you 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:

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

This option can also be used to activate the general principal stretch based models through user subroutines UPSTRECH and UOGDEN.

Note: The procedure used for the Ogden model is defined on the ELASTICITY parameter. If the Total Lagrange formulation is invoked, the elements in this case must be of the Herrmann formulation except for plane stress. If the updated Lagrange formulation is invoked, the elements must be conventional displacement formulation. (Near-incompressibility is imposed using mixed approach and condensing out pressure degrees of freedom.) For plane stress, displacement elements are always used.

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

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word OGDEN.
2nd data block			
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 file.

Format Fixed	Format Free	Data Type	Entry
Data block 3, 4, 5, 6, and 7 are repeated for each data set.			
3rd data block			
1-5	1st	I	Enter the material identification.
6-10	2nd	I	Enter the number of terms (N) which defines the strain energy function.
11-15	3rd	I	Enter 1 for Ogden model (default). Enter 2 for generalized principal stretch based model.
4th data block			
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.
5th data block			
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).
31-40	4th	F	Electrical resistivity.
41-50	5th	F	Emissivity.
Data block 6 is repeated once for each term specified in the 3rd data block. Not used if generalized stretch based model.			
6th data block			
1-10	1st	F	Enter the modulus.
11-20	2nd	F	Enter the power.
7th data block			
			Enter a list of element numbers associated with this particular elastomeric material.

FOAM

Define Data for Foam Material Model

Description

This option allows you to define the data associated with the foam model for highly compressible rubber material. The foam model can be used for plane strain, axisymmetric, and solid elements using the conventional displacement elements.

The strain energy function for this model has the form.

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

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

If the bulk modulus is entered, then $\beta i = 0$ for all values of i.

If the bulk modulus is zero and all βi are 0, then the material is treated as an Ogden material.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word FOAM.
2nd data block			
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 file.

Data blocks 3, 4, 5, 6, and 7 are repeated for each data set.

3rd data block			
1-5	1st	I	Enter the material identification.
6-10	2nd	I	Enter the number of terms (N) which define the strain energy function.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-10	1st	F	Not used; enter 0
11-20	2nd	F	Enter the mass density.
21-30	3rd	F	Enter the coefficient of thermal expansion.
5th data block			
(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).
31-40	4th	F	Electrical resistance.
41-50	5th	F	Emissivity.
Data block 6 is repeated once for each term specified in the 3rd data block.			
6th data block			
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 data block			
			Enter a list of element numbers associated with this particular elastomeric material.

GASKET

Define Material Data for Gasket Materials

Description

This option allows you to specify the material properties for a gasket material, such as the loading and unloading paths, the yield pressure, the transverse shear modulus, and the membrane behavior of the gasket (see also *MSC.Marc Volume A: Theory and User Information*). The loading and unloading paths should be defined using the TABLE option and must be given as a relation between the pressure on the gasket and the gasket closure (independent variable type 37). The reference value of these tables is always 1.

The membrane behavior should be specified using the ISOTROPIC option.

Note: This option can be used only with the lower-order solid composite element types 149, 151 and 152.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word GASKET.
2nd data block			
1-5	1st	I	Enter the number of sets of gasket material data to follow (optional).
6-10	2nd	I	Enter the unit number for input. Defaults to input file.

Data blocks 3 through 7 are repeated as a set, once for each set of gasket material to be input.

3rd data block			
1-5	1st	I	Enter the gasket material identification number.
6-10	2nd	I	Enter the material identification number of the isotropic material to be used for the membrane behavior of the gasket.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-5	1st	I	Enter gasket behavior type in the thickness direction. Currently only type 0 (elastic-plastic) is supported.
6-10	2nd	I	Enter the table identification number of the loading path of the gasket.
11-15	3rd	I	Enter the number of unloading paths to be read through data block 4a. If a 0 is entered here, the gasket behavior will be fully elastic. Up to 10 unloading paths may be specified.
4a data block			
Necessary only if the number of unloading paths entered in the 3rd field of the 4th data block is nonzero.			
1-5	1st	I	Enter the table identification number of the first unloading path of the gasket.
6-10	2nd	I	Enter the table identification number of the second unloading path of the gasket if present.
11-15	3rd	I	Enter the table identification number of the third unloading path of the gasket if present.
16-20	4th	I	Enter the table identification number of the fourth unloading path of the gasket if present.
21-25	5th	I	Enter the table identification number of the fifth unloading path of the gasket if present.
26-30	6th	I	Enter the table identification number of the sixth unloading path of the gasket if present.
31-35	7th	I	Enter the table identification number of the seventh unloading path of the gasket if present.
36-40	8th	I	Enter the table identification number of the eighth unloading path of the gasket if present.
41-45	9th	I	Enter the table identification number of the ninth unloading path of the gasket if present.
46-50	10th	I	Enter the table identification number of the tenth unloading path of the gasket if present.

Format Fixed	Format Free	Data Type	Entry
5th data block			
1-10	1st	F	Enter the yield pressure.
11-20	2nd	F	Enter the tensile modulus (pressure per unit length).
21-30	3rd	F	Enter the transverse shear modulus (force per unit area).
31-40	4th	F	Enter the initial gap.
6th data block			
1-5	1st	I	Enter the table identification number associated with the yield pressure (currently ignored).
6-10	2nd	I	Enter the table identification number associated with the tensile modulus (currently ignored).
11-15	3rd	I	Enter the table identification number associated with the transverse shear modulus (currently ignored).
11-15	3rd	I	Enter the table identification number associated with the initial gap (currently ignored).
7th data block			
			Enter a list of elements associated with this material. The elements have to be of lower-order solid composite type (element types 149, 151 or 152).

TABLE

Define Table

Description

This option defines the data associated with a function. These tables are referenced when defining material properties. A quantity may be a function of only one independent variable.

The value used in the analysis is the value of the evaluated table obtained by linear interpolation multiplied by the reference value, given in the input. If the reference value is entered as 0.0, it is taken as 1.0 and, hence, the table value is used. When the independent variable is out of range of the table, the function is evaluated at the endpoint of the table. No extrapolation is performed.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word TABLE.
2nd data block			
1-5	1st	I	Table id.
6-10	2nd	I	Number of independent variables.
11-15	3rd	I	Unit number to read data.
16-20	4th	I	Set to 1 to suppress printout.
21-25	5th	I	Enter 2 - read X_1, function pairs
26-30	6th	I	Enter the independent variable type - 37 gasket closure.
31-35	7th	I	Enter the number of X_1 data points (nw1).

Format Fixed	Format Free	Data Type	Entry
36-40	8th	I	Not used: enter 0.
41-45	9th	I	Not used: enter 0.
46-50	10th	I	Not used: enter 0.
51-55	11th	I	Not used: enter 0.
56-60	12th	I	Not used: enter 0.
61-65	13th	I	Not used: enter 0.

3rd data block

1-32	1st	A	Enter the table name.

4th data block

Enter nw1 lines each with the value of independent variable and function.

1-10	1st	E	Enter value of independent variable
11-20	2nd	E	Enter value of function.

DAMAGE

Define Properties for Damaging Materials

Description

This option allows you 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.

Gurson Model

For elasto-plastic materials, the damage model is based on a Gurson model for the yield surface definition for materials with voids:

$$F = \left(\frac{\bar{\sigma}}{\sigma_y}\right)^2 + 2q_1 f^* \cosh\left(\frac{q_2 \sigma_{kk}}{2\sigma_y}\right) - [1 - (q_1 f^*)^2] = 0$$

Void nucleation and void growth are based on a model by Tvergaard and Needleman. Here, f^* is introduced to model the rapid decrease in load carrying capacity if void coalescence occurs:

$$f^* = f \quad \text{if } f \le f_c$$

$$f^* = f_c + \left[\frac{f_u - f_c}{f_F - f_c}\right](f - f_c) \text{ if } f > f_c$$

the nucleation can be either stress or strain controlled. The strain controlled nucleation is given by:

$$\dot{f}_{\text{nucleation}} = \frac{f_N}{S\sqrt{2\pi}} \exp\left[-\frac{1}{2}\left(\frac{\varepsilon_m^p - \varepsilon_n}{S}\right)^2\right]\dot{\varepsilon}_m^p$$

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.

Rubber Models

For elastomeric materials, the model is based on the undamaged strain energy function W° multiplied by a Kachanov damage factor, . The damage capability is available for Ogden strain energy model using the total Lagrange formulation as well as the Mooney-Rivlin, Ogden, and general principal stretch based model using the updated Lagrange formulation.

$$W = K \cdot W^{\circ}$$

Both continuous damage (Miehe's formulation) as well as the discontinuous damage (Mullin's effect) can be modeled within an additive:

$$K = d^{\infty} + \sum_{n=1}^{N} d_n^{a} \exp\left(-\frac{\alpha}{\eta_n}\right) + \sum_{m=1}^{N} d_m^{\beta} \exp\left(-\frac{\beta}{\lambda_m}\right) \qquad (N = 1, 2)$$

or a multiplicative format:

$$K = d^{\infty} + \sum_{n=1}^{N} d_n \exp\left(-\frac{\alpha + \delta_n \beta}{\eta_n}\right) \qquad (N = 1, 2)$$

d^{∞} is automatically calculated by Marc.

Simplified Model

For elastic, elastic-plastic, or rigid-plastic materials, there is the option for you to define a simplified damage model. You define the damage factor (df) in the user subroutine UDAMAG.

If model 9 is used, then:

$$\sigma_y = \sigma_y(\bar{\varepsilon}^p, \dot{\bar{\varepsilon}}^p, T)*(1.0 - df)$$

If model 10 is used, then:

$$\sigma_y = \sigma_y(\bar{\varepsilon}^p, \dot{\bar{\varepsilon}}^p, T)*(1.0 - df) \quad \text{and} \quad \mathrm{E} = \mathrm{E}(T)*(1.0 - df)$$

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.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words DAMAGE.
2nd data block			
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 data blocks are entered as pairs, once for each distinct data set.

3rd data block			
1-5	1st	I	Material type identification (1, 2, 3, etc.) for cross-referencing to ISOTROPIC or OGDEN options.
6-10	2nd	I	Damage Model: 0 – Gurson Model, with no nucleation. 1 – Gurson Model, with plastic-strain controlled nucleation. 2 – Gurson Model, with stress controlled nucleation. 3 – Gurson Model, with nucleation controlled by the UVOIDN user subroutine. 4 – Elastomeric damage model; additive decomposition of the Kachanov factor. 5 – Elastomeric damage model; multiplicative decomposition of the Kachanov factor. 6 – Elastomeric damage model controlled by the UELDAM user subroutine. 9 – Simplified damage model, damage applied to yield stress. 10 – Simplified model, damage applied to yield stress and Young's modulus.

Format Fixed	Format Free	Data Type	Entry
4a data block			
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 coalescence of voids start (f_c).
41-50	5th	F	Failure void volume fraction (f_F). This is the value of the void volume fraction at which the stiffness of the material has reduced to zero.
51-60	6th	F	If strain controlled, enter the mean strain nucleation. If stress controlled, enter the mean stress for nucleation. Not needed if UVOIDN is used.
61-70	7th	F	Standard deviation in nucleation relation (S). Not needed if UVOIDN is used.
71-80	8th	F	Volume fraction of void nucleating particles f_N. Not needed if UVOIDN is used.
			Note: The presence of these blocks in the model definition option automatically overwrites the yield criterion specified for a specific material on the ISOTROPIC option. Currently, the model can only be used for isotropic hardening materials.
4b data block			
Use only for elastomeric damage model, additive decomposition, two term Prony series.			
1-10	1st	F	First scalar factor, continuous damage (d_1^{β}).
11-20	2nd	F	First relaxation parameter, continuous damage (λ_1).
21-30	3rd	F	Second scalar factor, continuous damage (d_2^{β}).

Format Fixed	Format Free	Data Type	Entry
31-40	4th	F	Second relaxation parameter, continuous damage (λ_2).
41-50	5th	F	First scalar factor, discontinuous damage (d_1^{α}).
51-60	6th	F	First relaxation parameter, discontinuous damage (η_1).
61-70	7th	F	Second scalar factor, discontinuous damage (d_2^{α}).
71-80	8th	F	Second relaxation parameter, discontinuous damage (η_2).

4c data block

Use only for elastomeric damage model, multiplicative decomposition, two term Prony Series.

1-10	1st	F	First scalar factor (d_1).
11-20	2nd	F	First proportioning term (δ_1).
21-30	3rd	F	First relaxation rate constant (η_1).
31-40	4th	F	Second scalar factor (d_2).
41-50	5th	F	Second proportioning term (δ_2).
51-60	6th	F	Second relaxation rate constant (η_2).

REBAR

Define Rebar Positions, Areas, and Orientations

Description

This option allows the rebar positions, areas, and orientations to be read in by means of input file instead of user subroutine REBAR. See Figures 3-19 and 3-20 for a description of a single rebar layer within an element.

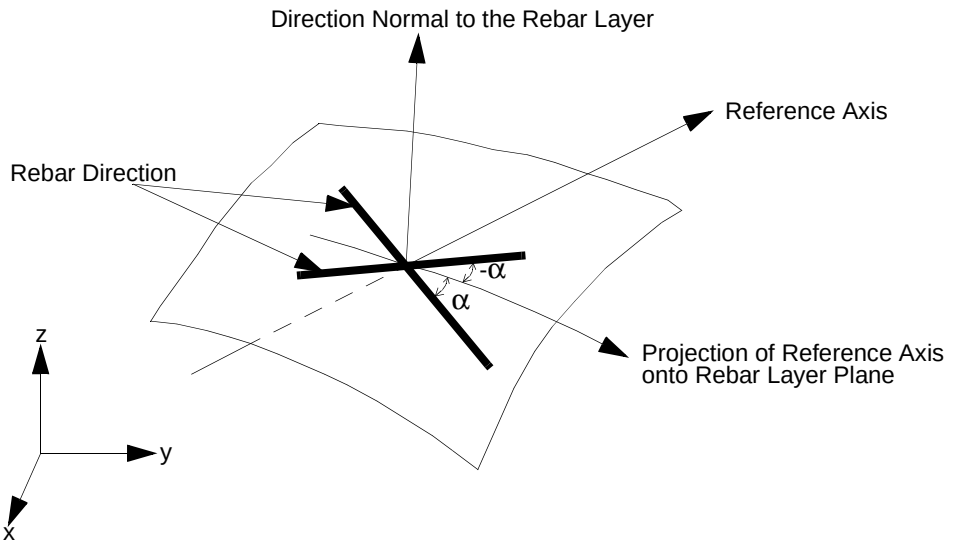
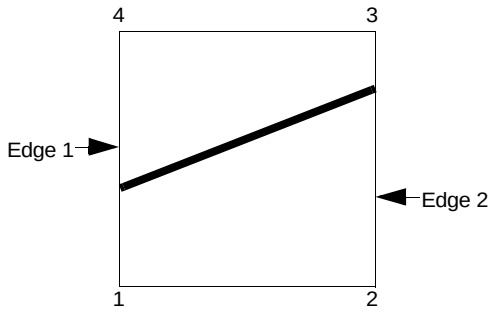


Figure 3-19 Description of Rebar Orientation on a Single Rebar Layer

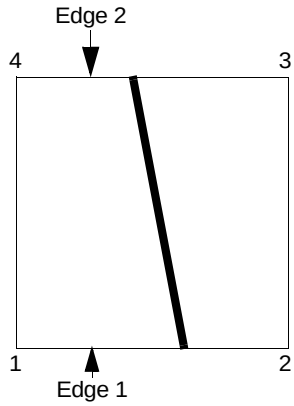
Rebar Postprocessing Results

You can output the stress tensors of specific rebar layers into a post file. Please see the description of post codes 471 and 481 in the POST model definition option. In the second field of the 3rd data block of the POST option is the global layer identification number which is defined by the REBAR option.. The stress and the direction of rebar can be viewed through postprocessing the rebar stress tensor. The magnitude of the vector of the maximum principal value of the stress tensor is the value of stress of the rebar layer at each element. The vector direction is the averaged rebar direction within the element.

2-D

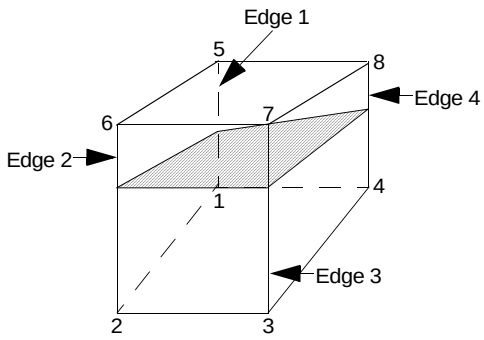


Rebar Orientation Type 1

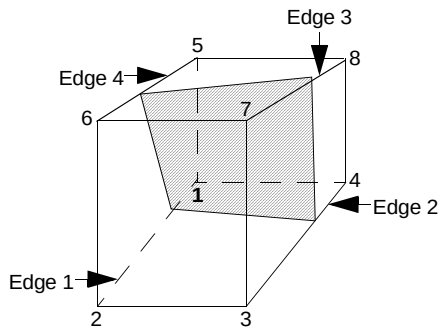


Rebar Orientation Type 2

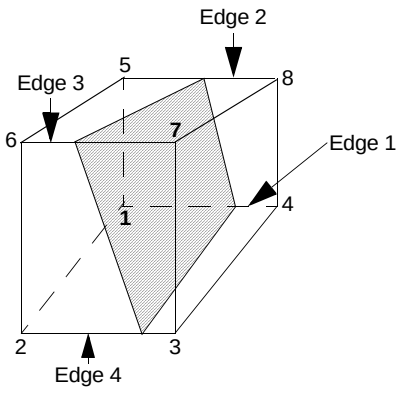
3-D



Rebar Orientation Type 1



Rebar Orientation Type 2



Rebar Orientation Type 3

Figure 3-20 Description of a Single Rebar Layer Within an Element

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-5	1st	A	Enter the word REBAR.
2nd data block			
1-5	1st	I	Number of data sets to be read in.
6-10	2nd	I	Unit number for input; defaults to standard input (unit 5).
Data blocks 3, 4, and 5 are given for each data set.			
3rd data block			
1-5	1st	I	Rebar data set id.
6-10	2nd	I	Number of rebar layers to be read in. Maximum is 5.
11-15	3rd	I	Enter 2 if the considered structure is an axisymmetric expansion of cylinders made of bias plies and the cords are nearly inextensible relative to matrix materials. In this case, all parameters below describe the cylinders from which the considered structure was made. The rebar positions, areas, and orientations for real structure is calculated by Marc. When using cylinder expansion option in rebar data definition, the reference axis needs to be the symmetric axis of the original cylinder, and this needs to pass through the origin of the coordinates.
16-25	4th	F	First direction cosine of reference axis.
26-35	5th	F	Second direction cosine of reference axis.
36-45	6th	F	Third direction cosine of reference axis. The default reference axis is (1, 0, 0). **Note:** (1.) Reference axis should not be perpendicular to rebar layer. (2.) If AXITO3D option is used, the reference axis must be (1,0,0). If the rebar layer is vertical to (1,0,0), the program internally switches the reference axis to (0,1,0).
46-50	7th	I	Enter 0 (default) if no skew type of rebar layers are to be defined. Enter 2 for 2-D. Enter 3 for 3-D and for membrane elements. Set to 0 if the third field of the third data block is 2.

Format Fixed	Format Free	Data Type	Entry
Data block 4 is given for each rebar layer.			
4th data block			
1-5	1st	I	Material id.
6-15	2nd	F	$\frac{p \cdot r}{T}$, relative position of the rebar layer at edge 1 (ratio of the distance between the reference surface (edge) and the rebar layer to the distance across the element); not used for membrane elements.
16-25	3rd	F	A, area of cross section of each rebar at edge 1.
26-35	4th	F	S, number of rebars per unit length in each layer at edge 1. Equivalent thickness of the rebar layer is $A \cdot S$.
36-45	5th	F	Angle (α) between the rebar and the projection of the reference axis on rebar layer plane [-90, 90] at edge 1. See Figure 3-19.
46-55	6th	F	Radius of the cylinder; only used when the third field of the 3rd data block is 2.
56-60	7th	I	Enter the global identification number of the rebar layer. Use for postprocessing only.
61-65	8th	I	Enter the rebar layer orientation type. **2-D** Enter 1 if this rebar layer is similar to the 1,2 and 3,4 edges of the element; the “thickness” direction is from the 1,2 edge to 3,4 edge of the element. Enter 2 if this rebar layer is similar to the 1,4 and 2,3 edges of the element; the “thickness” direction is from the 1,4 edge to 2,3 edge of the element. **3-D** Enter 1 if this rebar layer is similar to the 1,2,3,4 and 5,6,7,8 faces of the element; the “thickness” direction is from the 1,2,3,4 face to 5,6,7,8 face of the element. Enter 2 if this rebar layer is similar to the 1,4,8,5 and 2,3,7,6 faces of the element; the “thickness” direction is from the 1,4,8,5 face to 2,3,7,6 face of the element.

Format Fixed	Format Free	Data Type	Entry
			Enter 3 if this rebar layer is similar to the 2,1,5,6 and 3,4,8,7 faces of the element, the “thickness” direction is from the 2,1,5,6 face to 3,4,8,7 face of the element. Not used for membrane elements.

5th data block is only needed when the 7th field of the third data block is not zero.

5th data block

1-10	1st	F	Relative position of the rebar layer at edge 2. Not used for membrane elements.
11-20	2nd	F	Area of cross-section of rebar at edge 2.
21-30	3rd	F	Number of rebars per unit length at edge 2.
31-40	4th	F	Angle between the rebar and the projection of the reference axis on rebar layer plane (-90,90) at edge 2.

6th and 7th data blocks are only needed when the 7th field of the third data block is 3.

6th data block

1-10	1st	F	Relative position of the rebar layer at edge 3. Not used for membrane elements.
11-20	2nd	F	Area of cross-section of rebar at edge 3.
21-30	3rd	F	Number of rebars per unit length at edge 3.
31-40	4th	F	Angle between the rebar and the projection of the reference axis on rebar layer plane (-90,90) at edge 3.

7th data block

1-10	1st	F	Relative position of the rebar layer at edge 4. Not used for membrane elements.
11-20	2nd	F	Area of cross-section of rebar at edge 4.
21-30	3rd	F	Number of rebars per unit length at edge 4.
31-40	4th	F	Angle between the rebar and the projection of the reference axis on rebar layer plane (-90,90) at edge 4.

8th data block

Enter a list of elements.

ORIENTATION

Define Orientation of Elements

Description

The ORIENTATION option is used to specify orientation angle data as follows:

1. Edge orientation types (`EDGE i-j`). For two-dimensional elements (both continuum and shells), you choose a particular element edge with respect to which the preferred coordinates are specified. With these types, the direction vector along the edge from the first to the second edge node is projected onto the surface tangent plane (xy plane if continuum, or $\tilde{V}^1\tilde{V}^2$ plane if shell) at each integration point.

   The first preferred direction is given by a rotation about the surface normal (z axis if continuum, $\tilde{V}^3$ axis if shell) equal to the orientation angle. The third preferred direction is given by the surface normal, and the second preferred direction is given by a cross product of the third and first directions. See Figure 3-21.
2. Global intersecting plane types (`ij PLANE`). These types are also for two-dimensional elements. Here, a particular global coordinate plane (selected by the orientation type) is intersected with the surface tangent plane.
3. The first preferred direction is given by a rotation about the surface normal from this intersection by an amount equal to the orientation angle. The third preferred direction is given by the surface normal and the second direction by a cross product of the third and first. See Figure 3-22.
4. User-defined intersecting plane. These types are also for two-dimensional elements. Here, a plane, defined by you, with one coordinate direction and a user-defined vector or by two user-defined vectors is intersected with the surface tangent plane.
5. The first preferred direction is given by a rotation about the surface normal from this intersection by an amount equal to the orientation angle. The third preferred direction is given by the surface normal and the second direction by a cross product of the third and first. See Figure 3-23.
6. Three-dimensional orientation types (3D ANISO). For three-dimensional elements, you directly enter vectors in the first and second preferred directions. The third preferred direction is given by a cross product of the first and second direction. See Figure 3-24.

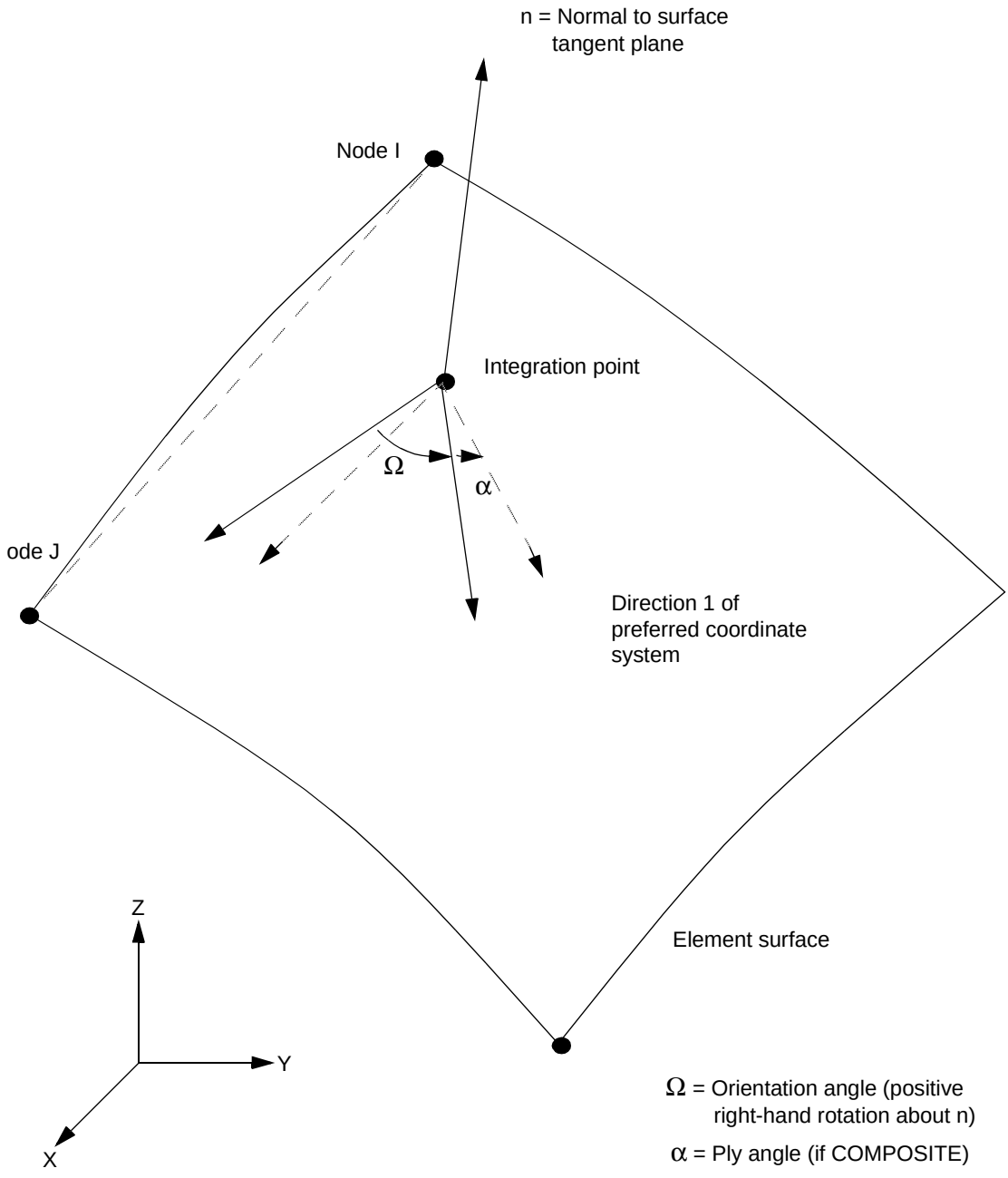


Figure 3-21 Edge I-J Orientation Type

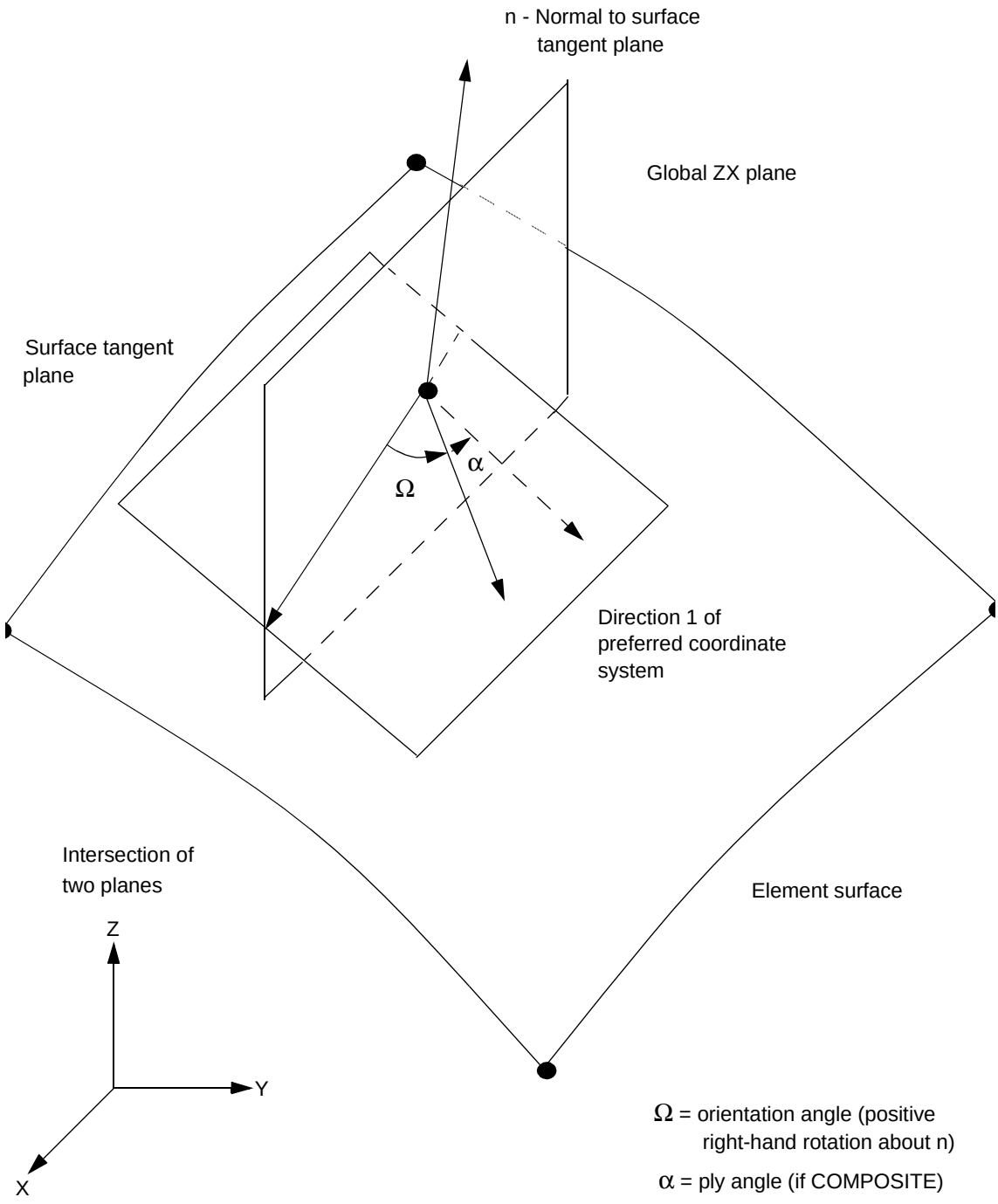


Figure 3-22 ZX Plane Orientation Type

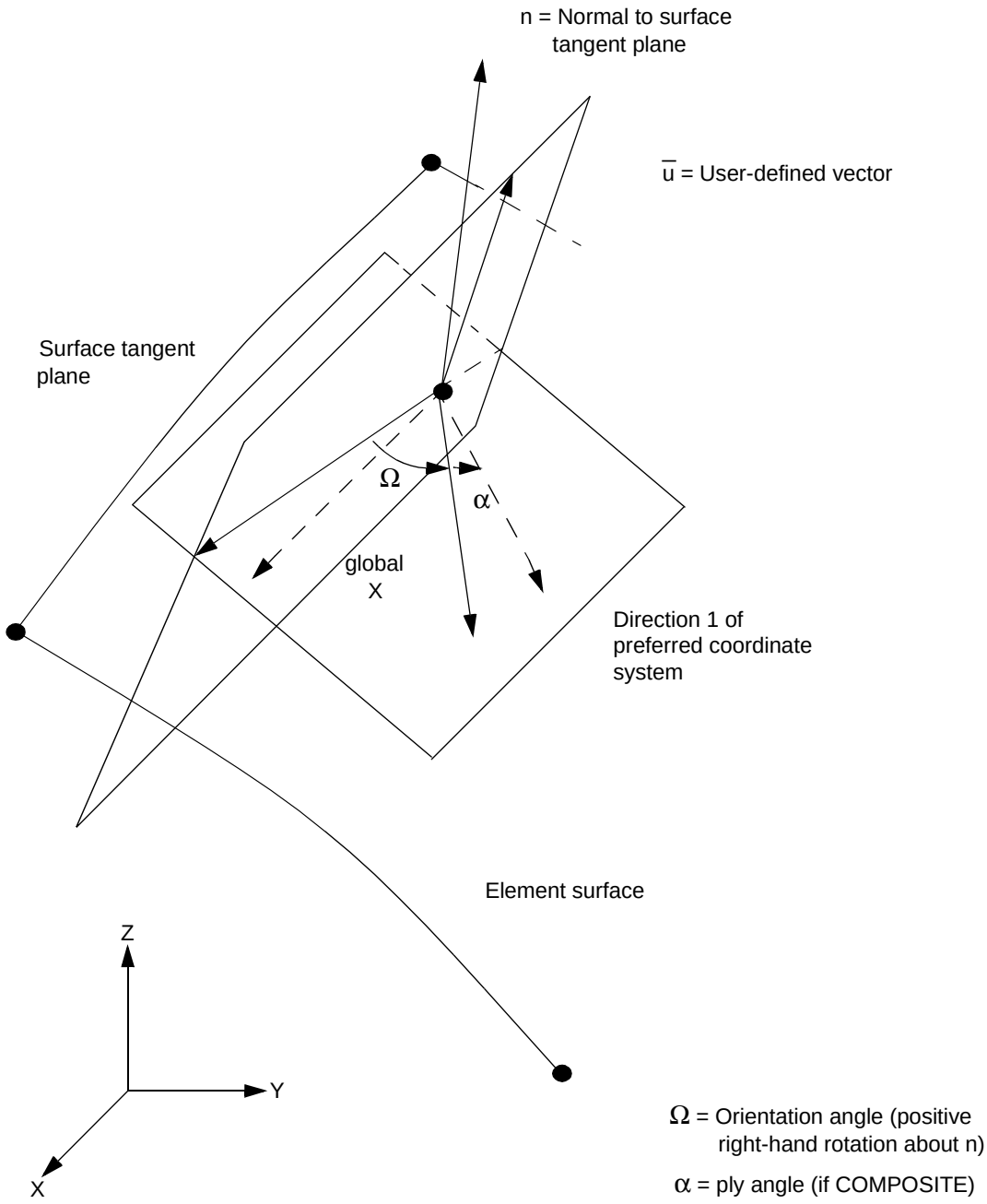


Figure 3-23 XU Plane Orientation Type

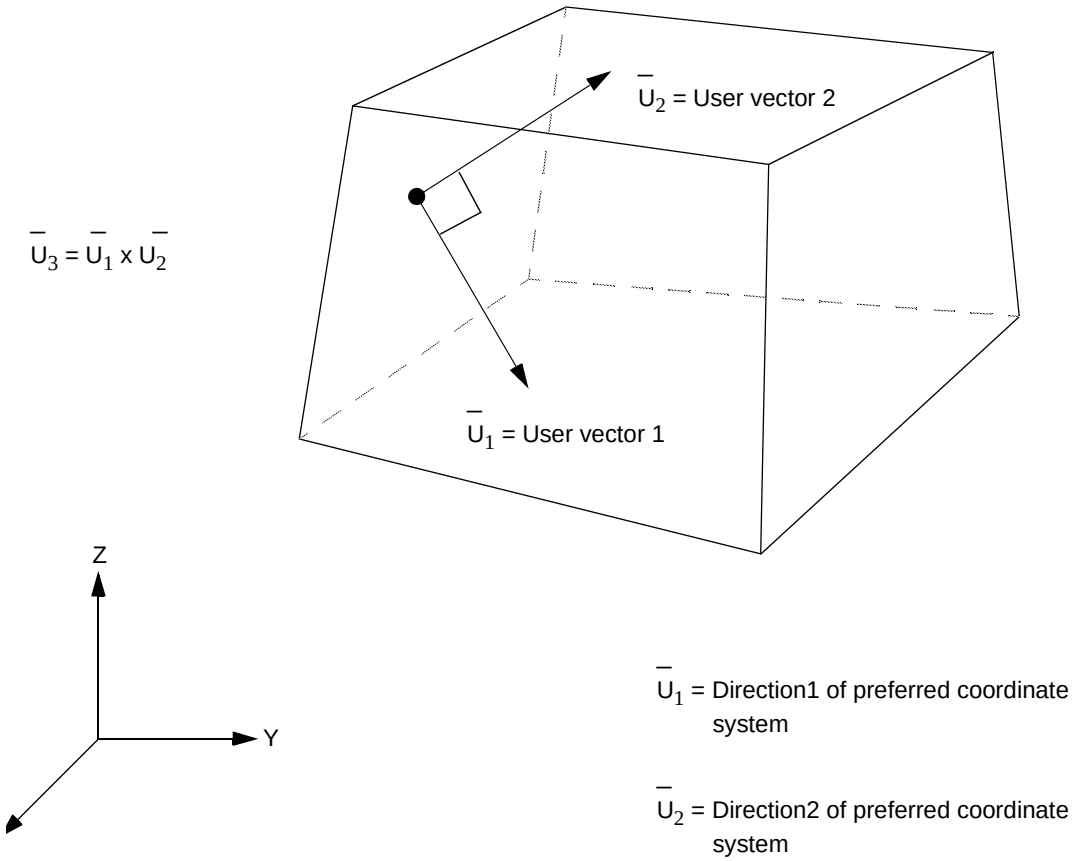


Figure 3-24 Three-dimensional ANISO Orientation Type

7. UORIENT orientation type. Here, you define the transformation matrix between global coordinates (if continuum elements) or local coordinates (if beams, plates or shells) directly in user subroutine ORIENT.

Notes: The ORIENTATION option is ignored for 1-D elements, gaps, pipe bend, shear panel and cable elements.

The ORIENTATION option is turned on for composite elements. If no ORIENTATION data is given for these elements, the default is no preferred orientation; that is, the default material orientation of the element.

The ORIENTATION option, UORIENT, is turned on for particular material numbers if the IANELS flag is set during data input (see ISOTROPIC, ORTHOTROPIC,ANISOTROPIC, MOONEY and HYPOELASTIC options). You can override this default by entering your own ORIENTATION option.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word ORIENTATION.
2nd data block			
1-5	1st	I	Enter the number of orientation angle data sets to follow.
6-10	2nd	I	Unit number for input. Defaults to input file.

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

3rd data block			
1-10	1st	A	Enter one of the following to specify orientation angle type. EDGE 1-2 EDGE 2-3 EDGE 3-4 EDGE 3-1 EDGE 4-1 XY PLANE YZ PLANE ZX PLANE XU PLANE YU PLANE ZU PLANE UU PLANE UORIENT 3D ANISO
11-20	2nd		Orientation angle

Format Fixed	Format Free	Data Type	Entry	
For XU PLANE, YU PLANE, ZU PLANE, UU PLANE, and 3D ANISO, complete the following:				
21-30	3rd	F	1	
31-40	4th	F	2	component of user vector 1 w.r.t. global coordinates.
41-50	5th	F	3	
For UU PLANE and 3D ANISO, complete the following:				
51-60	6th	F	1	
61-70	7th	F	2	component of user vector 2 w.r.t. global coordinates.
71-80	8th	F	3	

4th data block

Enter a list of elements to be associated with this orientation angle.

INITIAL TEMP

Define Initial Temperatures

Description

This option provides initial temperatures for heat transfer problems.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words INITIAL TEMP.
2nd data block			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional). Enter -1 if user subroutine USINC is used. In this case, data blocks 3 and 4 are not used.
6-10	2nd	I	Enter unit number for input of prescribed temperatures data, defaults to input.
11-15	3rd	I	Flag to indicate that initial conditions read from previously generated post file. Set to 1.
16-20	4th	I	Only nonzero 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 read. Defaults to unit 24 for a formatted post file, and to unit 25 for a binary post file.
21-25	5th	I	Enter step number to be read. If -1 is entered, the last step of the post file is used.
26-30	6th	I	Enter 1 if a formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file.

Format Fixed	Format Free	Data Type	Entry
3a data block			
For analyses which do not include heat transfer shell elements:			
1-10	1st	F	Initial temperature.
3b data block			
For analyses which include heat transfer shell elements:			
1-10	1st	F	Initial temperature in first degree of freedom.
11-20	2nd	F	Initial temperature in second degree of freedom.
21-30	3rd	F	Initial temperature in third degree of freedom. **Note:** See MSC.Marc Volume B: Element Library for the definition of nodal degrees of freedom.
4th data block			
			Enter list of nodes for which the above initial temperature is applied.

Chapter 4
History Definition Options

NEW

Use New Format

Description

This option allows the input of data in the K style format. Input is interpreted to be in this format until an OLD option is encountered. This option must not appear embedded inside a model definition or history definition option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word NEW.
11-15	2nd	I	Enter 1 if the default width of the data fields is used for input.This will override the EXTENDED parameter. Enter 2 if the double width of the data fields is used for input.

OLD

Use Old Format

Description

This option allows the input of data in the old (G, H, or J) style format. Input is interpreted to be in this format until a NEW option is encountered. This option must not appear embedded inside a model definition or history definition option.

Formats

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word OLD.

CONTROL

Define History Controls

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

For coupled thermal-stress analysis data block 4 must be used in addition to the 3rd data block.

For nonlinear static analysis, the controls are described in *MSC.Marc Volume A: Theory and User Information.* They do not appear on the restart file, and so must be re-entered on a restart run.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
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 the ELASTIC parameter 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, Marc does a maximum of the absolute value entered. If convergence has not been obtained, a warning is given and Marc proceeds to the next increment. This is not recommended.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Minimum number of recycles during an increment for plasticity or other tangent modulus nonlinearities. Default is 0. **Note:** This data field forces this number of recycles to take place at all subsequent increments. **CAUTION:** This value is overwritten by the PROPORTIONAL INCREMENT option.
16-20	4th	I	Flag for convergence testing. 0 or left blank Convergence is achieved when residuals satisfy the criteria. 1 Convergence is achieved when displacements satisfy the criteria. 2 Convergence is achieved when strain energy satisfies the criteria. 4 Convergence is achieved when either residual or displacement satisfies the criterion. 5 Convergence is achieved when both residual and displacement satisfies the criterion. **Notes:** Testing on relative displacements or strain energy always requires at least one iteration. If nonlinear analysis is done with the CENTROID parameter, the residuals are not calculated and testing is always done on displacements. Nonlinear analysis with the CENTROID parameter is not recommended. If the fields are set as 0, 1, or 2, only the 3rd data block is needed. If the fields are set as 4 or 5, the 3a data block is also needed. In this case, the 3rd data block is set for residual testing and 3a data block is set for displacements check only.
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 in which case absolute tolerances testing is used.

Format Fixed	Format Free	Data Type	Entry
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 *MSC.Marc Volume A: Theory and User Information*). 8. Secant method.
31-35	7th	I	Nonpositive definite flag. If set to 1, solution of nonpositive definite system is forced. **Note:** With use of gap and Herrmann elements, the matrix always is nonpositive definite, and this entry has no significance.
36-40	8th	I	No longer used; enter 0.
41-45	9th	I	To print convergence control messages to log file, enter 1.
46-50	10th	I	Control on initial stress stiffness 0 Normal-full contribution. 1 For Mooney material, reduce contribution of hydrostatic pressure on initial stress stiffness according to: $\sigma^{initial} = \sigma - f_r \cdot p \cdot I$ where $\sigma^{initial}$ is the stress tensor used in the initial stress stiffness matrix, σ is the current stress tensor, f_r is entered through the PARAMETERS option, p is the hydrostatic pressure and I is a unit tensor. 2 No initial stress stiffness. 3 Use stress at beginning of increment, not last iteration. 4 Results in the inclusion of only the positive stresses in the initial stress stiffness during the equilibrium iteration. Besides faster convergence, this leads to a stable analysis of very thin shell structures.

Format Fixed	Format Free	Data Type	Entry
51-55	11th	I	Control parameter: 0 Do not allow switching of convergence testing between residuals and displacements. 1 Allow switching of convergence testing between residual and displacements if reaction forces or displacements become extremely small. **Note:** Set this parameter as 0 if any kind of absolute value testing has been set as convergence tolerance.
3rd data block			
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 is bypassed or absolute testing is performed. If relative displacement checking: Minimum displacement, if displacement increment is less than this value, checking is bypassed or absolute testing is 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 is bypassed or absolute testing is performed. If relative displacement checking: Minimum rotation: if rotation increment is less than this value, checking is bypassed.
41-50	5th	F	If absolute residual testing: Maximum value of residual force. Default is 0.0 in which case, no checking on residual force takes place. If absolute displacement tasking Maximum value of displacement increment. Default is 0.0; in which case, no checking on displacements takes place.
51-60	6th	F	If absolute residual testing: Maximum value of residual moment. Default is 0.0 in which case, no checking on residual moments takes place. If absolute displacement tasking: Maximum value of rotation increment. Default is 0.0; in which case, no checking on rotations takes place.

3a data block

Necessary only if the 4th field of 2nd data block is set to 4 or 5.

1-10	1	F	Maximum allowable value of the change in displacement increment divided by the displacement increment. Default is 0.10.
11-20	2nd	F	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	Minimum displacement, if displacement increment is less than this value, checking is bypassed or absolute testing is performed.
31-40	4th	F	Minimum rotation: if rotation increment is less than this value, checking is bypassed.

Format Fixed	Format Free	Data Type	Entry
41-50	5th	F	Maximum value of displacement increment. Default is 0.0; in which case, no checking on displacements takes place.
51-60	6th	F	Maximum value of rotation increment. Default is 0.0; in which case, no checking on rotations takes place.

4th data block

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 (for example, latent heat, radiation boundary conditions). Default is 0, which bypasses this test. Set to maximum temperature error which is considered acceptable.

PARAMETERS

Definition of Parameters used in Numerical Analysis

Description

There are many parameters that are used in the finite element calculations. These parameters can be customized for your particular application. Some of these constants can be entered in other input blocks as well. The last nonzero value is used for the calculation.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word PARAMETERS.
2nd data block			
1-10	1st	E	Enter the scale factor which, when multiplied with the incremental strain, is used to predict the incremental strain in the next increment. Default is 1.0.
11-20	2nd	E	Enter the multiplier used to calculate the penalty used to impose boundary conditions. Default is 1.e9. The penalty used is, hence, 1.e9 times the maximum diagonal of the stiffness matrix. If the APPBC parameter is used, this option is not used.
21-30	3rd	E	Enter the penalty factor used to satisfy incompressibility in rigid plastic analysis for plane strain, axisymmetric, or solid analysis when displacement elements are used. Default is 100.
31-40	4th	E	Enter the penalty factor used to satisfy incompressibility in fluid analysis when displacement elements are used. Default is 1.e6.

Format Fixed	Format Free	Data Type	Entry
41-50	5th	E	Beta parameter used in transient dynamic analysis using Newmark-beta procedure Default is 0.25.
51-60	6th	E	Gamma parameter used in transient dynamic analysis using Newmark-beta procedure. Default is 0.50.
61-70	7th	E	Gamma1 parameter used in transient dynamic analysis using Single Step Houbolt procedure. Default is 1.5.
71-80	8th	E	Gamma parameter used in transient dynamic analysis using Single Step Houbolt procedure. Default is -0.5.
3rd data block			
1-10	1st	E	Enter the angle at which a node separates from a convex corner or becomes stuck in a concave corner for two-dimensional contact. Default is 8.625°.
11-20	2nd	E	Enter the angle at which a node separates from a convex corner or becomes stuck in a concave corner for three-dimensional contact. Default is 20.0°.
21-30	3rd	E	Enter the initial strain rate for rigid plastic analysis. Default is 1.e-4.
31-40	4th	E	Enter the cutoff strain rate for rigid plastic analysis. Default is 1.e-12.
41-50	5th	E	Enter the fraction of the mean strain that is subtracted from the stress tensor in the initial stress calculation. See the tenth field of the CONTROL option. Default is 0.0

Format Fixed	Format Free	Data Type	Entry
51-60	6th	E	Enter the factor used to calculate the drilling mode for shell elements type 22, 75, 138, 139, and 140. Default is 0.0001.
61-70	7th	E	Enter the scale factor to the incremental displacements for the increment after the rezoning increment. A value of 1 improves friction convergence, but may result in an element inside-out. Default is 1.0.
4th data block (Optional)			
1-10	1st	E	Universal gas constant (R). Default is 8.314 J $mol^{-1}K^{-1}$.
11-20	2nd	E	Offset temperature between user units and absolute temperature. Default is 273.15°; that is, user input in Centigrade.

ADAPT GLOBAL

Define Meshing Parameters Used in Global Remeshing

Description

This model definition option provides parameters used for the global remeshing.

This feature is available only for 2-D problems.

Remeshing Criteria

It is possible to choose one, two, three, or four remeshing criterias simultaneously.

Note: In general, very frequent remeshing should be avoided for effective and computationally efficient analysis. Also, since every remeshing and subsequent rezoning step involves interpolation and extrapolation of element variables, a possibility of error accumulation exists as the analysis progresses when remeshing occurs too frequently.

Increment

Remeshing occurs at specified increment frequency.

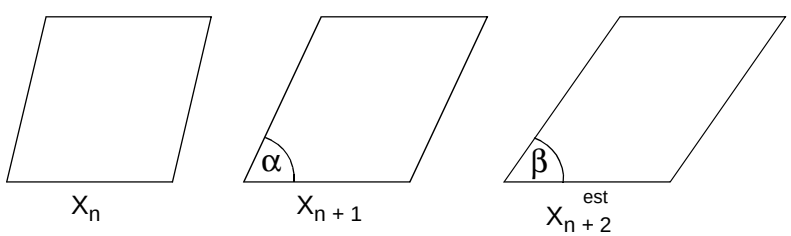
Element Distortion

The identified body is remeshed when the distortion in the elements becomes large, or it is anticipated that the distortion will become large.

The element distortion criteria is based upon examining the angles of the elements at the end of the increment and an estimate of the change in the angle in the next increment.

Given that X_n are the coordinates in the beginning of the step and that ΔU_n are the displacements in the increment, then:

$$X_{n+1} = X_n + \Delta U_n \text{ and } X_{n+2}^{est} = X_{n+1} + \Delta U_n$$



If $\cos \alpha > 0.8$ and $\cos \beta > 0.9$, remesh, or if $\cos \alpha > 0.9$ and $\cos \beta > \cos \alpha$, remesh

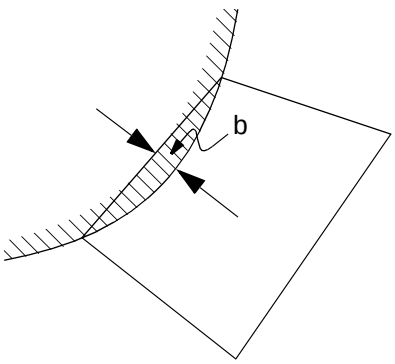
which is equivalent to

$0 < \alpha < 36°$ and $0 < \beta < 25°$ remesh or
$144° < \alpha < 180°$ and $155° < \beta < 180°$ remesh or
$0 < \alpha < 25°$ and $\beta < \alpha$ or
$155° < \alpha < 180°$ and $\beta > \alpha$

Contact Penetration

The identified body is remeshed when the curvature of the contact body is such that the current mesh cannot accurately detect penetration.

The penetration remeshing criteria is based upon examining the distance between the edge of the element and the representation of the other contact surface.



If b> penetration limit, remeshing is required. By default, penetration is 2 * (contact tolerance). The contact tolerance can be input by you or computed internally by the program

Please note that this check does not apply to the self-contact situation.

Angle Deviation

The identified body remeshes when the angles in the element have a deviation from the ideal angle greater than a specified amount. The ideal angle for quadrilaterals or hexahedrals is 90°. The ideal angle for triangles and tetrahedrals is 60°. The default of 40° indicates that any angle in the range $50 \leq \alpha \leq 130$ is acceptable for quadrilaterals.

Immediate

The identified body is remeshed before performing any analysis.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-8	1st	A	Enter the words ADAPT GLOBAL.
2nd data block			
1-5	1st	I	Enter the number of bodies to be remeshed.
6-10	2nd	I	Enter 0 for default. Enter the unit number to read data; defaults to input.
11-15	3rd	I	Not used; enter 0.

The third, fourth, and fifth data blocks are repeated as a set for each criteria selected based on the number of remeshing bodies selected in the 2nd data block.

3rd data block			
1-5	1st	I	Enter 2 for Advancing Front 2-D quadrilateral or triangular mesher. Enter 3 for Delaunay 2-D triangular mesher Enter 4 for 2-D Overlay quadrilateral mesher. A valid mesher type (2-4) must be used.
6-10	2nd	I	Not used; enter 0.
11-15	3rd	I	Enter the contact body to be remeshed (default = 1) where the contact body number is defined under the CONTACT option.
4th data block			
1-5	1st	I	Enter 1 if increment frequency used as remesh criterion.
6-10	2nd	I	Enter 1 if element distortion used as remesh criterion.
11-15	3rd	I	Enter 1 if penetration is used as remesh criterion.
16-20	4th	I	If the first field is 1, enter the frequency in increments.
21-30	5th	E	Not used.
31-35	6th	I	Enter 1 if angle criterion is used.

Format Fixed	Format Free	Data Type	Entry
36-45	7th	E	If the 6th field = 1, enter maximum change in angle from the reference angle (90°) for quadrilaterals.
46-60	8th	E	If the 6th field = 1, enter the maximum change in angle from reference angle (60°) for triangles.
61-65	9th	I	Enter 1 to force remeshing before any analysis if no other data in this load case, or at the end of this increment if incremental load data given.
66-75	10th	E	Penetration limit default = 2 * (contact tolerance).
5th data block (Two-dimensional Marc Advancing Front All Quadrilateral or Triangular Mesher) Mesher type = 2			
1-5	1st	I	Enter 0 for all quadrilateral mesh. Enter 2 for all triangular mesh.
6-15	2nd	E	Enter the target element size. Default means the target number of elements in the 4th field are used.
16-20	3rd	E	Not used; enter 0.
21-25	4th	I	Target number of elements after remeshing; default means no such control. If both the 2nd and 4th fields are default, the number of elements in the previous mesh are used.
26-30	5th	I	Curvature outline control Enter number of divisions of line segments to fit a curvature circle; default = 36. Enter number of divisions of line segments to fit a curvature circle; default = 36. Enter -1 to obtain uniform outline points.
31-40	6th	E	Outline smoothing ration range 0 - 1.0; default = 0.8.
41-50	7th	E	Minimum target element size; default - 1/3 * target element size.
51-60	8th	E	Percentage of change allowed for the new number of elements created. Default means no such control. Total of 5 remeshing trials will be used to create the next to meet the requirement. Not to be used for the remeshing with the automatic-stop-and-restart option.

Format Fixed	Format Free	Data Type	Entry
5th data block (Two-dimensional Delaunay Triangular Mesher) Mesher type = 3			
1-5	1st	I	Not used; enter 0.
6-15	2nd	E	Enter the target element size.
16-25	3rd	E	Not used; enter 0.
26-30	4th	I	Target number of elements after remeshing; default means no such control.
31-35	5th	I	Curvature outline control. Enter number of divisions of line segments to fit a curvature circle. Default = 36. Enter -1 to obtain uniform outline points.
36-45	6th	E	Outline smoothing ratio (range 0-1.0); default = 0.8.
46-55	7th	E	Minimum target element size; default = 1/3 * target element size.
56-65	8th	E	Percentage of change allowed for the new number of elements created. Default means no such control. Totally 5 remeshing trials will be used to create the mesh to meet the requirement. Not to be used for the remeshing with the automatic-stop-and-restart option.
5th data block (Two-dimensional Overlay Quadrilateral Mesher) Mesher type = 4			
1-10	1st	E	Enter the element target length.
11-15	2nd	I	Enter 1 if elements on the boundary in contact are to be refined one level if necessary. Enter 2 if elements on the boundary in contact are to be refined two levels if necessary.
16-20	3rd	I	Enter 1 if elements in the interior can be merged together. Four elements at a time will be merged.
21-25	4th	I	Target number of elements after remeshing; default means no such control.
26-30	5th	I	Not used; enter 0.
31-40	6th	E	Not used; enter 0.
41-50	7th	E	Not used, enter 0.
51-60	8th	E	Percentage of change allowed for the new number of elements created. Default means no such control. Totally 5 remeshing trials will be used to create the mesh to meet the requirement. Not to be used for the remeshing with the automatic-stop-and-restart option.

SOLVER

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 Marc, based upon the amount of workspace required and the number given on the SIZING parameter. You can also select whether a symmetric or nonsymmetric solver is used. Additionally, you can specify if the solution of a nonpositive 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.

You control the maximum number of iterations allowed. If this is a positive number, Marc stops if this is exceeded. If this is a negative number, Marc prints a warning and continues to the next Newton-Raphson iteration or increment.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word SOLVER.
2nd data block			
1-5	1st	I	0 = Profile Direct Solver. 2 = Sparse Iterative. 4 = Sparse Direct Solver 6 = Hardware Provided Direct Sparse Solver 8 = Multifrontal Direct Sparse Solver.
6-10	2nd	I	Enter 1 for solving a nonsymmetric system. Only available for solvers type 0 and 8.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Enter 1 if the solution of nonpositive definite system is to be obtained.
31-35	4th - 7th	I	Not used.
36-40	8th	I	Enter, in millions, the number of four-byte words to be used by solver type 6 or 8 before going out-of-core. Default is the same behavior as for other solvers. For solver type 6, this option is only available on SGI. For solver type 8, it is available on all platforms.

3rd data block

Only necessary if the sparse iterative solver is used.

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

4th data block

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.

AUTO LOAD

Define Equal Load Increments

Description

This option is useful for nonlinear analysis with proportional loads. It generates a specified number of increments; all having the same load increment. This load increment is the net result of the changes and scalings made to the load increment in all the previous increments, plus the effect of any tractions, proportional increment options, etc., in the current increment. These options should come after the AUTO LOAD option. If the proportional increment option is not included, AUTO LOAD sets the proportionality factor to a default of 1.

AUTO LOAD primarily controls mechanical loads and kinematic boundary conditions. For uncoupled thermal stress analysis, AUTO LOAD can also be used to control state variables specified by the THERMAL LOADS/CHANGE STATE options or point temperatures specified by the POINT TEMP option. If a restart is made from one of the increments generated by an AUTO LOAD, the rest of the increments associated with this AUTO LOAD are automatically completed before reading of new input. This can be avoided by using the REAUTO option. The completion of an AUTO LOAD by a restart is done using the control parameters specified by Marc used with the RESTART option.

For contact analysis, the TIME STEP history definition option is also needed with AUTO LOAD, so that the increment displacement of the contact body can be obtained from the velocity.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-9	1st	A	Enter the words AUTO LOAD.
2nd data block			
1-5	1st	I	Number of times this load increment is to be applied.
6-10	2nd	I	Reassembly interval for stiffness matrices. Defaults to whenever nonlinearity occurs.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Maximum number of allowable time step cuts. = 0 means no automatic restart from the previously converged step. > 1 means maximum number of time step cutbacks allowed. Marc automatically restart the analysis after each cutback until the maximum number is reached.

AUTO INCREMENT

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; hence, the post-buckling behavior of structures can be analyzed. However, the option cannot be used for thermal loading.

You have to specify in the DIST LOADS, POINT LOAD, and/or DISP CHANGE options the total loading for a sequence of load steps, and Marc automatically generates 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 ($C = \Delta u^T \Delta u$) 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 data, the AUTO INCREMENT option can be used as often as desired. For more details, see *MSC.Marc Volume A: Theory and User Information*.

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

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

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-14	1st	A	Enter the words AUTO INCREMENT.

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-10	1st	F	Fraction (α) of the total load increment that should be applied in the first cycle of the first increment of this AUTO INCREMENT session resulting in $\Delta u_1 = K^{-1}\alpha P$. $C_{in} = \Delta u_1^T \Delta u_1$.
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 3. Please allow for more recycles via CONTROL model definition option.
21-30	4th	F	Maximum fraction of the total load that can be applied in any increment of this AUTO INCREMENT session. Default is 1 if no contact is present.
31-40	5th	F	Maximum multiplier of applied arc length in norm of displacement vector to initial arc length. C_{max}/C_{in} 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.
51-60	7th	F	Fraction of the initial arclength to define a minimal arclength. C_{min}/C_{in} Default is 0.01.

Format Fixed	Format Free	Data Type	Entry
61-65	8th	I	Arclength root procedure: 1 = Crisfield (quadratic constraint) 2 = Riks/Ramm (linear constraint) 3 = Modified Riks/Ramm (linear constraint) (default) 4 = Crisfield; switch to Modified Riks/Ramm if no real root found

Note: 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 option.

66-70	9th	I	Maximum number of allowable time step cuts. = 0 means no automatic restart from the previously converged step. > 1 means maximum number of time step cutbacks allowed. Marc automatically restart the analysis after each cutback until the maximum number is reached.

AUTO STEP

Adaptive Load Step Control

Description

This option allows control of the automatic time/load stepping procedure. In this procedure, the time step is adjusted based upon the calculated value of a parameter (strain increment, plastic strain increment, creep strain increment, stress increment, strain rate, strain energy increment, temperature increment, displacement increment, rotation) versus a user-defined maximum. More than one criterion can be specified. If the criteria is not satisfied within an increment, recycling occurs with a reduced time/load applied. After the increment has converged based upon tolerances specified on the CONTROL values, the data given here controls the next increment.

The enhanced variant (flagged with a 1 in the 9th field of the second data block) allows the scheme to be used with or without a user specified criterion. The time step is adjusted based upon the number of recycles in addition to the user criteria. If the user-specified desired number of recycles is exceeded, the time step is divided by the factor specified in the third card, 6th field below. If the increment converges in less than the desired number of recycles, the time step is scaled up using the same factor. The increment is started from the beginning again if any of the following occurs: maximum number of iterations reached, elements going inside out, or a contact node slides off the end of a rigid body. In this case, the time step is divided by 2. The enhanced variant is available for mechanical, thermal and thermo-mechanically coupled analyses.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words AUTO STEP.
2nd data block			
1-10	1st	E	Enter the initial time step. Defaults to 1% of total time period for enhanced scheme. Input is required for old scheme.
11-20	2nd	E	Enter the total time period. Defaults to 1.0 for enhanced scheme.

Format		Data	
Fixed	Free	Type	Entry
21-30	3rd	E	Enter the smallest ratio between steps. Default is 0.1.
31-40	4th	E	Enter the largest ratio between steps. Default is 10.0.
41-50	5th	E	Enter the minimum time step. Defaults to total time/maximum number of steps.
51-60	6th	E	Enter the maximum time step. Defaults to total time.
61-65	7th	I	Enter the maximum steps allowed.
66-70	8th	I	Enter the desired number of recycles per increment.
71-75	9th	I	Enter 1 to use enhanced scheme. In this case read an extra data block below.
76-80	10th	I	Enter 1 to use artificial damping scheme for statics.

The 3rd data block is only present if the 9th field of the 2nd data block is equal to 1.

3rd data block

1-5	1st	I	Enter the number of states to put on the post file. These states will be equally spaced in time. By default, the increment based variant as given by the POST option will be used.
6-10	2nd	I	Enter the maximum number of times to cut down the time step in each increment. Defaults to 10.
11-15	3rd	I	Enter 0 to treat user criteria as limits. Enter 1 to treat user criteria as targets.
16-20	4th	I	Indicate finish criterion for thermal or coupled analysis. Set to 1 to finish the transient when all nodal temperatures fall below the value given in the fifth field (see below). Set to -1 to finish the transient when all nodal temperatures exceed the value given in the fifth field (see below). Set to 0 to complete transient time period without any check on temperatures reached.

Format Fixed	Format Free	Data Type	Entry
21-30	5th	E	Finish temperature value to be used in conjunction with flag set above.
31-40	6th	E	Enter scale factor for time step changes other than changes due to user criteria. Defaults to 1.2.
41-45	7th	I	Enter flag to override CREEP and DYNAMIC parameters as specified in the parameter section for this load case. 0 Do not override parameters. 1 Turn off CREEP and DYNAMICS. 2 Turn off CREEP. 3 Turn off DYNAMICS.

Repeat 4rd and 5th data blocks in pairs for each criterion.

4rd data block

1-5	1st	I	Enter the criterion id: Enter 1 for strain increment. Enter 2 for plastic strain increment. Enter 3 for creep strain increment. Enter 4 for normalized creep strain increment. Enter 5 for stress increment. Enter 7 for strain energy increment. Enter 8 for temperature increment. Enter 9 for displacement increment. Enter 10 for rotation increment.
6-80	2nd	I	Enter set name of elements/nodes to which this criteron is to be applied.

Format Fixed	Format Free	Data Type	Entry
5th data block			
1-10	1st	E	ΔYI.
11-20	2nd	E	XMAX1.
21-30	3rd	E	ΔY2.
31-40	4th	E	XMAX2.
41-50	5th	E	ΔY3.
51-60	6th	E	XMAX3.
61-70	7th	E	ΔY4.
71-80	8th	E	XMAX4.

For criteria 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, the time step is adjusted based upon:

if $X \leq XMAX1 \Delta Y$ calculated/$\Delta Y1$

if $XMAX1 < X < XMAX2 \Delta Y$ calculated/$\Delta Y2$

if $XMAX2 < X < XMAX3 \Delta Y$ calculated/$\Delta Y3$

if $XMAX3 < X \Delta Y$ calculated/$\Delta Y4$

where

Criterion	X	ΔY
1	strain	strain increment
2	plastic strain	plastic strain increment
3	creep strain	creep strain increment
4	creep strain	creep strain increment/elastic strain
5	stress	stress increment
7	strain energy	strain energy increment
8	temperature	temperature increment
9	displacement	displacement increment
10	rotation	rotation increment

CHANGE STATE

Redefine State Variables

Description

This option provides various ways of changing the state variables throughout the model. State variables are initialized in the INITIAL STATE model definition set. The number of state variables per point is defined in the STATE VARS parameter. The default is one, with temperature always being the first state variable at a point. If more than one state variable per point has been assigned, this option can be used repeatedly to change the values of all state variables. The default value is no change if this option is not used. In this option, the values of the state variable at the end of the current increment are read in. When the temperature is being defined, the following points should be noted:

- For “history following analysis”, the thermal strains are based on temperature change during this step.
- For elastic re-analysis (ELASTIC parameter) the thermal strains are always based on temperature change between the initial, stress free temperature field and the values read in here.
- The AUTO LOAD option is available for specifying a time-varying history of state variables. The value of the total state variable at the end of each increment is specified.
- The AUTO THERM option is available for automatic control of a nonlinear (elastic-plastic) temperature loaded stress problem, to be used in conjunction with this option.
- The THERMAL LOADS option can be used as an alternate to input the change of temperature. Either incremental or total temperatures can be specified using this option.
- The AUTO THERM CREEP option is available for automatic control of a thermally loaded elastic-plastic-creep problem, to be used in conjunction with this option.

Four ways of changing any state variable through CHANGE STATE are possible:

- Read a range of elements, integration points and layers and a corresponding state variable value for the end of the current step.
- Read the state variable values for the end of the current step through user subroutine NEWSV.

- Read the state variable values for the end of the current step from a named step of the post file output from a previous heat transfer analysis with Marc. With this option, Marc 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. For AUTO LOAD, a one-to-one correspondence between the thermal increments on the post file and the mechanical increments is assumed between the user-defined starting and ending post increments. For AUTO THERM or AUTO THERM CREEP, based on the user-defined allowable temperature change, the thermal increments on the post file can be subdivided into many mechanical increments. Providing state variables through the thermal post file is not currently supported for other adaptive stepping procedures.
- Read a list of elements, integration points, and layers and a corresponding state variable value.

It should be noted that the end of the current step is interpreted as the end of the current increment for fixed stepping procedures (AUTO LOAD, DYNAMIC CHANGE, CREEP INCREMENT) and as the end of the loadcase for adaptive stepping procedures (AUTO STEP, AUTO THERM, AUTO TIME, AUTO INCREMENT, AUTO CREEP).

Note: Using this option, total state variable values are input. From MSC.Marc 2001 onwards, the incremental change in the state variables is reset to 0 bevore each new increment for AUTO LOAD.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-12	1st	A	Enter the words CHANGE STATE.
2nd data block			
1-5	1st	I	Enter the state variable identifier for the state variable being changed (1,2,3,etc.) 1 = temperature. If more than one state variable is being used, the STATE VARS parameter must be included.

Format Fixed	Format Free	Data Type	Entry
6-10	2nd	I	Enter 1 to change the state variable via the 3rd and 4th data blocks below. In this case, the third field must also be defined. Enter 2 to change the state variable via user subroutine NEWSV. This subroutine is now called in a loop on all the elements in the mesh. Enter 3 to read the new values of the state variable from a post file written by a previous heat transfer analysis. In this case, the fourth and fifth field must be defined. Enter 4 to change the state variable via data blocks 5, 6, 7, and 8 below.
11-15	3rd	I	Only nonzero if the second field is set to 1 or 4. In that case, this entry gives the number of data blocks set in data blocks 3 and 4 used to input the new value of the state variable (optional).
16-20	4th	I	Only nonzero 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 read. Defaults to unit 24 for a formatted post file, and to unit 25 for a binary post file.
21-25	5th	I	Only nonzero if the second field is set to 3. In that case, this entry defines the step number on the heat transfer run post file to be read as the definition of the new value of the state variable at the end of the current step. This is currently only supported for AUTO LOAD and AUTO THERM.
26-30	6th	I	Only used if the AUTO LOAD or AUTO THERM options are in use. Give the number of sets of input to be read to define the temperature history.
31-35	7th	I	Enter 1 if formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file.

Format Fixed	Format Free	Data Type	Entry
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 defined in NEWSV.
41-45	9th	I	Enter the post code number to be read into this state variable, default is 9 (temperature).

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of sets is equal to the number given in the third field above.

3rd data block

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with this value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value.
21-25	5th	I	First layer or cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value can only be bigger than 1 for beam or shell elements.

4th data block

1-10	1st	F	New value of this state variable for the above range of points at the end of the current step.

Data blocks 5, 6, 7, and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	New total value of this state variable for the points given below at the start of the zeroth increment.

6th data block

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

Format		Data	
Fixed	Free	Type	Entry

7th data block

This data block is not necessary if the CENTROID parameter is used.

Enter a list of integration points to which the above state variable is applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above state variable is applied.

POINT TEMP

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 is used through the thickness direction.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words POINT TEMP.
2nd data block			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional). Enter -1 if USINC user subtroutine is used. In this case, the 3rd and 4th data blocks are not used.
6-10	2nd	I	Enter file 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 file. Set to 1.
16-20	4th	I	Only nonzero if the third field is set to 1. This entry defines the unit number from which the post file information is read.
21-25	5th	I	Enter step number to be read.
26-30	6th	I	Enter 1 if a formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3 and 4 are given in NSET pairs, only if the flag in the third field is not equal to 1.			
3rd data block			
1-10	1st	E	Temperatures at the end of the increment.
4th data block			
			Enter list of nodes for which the above initial temperature is applied.

CHANGE PORE

Define Pore Pressures for Uncoupled Soil Analysis

Description

This option provides various ways of changing the pore pressure throughout the model. This option is only used in uncoupled soil analysis.

Given below are four ways of providing the pore pressures.

1. Read a range of elements, integration points and layers, and corresponding pore pressures for the end of the current step.
2. Read the pore pressure values for the end of the current step through user subroutine NEWPO.
3. Read the pore pressure values for the end of the current step from a named step of the post file output from a previous pore pressure analysis with Marc. With this option, Marc assumes direct correspondence of the post file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by you.
4. Read a list of elements, integration points and layers, and corresponding pore pressure.

   Note: On this option, total pore pressures are input.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words CHANGE PORE.
2nd data block			
1-5	1st	I	Enter 1.
6-10	2nd	I	Enter 1 to change the pore pressure via data block 3 below. In this case, the third field must also be defined. Enter 2 to change the pore pressure via user subroutine NEWPO. This subroutine is then called in a loop on all the elements in the mesh.

Format Fixed	Format Free	Data Type	Entry
			Enter 3 to read the initial values of the pore pressure from the post file written by a previous pore pressure analysis. In this case, the fourth and fifth fields must also be defined. Enter 4 to change the pore pressure via data blocks 5, 6, 7 and 8 below.
11-15	3rd	I	Only nonzero if the second field is set to 1 or 4. In that case, this entry gives the number of blocks set in data blocks 3 and 4 used to input the new value of the pore pressure (optional).
16-20	4th	I	Only nonzero if the second field is set to 3. Then this entry defines the unit number from which the post file information from the previous pore pressure run is read.
21-25	5th	I	Only nonzero if the second field is set to 3. In that case this entry defines the step number on the pore pressure run post file to be read as the definition of the new value of the pore pressure at the end of the current step.
26-30	6th	I	Not used; enter 1.
31-35	7th	I	Enter 1 if a formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file.
36-40	8th	I	Only nonzero if the second field set to 2. Set to 1 to suppress printout of pore pressure values that are defined in user subroutine NEWPO.

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of data sets is equal to the number given in the third field above.

3rd data block

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with this value.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value can only be bigger than 1 if ALL POINTS parameter is used.
21-25	5th	I	First layer or cross-section point with this value.
26-30	6th	I	Last layer or cross-section point with this value.

4th data block

1-10	1st	F	New value of the pore pressure for the above range of points at the end of the current step.

Data blocks 5, 6, 7, and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	Pore pressure for the points given below at the end of the current increment.

6th data block

Enter a list of elements to which the above pore pressure is to be applied.

7th data block

This data block is not necessary if the CENTROID parameter is used.

Enter a list of integration points to which the above pore pressure is to be applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above pore pressure is to be applied.

CONTACT TABLE

Define Contact Table

Description

This option is useful for deactivating or activating bodies when the CONTACT option is used. To avoid unnecessary detection of contact between bodies, you can control the detection of contact. The default for contact analysis is that every body detects the possibility of contact relative to all other bodies and itself if it is a flexible body. When the CONTACT TABLE option is entered, the default of detection for every body is overridden. Instead, you specify the relationship of detection between bodies for contact. The touching body does not contact itself unless you request it. Whenever the touched body is a flexible one, by default, the capability of double-sided contact is applied between the contacting bodies. This can be switched off by selecting single-sided contact on the CONTACT option or by setting the searching order in the CONTACT TABLE option. A positive value of the interference closure implies that there is an overlap between the bodies; a negative value implies that a gap exists.

The following control variables of contact between bodies can be modified throughout the table: contact tolerance, separation force, friction coefficient, interference closure, and contact heat transfer coefficient (for coupled thermal-stress-contact analysis). In addition, you can invoke the glue option. The previous values of those control variables are not overridden unless nonzero values are entered here. For an acoustic-solid analysis, you can also modify the reactive boundary coefficients.

In the glue option, when an node contacts a rigid body, the relative tangential displacement is zero. When a node contacts a deformable body, all the translational degrees of freedom are tied.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word CONTACT TABLE.
2nd data block			
1-5	1st	I	Enter the number of sets of bodies to be input.

Format Fixed	Format Free	Data Type	Entry
The 3rd, 4th, and 5th data blocks are entered once for each set of bodies to be input.			
3rd data block			
1-5	1st	I	Enter the touching body number.
6-15	2nd	F	Enter the contact tolerance (ERROR).
16-25	3rd	F	Enter the contact separation force (FNTOL).
26-35	4th	F	Enter the friction coefficient.
36-45	5th	F	Enter the interference closure amount, normal to the contact surface.
46-55	6th	F	Enter the contact heat transfer coefficient (coupled analysis only).
56-65	7th	F	Enter 1.0 to invoke the condition that, when a node comes into contact with this body, there is no relative normal or tangential displacement. Enter 2.0 to invoke the condition that a node is initially in contact with this body (there can be a small gap or overclosure) and there is no relative normal or tangential displacement. The node is not projected onto the surface.
65-70	8th	I	Enter 1 to indicate that the searching order for deformable contact bodies is from the touching body to the touched bodies on the 5th data block. This might change the default order for deformable bodies, which is from bodies with a lower number to bodies with a higher number. Enter 2 to let the program decide which searching order is optimal for deformable bodies. This order is set up such that searching is done starting with the body having the smallest element edge. This option forces single-sided contact between the touching and touched bodies: searching is done only from one body to another and not the other way around.
71-75	9th	I	Enter 2 or 3 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment.

Format Fixed	Format Free	Data Type	Entry
4th data block (Only necessary for harmonic acoustic analysis)			
1-10	1st	F	Enter the $\frac{1}{k_1}$ reactive boundary coefficient.
11-20	2nd	F	Enter the $\frac{1}{c_1}$ reactive boundary coefficient.
5th data block			
			Enter a list of bodies for which the above body detects contact with the specified parameters.

Chapter 5
Rezoning Options

CONTACT CHANGE

Change Surface Contact after Rezoning

Description

This option allows changes to a deformable surface definition (rezoned mesh) in 2-D and 3-D contact problems after rezoning occurs. The rigid surface definition cannot be changed. It also allows changes to friction type, choice of Coulomb friction calculation, maximum number of separations in each increment, suppression of splitting of increment. In addition, relative sliding velocity for sticking condition, contact tolerance, separation force, as well as average and cut-off strain rates in rigid-plastic analysis, can also be changed.

Note: See the SPLIT BODIES option if the analysis has deformable-to-deformable contact.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-14	1st	A	Enter the words CONTACT CHANGE.
2nd data block			
1-5	1st	I	Number of surfaces to be defined. (Must be same value as before rezoning.)
6-10	2nd	I	Maximum number of entities to be created for any surface. (Same value as before rezoning.)
11-15	3rd	I	Upper bound to the number of nodes that lie on the periphery of any deformable surface. (Same value as before rezoning.)
16-20	4th	I	Friction type 0: No Friction 1: Shear Friction 2: Coulomb 3: Shear Friction for Rolling 4: Coulomb Friction for Rolling 5: Stick-slip Coulomb Friction (Same value as before rezoning.)

Format Fixed	Format Free	Data Type	Entry
21-25	5th	I	Enter 1 for the calculation of Coulomb friction based on nodal force instead of nodal stress. Default is 0. (Same value as before rezoning.)
26-30	6th	I	Maximum of separations allowed in each increment. Default is 9999.
31-35	7th	I	Enter 1 for the suppression of the splitting of an increment in fixed time step procedure. Enter 2 for adaptive time step procedure. Default is 0. Enter 3 to use contact procedure which does not require increment splitting.
36-40	8th	I	Enter 1 for the interference kinematic check. Enter 2 to suppress bounding box checking (this might eliminate penetration, but slows down the solution). Enter 3 to not reset NCYCLE=0; this speeds up the solution but might result in instabilities. Enter 4 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 0 is entered, if the force on a node is greater than the separation force, the node separates and an iteration occurs. When 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 does not separate in this increment, but separates at the beginning of the next increment. When 2 is entered, if a new node comes into contact during this increment, it is not allowed to separate during this increment (prevents chattering). When 3 is entered, both (1) and (2) above are in effect.

Format Fixed	Format Free	Data Type	Entry
46-50	10th	I	Parameter governing normal direction/thickness contribution of shell (ISH). Enter 0 – Check Node Contact with top and bottom surface Enter 1 – Nodes only come into contact with bottom layer Enter 2 – Nodes only come into contact with bottom layer and ignore shell thickness Enter -1 – Nodes only come into contact with top layer Enter -2 – Nodes only come into contact with top layer and ignore shell thickness
51-55	11th	I	Enter 1 to reduce printout of surface definition.
56-60	12th	I	Enter 1 to have separation based upon stresses not forces.
61-65	13th	I	Enter 1 to activate beam-beam contact.
3rd data block			
1-10	1st	F	For friction types 1, 2, 3, or 4 enter the relative sliding velocity between surfaces below which sticking is simulated (RVCNST). Default = 1.0. For friction type 5, enter the slip-to-stick transition region (β); Default is 1.e-6.
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	Not available.

Format Fixed	Format Free	Data Type	Entry
31-40	4th	F	Not available.
41-50	5th	F	Separation force above which a node separates from a surface (FNTOL). Default is the maximum residual force. If the 12th field of the second line is 1, enter the separation stress.
51-60	6th	F	Contact tolerance BIAS factor. (0-1)
61-70	7ht	F	For stick-slip model, enter the friction coefficient multiplier (α). Defaults to 1.05
71-80	8th	F	For stick-slip model, enter the friction force tolerance (e). Defaults to 0.05.

For two- and three-dimensional contact problems

The data blocks 4 and 5 are repeated once for each data set.

4th data block

1-5	1st	I	Surface number.
6-10	2nd	I	Number of sets of geometrical data to be input for this surface. Enter 0 if flexible surface.
11-15	3rd	I	For rigid surfaces, enter 1 if surface is a symmetry plane. For deformable bodies, enter 1 if single-sided deformable-deformable contact is used. Note that, in this case, results are dependent upon the order in which contact bodies are defined.
16-20	4th	I	Not used; enter 0.
21-25	5th	I	Enter 1 if analytic form is to be used.
26-30	6th	I	Enter -1 if body is position controlled. Enter 0 (default) if body is velocity controlled. Enter a positive number if load controlled. The number entered is the node number which has the displacement degrees of freedom of the body. The position of this node is at the center of rotation given in the 5th data block.

Format Fixed	Format Free	Data Type	Entry
31-35	7th	I	Enter a positive number if load controlled and rotations are allowed. The number is the node number which has the rotation(s) of the body as degrees of freedom. The position of this node is at the center of rotation given in the 5th data block.
36-40	8th	I	Contact body type (optional): 1: rigid body; 2: deformable body; 3: symmetry body; 4: heat-rigid body; 5: workpiece; 6: acoustic body.
41-64	9th	A	Contact body name (optional)

The 5th data block is only necessary if the surface is deformable; if the surface is rigid, no additional data is required.

5th data block

	Enter a list of elements of which the surface is comprised.

Appendix A
Program Messages

Exit Numbers 1001-2000

Exit Number	Explanation
1001	Connectivity exceeded at the node given in the message: `MORE THAN MAXNP JOINED TO NODE...` during in-core assembly of elements. This aborts Marc at that point. If this occurs during a contact analysis, try to activate single sided (K3 style) contact on the CONTACT option. Otherwise Marc system error; consult MSC.Software Corporation analyst.
1002	Connectivity exceeded at the node given in the message: `MORE THAN MAXNP JOINED TO NODE...` during out-of-core assembly of elements. This aborts Marc at that point. If this occurs during a contact analysis, try to activate single sided (K3 style) contact on the CONTACT option. Otherwise Marc system error; consult MSC.Software Corporation analyst.
1003	Too many nodes joined to node in forming fluid coupling matrix. Marc system error; consult MSC.Software Corporation analyst.
1005	Errors during stiffness or mass matrix generation. The output reveals which element has a particular problem. If this occurs during the first assembly, it is due to input errors associated with the COORDINATES, GEOMETRY, or CONNECTIVITY options. If this occurs during a later increment, it is due to excessive deformation in the element. This can occur during the iterative process, so that it is not always possible to visualize the excessive deformation. Check the material behavior and the magnitude of the incremental loads.
1006	Elastic reanalysis attempted with nonzero displacement boundary conditions and boundary condition enforcement by row/column elimination. This is not possible. Remove the APPBC parameter.
1009	Error encountered in stress recovery. The output reveals which element has a particular problem. The error is usually due to excessive deformation in the element. This can occur during the iterative process, so that it is not always possible to visualize the excessive deformation. Check the material behavior and the magnitude of the incremental loads.
1021	Error in adding fluid mass to node. Marc system error; consult MSC.Software Corporation analyst.
1030	Reference vector has zero length; cannot determine preferred orientation. Check input on ORIENTATION option.

Exit Number	Explanation
1031	Error while determining view factors in cavity. Make sure that node numbers defining cavity are in consecutive order in the input file.
1040	Maximum number of element groups exceeded while using EBE iterative solver. Increase the maximum number of groups allowed on the SOLVER option.
1041	Error in calculation of the SPLINE constants during the reading of the material data base.
1042	Material data base file could not be opened.
1043	Too many materials from data base used.
1044	Curves in material data base have too many points.
1045	Error in calculation of the SPLINE constants during the reading of the material data base. Total number of curves read from material data base is too large.

Appendix B Workspace Definition and the Sizing Option

Table B-1 FORTRAN File Units Used by the UNIX Version of Marc

File name	Unit	Description	Comments
jidname.log	0	Error message output unit	Except HP
jidname.t01	1	Formatted data file	Usually contains mesh
jidname.t02	2	OOC* solver scratch file	random access binary file
jidname.t03	3	ELSTO file	sequential access binary file
jidname.t04	4	Neutral plot file	sequential access binary file
jidname.dat	5	Formatted data input file	formatted FORTRAN file
jidname.out	6	Printed output file	formatted FORTRAN file
jidname.log	7	Error message output file	HP only
jidname.t08	8	New restart file	sequential access binary file
ridname.t08	9	Old restart file	sequential access binary file
jidname_i_.dat	10	Temporary input file after jump back recycling	formatted FORTRAN file
jidname.t11	11	OOC* solver scratch file	sequential access binary file
jidname.t12	12	OOC* solver scratch file	sequential access binary file
jidname.t13	13	OOC* solver scratch file	sequential access binary file
jidname.t14	14	OOC* solver scratch file	random access binary file
jidname.t15	15	OOC* solver scratch file	sequential access binary file
jidname.t16	16	New post file (FORTRAN file)	sequential access binary file
ridname.t16	17	Old post file (FORTRAN file)	sequential access binary file
jidname.t18	18	Formatted data file, optimization table	formatted FORTRAN file
jidname.fem	18	From Marc to 3-D mesher	formatted FORTRAN file
jidname.t19	19	New post file	formatted FORTRAN file
ridname.t19	20	Old post file	formatted FORTRAN file
jidname_j_.dat	21	Temporary input file when cut-back is used.	formatted FORTRAN file
jidname.t22	22	Subspace iteration scratch file	sequential access binary file
jidname.t23	23	Fluid-solid interaction file	sequential access binary file
pidname.t19	24	Heat data input file	formatted FORTRAN file
pidname.t16	25	Heat data input file (FORTRAN file)	sequential access binary file
sidname.t31	31	Substructure master data file	random access binary file
jidname.t32	32	Secant method file	sequential access binary file
jidname.t34	34	Neutral plot file	formatted FORTRAN file
sidname.t35	35	Substructure file	sequential access binary file
*OOC denotes Out-Of-Core solution.			

Table B-1 FORTRAN File Units Used by the UNIX Version of Marc (Continued)

File name	Unit	Description	Comments
sidname.t36	36	Substructure file	sequential access binary file
jidname.t41	41	New Post file – Domain Decomposition	sequential access binary
ridname.t42	42	Post file – Domain Decomposition	formatted FORTRAN file
jidname.t45	45	Optimization data file	formatted FORTRAN file
jidname.t46	46	Optimization scratch file	formatted FORTRAN file
jidname.trk	47	New particle tracking file	formatted FORTRAN file
ridname.trk	48	Old particle tracking file	formatted FORTRAN file
jidname.cntl	52	Dynamic control file	formatted FORTRAN file
jidname.seq	53	Sequence option	formatted FORTRAN file
jidname.rst	54	Load case data	formatted FORTRAN file
jidname.mesh	55	User supplied mesh	formatted FORTRAN file
jidname.feb	55	From 3-D mesher to Marc	formatted FORTRAN file
jidname.pass	56	Auto restart command line	formatted FORTRAN file
jidname.rms	57	2D outline file for remeshing	formatted FORTRAN file
jidname.domesh jidname.donemesh	59	Lock files indicating meshing status	formatted FORTRAN file
jidname.sts	67	Analysis progress reporting file	formatted FORTRAN file
USRDEF	98	Global default file	formatted FORTRAN file
EXITMSG	99	Exit messages	formatted FORTRAN file
*OOC denotes Out-Of-Core solution.			

Appendix F
Material Database

APPENDIX

F Material Database

To enter a new material into the database, the following steps would be performed.

Please use the name convention followed in Marc to be certain that Marc reads your material data file properly. Using the convention, Marc will exit with error if the file with the material name does not exist in the directory. The user-defined material data should use the prefix of `usr_` and the file extension for the flow stress data should be `.mat`. Eight characters are allowed after `usr_`. For example, if you name your material as `usr_material`, you will create one `use_material.mud` or `use_material.mfd` file for Mentat database and one flow stress data file, `usr_material.mat` for the analysis. You have to place the Mentat database file in *mentat/materials/directory.* You can put the flow stress file in your current job directory or in *marc/AF_flowmat* directory if you want to share it with other users. Once you have done the preparation for the material database, you can use it in setting up your finite element model file in the future.

Step 1

Collect all experimental data for the new material. This includes the following data:

Young's modulus
Poisson's ratio
mass density
instantaneous coefficient of thermal expansion
yield stress
conductivity
specific heat
latent heats (if applicable)

The above properties may be dependent on the temperature. For the yield stress, it is dependent upon the equivalent plastic strain and optionally with strain rate.

For the yield behavior of materials, the data must be in the form Cauchy (true) stress versus logarithmic (true) strain. Therefore, if your data is in any other form, it is necessary to convert the data. Make sure your material data is in a consistent set of units.

Step 2

Begin with a new database, use the `FILES>NEW` to clear out old data. Use `SAVE AS` to create a database with the material name you desire. Material name must be no more than eight characters.

Step 3

Use the MATERIAL and TABLE menus to define the material properties except for the flow stress. Enter a 1 for the initial yield stress.

Step 4

Save database for this material. This file will be called `usr_material.mfd`, or `usr_material.mud`.

Step 5

If you have the write permissions, move this database into `mentat/materials/`. Be sure that you don't overwrite a material that has been previously defined.

Step 6

Use the editor to define a file for the flow stress. This file should be called `usr_material.dat`.

The structure of this file is as follows:

Data structure in x_xxxxcc.dat

1. data card: Material name beginning in column one with the material identification number

   Format : character*40
   Example : 3.2318 AlMgSi 1

2. data card: ncurves, npoints, ntemps, nerates, number of :
   where
   ncurves : number of curves in input (Max. 400)
   npoints : number of data points in each curve (Max. 100)
   ntemps : number of reference temperatures (Max. 20)
   nerates : number of reference strain rates (Max. 20)

   Format : four integer in free format
   Example : 30, 13, 5, 6

3. data card: eqpemin, eqpemax, equivalent plastic strain range of the material described in this input eqpemin must be = 0.0, logarithmic strain measure

   Format : two real in free format
   Example : 0.0, 7.0,

4. data card: eratmin, eratmax, equivalent plastic strain rate range of the material described in this input

   Format : two real in free format
   Example : 0.2, 10.0,

5. data card: eratmin, eratmax, Temperature range of the material described in this input

   Format : two real in free format
   Example : 350.0, 550.0,

The following data are repeated " ncurves " time (See card 2)

6. 0 data card : documentation text, character*80

6. 1 data card : icurve, temp., erate, sequential curve identification number, Temperature and equivalent plastic strain rate

6. 2 data card : eqpmin, eq_stress, logarithmic equivalent plastic strain and equivalent von mises (true) stress (first point)

6. n data card : eqpmax, eq_stress, logarithmic equivalent plastic strain and equivalent von mises (true) stress (npoint'th point, see card 2)

6.0 : Format : character*80
Example : === CURVE_01 Sig_Yiel, T=350. C, Eps_dot=0.2 1/s

6.1 : Format : one integer, two real in free format
Example : 1, 350.0, 0.20,

6.2 : Format : two real in free format
Example : 0.00, 78.0,

6.i : Format : two real in free format
Example : 11.75, 59.0,0,

6.n : Format : two real in free format
Example : 7.00, 52.0,

-
-
-

Step 7

Save file and move file into `marc/AF_flowmat/`.

Step 8

Your new material can now be read from:

`material properties>Read>Read other materials`.

Volume D: User Subroutines and Special Routines

Chapter 2
User-defined Loading Boundary Conditions and State Variables User Subroutines

USINC

Input of Initial Conditions

Description

This user subroutine allows you to input initial displacements, velocities,and accelerations for dynamic stress analysis, initial temperatures for heat transfer analysis, or thermal stress analysis, print temperature history for thermal stress analysis, or a spatially varying interference fit for contact analysis. You give the values for all degrees of freedom in vector F. This user subroutine is used with either the INITIAL DISP, INITIAL VEL, INITIAL TEMP model definition options, or the POINT TEMP model and history definition options. This user subroutine is called for every node in the structure if it is used.

Format

User subroutine USINC is written with the following headers:

```
SUBROUTINE USINC(F,N,NDEG,IFLAG)
IMPLICIT REAL *8 (A-H, O-Z)
DIMENSION F(NDEG)
   user coding
RETURN
END
```

where:

F	is the vector of initial conditions or point temperatures to be given by you.
N	is the node number.
NDEG	is the number of degrees of freedom per node.
IFLAG	is the type flag. = 1 initial displacement. = 2 initial velocities. = 3 initial temperatures. = 4 initial acellerations = 5 point temperatures (only for thermal stress analysis)

MOTION

Definition of Rigid Surface Motion for 2-D Contact

Description

This user subroutine allows the definition of nonuniform rigid surface motions, in conjunction with the CONTACT option. Its call is triggered by the model definition option UMOTION. This user subroutine should only be used with velocity controlled rigid surfaces.

User subroutine MOTION is called during the calculations at the beginning of each time increment and you return the surface velocities for that increment. Imposed displacement increments at nodal points in contact with rigid surfaces are obtained from the velocity multiplied by the time increment. The surface path becomes an explicit forward integration of velocities. Therefore, caution should be taken when there are abrupt changes in surface path direction or abrupt changes in velocity by making time increments as small as necessary.

CAUTION: The increment number (`inc`), which is passed from the calling routine, is useful in making sure the data given is appropriately attributed to the relevant increments. For example, a center of rotation given for increment 0

```
if(inc.eq.0) then
   x(1)=...
   etc.
endif
```

will be updated internally as motion and deformation take place. However, if the increment is not specified, this center of rotation will be reset each time this routine is read; that is, every increment. Obviously, the results will be different for the two cases.

The same caution is also applicable to the surface number (`nsurf`). Thus, the data should be coupled only with the surface number(s) for which it is meant. For example:

```
if(nsurf.eq.2 .or. nsurf.eq.4) then
   x(1)=...
   ...
   v(1)=...
   etc.
endif
```

Important: The correct use of increment numbers and surface numbers may give rise to more complex (or nested) `if` statements. For example:

```
if(inc.eq.2 .and. nsurf.eq.3) then
   ...
elseif(....) then
   if(....) then
      ...
   endif
...
endif
```

If, at the start of the analysis, a surface is placed apart from the body to be deformed, user subroutine MOTION is also used in the approaching phase.

If two-dimensional elements are being used, the surfaces have rigid body motions in two dimensions. It is assumed that such motions can be defined by a translation of a point (the center of rotation), plus a rotation around that point.

Format

User subroutine MOTION is written with the following headers:

```
SUBROUTINE MOTION (X,F,V,TIME,DTIME,NSURF,INC)
IMPLICIT REAL *8 (A-H, O-Z)
DIMENSION X(*),V(*),F(*)
    user coding
RETURN
END
```

where:

Input:

`X(3)`	is the array of current die defining coordinates. `X(1)` = first coordinate of center of rotation. `X(2)` = second coordinate of center of rotation. `X(3)` = angle rotated around z-axis.
`F(3)`	is the array of current surface loads. `F(1)` = first component of load. `F(2)` = second component of load. `F(3)` = moment.
`TIME`	is the time at which data is requested.
`DTIME`	is the current time increment.
`NSURF`	is the surface number for which data is requested.
`INC`	is the increment number.

Output:

V(3)	is the array of current surface velocities. V(1) = first component of the velocity at the center of rotation. V(2) = second component of the velocity at the center of rotation. V(3) = angular velocity.

Example

Assume that a rigid surface is identified as surface number 1, and is moving in the negative x-direction with a velocity of 1.0. The user subroutine MOTION can be written as follows:

```
SUBROUTINE MOTION(X,F,V,TIME,DTIME,NSURF,INC)
IMPLICIT REAL *8 (A-H, O-Z)
DIMENSION X(*),V(*),F(*)
IF(NSURF.EQ.1) THEN
   V(1)=-1.
   V(2)=0.
   V(3)=0.
ENDIF
RETURN
END
```

MOTION

Definition of Rigid Surface Motion for 3-D Contact

Description

This user subroutine allows the definition of nonuniform rigid surface motions in conjunction with the CONTACT option. Its call is triggered by the model definition option UMOTION. This user subroutine should only be used with velocity controlled rigid surfaces.

User subroutine MOTION is called during the calculations at the beginning of each time increment and your return surface velocities for that increment. Imposed displacement increments at nodal points in contact with rigid surfaces are obtained from the velocity multiplied by the time increment. The surface path becomes an explicit forward integration of velocities. Therefore, caution should be taken when there are abrupt changes in surface path direction or abrupt changes in velocity by making time increments as small as necessary.

CAUTION: The increment number (`inc`), which is passed from the calling routine, is useful in making sure the data given is appropriately attributed to the relevant increments. For example, a center of rotation given for increment 0

```
if(inc.eq.0) then
   x(1)=...
   etc.
endif
```

will be updated internally as motion and deformation take place. However, if the increment is not specified, this center of rotation will be reset each time this routine is read; that is, every increment. Obviously, the results will be different for the two cases.

The same caution is also applicable to the surface number (`nsurf`). Thus, the data should be coupled only with the surface number(s) for which it is meant. For example:

```
if(nsurf.eq.2 .or. nsurf.eq.4) then
   x(1)=...
   ...
   v(1)=...
   etc.
endif
```

Important: The correct use of increment numbers and surface numbers may give rise to more complex (or nested) `if` statements. For example:

```
if(inc.eq.2 .and. nsurf.eq.3) then
   ...
elseif(....) then
   if(....) then
      ...
   endif
...
endif
```

If, at the start of the analysis, a rigid surface is placed apart from the deformable body, user subroutine MOTION is also used in the approaching phase.

If three-dimensional elements are used, the surfaces have rigid body motions in three dimensions. It is assumed that such motions can be defined by a translation of a point (the center of rotation), plus a rotation about the axis of rotation through that point.

Format

User subroutine MOTION is written with the following headers:

```
SUBROUTINE MOTION (X,F,V,TIME,DTIME,NSURF,INC)
IMPLICIT REAL *8 (A-H, O-Z)
DIMENSION X(*),V(*),F(*)
    user coding
RETURN
END
```

where:

Input:

`X(6)`	is the array of current die defining coordinates. `X(1)` = first coordinate of center of rotation. `X(2)` = second coordinate of center of rotation. `X(3)` = third coordinate of center of rotation. Axis for specifying angular velocity: `X(4)` = first component of direction cosine. `X(5)` = second component of direction cosine. `X(6)` = third component of direction cosine.
`F(6)`	is the array of current surface loads. `F(1)` = first component of load. `F(2)` = second component of load. `F(3)` = third component of load. `F(4)` = first component of moment . `F(5)` = second component of moment. `F(6)` = third component of moment.

TIME	is the time at which data is requested.
DTIME	is the current time increment.
NSURF	is the surface number for which data is requested.
INC	is the increment number.
Output:	
V(4)	is the array of current surface velocities. V(1) = first component of the velocity at the center of rotation. V(2) = second component of the velocity at the center of rotation. V(3) = angular velocity. V(4) = angular velocity around axis defined above with X(4), X(5), and X(6).

Example

Assume that a rigid surface is identified as surface number 2 and is moving in the negative x-direction with a velocity of 1.0. The user subroutine MOTION can be written as follows:

```
SUBROUTINE MOTION(X,F,V,TIME,DTIME,NSURF,INC)
IMPLICIT REAL *8 (A-H, O-Z)
DIMENSION X(*),V(*),F(*)
IF(NSURF.NE.2) THEN
   V(1)=-1.
   V(2)=0.
   V(3)=0.
   V(4)=0.0
ENDIF
RETURN
END
```

UHTCOE

Definition of Environment Film Coefficient

Description

This user subroutine allows the definition of variable film coefficients and sink temperatures of free surfaces, in conjunction with the CONTACT option and COUPLE parameter. Its call is triggered by the UHTCOEF option.

User subroutine UHTCOE is called at every element surface containing nodes that are on a free body boundary and for each surface at the trapezoidal rule integration points (that is, the nodes). These calls are made every iteration both during the assembly phase and the recovery phase of the heat transfer pass of a coupled analysis.

A distributed heat flux is being calculated according to the equation:

$$q = H(T - TS)$$

where:

q	is the heat flux entering the surface.
T	is the surface temperature.
TS	is the sink temperature.
H	is the film coefficient.

By modifying H and TS, you can model varying heat transfer conditions along the boundary. Special attention has been given to provide you the capability of simulating radiation heat transfer, by making available the location and temperatures of all the surfaces in the environment.

You can either specify H and TS or specify the flux q directly which is treated strictly as such.

Format

User subroutine UHTCOE is written with the following headers:

```
      SUBROUTINE UHTCOE(MIBODY,XP,TEMP,IBODY,ICONNO,XORD,NVXORD,XT,
     +NVXT,DXT,NVDXT,TMPALL,NVTMP,TMPALO,NVTMPO,TOTINC,TIMINC,INC,
     +NCRD,NDEG,NDEGH,NCRDMX,NDEGMX,NBCD,NBCN,TSINK,HTCOEF,IFLAG)
      IMPLICIT REAL *8 (A-H, O-Z)
      DIMENSION MIBODY(*),XP(*),ICONNO(NBCN,*),XORD(*),XT(*),DXT(*),
     +TMPALL(*),NBCD(*), TMPALO(*)
           user coding
      RETURN
      END
```

where:

MIBODY(1)	is the user element number for where the surface flux is being calculated.
MIBODY(2)	is the side of said element.
MIBODY(3)	is the integration point of said side.
MIBODY(4)	is the internal element number.
XP(NCRD)	are the coordinates of point where calculation is being made, updated to end of increment.
TEMP(2)	is the temperature of said point.
IBODY	is the flexible surface to which point belongs.
ICONNO(*)	is the nodal points that make the boundary of deformable surfaces declared in the CONTACT option.
NBCN	is the upper bound to the number of nodes on a flexible surface boundary.
NBCD(*)	is the array of actual number of boundary nodes on flexible surfaces.
XORD(*)	is the array of original nodal point coordinates.
NVXORD	is the Marc coordinate vector number.
XT(*)	is the array of nodal point displacements.
NVXT	is the Marc displacement vector number.
DXT(*)	is the array of nodal displacement increments.
NVDXT	is the Marc displacement increment vector number.
TMPALL(*)	is the array of nodal temperatures (current estimate at end of increment).
NVTMP	is the Marc temperature vector number.
TMPALO(*)	is the array of nodal temperatures (at beginning of increment).
NVTMPO	is the Marc temperature vector number.
TOTINC	is the current accumulated time.

TIMINC	is the time increment.
INC	is the increment number.
NCRD	is the number of coordinates per node.
NDEG	is the number of degrees of freedom per node.
NDEGH	is the number of degrees of freedom for temperature (usually 1).
NCRDMX	= NCRD unless different element types are in the model.
NDEGMX	= NDEG unless different element types are in the model.
TSINK	is the sink temperature declared in contact option for this flexible surface.
IFLAG	=0 HTCOEF is a heat transfer coefficient =1 HTCOEF is a flux.
HTCOEF	is the heat transfer coefficient between surface and environment, such that the heat flux per unit area that leaves the surface is: Q = HTCOEF (TEMP - TSINK) or the heat flux per unit area that leaves the surface. HTCOEF is to be defined here.

Chapter 3 User-defined Anistropy and Constitutive Relations User Subroutines

Table 3-1 User-defined Anisotropy and Constitutive Relations User Subroutine Requirements

User Subroutine	Required Parameters or Model Definition Options	Purpose
ANELAS	ORTHOTROPIC or ANISOTROPIC	Definition of factors to scale elastic stress strain law.
ANEXP	ORTHOTROPIC or ANISOTROPIC	Definition of thermal strain increment.
ANKOND	ORTHOTROPIC or ANISOTROPIC	Definition of thermal conductivity or electrical resistance in Joule heating.
ANPLAS	ORTHOTROPIC or ANISOTROPIC	Definition of parameters for Hill yield criteria
CRPLAW	CREEP	Definition of function to describe creep strain rate.
GAPU	GAP DATA	Definition of contact gap closure distance
GENSTR	SHELL SECT	Definition of generalized stress-strain law for shells.
HOOKLW	ORTHOTROPIC or ANISOTROPIC	Definition of elastic stress-strain or compliance relation.
HYPELA	HYPOELASTIC	Definition of nonlinear stress-strain relationship.
HYPELA2	HYPOELASTIC	Definition of nonlinear stress-strain relationship.
ORIENT	ORIENTATION	Definition of preferred material orientation for orthotropic or anisotropic behavior.
TENSOF	ISOTROPIC CRACK DATA	Definition of tension softening modulus.
UACOUS	ACOUSTIC CONTACT (2-D) CONTACT (3-D)	Definition of material properties for an acoustic medium.
UARRBO	ARRUDBOYCE	Definition of constants in strain energy function.
UBEAM	HYPOELASTIC	Definition of nonlinear generalized stress-strain law for element types 52 or 98.
UCOMPL	HARMONIC	Definition of stress-strain rate relationship for harmonic analysis.
UCRACK	ISOTROPIC CRACK DATA	Definition of ultimate stress for cracking analysis.
UDAMAG	DAMAGE	Definition of the Kachanov damage factor to be applied to the material properties
UELDAM	OGDEN DAMAGE	Definition of damage parameters for Ogden rubber model.
UENERG	MOONEY	Definition of strain energy function.
UEPS	ELECTRO or EL-MA ORTHOTROPIC	Definition of anisotropic electrical permittivity.
UFAIL	FAIL DATA	Definition of composite failure criteria.

Table 3-1 User-defined Anisotropy and Constitutive Relations User Subroutine Requirements (Continued)

User Subroutine	Required Parameters or Model Definition Options	Purpose
UFINITE	PLASTICITY or ELASTICITY	Definition of finite deformation isotropic material models.
UGENT	ARRUDBOYCE	Definition of constants in strain energy function.
UGRAIN	GRAIN SIZE	Definition of typical grain size calculation based upon the state of material
UMOONY	MOONEY	Definition of temperature dependent Mooney-Rivlin constants.
UMU	MAGNETO or EL-MA ORTHOTROPIC	Definition of anisotropic magnetic permeability.
UNEWTN	R-P FLOW or FLUID	Definition of material viscosity.
UOGDEN	OGDEN	Definition of Ogden material parameters.
UPERM	PORE	Definition of soil permeability.
UPHI	HARMONIC MOONEY PHI-COEFFICIENTS	Definition of phi coefficients for rubber-viscoelastic harmonic analysis.
UPOWDR	POWDER	Definition of powder material data.
UPSTRECH	ODGEN	Definition of generalized principal stretch based elasticity models.
URPFLO	R-P FLOW	Definition of yield surface for rigid plastic flow.
USELEM	USER	Definition of consistent nodal loads, mass matrix, stiffness matrix, and residuals for user-defined element.
USHRET	ISOTROPIC CRACK DATA	Definition of shear retention factor for elements that have cracks.
USIGMA	EL-MA	Definition of anisotropic electrical conductivity.
USPCHT	HEAT or COUPLE or FLUID	Definition of specific heat.
USPRNG	SPRINGS or FOUNDATION	Definition of nonlinear spring or foundation stiffness.
USSUBS	SUPER SUPERINPUT	Definition of superelements not generated by Marc.
UVOID	DAMAGE	Definition of initial void fraction for Gurson damage model.

Table 3-1 User-defined Anisotropy and Constitutive Relations User Subroutine Requirements (Continued)

User Subroutine	Required Parameters or Model Definition Options	Purpose
UVOIDN	DAMAGE	Definition of void nucleation for Gurson damage model.
VSWELL	CREEP	Definition of volumetric swelling.
WKSLP	ISOTROPIC or ORTHOTROPIC or ANISOTROPIC WORK HARD	Definition of work hardening or strain hardening data.

USPRNG

Input of Nonlinear Spring, Dashpot and Foundation Stiffness

Description

User subroutine USPRNG permits the introduction of nonlinear spring constants for use with the SPRINGS and/or FOUNDATION options and input of nonlinear damping if the DASHPOT option is to be used. Your coding must supply both the ratio of the current value of spring stiffness to the data input value and the total spring force. For dynamic analysis, the ratio of damping coefficient can also be provided. The data value of the spring/dashpot constant, total time, and the element or spring number are made available to the user subroutine. For harmonic analysis, the spring/dashpot constants can be a function of the frequency. User subroutine USPRNG is accessible whenever either the SPRINGS or the FOUNDATION option is used.

Format

User subroutine USPRNG is written with the following headers:

```
SUBROUTINE USPRNG(RATK,F,DATAK,U,TIME,N,NN,NSPRNG)
IMPLICIT REAL *8 (A-H, O-Z)
DIMENSION RATK(2),DATAK(2),U(2),TIME(2),N(2),F(2)
    user coding
RETURN
END
```

where:

RATK(1)	is the ratio of the present value of spring stiffness to the data value given in the option input; to be defined by you.
RATK(2)	is the ratio of the present value of the damping coefficient to the data value given in the input; to be defined by you. This applies to SPRINGS in dynamic analysis only.
F(1)	is the force to be defined by you. (a) For springs: F(1) = spring force. (b) For elastic foundation: F(1) = pressure per unit area. (c) For harmonics: F(1) = real part of harmonic force.
F(2)	(a) For springs: F(2) = the damping force. (b) For harmonics: F(2) = imaginary part of harmonic force.
DATAK(1)	is the data value of spring constant (or foundation stiffness) as defined by you in SPRINGS/FOUNDATION options data input.

DATAK(2)	is the data value of the damping constant as defined by you in the SPRINGS option data input.
For elastic foundation (only static contribution):	
U(1)	For elastic foundation: U(1) = U_n. (positive in the direction specified by face identification given in the FOUNDATION option).
U(2)-U(4)	not used
For springs/dashpots (static and/or dynamic contribution):	
U(1)	For springs: $U(1) = U_2 - U_1$.
U(2)	For dynamic spring/dashpot $U(2) = \dot{U}_2 - \dot{U}_1$.
U(3)-U(4)	not used
For springs/dashpots (harmonic analysis):	
U(1)	$U(1) = U_2 - U_1$ static predeformation
U(2)	not used
U(3)	$U(3) = U_2 - U_1$ real part of harmonic deformation.
U(4)	$U(4) = U_2 - U_1$ imaginary part of harmonic deformation
TIME(1)	is the total time (for dynamic or creep analysis).
TIME(2)	is the frequency (for harmonic analysis with spring/dashpot).
N(1)	is the element number (only for elastic foundation).
N(2)	is the face number (only for elastic foundation).
NN	is the integration point number (only for elastic foundation).
NSPRNG	is the spring number, the position of the spring in the input data list (only for springs).

If the user subroutine is called for an elastic foundation point, NSPRNG is zero.

If the user subroutine is called for a spring, N and NN are zero.

Note that if you prefer to give the absolute value of the spring constant rather than a ratio, the corresponding value in the SPRINGS or FOUNDATION option should be set to 1. The same applies for a damping constant.

UVOIDN

Definition of the Void Nucleation Rate

Description

This user subroutine allows the definition of the void nucleation rate in a material using the Gurson model. This user 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

User subroutine UVOIDN is written with the following headers:

```
      SUBROUTINE UVOIDN(A,B,M,NN,KC,MATS,EPL,EPLAS,S,NDI,NSHEAR,DT,
     +DTDL)
      IMPLICIT REAL *8 (A-H, O-Z)
      DIMENSION M(2),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 your element number.
M(2)	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 user subroutine, the following type of stress controlled nucleation rate can be specified:

$$\dot{f} = A\dot{\bar{\sigma}} + B\frac{\dot{\sigma}_{kk}}{3}$$

where $\dot{\bar{\sigma}}$ is the von Mises equivalent stress rate, and $\dot{\sigma}_{kk}$ is the hydrostatic stress rate.

UDAMAG

Prediction of Material Damage

Description

This user subroutine provides you with the mechanism for providing a Kachanov damage factor to be applied to the material properties. UDAMAG is used in conjunction with the DAMAGE model definition option. You define the damage factor (df). $0 \le df \le 1$ where $df = 0$ implies a fully damaged material. If model 9 is used, then:

$$\sigma_y = \sigma_y(\bar{\varepsilon}^p, \dot{\bar{\varepsilon}}^p, T)*(1.0 - df)$$

If model 10 is used, then:

$$\sigma_y = \sigma_y(\bar{\varepsilon}^p, \dot{\bar{\varepsilon}}^p, T)*(1.0 - df) \quad \text{and} \quad E = E(T)*(1.0 - df)$$

Format

User subroutine UDAMAG is written with the following header lines:

```
 SUBROUTINE UDAMAG(M,N,NN,KC,MATS,EPLAS,ERATE,DT,DTDl,
*DAMDAT,DAMFAC,TIME,DELTIME)
 IMPLICIT REAL*8 (A-H,O-Z)
 DIMENSION DT (*),DAMDAT(*)
    user coding
 RETURN
 END
```

where:

`M`	is the element number.
`N`	is the elsto number.
`NN`	is the integration point number.
`KC`	is the layer.
`MATS`	is the material id.
`EPLAS`	is the equivalent plastic strain.
`ERATE`	is the equivalent plastic strain rate.
`DT`	is the state variables at beginning of increment.
`DTDL`	is the incremental state variables.
`DAMDAT`	is the material data, `DAMDAT (1)` is the initial damage factor.

DAMFAC	is the current damage factor.
TIME	is the time - beginning of increment.
DELTIME	is the incremental time.

USELEM

User-defined Element

Description

This user subroutine allows you to calculate your own finite element stiffness or mass matrix. This can 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 user subroutine is called a multiple number of times. The calls and your requirements are defined as follows:

`IFLAG=1`	Return the equivalent nodal loads (`F`) given distributed surface or body loads. If the ELASTIC, FOLLOW FOR parameters or the AUTO STEP, AUTO TIME, AUTO INCREMENT 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 parameter 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, you do not need to return any values.
`IFLAG=5`	Output element results if so desired.

To use this option, the USER parameter 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

User subroutine USELEM calls for the following headers:

```
      SUBROUTINE USELEM(M,XK,XM,NNODE,NDEG,F,R,
     * JTYPE,DISPT,DISP,NDI,NSHEAR,IPASS,NSTATS,NGENEL,
     * INTEL,COORD,NCRD,IFLAG,IDSS,T,DT,ETOTA,GSIGS,DE,
     * GEOM1,GEOM2,GEOM3,GEOM4,GEOM5,GEOM6, SIGXX, NSTRMU)
      IMPLICIT REAL *8 (A-H, O-Z)

      DIMENSION
     XK(IDSS,IDSS),XM(IDSS,IDSS),DISPT(NDEG,*),DISP(NDEG,*)
      DIMENSION T(NSTATS,*),DT(NSTATS,*),COORD(NCRD,*)
      DIMENSION ETOTA(NGENEL,*),GSIGS(NGENEL,*),DE(NGENEL,*)
      DIMENSION F(NDEG,*),R(NDEG,*),SIGXX(NSTRMU,*)
       user coding
      RETURN
      END
```

where:

M	is your 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. =0 during an uncoupled analysis. =1 during a stress analysis pass. =2 during a heat transfer pass.
NSTATS	is the number of state variables.
NGENEL	is the number of generalized strains.
INTEL	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 you. =1 Called by OPRESS during formation of load vector. You return F. =2 Called by OASEMB during formation of stiffness matrix. You return XK,R. =3 Called by OASMAS during formation of mass matrix. You return XM. =4 Called by OGETST during stress recovery. You return R,GSIGS,DE,ETOTA,SIGXX. =5 Called by SCIMP during output phase. You print 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.
NSTRMU	is the number of stresses per integration points.

Note that the stiffness matrix is normally symmetric. If a nonsymmetric formulation is used, the SOLVER option should be used to indicate this.

USSUBS

Superelements Not Generated by Marc

Description

The user subroutine USSUBS can be used to enter stiffness matrix, mass matrix, conductivity matrix, capacity matrix, load vector, internal force vector, or output for superelements not generated by MSC.Marc.

The number of superelements and the dimension are given via SUPER parameter and the connectivity of the superelements is given via the SUPERINPUT model definition option.

Format

User subroutine USSUBS is written with the following headers:

```
      SUBROUTINE USSUBS(NLEVl,NSS,IC,NODSUB,NDEG,LMI,LM,
     *                  TIME,TIMINC,INC,IPASS,
     *                  XLOAD,XDISP,XRESI,XMAT)
      IMPLICIT REAL*8 (A-H,O-Z)
      DIMENSION LMI(NODSUB),LM(NODSUM)
      DIMENSION XLOAD(NDEG,NODSUB),XDISP(NDEG,NODSUB),
                XRESI(NDEG,NODSUB)
      DIMENSION XMAT(NDEG*NODSUB,NDEG*NODSUB)

         user coding

     RETURN
     END
```

where:

NLEV	is the superelement level = 1.
NSS	is the superlelement number.
IC	= 1 : return XLOAD (XDISP, XRESI, XMAT not filled) = 2 : return XMAT (stiffness matrix) (XLOAD, XDISP, XRESI not filled) = 3 : return XRESI or if IC is reset to -3 return XMAT, the program will calculate XRESI=XMAT*XDISP XLOAD, MXAT not filled XDISP filled = 4 : output phase XLOAD, XRESI, XMAT not filled XDISP filled = 5 : return Xmat (mass matrix) XLOAD, XDISP, XRESI not filled = 6 : return XMAT (damping matrix) XLOAD, XDISP, XRESI not filled

NODSUM	is the number of nodes in the superelement.
NDEG	is the number of degrees of freedom per node in the superelement.
LIM()	is the node id's of the superelement.
LM()	is the future expansion.
TIME	is the transient time at the start of the increment.
TIMINC	is the incremental time period.
INC	is the increment number.
IPASS	0 - not a coupled analysis. 1 - coupled analysis, stress pass 2 - coupled analysis, heat transfer pass
XLOAD()	is the external load vector on the superelement; this is total external force.
XDISP()	is the displacement of the superelement.
XRESI()	in the internal force vector for the superelement XRESI-XMAT*XDISP (if linear)
XMAT()	is the stiffness, mass, damping matrix of the superelement.

Chapter 6
Geometry Modifications User Subroutines

UACTIVE

Activate or Deactivate Elements

Description

The user subroutine UACTIVE can be used to either activate or deactivate elements in the model. The user subroutine is called at the beginning of the analysis and at the end of each increment. A deactivated element does not contribute to the load, mass, stiffness, or internal force calculation. If an element is activated after previously being deactivated, you can specify if the material is to come back in its previous state or in a modified state.

Format

User subroutine UACTIVE is written with the following headers:

```
SUBROUTINE UACTIVE(M,N,MODE,IRSTSTR,IRSTSTN,INC,TIME,TIMINC)
IMPLICIT REAL*8 (A-H,O-Z)
DIMENSION M(2)
    user coding
RETURN
END
```

where:

M(1)	is the element number.
M(2)	is the master element number in an adaptive analysis
NN	is the internal element number.
MODE	-1 deactivate element. 2 leave in current status. 1 activate element.
IRSTSTR	set to 1 to reset stresses to zero.
IRSTSTN	set to 1 to reset strains to zero.
INC	is the increment number.
TIME	is the time at the beginning of the increment.
TIMINC	is the incremental time.

Chapter 7
Output Quantities User Subroutines

PLOTV

User-selected Postprocessing of Element Variables

Description

User subroutine PLOTV is used in conjunction with either element code 19 or a negative code entered in the POST option. This allows you to define an element variable to be written to the post file.

Format

User subroutine PLOTV is written with the following headers:

```
 SUBROUTINE PLOTV(V,S,SP,ETOT,EPLAS,ECREEP,T,M,NN,LAYER,NDI,
+NSHEAR,JPLTCD)
 IMPLICIT REAL *8 (A-H, O-Z)
 DIMENSION S(1),SP(1),ETOT(1),EPLAS(1),ECREEP(1),M(2)
  user coding
 RETURN
 END
```

where:

V	is the variable to be plotted or put onto the post file, to be defined in this routine.
S	is the array of stresses at this integration point. For heat transfer analysis, S contains $\partial T/\partial X_i$ and $K_i \partial T/\partial X_i$. For a magnetostatic analysis, S contains the magnetic induction (B) (positions 1, 2, 3 for x, y,z) and the magnetic field intensity (H) (positions 5, 6, 7 for x, y, z).
SP	is the array stresses in the preferred direction if ORIENTATION is used.
ETOT	is the total strain (generalized) at this integration point.
EPLAS	is the total plastic strain at this integration point.
ECREEP	is the total creep strain at this integration point.
T	is the temperature at this integration point.
M(1)	is your element number.
M(2)	is the internal element number.
NN	is the integration point number.
LAYER	is the layer number (for beams or shells).

NDI	is the number of direct stresses.
NSHEAR	is the number of shear stresses.
JPLTCD	is the absolute value of your entered code.

Example

For example, suppose you wish to output the sum of the squares of the two shear stresses in the friction theory. These are S(2) and S(3), so the user subroutine would appear as:

```
 SUBROUTINE
PLOTV(V,S,SP,ETOT,EPLAS,ECREEP,T,M,NN,LAYER,NDI,NSHEAR,
+JPLTCD)
 IMPLICIT REAL *8 (A-H, O-Z)
 DIMENSION S(1),SP(1),ETOT(1),EPLAS(1),ECREEP(1)
 V=SQRT(S(2)**2 + S(3)**2)
 RETURN
 END
```

This quantity could then be postprocessed using MSC.Marc Mentat (Mentat).

For electromagnetics, the PLOTV variables are:

V	is the variable.
ERI	are the real and imaginary components of the electric field intensity.
DRI	are the real and imaginary components of the electric displacement.
BRI	are the real and imaginary components of the magnetic induction.
HRI	are the real and imaginary components of the magnetic field intensity.
CRI	are the real and imaginary components of the current density.
T	is the current temperature; not used.
M	is the element number.
NN	is the integration point number.
LAYER	is the layer number = 2.
NDI	is the number of components = 3
NSHEAR	is not used.
JPLTCD	is the absolute value of your post code.

Chapter 9 Special Routines — Marc Post File Processor

```
BLOCK 514nn - contact geometry data
 **********************************

 if(ndie.gt.0) then

   read(formatted,'(a70)') blkbegin
    do ijk=1,ndie
      if(ipstk2.lt.8) then
       read(formatted,'(3i13)') ibody,itype,nitems
      else
       read(formatted,'(4i13)') ibody,itype,nitems,istruc
       read(formatted,'(a24)') bdname
       read(formatted,(6e13.6)')
(pos(ij1),ij1-1,3),(rot(ij1),ij1=1,3)
      endif
      if(itype.ne.0.or.ipstk2.ge.8) then
       if(ibody.eq.0) then
        read(formatted,'(i13)') nelem
        read(formatted,'(6i13)') (ielem(ijl),ijl=1,nelem)
       endif
       if(ibody.eq.1) then
        do ijl=1,nitems
         read(formatted'(2i13)') npatch,npoint
         do ijm=1,npatch
          read(formatted'(4i13)') ipatn,ipatt,ip1,ip2
         enddo
         do ijm=1,npoint
          read(formatted'(i13,2e13.6)') ipoint,xp,yp
         enddo
        enddo
       endif
       if(ibody.eq.2) then
        do ijl=1,nitems
         read(formatted'(2i13)') npatch,npoint
         do ijm=1,npatch
          read(formatted'(6i13)') ipatn,ipatt,ip1,ip2,ip3,ip4
         enddo
         do ijm=1,npoint
          read(formatted'(i13,3e13.6)') ipoint,xp,yp,zp
         enddo
        enddo
       endif
       if(ibody.eq.3) then
        do ijl=1,nitems
         read(formatted,'(6i13)') nurbid,kpt,idum3,kor,idum5,idum6
         do ijm=1,kpt
          read(formatted,'(3e13.6)') xp,yp,zp
         enddo
         read(formatted,'(6e13.6)') (homo(ijm),ijm=1,kpt)
         read(formatted,'(6e13.6)') (xnot(ijm),ijm=1,kpt+kor)
        enddo
       endif
```

```
        if(ibody.eq.4) then
         do ijl=1,nitems
          read(formatted,’(6i13)’) nurbid,nptu,nptv,noru,norv,itrim
          do ijm=1,nptu*nptv
           read(formatted,’(3e13.6)’) xp,yp,zp
          enddo
          read(formatted,’(6e13.6)’) (homo(ijm),ijm=1,nptu*nptv)
          read(formatted,’(6e13.6)’)
(xnot(ijm),ijm=1,nptu+noru+nptv+norv)
          do ijm=1,itrim
           read(formatted,’(6i13)’)
itriid,kpt,idum3,idum4,idum5,idum6
           do ijl=1,kpt
            read(formatted,’(3e13.6)’) xp,yp,zp
           enddo
          enddo
         enddo
        endif
       endif
     enddo
    read(formatted,’( a5)’) blkend

    read(binary) (ibeg(ijk),ijk=1,70)
    write(blkbegin,’(70a1)’) blkbegin
     do ijk=1,ndie
       if(ipstk2.lt.8) then
        read(binary) ibody,itype,nitems
       else
        read(binary) ibody,itype,nitems,istruc
        read(binary) (ibdname(ij1),ij1=1,24)
        write(bdname,’(24a1)’) (ibdname(ij1),ij1=1,24)
        read(binary) (pos(ij1),ij1=1,3),(rot(ij1),ij1=1,3)
       endif
       if(itype.ne.0.or.ipstk2.ge.8) then
        if(ibody.eq.0) then
         read(binary) nelem
         read(binary) (ielem(ijl),ijl=1,nelem)
        endif
        if(ibody.eq.1) then
         do ijl=1,nitems
          read(binary) npatch,npoint
          do ijm=1,npatch
           read(binary) ipatn,ipatt,ip1,ip2
          enddo
          do ijm=1,npoint
           read(binary) ipoint,xp,yp,zp
          enddo
         enddo
        endif
        if(ibody.eq.2) then
         do ijl=1,nitems
          read(binary) npatch,npoint
```

```
          do ijm=1,npatch
           read(binary) ipatn,ipatt,ip1,ip2,ip3,ip4
          enddo
          do ijm=1,npoint
           read(binary) ipoint,xp,yp,zp
          enddo
         enddo
        endif
        if(ibody.eq.3) then
         do ijl=1,nitems
          read(binary) nurbid,kpt,idum3,kor,idum5,idum6
          do ijm=1,kpt
           read(binary) xp,yp,zp
          enddo
          read(binary) (homo(ijm),ijm=1,kpt)
          read(binary) (xnot(ijm),ijm=1,kpt+kor)
         enddo
        endif
        if(ibody.eq.4) then
         do ijl=1,nitems
          read(binary) nurbid,nptu,nptv,noru,norv,itrim
          do ijm=1,nptu*nptv
           read(binary) xp,yp,zp
          enddo
          read(binary) (homo(ijm),ijm=1,nptu*nptv)
          read(binary) (xnot(ijm),ijm=1,nptu+noru+nptv+norv)
          do ijm=1,itrim
           read(binary) itriid,kpt,idum3,idum4,idum5,idum6
           do ijl=1,kpt
            read(binary) xp,yp,zp
           enddo
          enddo
         enddo
        endif
       endif
     enddo

endif

blkbegin = =beg=51400 (Contact Geometry Data)
ndie     = number of contact bodies (from BLOCK 502nn)
ibody    = number of body ijk
itype    = type of body ijk
           0 : deformable
           1 : 2d line elements (type   9)
           2 : 3d patch elements (type 18)
           3 : 2d curves
           4 : 3d surfaces
nitems   = number of entities in body ijk
istruc   = physical meaning of body ijk
           1 : rigid
           2 : deformable structural
```

```
            3 : symmetry
            4 : deformable heat-rigid
            5 : workpiece (Autoforge only)
            6 : deformable acoustic
bdnam    = name of body ijk
pos(i)   = position of center of body ijk
rot(i)   = rotation vector for body ijk
nelem    = number of elements in deformable body ijk
ielem(i) = user element numbers of deformable body ijk
npatch   = number of patches in body ijk entity ijl
npoint   = number of points  in body ijk entity ijl
ipatn    = patch number
ipatt    = patch type (9=line,18=surface)
ip1      = first  node of patch
ip2      = second node of patch
ip3      = third  node of patch
ip4      = fourth node of patch
ipoint   = point number
xp,yp,zp = x-, y- and z-coordinates of point
nurbid   = identifier of NURBS
kpt      = number of points for NURBS curve
kor      = order of NURBS curve
nptu     = number of points in u-direction for NURBS surface
nptv     = number of points in v-direction for NURBS surface
noru     = order of NURBS surface in u-direction
norv     = order of NURBS surface in v-direction
itrim    = number of trimming curves of NURBS surface
homo(i)  = homogeneous coordinates
xnot(i)  = knot vectors
itriid   = identifier of trimming curve of NURBS surface
blkend   = =end=
```

```
BLOCK 517nn - integer increment verification data
 ************************************************

   read(formatted,'(a70)') blkbegin
    read(formatted,'(6i13)') (lm(ijk),ijk=1,12)
   read(formatted,'( a5)') blkend

   read(binary) (ibeg(ijk),ijk=1,70)
   write(blkbegin,'(70a1)') blkbegin
    read(binary) (lm(ijk),ijk=1,12)
   read(binary) (iend(ijk),ijk=1,5)
   write(blkbegin,'(5a1)') blkend

 blkbegin = =beg=51701 (Integer Increment Verification Data)
 lm( 1)   = remeshing flag                    (newmo)
            0 : same mesh as before
            1 : new mesh
 lm( 2)   = increment number                  (inc)
 lm( 3)   = sub-increment number              (incsub)
 lm( 4)   = analysis type                     (jantyp)
           > 100 element variables are written for this increment
 lm( 5)   = number of nodal variables         (knod)
 lm( 6)   = number of design variables        (ndsvar)
 lm( 7)   = normal/harmonic/modal/buckle flag (ihresp)
            0 : normal
            1 : modal result
            2 : buckle result
            3 : real harmonic result
            4 : complex harmonic result
 lm( 8)   = number of recycles for this increment
 lm( 9)   = total number of separation recycles
 lm(10)   = total number of cutbacks
 lm(11)   = total number of increment splittings
 lm(12)   = not used
 blkend   = =end=
```

```
BLOCK 51800 - real increment verification data
 ********************************************

If post file revision number ≤ 9 (MARC 2000 and before)

   read(formatted,’(a70)’) blkbegin
    read(formatted,’(6e13.6)’) (xlm(ijk),ijk=1,6)
   read(formatted,’( a5)’) blkend

   read(binary) (ibeg(ijk),ijk=1,70)
   write(blkbegin,’(70a1)’) blkbegin
    read(binary) (xlm(ijk),ijk=1,6)
   read(binary) (iend(ijk),ijk=1,5)
   write(blkbegin,’(5a1)’) blkend

 blkbegin = =beg=51800 (Real Increment Verification Data)
 xlm( 1)   = transient time                      (time)
 xlm( 2)   =
             modal    result : frequency        (freq)
             harmonic result : frequency        (freq)
             buckle   result : buckle factor    (fact)
 xlm( 3)   =
             modal    result : generalized mass (gmas)
 xlm( 4)   =
             jantyp = 60 sensitivity check      (respon)
             jantyp = 61 objective function     (objec )
 xlm( 5)   =
             jantyp = 60 limiting value         (rsplim)
             jantyp = 61 critical constraint    (conval)
 xlm( 6)   = not used
 blkend   = =end=
```

```
BLOCK 51801 - real increment verification data
 ********************************************

If post file revision number ≥ 10 (MARC 2001 and later)

   read(formatted,'(a70)') blkbegin
   read(formatted,'(6i13)') nw
    read(formatted,'(6e13.6)') (xlm(ijk),ijk=1,nw)
   read(formatted,'( a5)') blkend

   read(binary) (ibeg(ijk),ijk=1,70)
   write(blkbegin,'(70a1)') blkbegin
   read(binary) nw
    read(binary) (xlm(ijk),ijk=1,nw)
   read(binary) (iend(ijk),ijk=1,5)
   write(blkbegin,'(5a1)') blkend

 blkbegin = =beg=51801 (Real Increment Verification Data)
 xlm( 1)   = transient time                      (time)
 xlm( 2)   =
             modal    result : frequency        (freq)
             harmonic result : frequency        (freq)
             buckle   result : buckle factor    (fact)
 xlm( 3)   =
             modal    result : generalized mass (gmas)
 xlm( 4)   =
             jantyp = 60 sensitivity check      (respon)
             jantyp = 61 objective function     (objec )
 xlm( 5)   =
             jantyp = 60 limiting value         (rsplim)
             jantyp = 61 critical constraint    (conval)
 xlm( 6)   = not used
 xlm( 7)   = total volume
 xlm( 8)   = total mass
 xlm( 9)   = total strain energy
 xlm(10)   = total plastic strain energy
 xlm(11)   = total creep strain energy
 xlm(12)   = total Kinetic energy
 xlm(13)   = total damping energy
 xlm(14)   = total work done by contact/external forces
 xlm(15)   = total thermal energy
 xlm(16)   = total elastic strain energy
 xlm(17)   = total work done by contact forces
 xlm(18)   = total work done by friction forces
 xlm(19)   = total work done by springs
 xlm(20)   = total work done by foundations
 xlm(21)   = not used
 xlm(22)   = not used
 xlm(23)   = not used
 xlm(24)   = not used
 blkend    = =end=

Note: nw = 18
```

Volume E: Demonstration Problems, Part II

Chapter 3
Plasticity and Creep

3.7 Elastic-Plastic Analysis of a Thick Cylinder

In this problem, a thick cylinder under the action of uniform internal pressure is loaded into the plastic region. A comparison with rigid plastic results is provided.

Element

The axisymmetric quadrilateral element, library element type 10, is used to model the wall of the cylinder. Details for this element are found in *MSC.Marc Volume B: Element Library.*

Model

Figure 3.7-1 shows the model geometry for this example. The cylinder wall has an inner radius of 1.0 inches and an outer radius of 2.0 inches.

The mesh is shown in Figure 3.7-2 and results in a model of the wall consisting of 20 elements, 42 nodes and 84 degrees of freedom.

Geometry

The geometry option is not required for this element.

Material Properties

The material data is: Young's modulus (E) of 30.0 x 10^6 psi, Poisson's ratio (ν) of 0.3, and von Mises yield stress (σ_y) of 45,000 psi. The material is assumed to behave elastic-perfectly plastic; that is, no strain hardening.

Boundary Conditions

Restraint boundary conditions are imposed in the axial direction on all nodes thus allowing only radial motion of the wall. This solution corresponds to a plane strain case.

Loading

An initial uniform pressure of 19,550 psi is applied using the DIST LOAD option. To investigate the plastic effects, SCALE is used to raise this pressure to a magnitude such that the highest stressed element (element 1) in the model has an equivalent yield stress (J2) which is equal to the specified yield stress of 45,000 psi. The resulting scale factor here is 1.045 which indicates the applied pressure for increment zero is 20,430 psi.

The data before END OPTION provides the elastic solution such that the highest stressed element is at first yield of 45,000 psi and any further loading is done incrementally into the plastic region.

Control

This option specifies a maximum of 15 increments in this example and a tolerance of 15% for convergence. (Only 11 increments are provided as the input data count for the zero increment.)

Incremental Loading

The data blocks following END OPTION are used to specify the incremental load step into the plastic region. The AUTO LOAD option is used to apply two load increments of equal size and the PROPORTIONAL INCREMENT option is used to provide a scaling factor of the load step size for each application of the AUTO LOAD option.

The PROPORTIONAL INCREMENT option as used here specifies a scaling factor to be applied to the previous load step size, and the minimum number of cycles through the prediction of plastic effects (NCYCM) was set to 2 to improve solution accuracy. The scaling factor is adjusted to give the necessary small load steps to keep the solution within the desired tolerance.

The incremental loads which are applied in this example are as follows:

Increment

0 P_0 $= sp = (1.03)(19550) = 20{,}136$ psi

1 P_1 $= P_0 + \Delta P_1$: $\Delta P_1 = fsp = (0.13)(1.03)(19{,}550)$

2 P_2 $= P_0 + \Delta P_1 + \Delta P_2$: $\Delta P_2 = \Delta P_1$

3 P_3 $= P_0 + \Delta P_1 + \Delta P_0 + \Delta P_3$: $\Delta P_3 = 0.8\Delta P_2$

5 P_5 $= P_0 + \Delta P_1 + ... + \Delta P_5$: $\Delta P_5 = 0.7\Delta P_4$

7 P_7 $= P_0 + \Delta P_1 + ... + \Delta P_7$: $\Delta P_7 = 0.667\Delta P_6$

10 P_{10} $= P_0 + \Delta P_1 + ... + \Delta P_{10}$

$\Delta P_{10} = \Delta P_9 = 0.5\Delta P_8 = 0.5\Delta P_7 = ... = 0.5(0.667)(0.7)(0.8)(0.13)\Delta p$
$= 1.04052 \times 10^{-2}\ \Delta p = 488$ psi

If a reverse load is desired, a negative scale factor should be used only once to reverse the sign of the load step.

If a load step is applied which is too large to allow the energy change tolerance to be satisfied, Marc, in this case, cycles through the predicted displacement iteration five times. On the last try, a message indicating NO CONVERGENCE TO TOLERANCE is printed out. Then the strains and stresses corresponding to the last iteration are printed in the output, and Marc exits with an appropriate exit message.

Restart

To protect against failure to meet tolerances, use of the restart capability available in the program is recommended. The RESTART option has been used in this example. Two input decks which follow this discussion illustrate the use of RESTART. The first run creates a restart file (unit 8) and writes the necessary data to this file so that the analysis can be restarted at any increment.

The initial deck is set up to run completely through the analysis while the second is used to restart the problem at a point in the middle of the analysis. The analysis was restarted at increment 7.

In general, this specification requires the program to read the next set of load data following END OPTION to be applied as the increment 8 load set. In this case, the program already has the required load data for the increment 8 solution because of the use of the AUTO LOAD option, and it will complete the step of the option before reading the additional data after END OPTION. The data supplied after END OPTION is only enough to complete increments 9 and 10.

Results

The results of this analysis are shown in Figure 3.7-3 through Figure 3.7-6. Comparison is made with the results of the finite difference solution given in Chapter 4 of *Theory of Perfectly Plastic Solids* by W. Prager and P. G. Hodge, Jr. (published by John Wiley and Sons, 1963).

Comparison is shown for two values of tolerance which varied from 0.5 to 0.1. The results did not vary appreciably as a function of the displacement tolerance.

The following terminology is used in Figure 3.7-4 through Figure 3.7-6:

a = inner radius

b = outer radius

ρ = radius of elastic-plastic boundary

σ_r = radial stress

$\sigma\theta$ = circumferential stress

σ_z = axial stress

Y = yield stress

k = $Y/\sqrt{3}$

The elastic-plastic boundary is shown as a function of the pressure, p, in Figure 3.7-3.

For the plane strain condition, a numerical solution obtained by finite difference methods was given in the reference. The radial stress distribution for two different positions of the elastic-plastic boundary ($\rho/a = 1.2$ and $\rho/a = 2.0$) are compared to the solution given in the reference in Figure 3.7-4. Excellent agreement is observed. The circumferential stress distribution in the partially plastic tube is similarly compared in Figure 3.7-5. A comparison of the axial stress distribution is given in Figure 3.7-6. The two solutions are seen to be in good agreement.

Parameters, Options, and Subroutines Summary

Example e3x7a.dat:

Parameters	Model Definition Options	History Definition Options
END	CONNECTIVITY	AUTO LOAD
SCALE	CONTROL	CONTINUE
SIZING	COORDINATES	PROPORTIONAL INCREMENT
TITLE	DIST LOADS	
	END OPTION	
	FIXED DISP	
	ISOTROPIC	
	PRINT CHOICE	
	RESTART	

Example e3x7b.dat:

Parameters	Model Definition Options	History Definition Options
END	CONTROL	AUTO LOAD
SCALE	DIST LOADS	CONTINUE
SIZING	END OPTION	PROPORTIONAL INCREMENT
TITLE	FIXED DISP	
	ISOTROPIC	
	PRINT CHOICE	

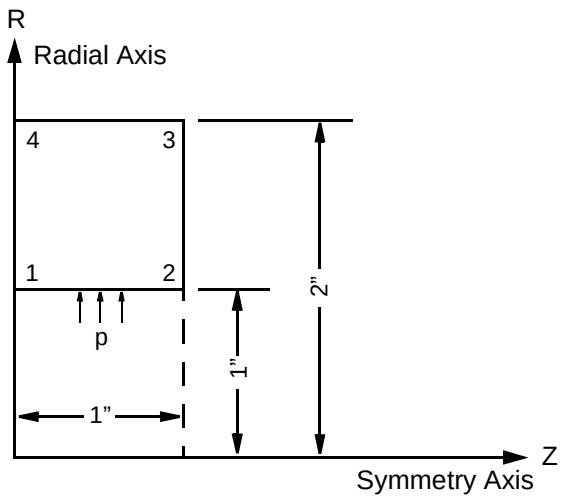


Figure 3.7-1 Cylinder Wall

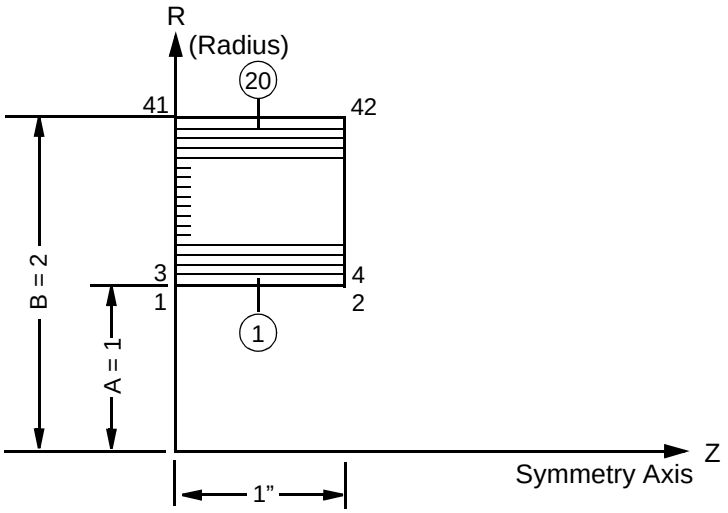


Figure 3.7-2 Cylinder Wall Generated Mesh

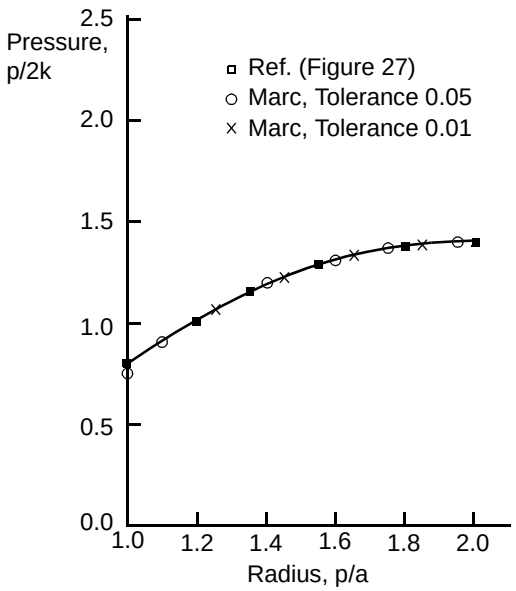


Figure 3.7-3 Pressure Versus Elastic-Plastic Boundary

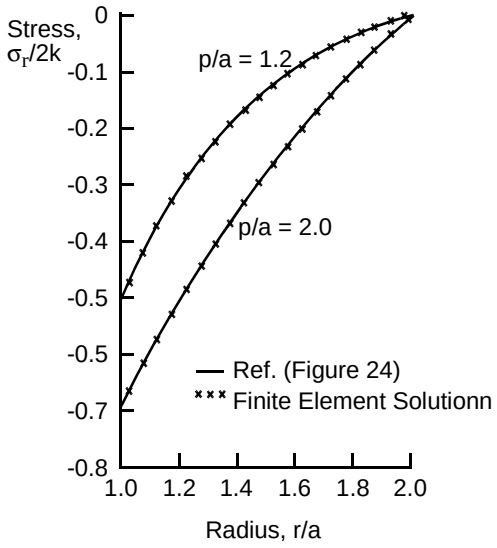


Figure 3.7-4 Radial Stress Distribution

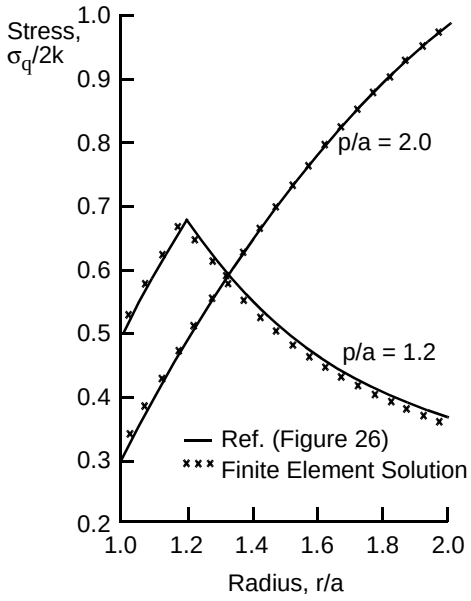


Figure 3.7-5 Circumferential Stress Distribution

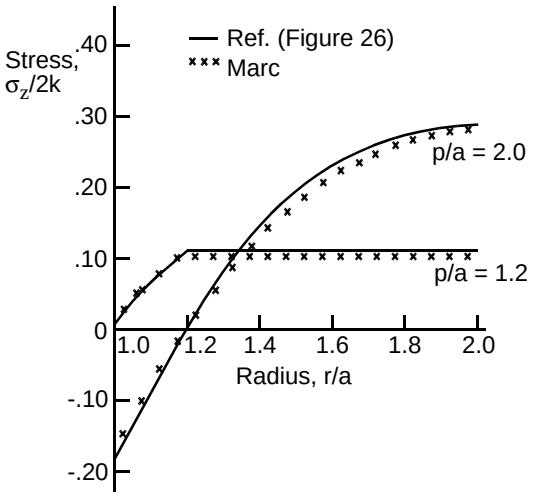


Figure 3.7-6 Axial Stress Distribution

3.32 Superplastic Forming of a Strip

This problem demonstrates how to form a shape with a superplastic material. A two-dimensional flat workpiece is pressurized into a die of the desired final shape. Three different models are constructed using two-dimensional plane strain elements, three-dimensional membrane elements, and three-dimensional shell elements.

This problem is modeled using the three element types and summarized below.

Data Set	Element Type(s)	Number of Elements	Number of Nodes
e3x32a	11	340	430
e3x32b	18	80	162
e3x32c	75	80	162

Mesh Generation

These three models consist of 340 4-node isoparametric quadrilateral plane strain elements, 80 membrane elements, and 80 shell elements. The workpiece is 2.3 inches long with an in-plane thickness of 0.078 inches.

Boundary Conditions

The sheet is fixed at both ends in the x-direction and the left end is fixed in the y-direction where the workpiece contacts the die. The membrane and shell models have similar boundary conditions and their out-of-plane displacements are fixed to simulate plane strain. Furthermore, the membrane model will have an initial in-plane tensile stress of 50 psi for the first five increments to avoid any instabilities. For the membrane elements, a prestress of 50 psi is applied for 5 increments to prevent numerical instabilities. The workpiece is subjected to a uniform pressure whose magnitude is determined automatically to maintain a target strain rate of 0.0002. The model with these boundary condition is shown in Figure 3.32-1 and Figure 3.32-2 for the plane strain and membrane (shell) models, respectively.

Analysis Controls

The SPFLOW parameter card is needed for the Superplastic analysis. This turns the PROCESSOR and FOLLOW FOR options on by default. The SUPERPLASTIC model definition option allows the control of prestress (and the number of increments it needs to be applied for), the process control parameters, process driving parameters as well as the analysis termination criterion.

Material Properties

The out-of-plane thickness is 1.0 inch for all models. The CONSTANT DILATATION parameter for the plane strain elements is used. Superplastic materials can be viewed as exhibiting time-dependent inelastic behavior with the yield stress a function of time, temperature, strain rate, total stress, and total strain. In this case, the yield stress is only a function of the effective strain rate and is represented as either a POWER LAW or RATE POWER LAW hardening in the ISOTROPIC model definition option:

POWER LAW: $\bar{\sigma} = A(\varepsilon_o + \bar{\varepsilon})^m + B\dot{\bar{\varepsilon}}^n$ with A = 0, B = 50000, n = 0.6, m = 0

or RATE POWER LAW: $\bar{\sigma} = A\bar{\varepsilon}^m\dot{\bar{\varepsilon}}^n + B$ with B = 0, A = 50000, m = 0, n = 0.6

where $\dot{\bar{\varepsilon}}$ = effective strain rate, $\bar{\sigma}$ = yield stress.

Contact

Each model has one rigid body and one deformable body. In increment 0, the rigid body is moved into first contact with the workpiece and held fixed thereafter. (In contact control, Coulomb friction with μ = 0.5, a separation force of 1.0e6 lbf, and a sliding velocity of 1.0e-5 inches/seconds are used. For the membrane and shell models, the contact zone tolerance is set equal to one-half of the element thickness and the bias factor is set to .99 to reduce the touching distance because of the large thickness. Because nodal based friction forces must be used with membrane and shell elements, it is used for all models.)

Loading

The load schedule consists of a single rigid plastic loadcase with a total time period of 2500 seconds and 500 steps with convergence testing on displacements.

Results

Although the total time period is 2500 seconds, the part forms in a little over 2000 seconds (34 minutes). As expected, the bending of the workpiece is insignificant and the results of all models are in close agreement. The membrane elements are superior in conforming to the die shape and are substantially faster (2.5 times faster) than the plane strain or shell models.

Figure 3.32-3 shows the final deformed shape of the plane strain model. The final average thickness is 0.0554 and the sheet has elongated to 3.231 inches, showing virtually no change in volume. The final average thickness can be estimated prior to

the finite element analysis since the original and final sheet length are known and the sheet is incompressible. The thickness for continuum elements is used by use of PLOTV subroutine.

The process pressure is available as a default history post variable. Figure 3.32-4 shows the pressure schedule for all models with very small differences. These small differences in the pressure schedule are caused when the sheet begins to fill the concave corner. As the sheet begins to fill the concave corner, the pressure must increase rapidly to maintain the target strain rate of 2.03-4/seconds. Note that the sliding velocity is 1/20 of the target strain rate which is a typical value (this is true for length units of inches and would need to be modified for other length units). The maximum pressure is physically limited and has a maximum value of 300 psi. Furthermore, Figure 3.32-4 also plots the vertical reaction on the die divided by the sheet area versus time. This die pressure leads the sheet pressure because of friction acting on the vertical portion of the die. Here more differences exist between the three models. The biggest difference is around 1800 seconds where the friction stops contributing to the die force because of the inability of the plane strain model to completely fill the concave corner.

Figure 3.32-5 shows the thickness profile over the deformed position along the sheet. The largest thinning in all models occurs at the 1.0 inch position or the concave corner. The significant area of difference occurs at the convex radius at the 1.9 inch position. This difference is because of transverse normal stresses caused by bearing on the radius, the plane strain elements thin more since the membrane and shell elements cannot support this deformation state. The membrane elements only thin because they are stretched and must maintain volume. The shell elements thin because they stretch and bend while maintaining volume. The plane strain elements thin because of stretching, bending, and transverse deformations while maintaining volume.

Since friction plays a large roll in the thinning of the sheet and the membrane model runs fastest, a frictionless case is run for the membrane elements. The thickness profiles of the friction and frictionless cases are shown in Figure 3.32-6. Without friction, the thinning is very uniform and the final thickness is almost the average thickness everywhere.

Figure 3.32-7 shows the balance between strain energy and the total work done by external forces with input data e3x32.dat.

Summary of Parameters, Options, and Subroutines Used

Example e3x32a.dat, e3x32b, e3x32c:

Parameters	Model Definition Options	History Definition Options
ELEMENTS	CONNECTIVITY	AUTO LOAD
END	CONTROL	CONTINUE
PRINT	COORDINATES	DIST LOADS
SIZING	DIST LOADS	MOTION CHANGE
SPFLOW	FIXED DISP	SUPERPLASTIC
TITLE	GEOMETRY	TIME STEP
	ISOTROPIC	
	NO PRINT	
	OPTIMIZE	
	POST	
	SOLVER	

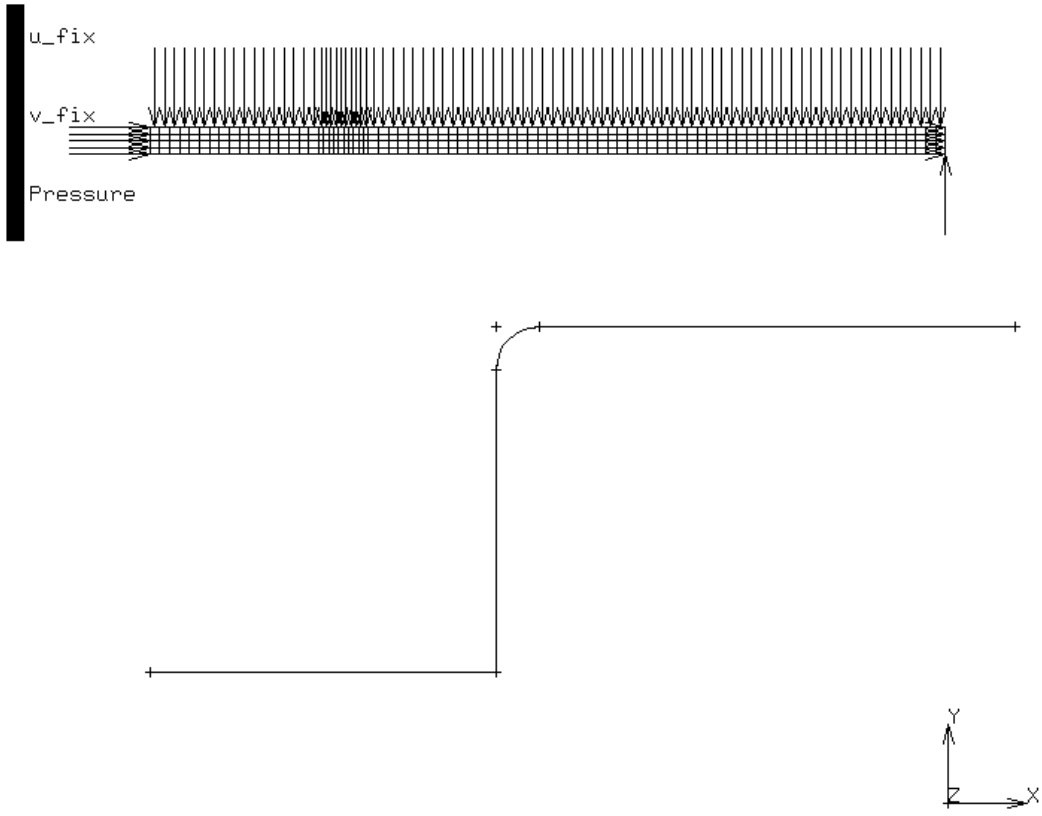


Figure 3.32-1 Plane Strain SPF Model with Boundary Conditions

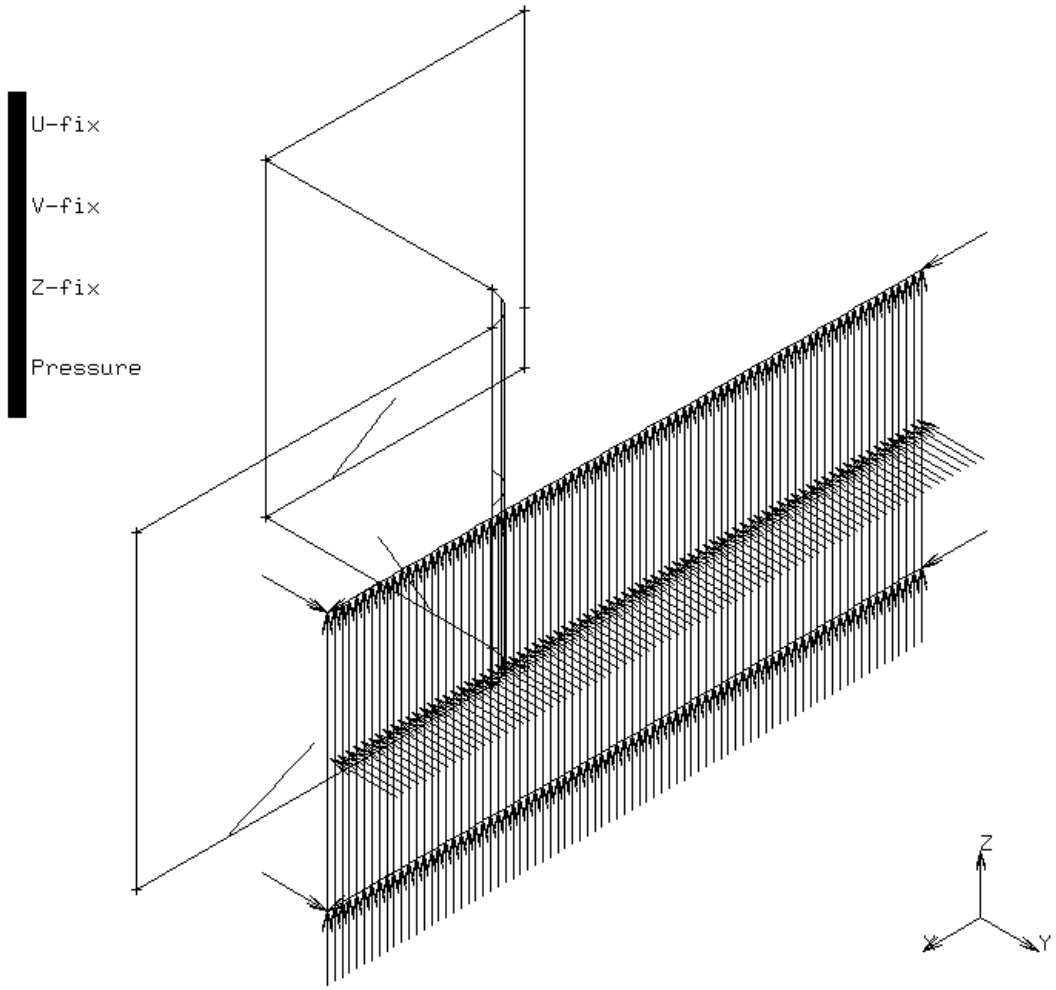


Figure 3.32-2 Membrane and Shell SPF Model with Boundary Conditions

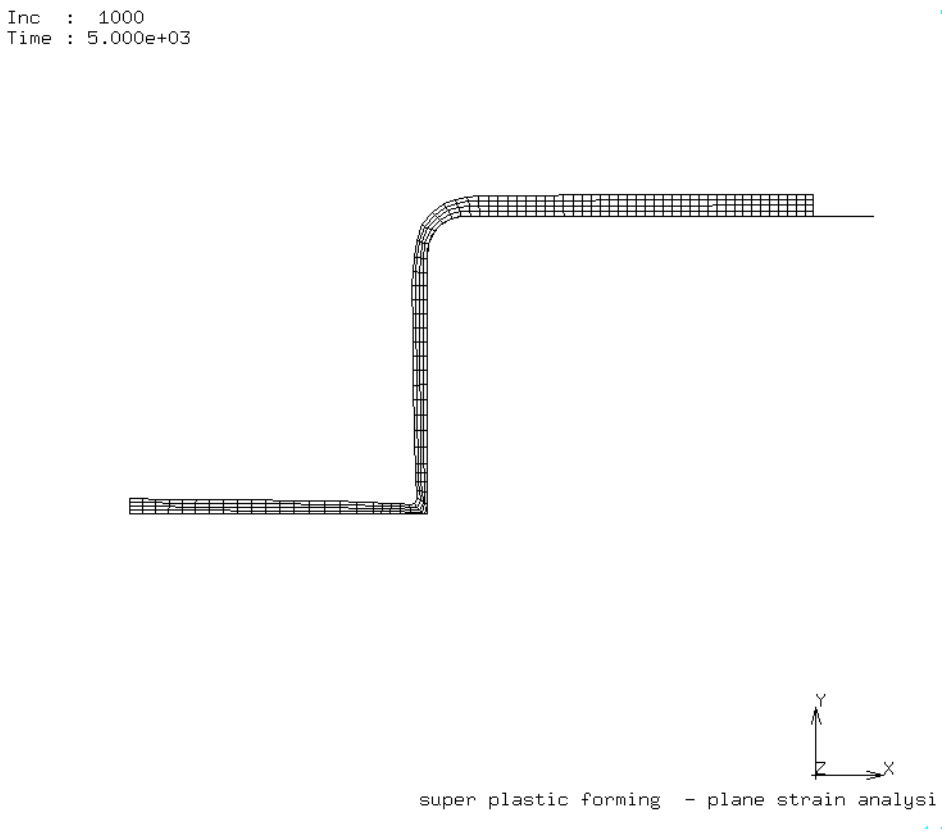


Figure 3.32-3 Plane Strain SPF Model Final Shape

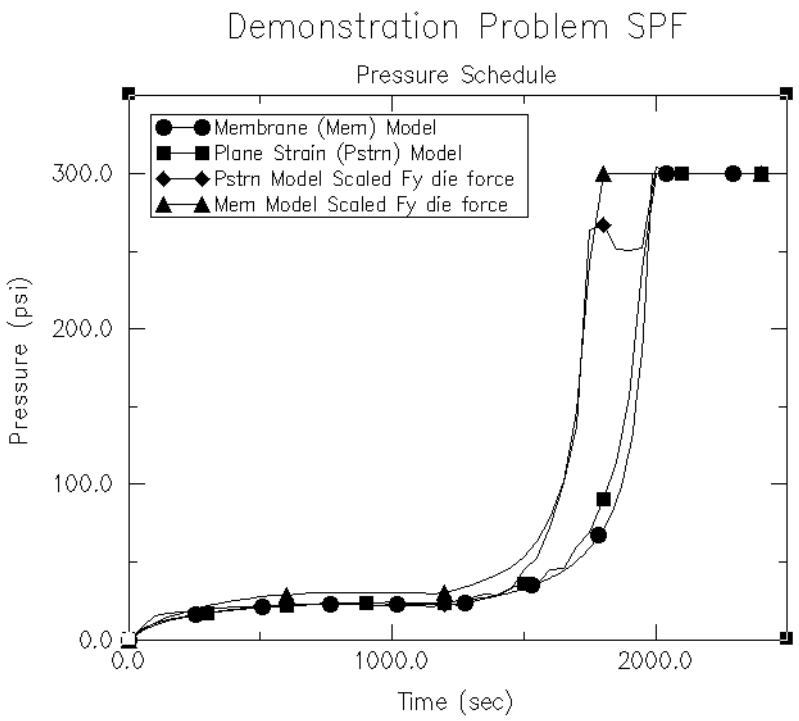


Figure 3.32-4 Pressure Schedule for all Models

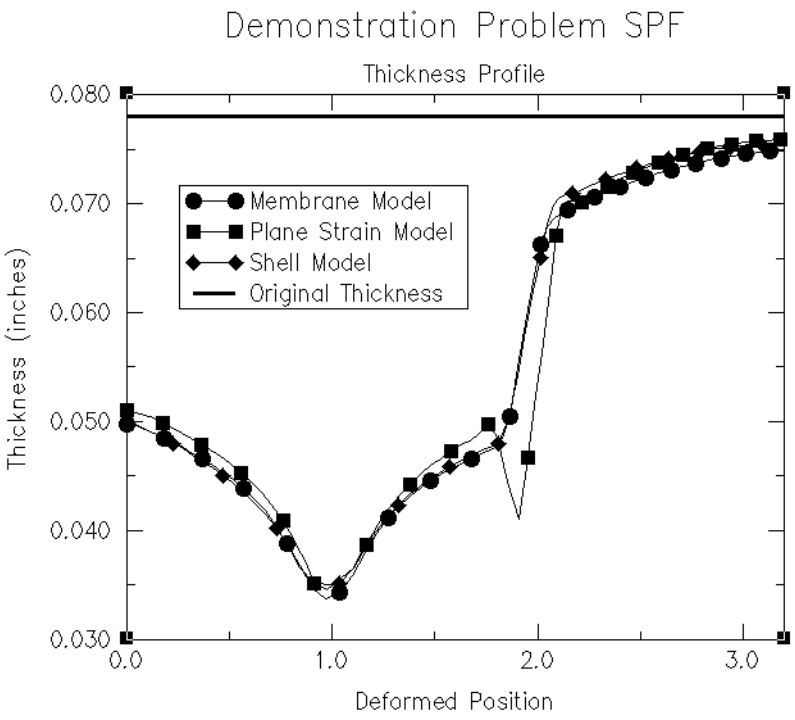


Figure 3.32-5 Thickness Profile for all Models

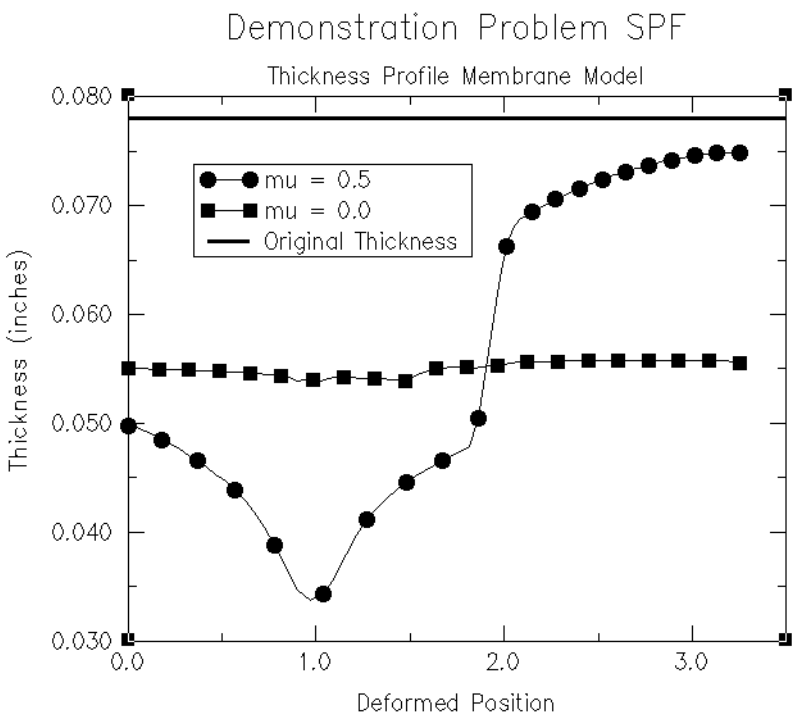


Figure 3.32-6 Thickness Profile Membrane Friction Effects

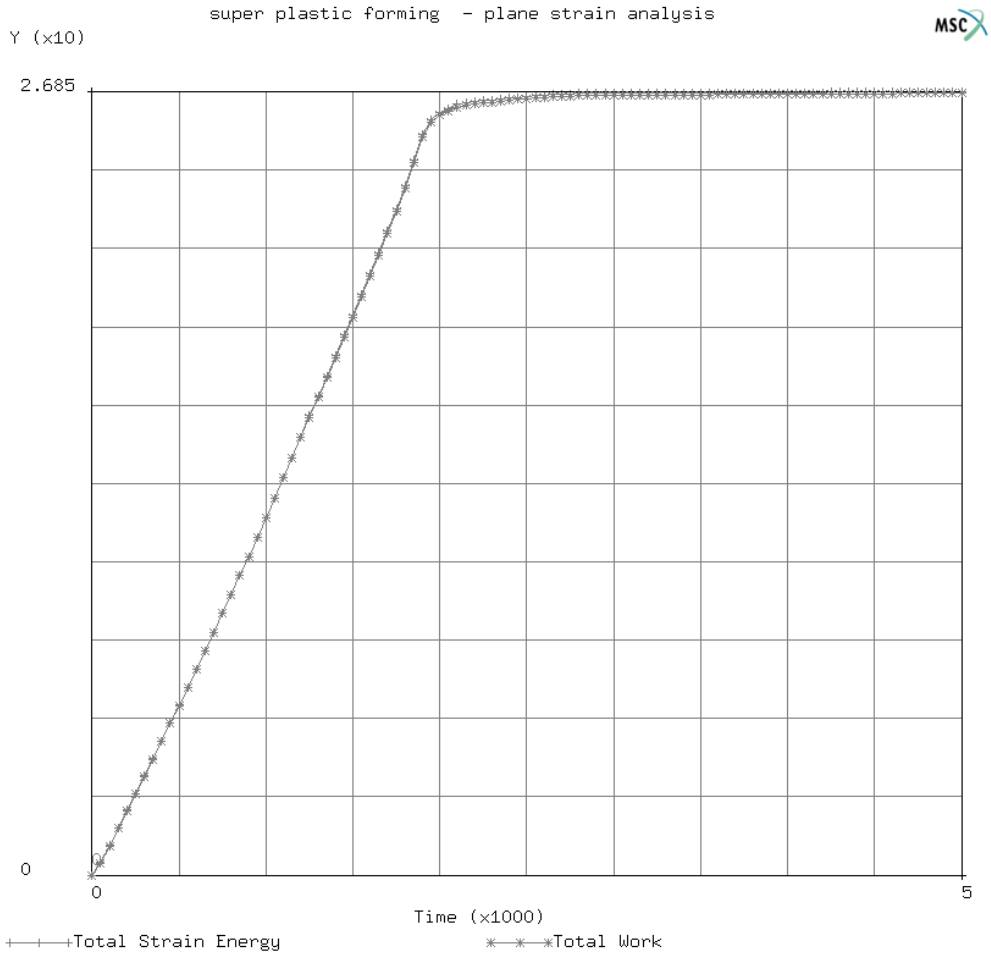


Figure 3.32-7 Energy Balance of e3x32a.dat

3.33 Large Strain Tensile Loading of a Plate with a Hole

This problem simulates the tensile loading of a plate with a hole to large strains. The example demonstrates the accuracy of the finite strain plasticity algorithm using F^eF^p formulation to simulate large strains. The multiplicative decomposition procedure is invoked using the PLASTICITY option.

This problem is modeled using two techniques summarized below.

Data Set	Element Type(s)	Number of Elements	Number of Nodes	Differentiating Features
e3x33	26	20	79	Plane Stress
e3x33b	27	20	79	Plane Strain

Element

This problem simulates two-dimensional plane stress and plane strain cases. For the plane stress case, an 8-node plane stress isoparametric element type 26 is used to construct a mesh while for the plane strain case, an 8-node plane stress isoparametric element type 27 is used. There are two degrees of freedom per node with a bi-quadratic interpolation and eight-point Gaussian quadrature for stiffness assembly.

Model

Due to symmetry of the geometry and loading, a quarter of the actual model is simulated. The finite element model is made up of 20 elements and 79 nodes. There is a total of 158 degrees of freedom. The model is shown in Figure 3.33-1.

Geometry

The model is assumed to be a square of side five units from which a quarter of a circle of radius one unit has been cut out. In the plane stress case, the initial thickness is one unit.

Material Properties

The material is assumed to be isotropic elastic plastic. The Young’s modulus is 3.0E+07 psi. Poisson’s ratio is 0.30. The initial yield stress is 5.0E+04 psi. The hardening behavior is given in Table 3.33-1.

Table 3.33-1 Hardening Behavior

Equivalent Plastic Strain	Workhardening Slope (psi)
0.00	14.30E+06
7.00E-04	3.00E+06
1.60E-03	1.90E+06
2.55E-03	6.70E+05
3.30E-03	3.00E+05
1.00	1.00E+05

Boundary Conditions

The loading is tensile. The lower edge of the model is restrained to have no y displacements, while the left edge the model is constrained to have no x displacements. The top edge is subjected to displacement increments in the y direction.

Results

The contours of effective plastic strain on the deformed model are shown in Figure 3.33-2. Plasticity initiates at the hole due to the stress concentration and accumulates with increasing strain. The maximum value is 186% at the end of the last increment. The history plot of x displacement at node 34 as a function of the increment is shown in Figure 3.33-3. Node 34 is the node on the hole edge and has specified zero y displacement. Figure 3.33-3 shows that the increments of x displacement at node 34 are initially negative, indicating that the hole is shrinking in dimension perpendicular to the loading direction. However, as plasticity accumulates, the x displacement increments become positive, indicating a growth in the hole dimension perpendicular to the loading direction. As the hole surface grows outward, the external surface continues to move inward. This reduces the ligament size available to carry load and necking results. This behavior is also seen for the plane strain case although with different numerical values.

The contours of effective plastic strain on the deformed model for e3x33b.dat are shown in Figure 3.33-4.

Parameters, Options, and Subroutines Summary

Examples e3x33a.dat and e3x33b.dat :

Parameters	Model Definition Options	History Definition Options
TITLE	CONNECTIVITY	AUTO STEP
SIZING	COORDINATES	DISP CHANGE
ELEMENTS	ISOTROPIC	CONTINUE
LARGE DISP	GEOMETRY	
PLASTICITY	WORK HARD	
	FIXED DISP	

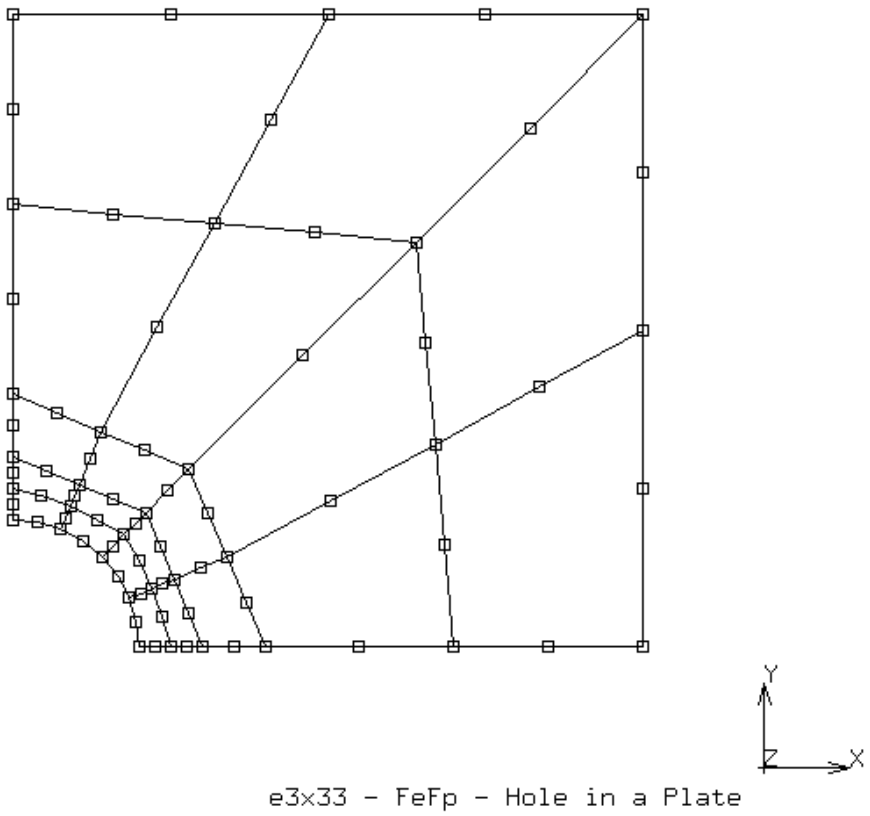


Figure 3.33-1 Initial Model for Hole-in-Plate

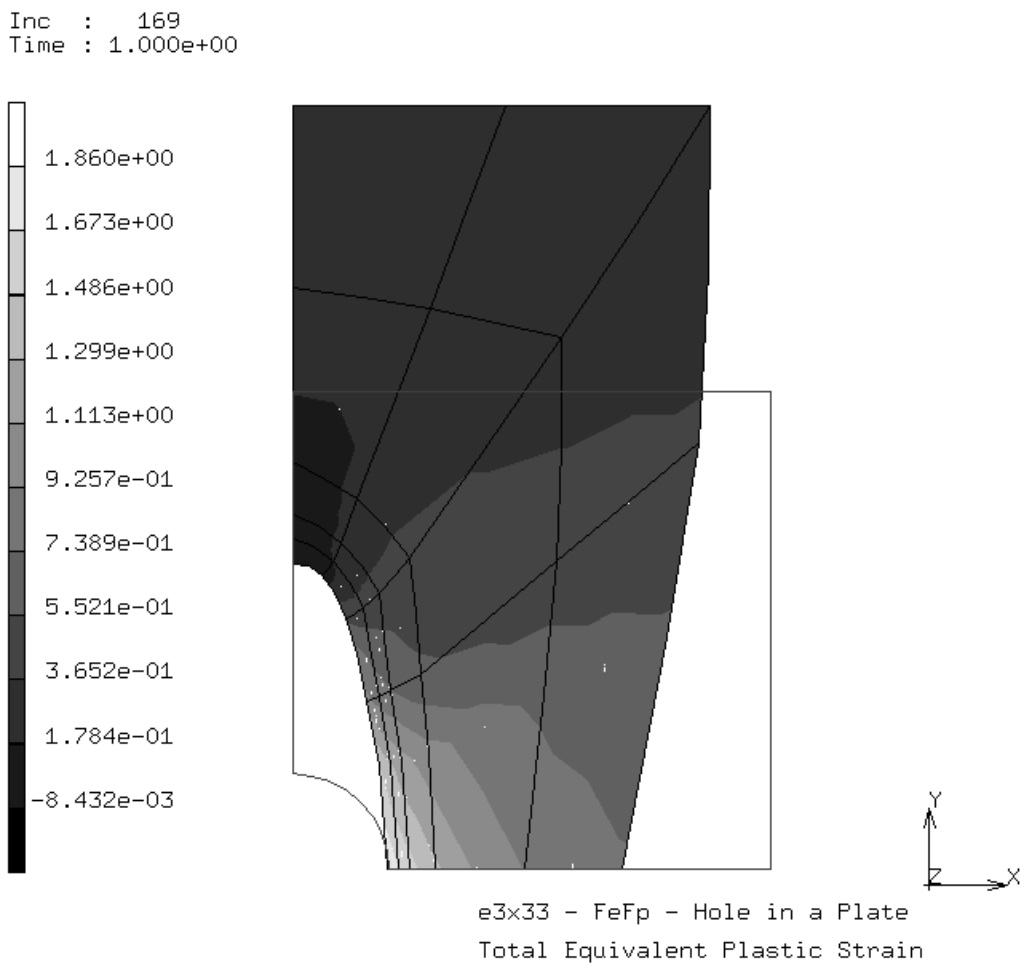


Figure 3.33-2 Equivalent Plastic Strain on the Deformed Model

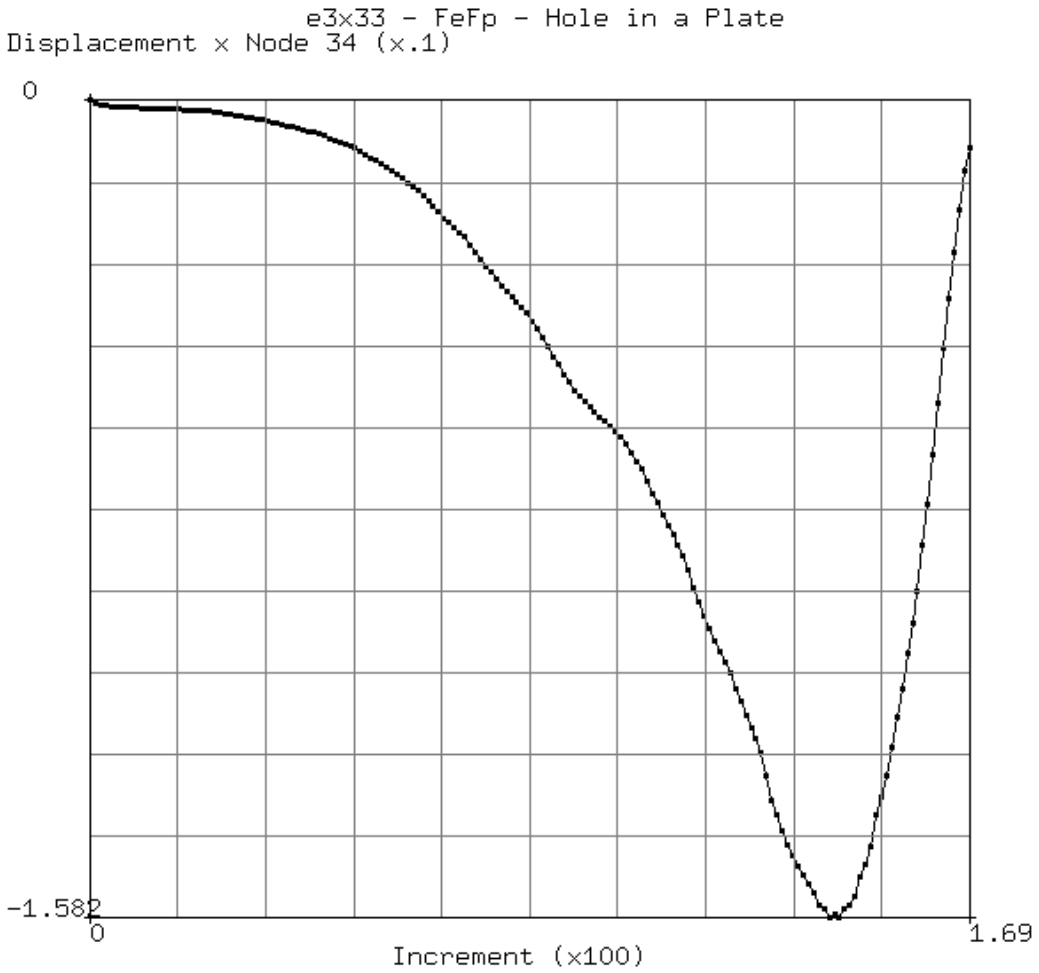


Figure 3.33-3 History Plot of x Displacement versus Increment for Node 34

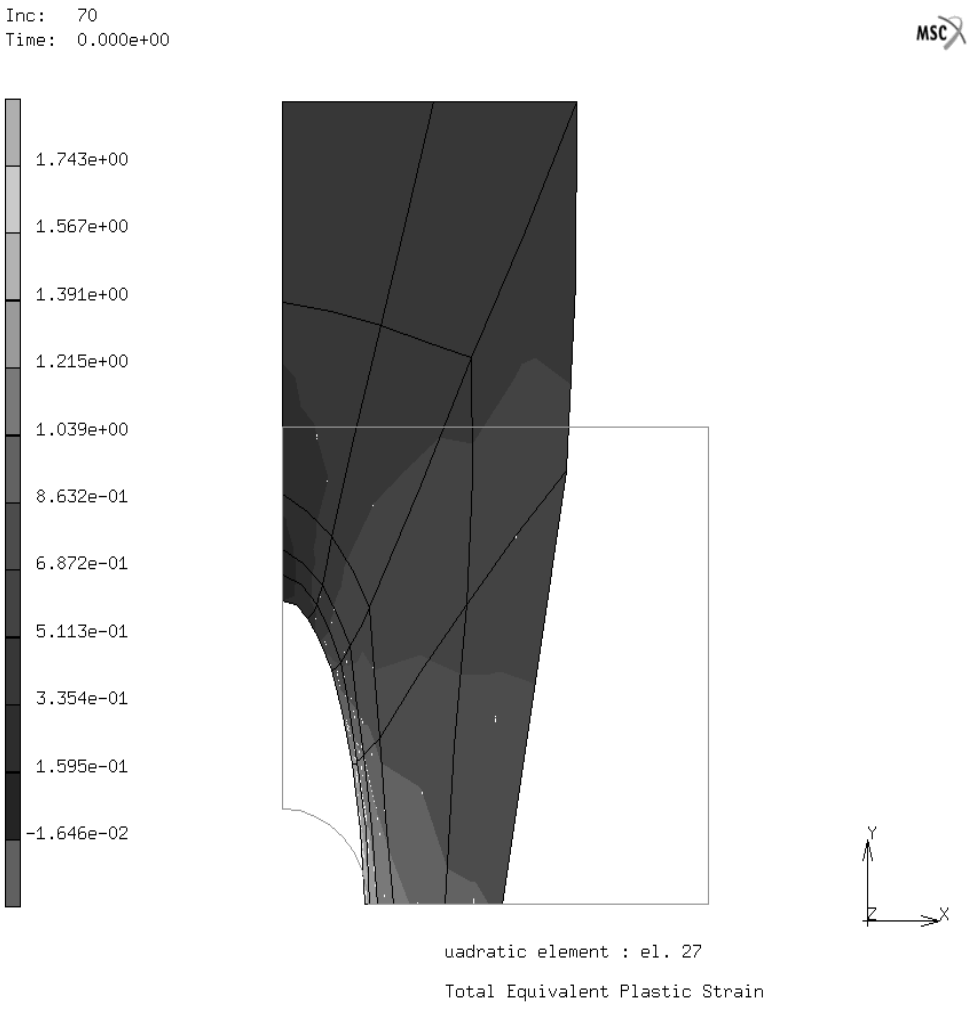


Figure 3.33-4 Equivalent Plastic Strain on the Deformed Model for e3x33b

Volume E: Demonstration Problems, Part III

Chapter 5
Heat Transfer

5.17 Cooling of Electronic Chips

This problem demonstrates the air cooling of an electronic chip at room temperature. The comparison of the no-inclusion of heat convection, e5x17a.dat, and the inclusion of the contribution of heat convection, e5x17b.dat, by air is made. The nonsymmetric solver is turned on automatically when heat convection is included.

This problem is modeled using the two techniques summarized below.

Data Set	Element Type(s)	Number of Elements	Number of Nodes	Differentiating Features
e5x17a	39	360	399	Exclude convection
e5x17b	39	360	399	Include convection

Element

Element type 39 is used for both the air region and the chip body. The model is shown in Figures 5.17-1 and 5.17-3.

Material properties

Room temperature thermal properties for air are used. The specific heat is 1.0057 kJ/kg.°C, the density is 1.177e-6 kg/cm^3, and thermal conductivity is 0.0002624 W/cm.°C. Thermal properties for pure copper are used for the chip. The specific heat is 0.3855 kJ/kg.°C, the density is 8.893e-3 kg/cm^3, and thermal conductivity is 3.8015 W/cm.°C. Assume the variation of properties with temperature is negligible.

Initial Conditions

The initial nodal temperature for chips is 40°C and for air is 10°C throughout.

Boundary Conditions

The temperature of the air far away from chips is fixed at 10°C and velocity of the air is kept at a constant 1400 cm/second. The velocity of the chips is zero.

Transient Nonauto

A fixed time step is used to simulate the cooling process near steady-state condition.

Results

The temperature distributions shown in Figures 5.17-2 and 5.17-4 indicate the effect of heat convection on the cooling of the chips. The chips have cooled down faster on the left side because, as heat convection of the air is included, more heat is carried away by the air. The effect of the boundary layer between the air and the surface of the chips is neglected. Because the Courant number is too large, numerical dispersion occurs at the air region far away from the chips. Figure 5.17-5 shows the thermal energy of the chips.

Parameters, Options, and Subroutines Summary

Example e5x17a.dat:

Parameters	Model Definition Options	History Definition Options
ALL POINTS	CONNECTIVITY	CONTINUE
COMMENT	CONTROL	TRANSIENT
DIST LOADS	COORDINATE	
END	DEFINE	
HEAT	END OPTION	
PRINT	FIXED TEMP	
SETNAME	INITIAL TEMP	
SIZING	ISOTROPIC	
TITLE	NO PRINT	
	POST	
	VELOCITY	

Example e5x17b.dat:

Parameters	Model Definition Options	History Definition Options
ALL POINTS	CONNECTIVITY	CONTINUE
COMMENT	CONTROL	
DIST LOADS	COORDINATE	
END	DEFINE	TRANSIENT
HEAT	END OPTION	
PRINT	FIXED TEMP	
SETNAME	INITIAL TEMP	
SIZING	ISOTROPIC	
TITLE	NO PRINT	
	POST	
	VELOCITY	

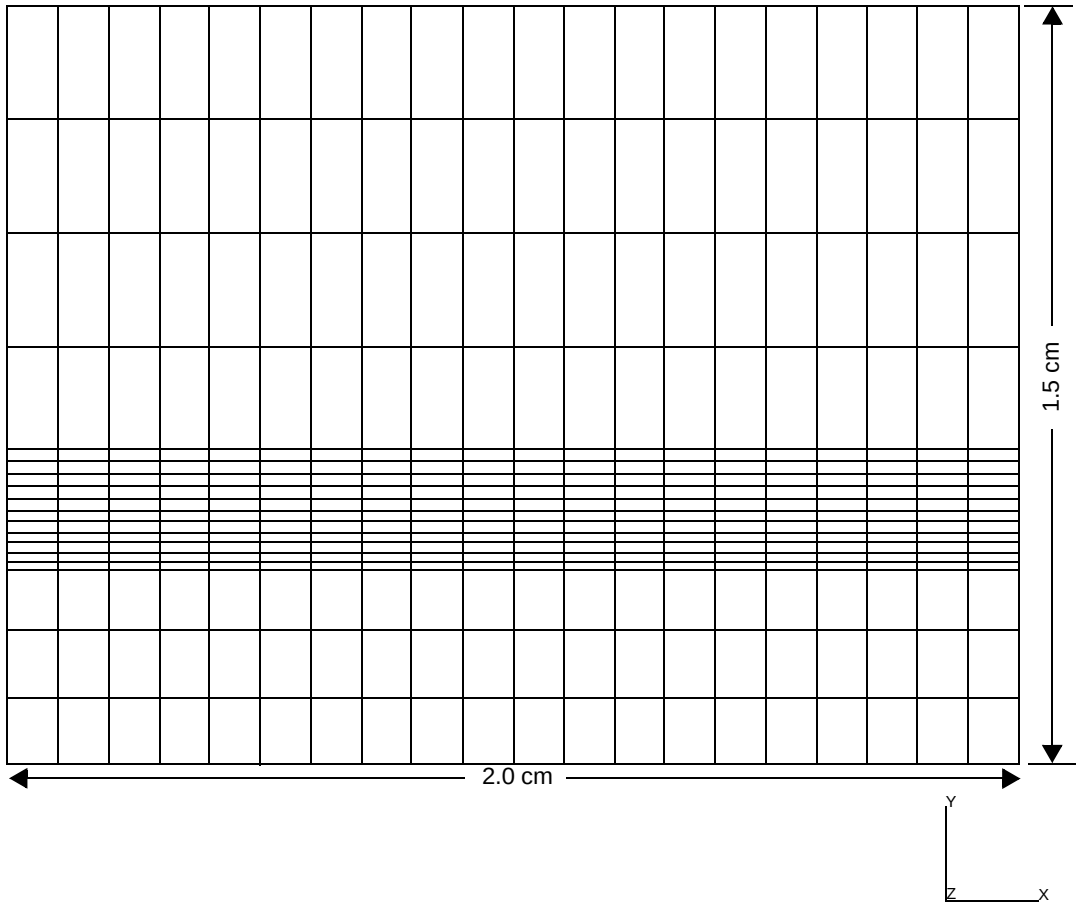


Figure 5.17-1 Complete Finite Element Mesh

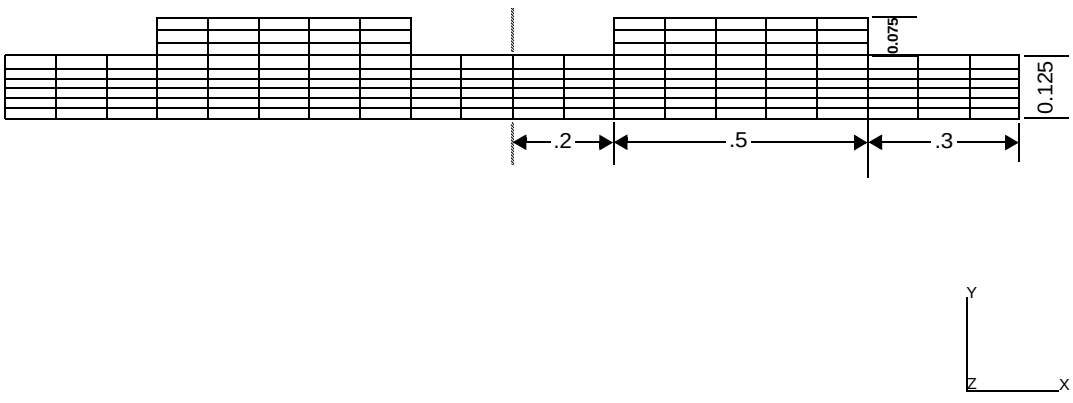


Figure 5.17-2 Finite Element Mesh of Chips and Board

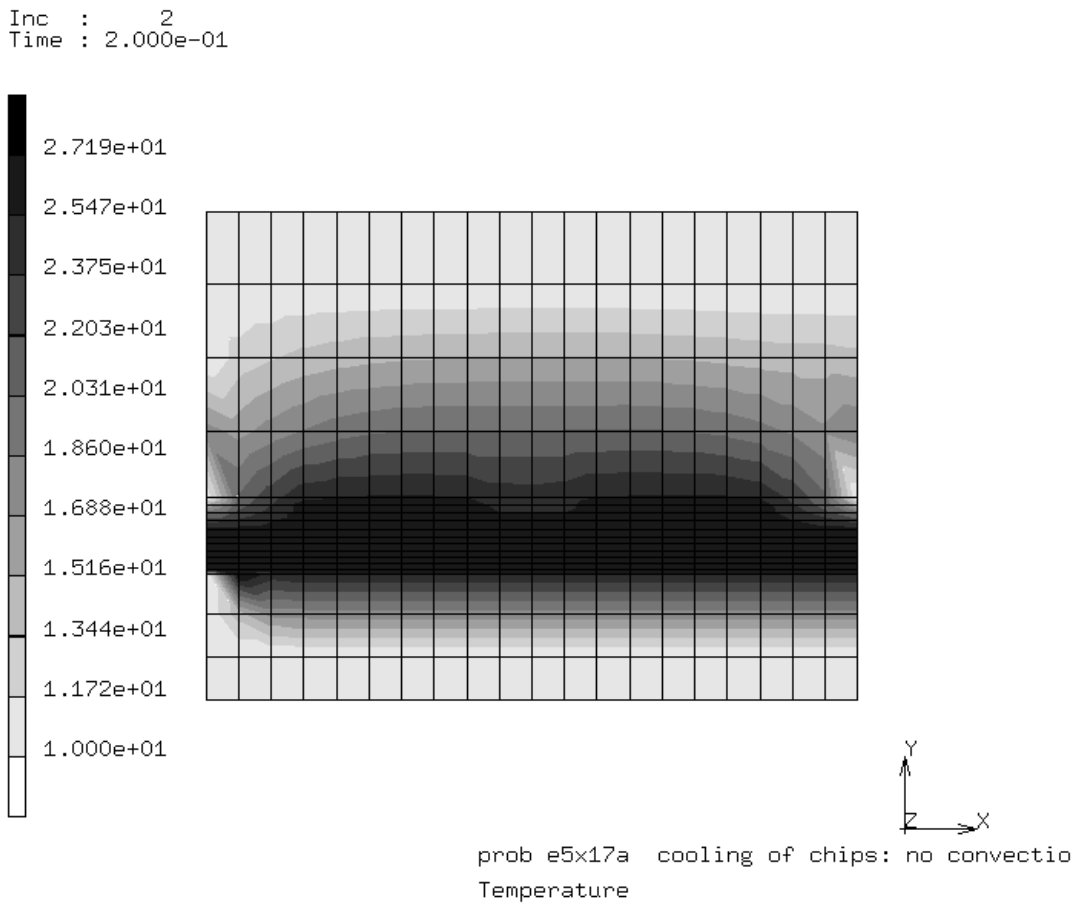


Figure 5.17-3 Temperature Distribution Excluding Heat Convection

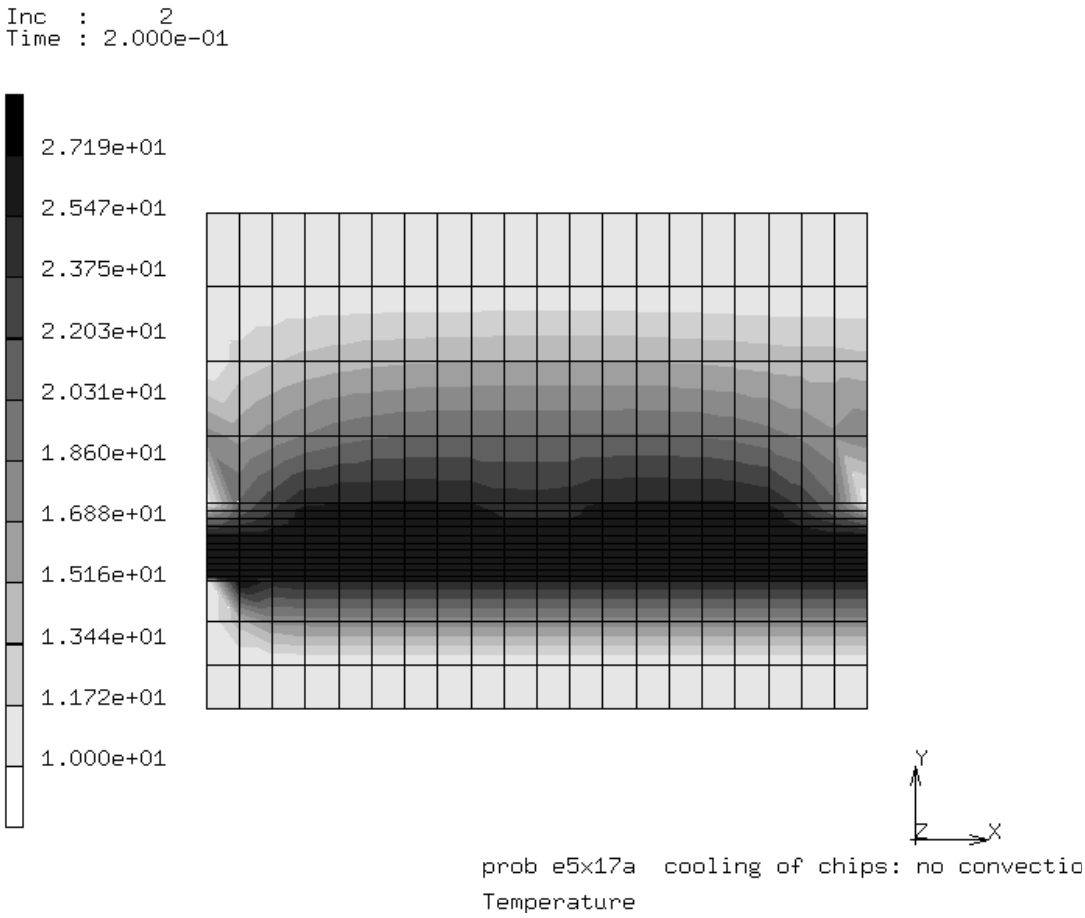


Figure 5.17-4 Temperature Distribution Including Heat Convection

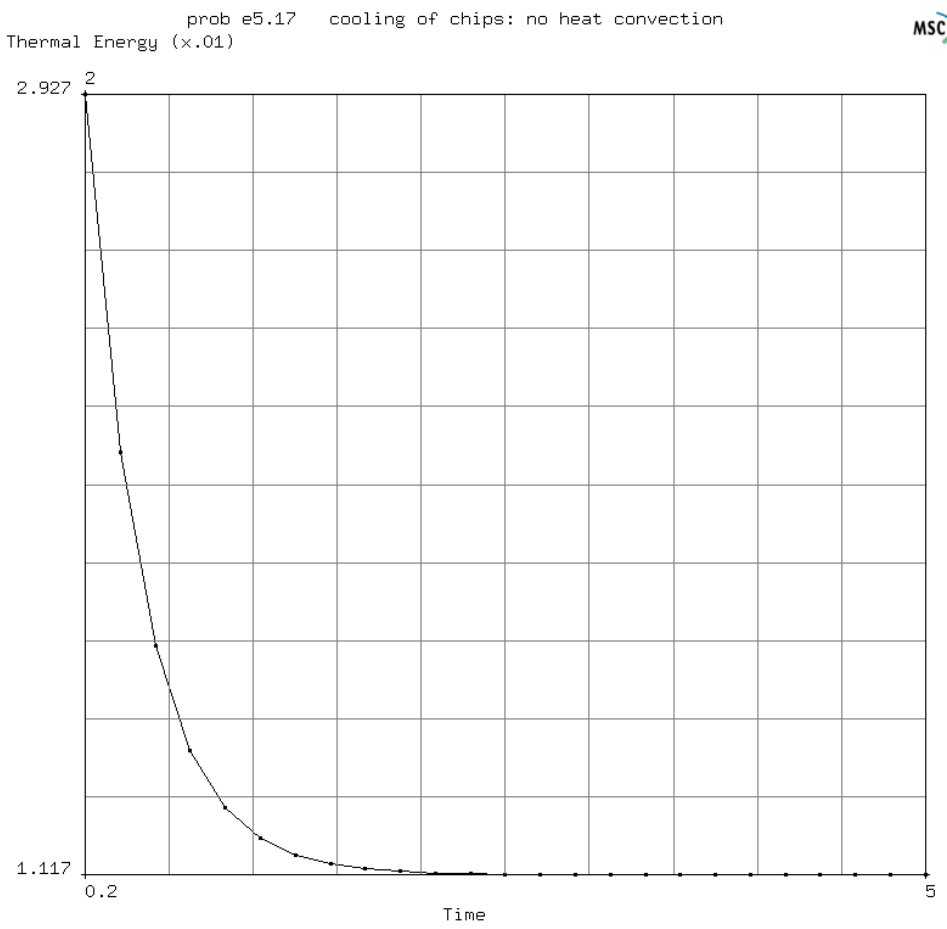


Figure 5.17-5 Thermal Energy Change During Cooling Process

Chapter 6
Dynamics

6.22 Dynamic Collapse of a Cylinder

In this example, the dynamic collapse of a cylinder is analyzed. The cylinder, with a radius of 0.02 m, a wall thickness of 0.00131 m and a length of 0.08 m, is compressed between two rigid bodies, of which one is fixed and one has a velocity of 50 m/s.

Element

Element type 10, a four-node axisymmetric isoparametric element with full integration is used to model the cylinder.

Constant Dilatation

The CONSTANT DILATATION parameter is used since large plastic deformations occur. In this way, the incompressibility of the material during plastic deformations can be accounted for without showing a too stiff behavior.

Dynamic

The Single Step Houbolt dynamic time integration method is activated using the DYNAMIC parameter. This method is especially recommended for dynamic contact problems, since is possesses high-frequency dissipation, so that undesired, numerically triggered, high-frequency oscillations may be damped out quickly.

Lump

The mass matrices are applied in a lumped form using the LUMP parameter.

Plasticity

The material behavior is based on small strain elasticity and large strain plasticity based on the additive decomposition of the strain tensor.

Isotropic

The elastic material properties are given by a Young's modulus of 1×10^{11} N/mm^2, a Poisson's ratio of 0.3 and a density of 7000 kg/mm^3. Plasticity is according to the von Mises criterion with an initial yield stress of 1×10^{8} N/mm^2.

Work Hard,Data

A linear hardening modulus of 3 x 10 N/m^2 is defined using the WORK HARD,DATA model definition option.

Contact

Three contact bodies are defined: one deformable body consisting of all the finite elements, and two rigid bodies, each consisting of a straight line (see also Figure 6.22-1). Friction between the cylinder and the first rigid body is entered based on a friction coefficient of 0.1 and the Coulomb (for rolling) friction model. The increment splitting method is set to the iterative one.

No Print

The NO PRINT model definition option is used to suppress print out.

Post

As element post file variables, the total equivalent plastic strain and the equivalent von Mises stress are selected (post codes 7 and 17). As nodal post file variables, the displacements, velocities, contact normal stress, contact normal force and contact status are selected (nodal post codes 1, 28, 34, 35, and 38). The possibility to select nodal variables allows you to reduce the size of the post file by selecting a limited number of nodal variables, or get more detailed information by selecting a large number of variables. The contact status (value 0 or 1) shows if a node is whether or not in contact.

Control

Convergence testing is based on relative displacement changes with a tolerance of 0.01. The solution of a nonpositive definite system is allowed.

Dynamic Change

A time integration is performed over a total time of 0.0008 s with 400 equally sized steps.

Motion Change

The velocity of one of the rigid bodies is set to 50 m/s in negative x-direction.

Results

The deformed mesh at increments 200 and 400 are shown in Figures 8.63-3 and 8.63-4. It should be noted that the deformed shape is affected by the fact that there is only friction with one of the rigid bodies. Finally, Figure 6.22-4 shows which nodes are in contact at the left-hand side of the cylinder by a symbol plot of the contact status at increment 300. Figure 6.22-5 shows the various energy changes during the collapsing process.

Parameters, Options, and Subroutines Summary

Parameter Options	Model Definition Options	History Definition Options
SIZING	SOLVER	TITLE
ELEMENTS	OPTIMIZE	CONTROL
PROCESSOR	CONNECTIVITY	DYNAMIC CHANGE
$NO LIST	COORDINATES	MOTION CHANGE
DYNAMIC	ISOTROPIC	CONTINUE
CONSTANT DILATATION	WORK HARD,DATA	
PLASTICITY	CONTACT	
ALL POINTS	NO PRINT	
LUMP	POST	
END	END OPTION	

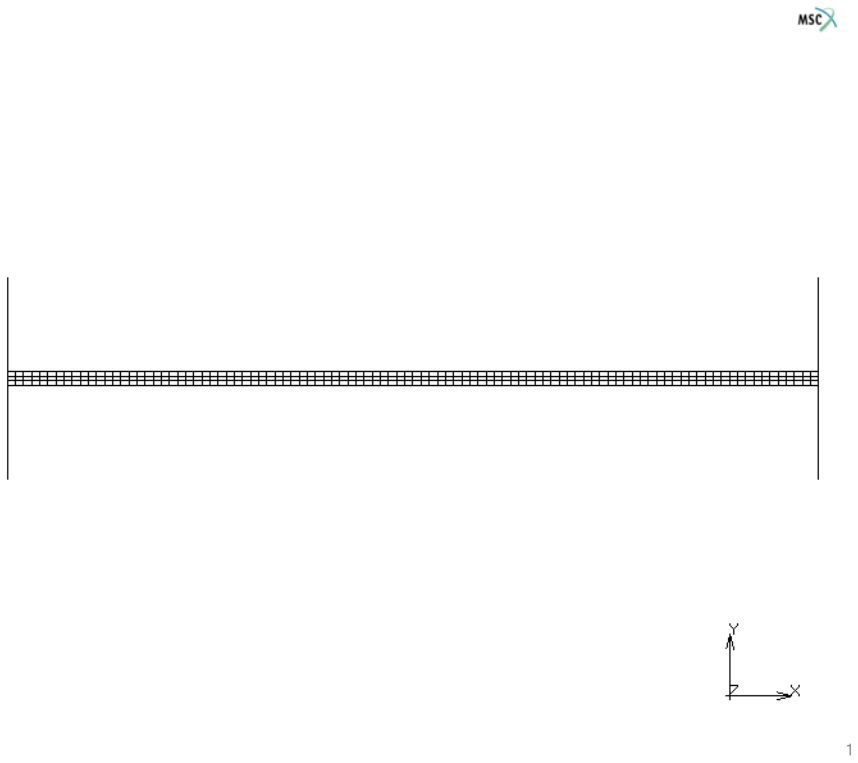


Figure 6.22-1 Finite Element Mesh and Rigid Contact Bodies

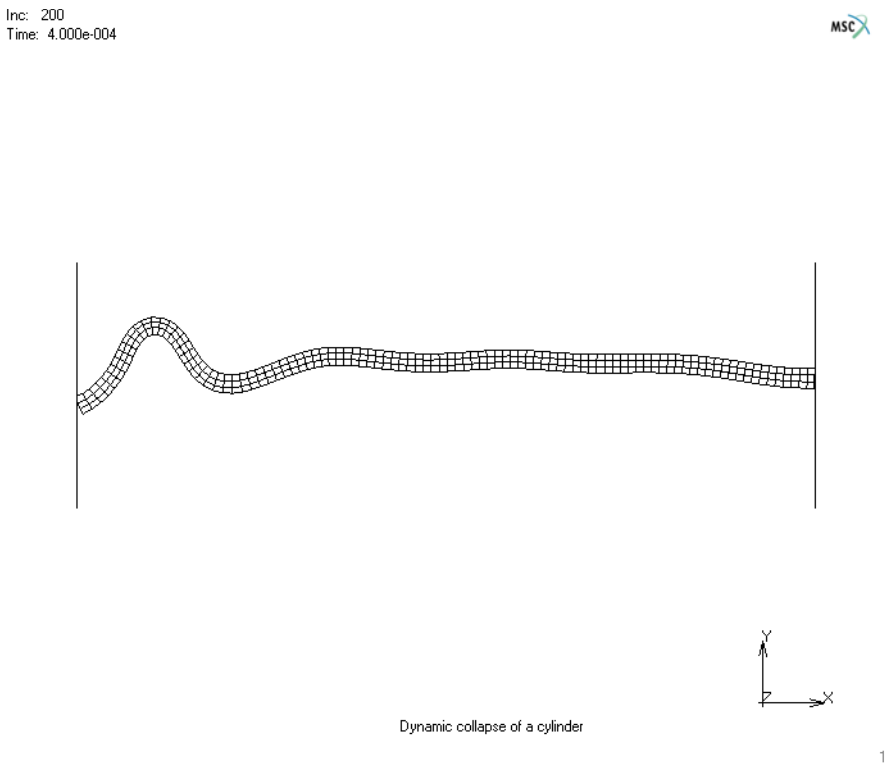


Figure 6.22-2 Deformed Configuration at Increment 200

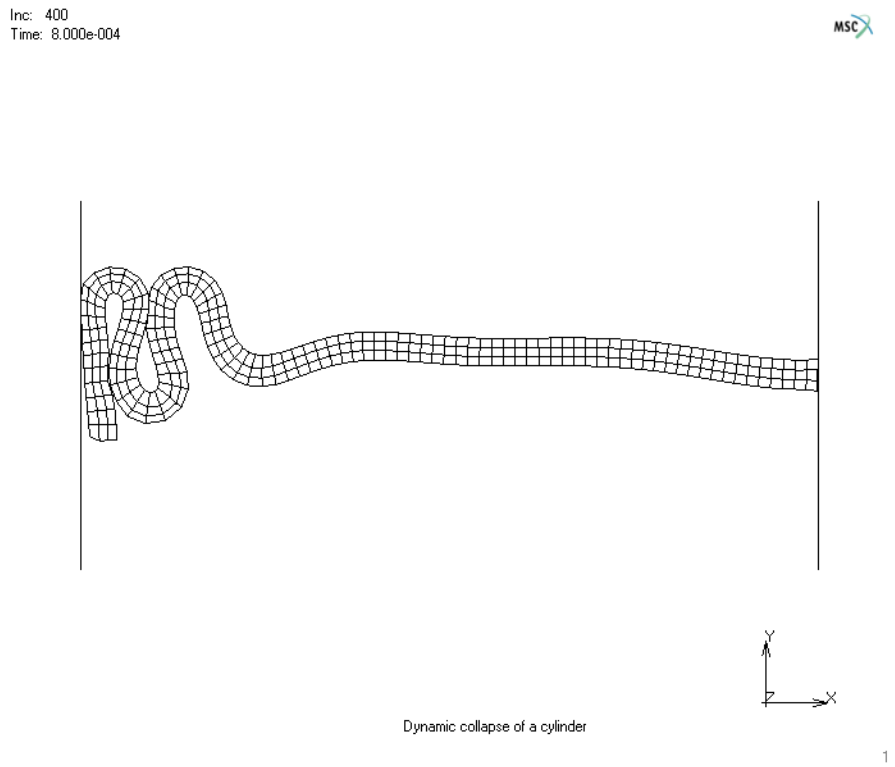


Figure 6.22-3 Deformed Configuration at Increment 400

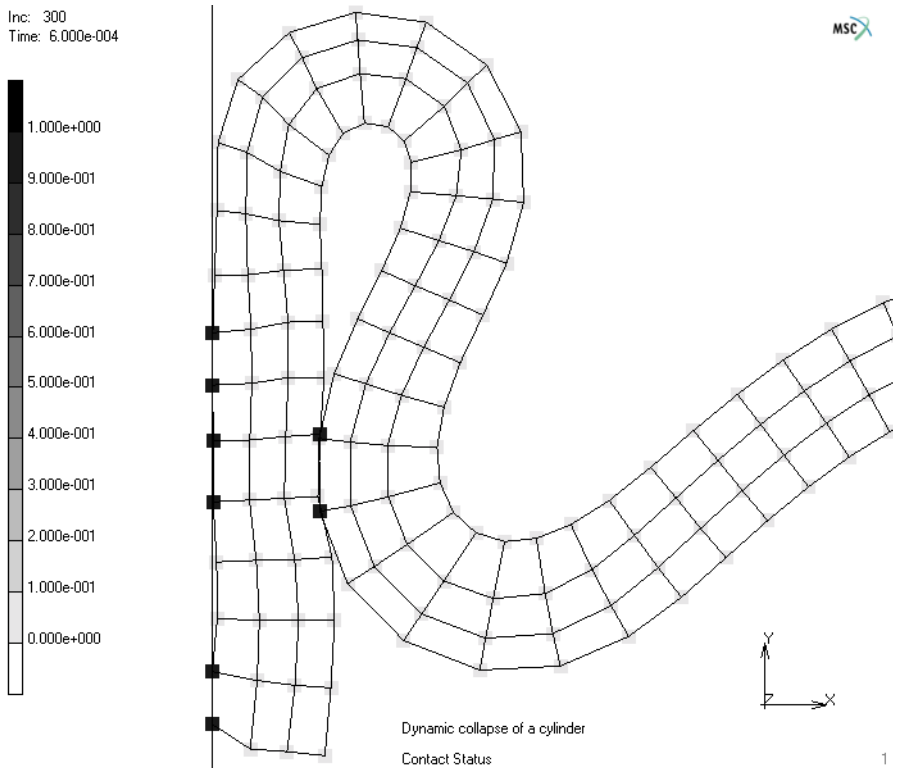


Figure 6.22-4 Contact Status at Increment 300

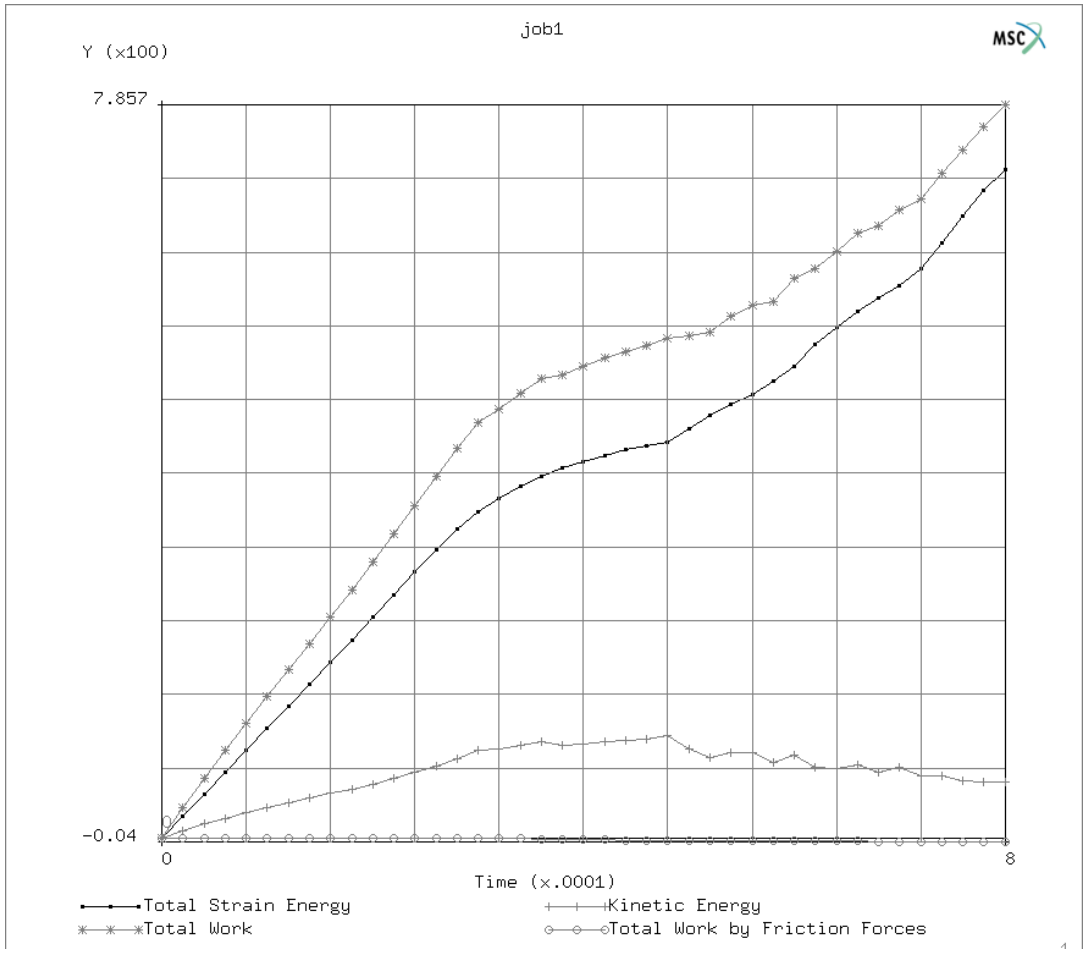


Figure 6.22-5 Various Energies During the Collapsing Process

Volume E: Demonstration Problems, Part IV

Chapter 7
Contact

7.8 Cylinder Under External Pressure (Fourier Analysis)

A solid cylinder in plane strain with radius (a) and external pressure (p_o) is elastically analyzed. Love [1] gives the solutions to the first and second modes of this problem as follows:

$\sigma_{rr} = p_o\, r \cos\theta$

$\sigma\theta\theta = 3p_o\, r \cos\theta$

$\sigma_r\theta = p_o\, r \sin\theta$

for the first mode, and

$\sigma_{rr} = p_o \cos 2\theta$

$$\sigma_{\theta\theta} = p_o\left(\frac{2r^2 - a^2}{a^2}\right)\cos 2\theta$$

$$\sigma_{r\theta} = p_o\left(\frac{r^2 - a^2}{a^2}\right)\sin 2\theta$$

for the second mode. It should be noted that for the first mode, the condition $\sigma_{rr}(a) = p_o \cos\theta$ requires that $\sigma_r\theta(a) = p_0 \sin\theta$, where “a” is 1 inch. Two Fourier series are used for expansion of the 100 psi pressure loading. One series is for the cosine terms and the other for the sine terms. Three different methods, as shown in Problems 7.8a, e7.8b and e7.8c are demonstrated in describing the series. Comparison of the results with Love’s [1] exact solution is presented.

This problem is modeled using the three techniques summarized below.

Data Set	Element Type(s)	Number of Elements	Number of Nodes	Differentiating Features
e7x8a	62	10	53	Fourier coefficients input
e7x8b	62	10	53	Define nonsymmetric
e7x8c	62	10	53	User subroutine UFOUR

Element

Element type 62, the axisymmetric quadrilateral element for arbitrary loading, is used here. Details on this element are found in *MSC.Marc Volume B: Program Input*.

Model

The geometry and mesh used are shown in Figure 7.8-1. The solid cylinder has a height of 0.1 inch and a radius of 1.0 inch. The mesh has 10 elements and 53 nodes.

Geometry

This option is not required for this problem.

Material Properties

The elastic material data assumed for this example is Young's modulus (E) is 30. x 10^6 psi and Poisson's ratio (ν) is 0.25.

Loading

The 100 psi external pressure is specified as a distributed load (IBODY=0) and associated with Fourier series number 1. The -100 psi shear is specified as a uniform load in the circumferential direction (IBODY=14) and associated with Fourier series number 2. Only element 10 is specified with the above loads using the DIST LOAD option.

Boundary Conditions

All nodes on the plane Z = 0. and Z = 0.1 are constrained in the axial direction such that only radial motion is permitted. Nodes 1, 2, and 3 on the plane R = 0 are also constrained in the radial direction due to symmetry.

Fourier

Three different ways are used to describe the series:

1. Specify the first two nonzero terms for series number 1 by evaluating the following integral:

$$a_n = \frac{1}{\pi}\int_o^{2\pi} \begin{pmatrix} \cos\theta \\ \cos 2\theta \end{pmatrix} \cos\, n\theta\, d\theta = \begin{cases} 0, \text{ all n except} \\ 1, n = 1, \text{ and } 2 \end{cases}$$

and the first nonzero term for series number 2 by evaluating the following integral:

$$b_n = \frac{1}{\pi}\int_o^{2\pi} \sin\,\theta\, \sin\, n\theta\, d\theta = \begin{cases} 0, \text{ all n except} \\ 1, n = 1 \end{cases}$$

2. Describe the function F(θ) which is to be expanded into a Fourier series by an arbitrary number (say 5) of [θ, F(θ)] pairs of data.
3. Use user subroutine UFOUR to generate an arbitrary number of [θ,F(θ)] pairs and let Marc calculate the Fourier series coefficients. Five pairs of [θ,F(θ)] are defined in this example.

It should be pointed out that five pairs of [θ,F(θ)] have been chosen for demonstration only. It is easy to add more by changing the number 5 in the user subroutine UFOUR. An increased number of [θ,F(θ)] pairs would yield better results in comparison with the exact coefficient evaluations.

Results

The results for the radial and circumferential stresses of Problem e7.8a and Love's exact solution are plotted in Figure 7.8-2 and Figure 7.8-3. They indicate that the finite element solutions are in good agreement with the exact solutions.

Reference

Love, A.E.H., *A Treatise on the Mathematical Theory of Elasticity*, Dover, New York.

Parameters, Options, and Subroutines Summary

Example e7x8a.dat:

Parameters	Model Definition Options
ELEMENT	CONNECTIVITY
END	CONTROL
FOURIER	COORDINATE
SIZING	DIST LOADS
TITLE	END OPTION
	FIXED DISP
	FOURIER
	ISOTROPIC
	RESTART

Example e7x8b.dat:

Parameters	Model Definition Options
ELEMENT	CONNECTIVITY
END	CONTROL
FOURIER	COORDINATE
SIZING	DIST LOADS
TITLE	END OPTION
	FIXED DISP
	FOURIER
	ISOTROPIC
	RESTART

Example e7x8c.dat:

Parameters	Model Definition Options
ELEMENT	CONNECTIVITY
END	CONTROL
FOURIER	COORDINATE
SIZING	DIST LOADS
TITLE	END OPTION
	FIXED DISP
	FOURIER
	ISOTROPIC
	RESTART

User subroutine in u7x8c.f:

UFOUR

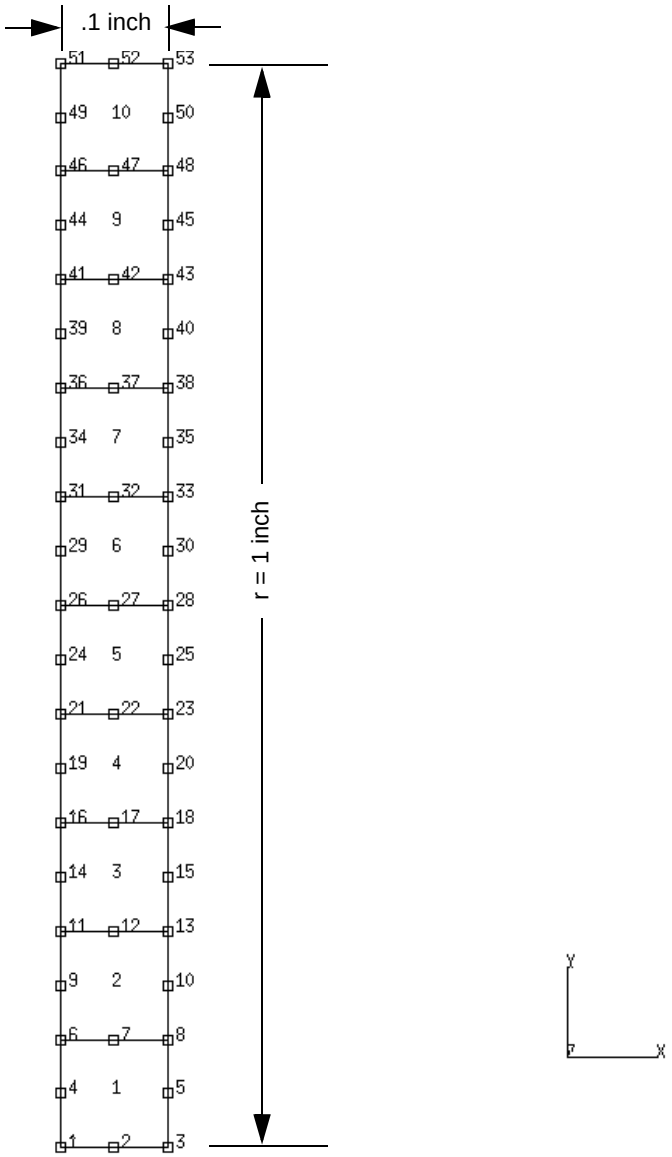


Figure 7.8-1 Cylinder and Mesh

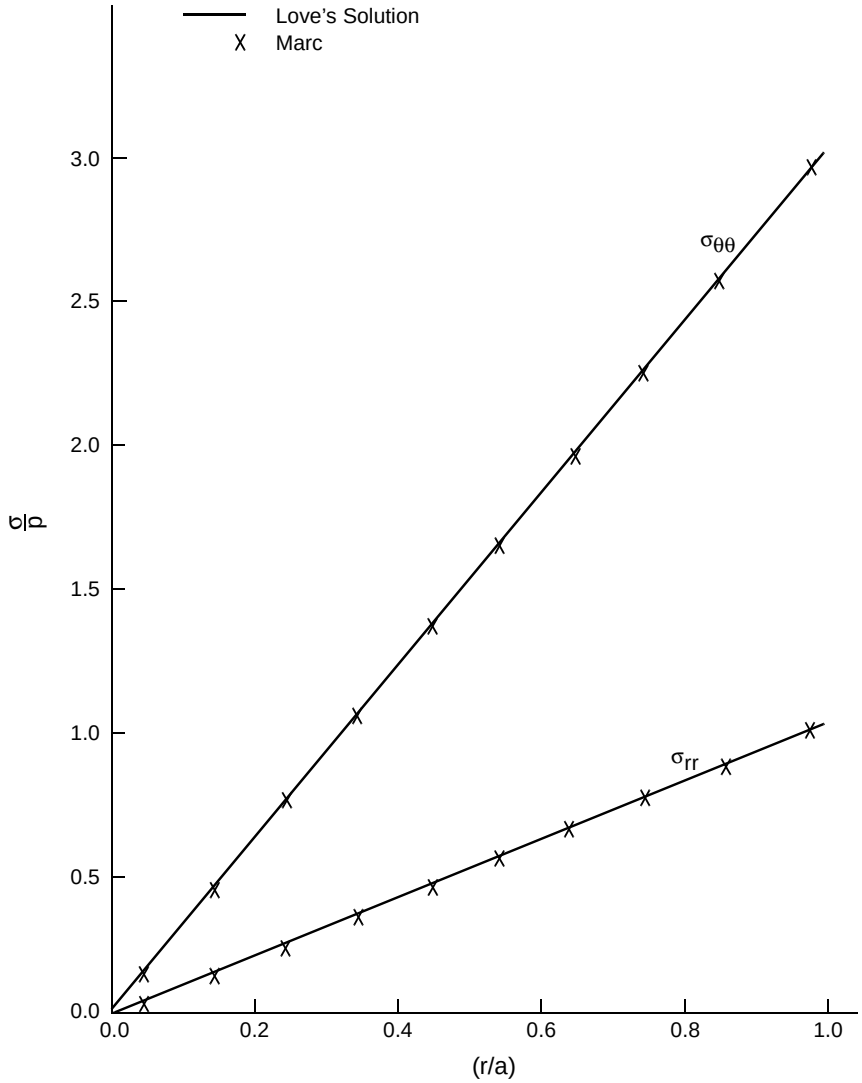


Figure 7.8-2 First Mode Solid Cylinder Plane Strain

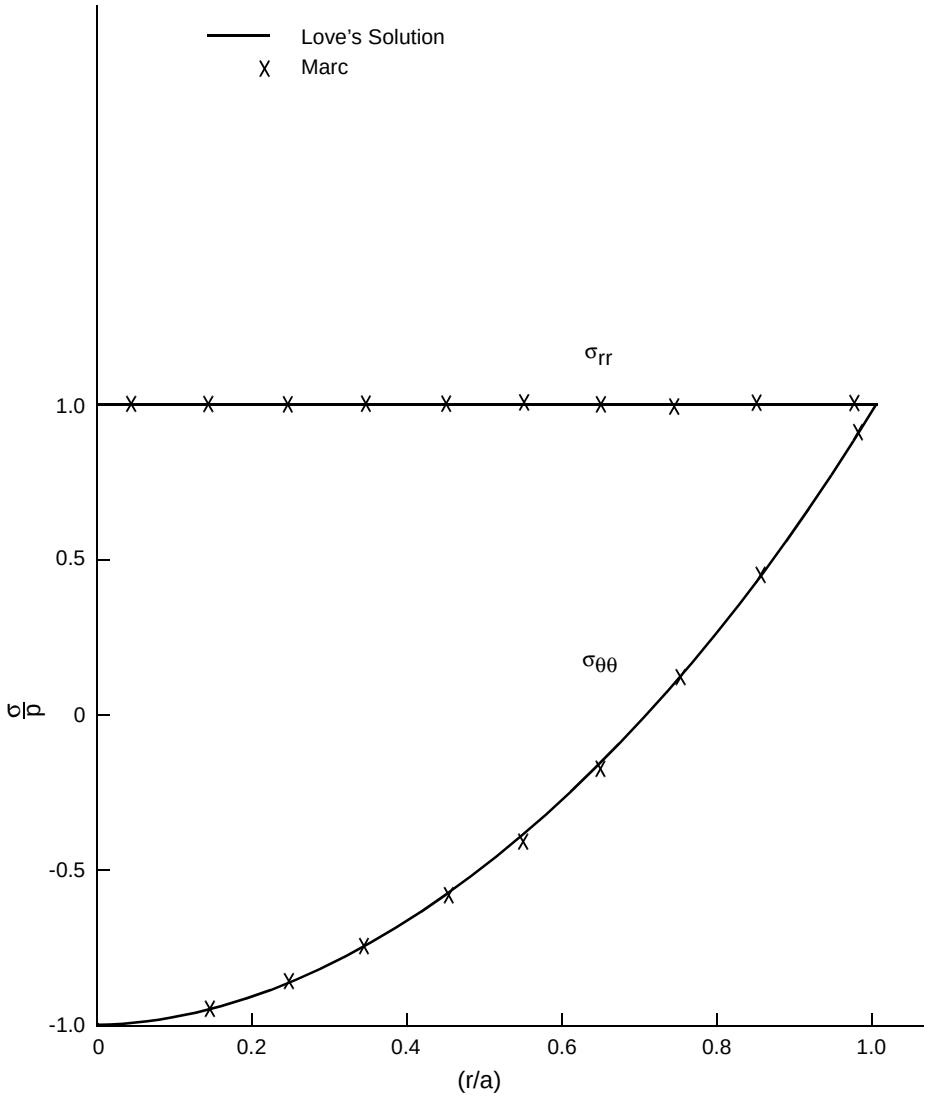


Figure 7.8-3 Second Mode Solid Cylinder Plane Strain

7.13 Analysis of Pipeline Structure

Marc beam element type 14 and pipe-bend element type 17 are utilized for the elastic analysis of a pipeline structure subjected to either in-plane or out-of-plane bending. The structure is loaded until the limit load is reached.

Model

There are a total of 20 elements in the model, of which 6 are type 14 and 14 are type 17. A total of 26 nodes are used. The dimension of the pipeline structure and a finite element mesh are shown in Figure 7.13-1.

Material Properties

The Young's modulus and Poisson's ratio of the pipeline material are 155.53×10^3 (ksi) and 0.3, respectively.

Tractions

An out-of-plane moment of 2.06×10^7 (in-kips) is applied at node 1 in the first analysis. As shown in Figure 7.13-1, the applied load is a moment about the y-axis (the fifth degree of freedom of node 1). The load is increased to a final load of 3.71×10^7 (in-kips) by increment 8. In the second analysis, an in-plane moment of 1.37×10^7 (in-kips) is applied at node 1. The applied load is about the z-axis (the sixth degree of freedom of node 1). The load is increased to a final load of 2.87×10^7 (in-kips) by increment 11.

Boundary Conditions

All degrees of freedom of node 26 are constrained for the fixed-end condition.

Geometry

The wall thickness and mean radius of the beam elements (element type 14) are:

For Elements 1, 2, 19, and 20:

Wall Thickness= 8.8 inches
Mean Radius = 275 inches

For Elements 3 and 18:

Wall Thickness= 10.4 inches
Mean Radius = 274.5 inches

For the pipe-bend elements (element type 17) the geometry data are:

Pipe thickness, t = 10.4 inches
The angular extent of the pipe-bend section, $\phi = 90°$
The radius to the center of the pipe in the r-z plane, R= 838.2 inches

Results

In both analyses, the load is scaled such that incipient yield occurs at increment 1. The loading was increased until the limit load was reached. This was due to an inability to obtain a convergent solution. At the limit load, plasticity had occurred through all 11 layers through the thickness of the elbow section. Figure 7.13-2 shows the load-displacement results of this analysis. The special pipe bend element (type 17) allows the analyst to examine the ovalization of the cross section of the pipe. Using the SECTIONING option in the plot description section, we can examine this effect. Figure 7.13-3 shows the ovalization due to the two types of loading conditions.

Parameters, Options, and Subroutines Summary

Example e7x13b.dat:

Parameters	Model Definition Options	History Definition Options
ELEMENT	CONNECTIVITY	AUTO LOAD
ELSTO	CONTROL	CONTINUE
END	COORDINATE	PROPORTIONAL INCREMENT
SCALE	END OPTION	
SIZING	FIXED DISP	
TITLE	GEOMETRY	
	ISOTROPIC	
	POINT LOAD	
	RESTART	
	TYING	

Example e7x13c.dat:

Parameters	Model Definition Options	History Definition Options
ELEMENT	CONNECTIVITY	AUTO LOAD
ELSTO	CONTROL	CONTINUE
END	COORDINATE	PROPORTIONAL INCREMENT
SCALE	END OPTION	
SIZING	FIXED DISP	
TITLE	GEOMETRY	
	ISOTROPIC	
	POINT LOAD	
	RESTART	
	TYING	

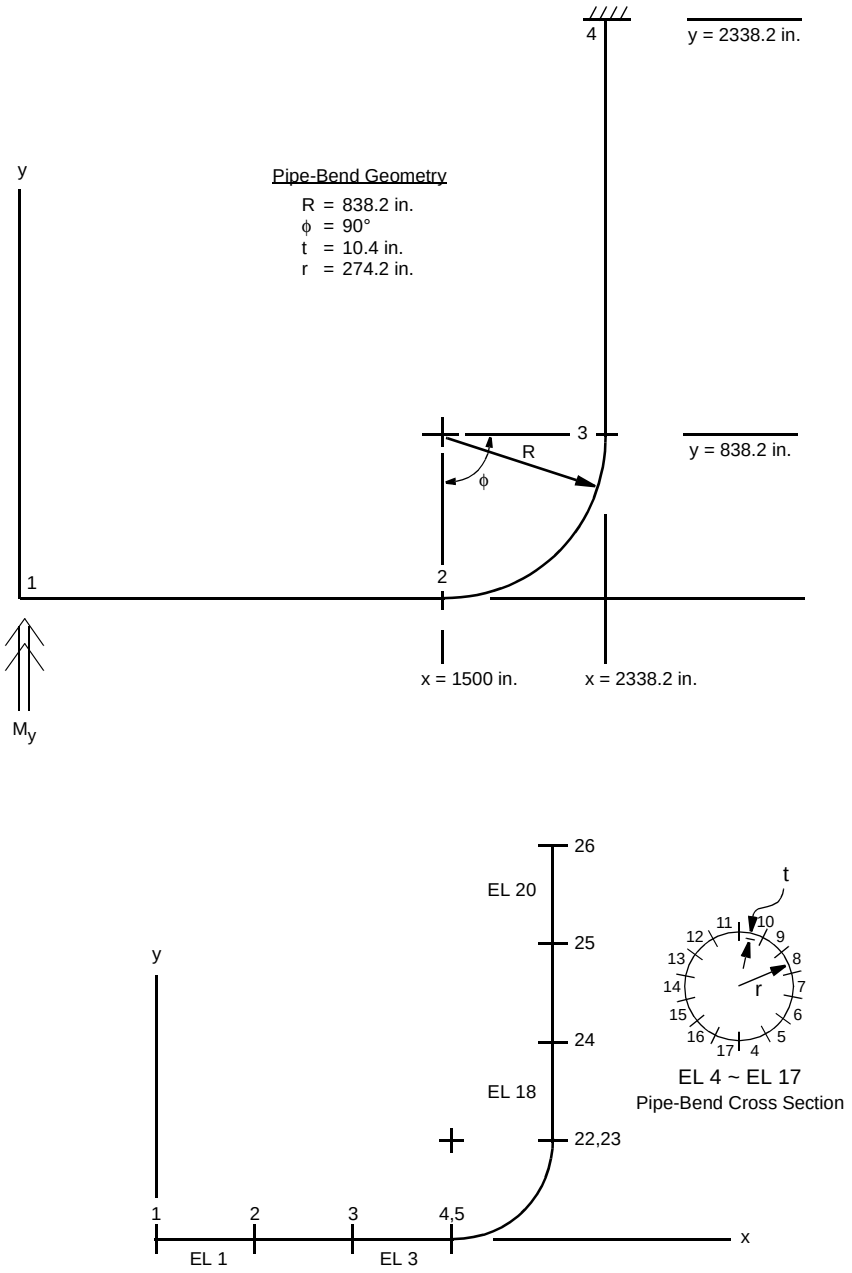


Figure 7.13-1 Pipe Line Geometry and Model

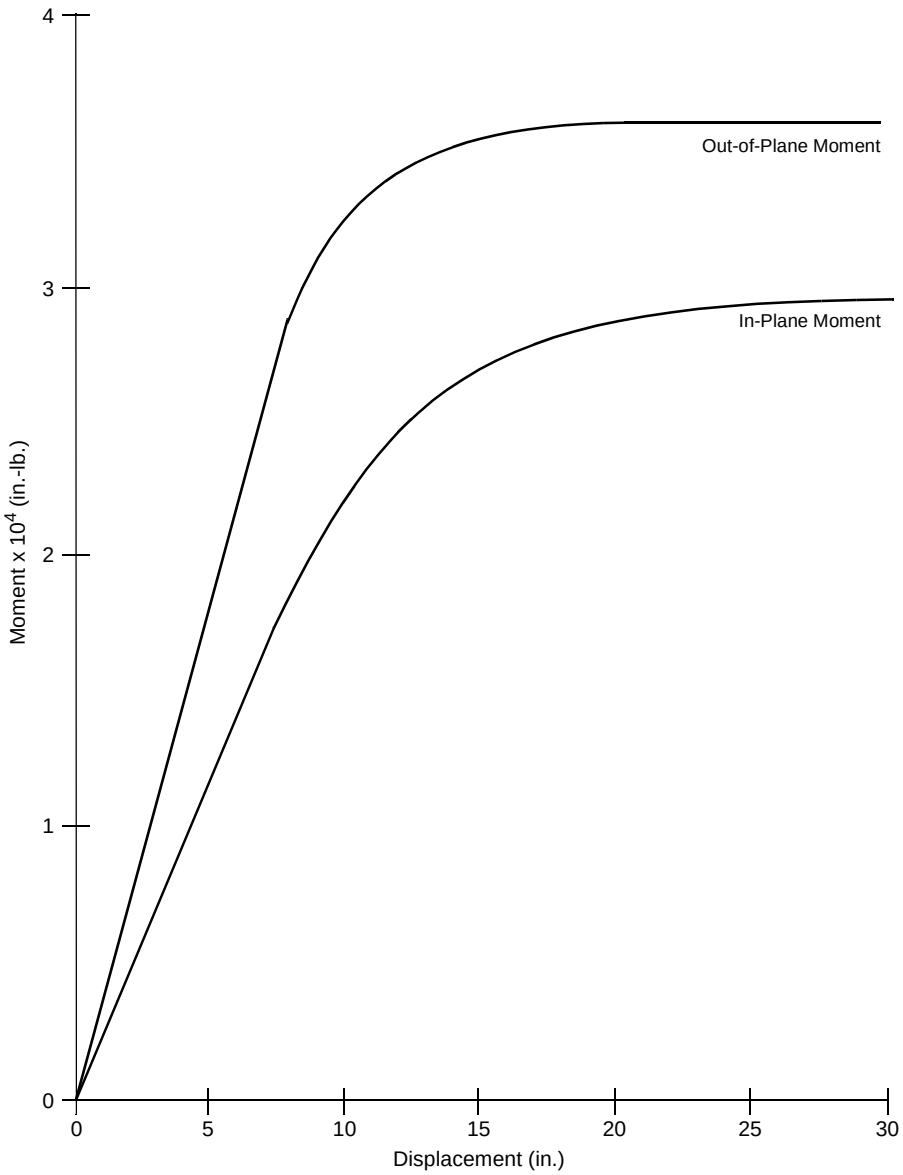
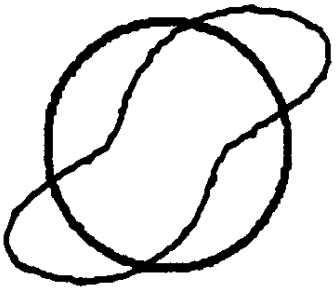
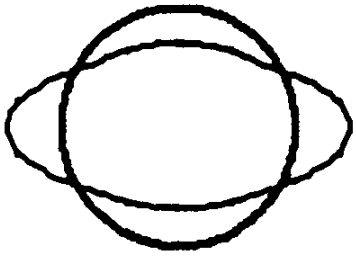


Figure 7.13-2 Load vs. Displacement



(a) Due to Out-of-Plane Moment



(b) Due to In-Plane Moment

Figure 7.13-3 Ovalization Behavior due to Out-of-Plane and In-Plane Moments

7.27 Twist and Extension of Circular Bar of Variable Thickness at Large Strains

This problem illustrates the use of Marc element 67, higher order axisymmetric with twist element, for a large strain elastic analysis of a circular bar of variable thickness. The bar is subjected to both a twist moment and an axial force at the free end of the circular bar. The tying constraint option is used to insure that the cross section at the small end of the bar remains flat. The material is modeled using Ogden model. The ELASTICITY,2 option is used to activate the updated Lagrangian formulation.

Element

Element type 67, an 8-node axisymmetric element with twist, is used in this example.

Model

There are 12 elements, with a total of 53 nodes. Dimensions of the circular bar and the finite element mesh are shown in Figure 7.27-1.

Material Properties

Ogden material properties are given as:

$\mu_1 = 16$ lb/in^2, $\alpha_1 = 2$, $\mu_2 = -4$ lb/in^2, $\alpha_2 = -2$.

Boundary Conditions

Degrees of freedom u and w are 0 at the fixed end (nodes 1-5). Symmetry conditions are imposed at r = 0 (v = 0).

Loading

In each increment, a 10 pound point load in the positive x-direction and a 4 inch per pound torque is applied at node 49. Due to the applied tying, the point load is distributed over the whole cross section.

Tying

Tying type 1 is used at the free end to simulate a generalized plane-strain condition in the z-direction. The tied nodes are 50, 51, 52, and 53 and the retained node is 49.

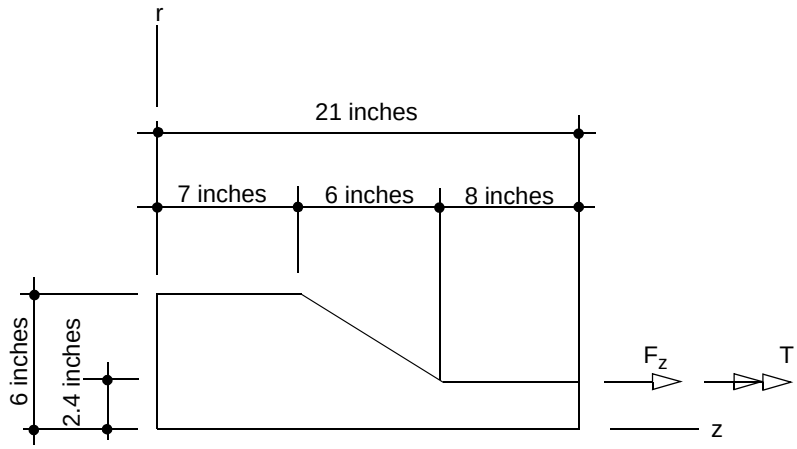
Results

The deformed mesh and the distribution of Equivalent von Mises stress is depicted in Figure 7.27-2.

Parameters, Options, and Subroutines Summary

Example e7x27.dat:

Parameters	Model Definition Options	History Definition Options
ALIAS	CONNECTIVITY	AUTO LOAD
ELASTICITY	COORDINATE	CONTINUE
ELEMENTS	END OPTION	CONTROL
END	FIXED DISP	POINT LOAD
LARGE DISP	ISOTROPIC	
SIZING	OGDEN	
TITLE	POINT LOAD	
	POST	
	TYING	



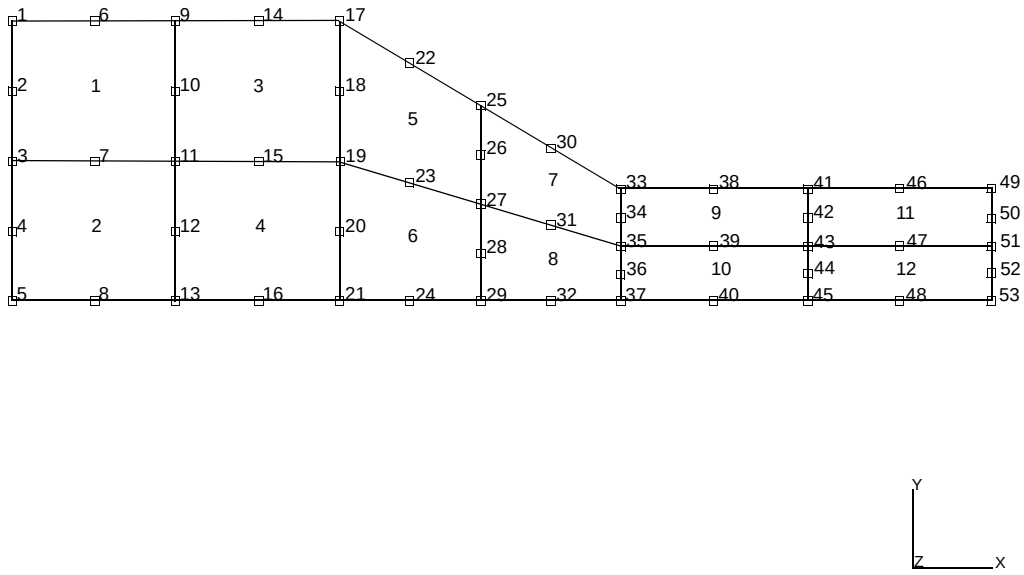


Figure 7.27-1 Circular Bar and Mesh

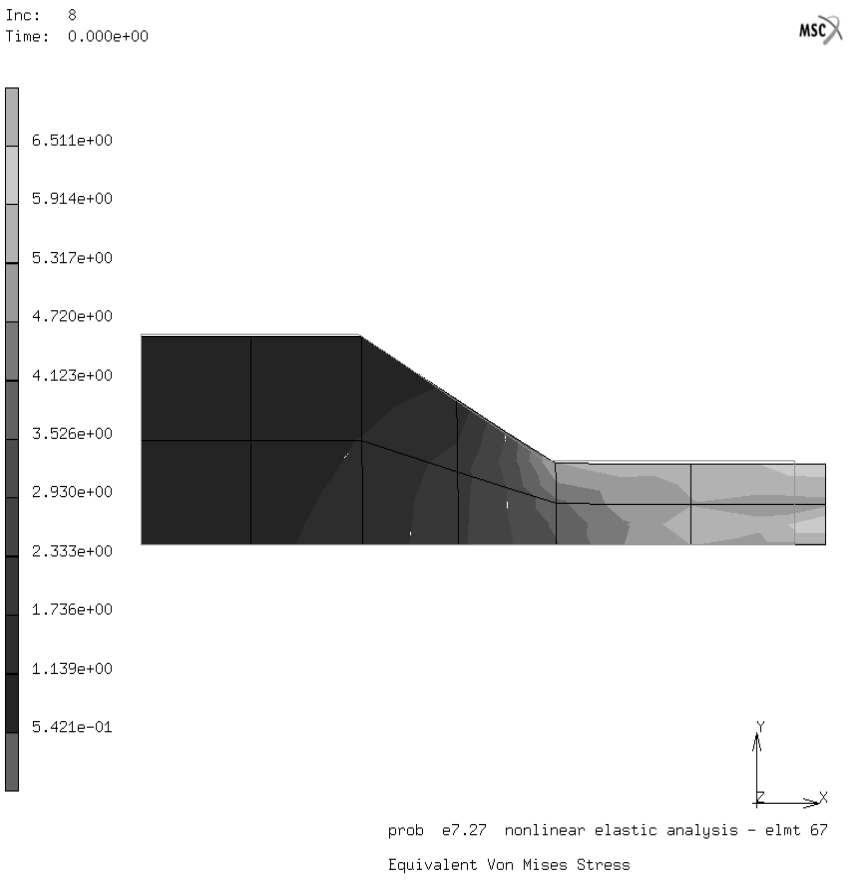


Figure 7.27-2 Deformed Mesh and Distribution of Equivalent von Mises Stress

7.28 Analysis of a Thick Rubber Cylinder Under Internal Pressure

In this example, the deformation of a thick rubber cylinder under internal pressure is modeled.

This problem illustrates the use of Marc elements types 10, 28, 55, and 116 (4- and 8-node axisymmetric elements with only displacement degrees of freedom at nodes) for rubber materials. Option ELASTICITY,2 is invoked to activate Marc updated Lagrangian formulation. The rubber material is modeled with either the Ogden or Mooney material models.

Data Set	Element Type(s)	Number of Elements	Number of Nodes	Differentiating Features
e7x28a	10	4	20	Odgen
e7x28b	116	4	10	Mooney
e7x28c	28	4	23	Ogden
e7x28d	55	4	23	Ogden

Element

Library element 10 is a 4-node bilinear axisymmetric element with displacements in radial and axial directions as degrees of freedom. Library element 116 is a 4-node bilinear, reduced integration, axisymmetric element with displacements in radial and axial directions as degrees of freedom. Library element 28 is a 8-node axisymmetric element with displacements in radial and axial directions as degrees of freedom. Library element 55 is a 8-node, reduced integration, axisymmetric element with displacements in radial and axial directions as degrees of freedom.

Model

The cylinder has an internal radius of 1 mm and an external radius of 2 mm. Figure 7.28-1 shows the initial mesh for the data sets using 8-noded elements.

Material Properties

The Mooney material properties are given as:

$C_1 = 8$ N/mm^2, $C_2 = 2$ N/mm^2;

The Ogden material properties are given as:

$\mu_1 = 16$ N/mm^2, $\alpha_1 = 2$, $\mu_2 = 4$ N/mm^2, $\alpha_2 = -2$.

The bulk modulus is chosen as 200000 N/mm^2, resulting in the ratio of K/G being 10000. The material is therefore highly incompressible. Both materials are equivalent.

Loads

A uniformly distributed internal pressure of 11.5 N/mm^2 is applied on element number 1. This load is applied in increment zero. In Marc, increment zero is treated as linear. So an additional increment, with no additional load, is used to bring the solution to the correct nonlinear state.

Boundary Conditions

u = 0 on the planes z = 0 and z = 1.0 to simulate a plane strain condition.

Results

A. 8-Node Model (Element Type 28 and 55)

After the linear elastic step (increment 0), the radial displacements of the inside nodes for both elements 28 and 55 are 0.3833 mm.

They are the same as the analytical solution which predicts a radial displacement of 0.3833 mm.

After ten iterations, the radial displacement at the inside node is 1.0057 mm and the corresponding pressure can be computed from the following expression:

$$P = (C_1 + C_2)\left[\log\left(\frac{B^2a^2}{A^2(B^2 - A^2 + a^2)}\right) + \frac{(a^2 - A^2)(B^2 - A^2)}{a^2(B^2 - A^2 + a^2)}\right]$$

where A and B are the inner and outer radii of the cylinder in the undeformed state, “a” is the inner radius in the deformed state, and C_1 and C_2 are material constants.

The computed pressure of 11.5 N/mm^2 is in very good agreement with the prescribed value of 11.5 N/mm^2.

B. 4-Node Model (Element Type 10 and 116)

After the linear elastic step (increment 0), the radial displacements of the inside nodes (nodes 1 and 6) are 0.3817 mm (for element type 10) and 0.3834 mm (for element type 116) respectively.

Agreement with analytical solution of 0.3833 mm is good. After ten iterations, the radial displacement at inside node is 1.0068 mm, and the corresponding pressure is 11.5 N/mm^2 for element 10. For element 116, the displacement at the inside node is 1.0063 mm and the corresponding pressure is 11.5 N/mm^2. Agreement with prescribed value of 11.5 N/mm^2 is excellent.

Parameters, Options, and Subroutines Summary

Example e7x28a.dat:

Parameters	Model Definition Options	History Definition Options
ALIAS	CONNECTIVITY	CONTINUE
ELASTICITY	CONTROL	DIST LOAD
ELEMENTS	COORDINATES	
END	DIST LOAD	
FOLLOW FOR	END OPTION	
LARGE DISP	FIXED DISP	
SIZING	OGDEN	
TITLE	POST	

Example e7x28b.dat:

Parameters	Model Definition Options	History Definition Options
ALIAS	CONNECTIVITY	CONTINUE
ELASTICITY	CONTROL	DIST LOAD
ELEMENTS	COORDINATES	
END	DIST LOAD	
FOLLOW FOR	END OPTION	
LARGE DISP	FIXED DISP	
SIZING	MOONEY	
TITLE	POST	

Example e7x28c and e7x28d.dat:

Parameters	Model Definition Options	History Definition Options
ALIAS	CONNECTIVITY	CONTINUE
ELASTICITY	CONTROL	DIST LOAD
ELEMENTS	COORDINATES	
END	DIST LOAD	
FOLLOW FOR	END OPTION	
LARGE DISP	FIXED DISP	
SIZING	OGDEN	
TITLE	NODE FILL	
	POST	

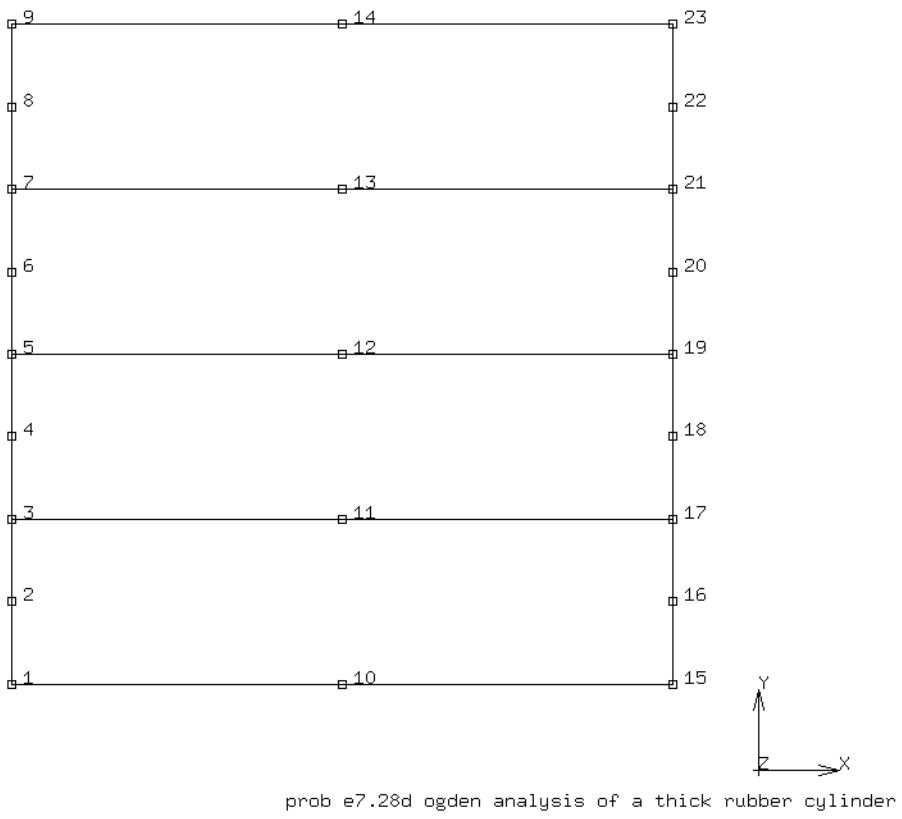


Figure 7.28-1 Cylinder Mesh (8-Node Model)

7.29 3-D Analyses of a Plate with a Hole at Large Strains

This problem simulates the tensile loading of a plate with a hole at large strains. In e7x29a.dat, the HYPOELASTIC option and the user subroutine HYPELA2 are used to define constitutive behavior. Element type 7 is used and the material here is compressible. This job demonstrates the use of kinematics in defining user-defined material behavior. In e7x29b.dat, Element type 117 is used to model the plate (with the user-defined defaults file). The material in e7x29b.dat is modeled using Ogden model and is nearly incompressible.

In e7x29c.dat, element type 157 is used to model the plate. The material in e7x29c.dat is the same as for e7x29b.dat.

This problem is modeled using the two techniques summarized below.

Data Set	Element Type(s)	Number of Elements	Number of Nodes	Differentiating Features
e7x29a	7	92	218	HYPOELASTIC HYPELA2
e7x29b	117	92	218	user default file
e7x29c	157	2208	2902	element type 157

Element

Library element 7 is a 8-node trilinear brick element with global displacements as degrees of freedom. Library element 117 is a 8-node trilinear brick element with reduced integration and global displacements as degrees of freedom. Library element type 157 is a 4+1-node, low-order tetrahedron using the Herrmann formulation.

Model

Due to symmetry of the geometry and loading, a quarter of the actual model is simulated. The finite element model is made up of 92 elements and 218 nodes. The finite element mesh is shown in Figure 7.29-1. The finite element mesh for e7x29c.dat is shown in Figure 7.29-2.

There are a total of 2902 nodes in the mesh. However, 2208 center nodes are condensed out on the element level and do not appear in the global matrix.

Geometry

The model is assumed to be a square of side 2 mm from which a quarter of a circle of radius 0.6 mm has been cut out. The initial thickness is 0.2 mm.

Material Properties

In e7x29a.dat, a quadratic-logarithmic, nonlinear elastic model with the initial bulk modulus of 21666.67 N/mm^2 and the initial shear modulus of 10000.00 N/mm^2 is defined using the HYPOELASTIC option and the user subroutine HYPELA2. In e7x29b.dat, the Ogden parameters are given as μ_1=0.586 N/mm^2, α_1=2.0, μ_2=-0.354N/mm^2, and α_2=-2.0. The initial bulk modulus is 666666.667 N/mm^2. The material properties for e7x29c.dat are the same as for e7x29b.dat.

Boundary Conditions and Loading

In addition to the boundary conditions due to symmetry, the third degree of freedom of the nodes located on the edges of the lower surface are fixed to avoid the rigid body motion in z-direction.

The loading is tensile. In e7x29a.dat, a uniform displacement of 1 mm is applied to one of the plate edges using 20 increments. The macroscopic total logarithmic strain is 40%. In e7x29b.dat, a uniform displacement of 2.5 mm is applied to one of the plate edges using 10 increments. The macroscopic total logarithmic strain is 81%.

In e7x29c.dat, the load in the form of the prescribed displacement is the same as for e7x29b.dat.

Results

The distribution of equivalent von Mises stress and the deformed model for e7x29a.dat after 20 increments is shown in Figure 7.29-3. The deformed model and the contour band plot of x displacements for e7x29b.dat and e7x29c.dat are shown in Figure 7.29-4 and Figure 7.29-5, respectively. Close agreement is observed.

Parameters, Options, and Subroutines Summary

Example e7x29a.dat:

Parameters	Model Definition Options	History Definition Options
ELEMENTS	CONNECTIVITY	AUTO LOAD
END	CONTROL	CONTINUE
LARGE DISP	COORDINATES	DISP CHANGE
PROCESS	END OPTION	
SETNAME	FIXED DISP	
SIZING	GEOMETRY	
TITLE	HYPOELASTIC	
UPDATE	OPTIMIZE	
	POST	

User subroutine in u7x29a.f:

HYPELA2

Example e7x29b.dat:

Parameters	Model Definition Options	History Definition Options
ELEMENTS	CONNECTIVITY	AUTO LOAD
END	CONTROL	CONTINUE
SETNAME	COORDINATES	DISP CHANGE
SIZING	END OPTION	
TITLE	FIXED DISP	
	GEOMETRY	
	OGDEN	
	OPTIMIZE	
	POST	

User-defined Default Input e7x29_def.dat:

Parameters	Model Definition Options
ALL POINTS	END OPTION
ELASTICITY	PARAMETER
END	
LARGE DISP	
PRINT	
PROCESS	

Example e7x29c.dat:

Parameters	Model Definition Options	History Definition Options
ELEMENTS	CONNECTIVITY	AUTO LOAD
END	CONTROL	CONTINUE
LARGE DISP	COORDINATES	DISP CHANGE
SIZING	END OPTION	
TITLE	FIXED DISP	
	NO PRINT	
	OGDEN	
	OPTIMIZE	
	POST	
	SOLVER	

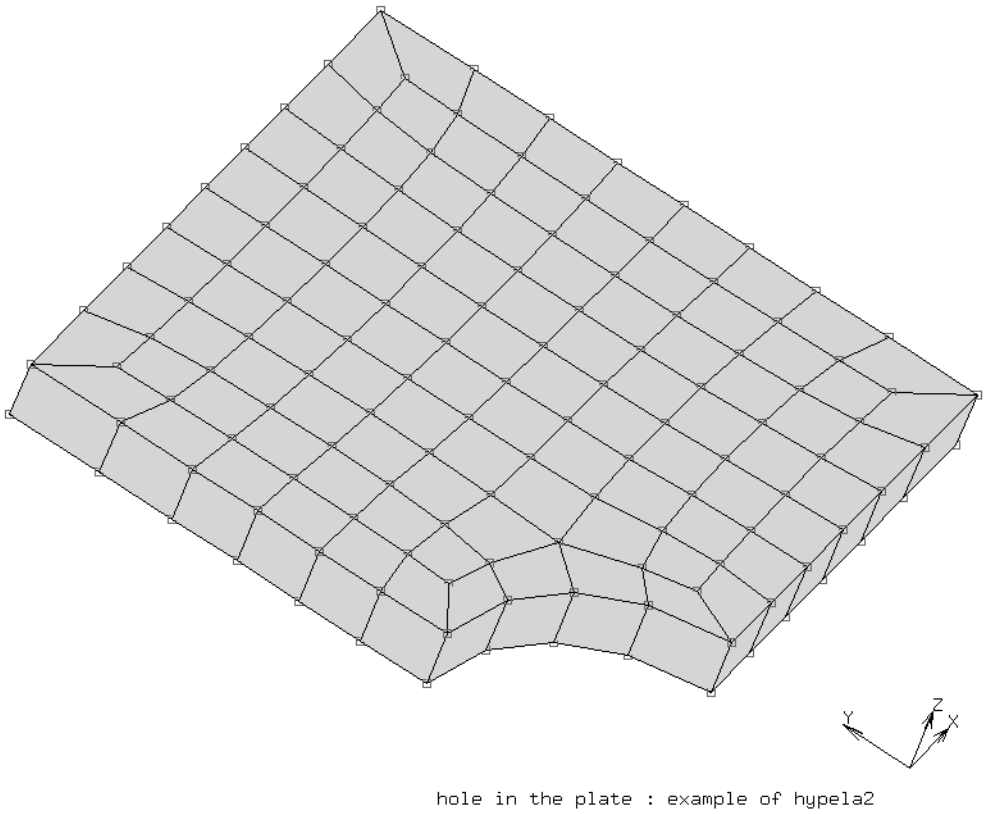


Figure 7.29-1 Initial Mesh for e7x29a.dat and e7x29b.dat

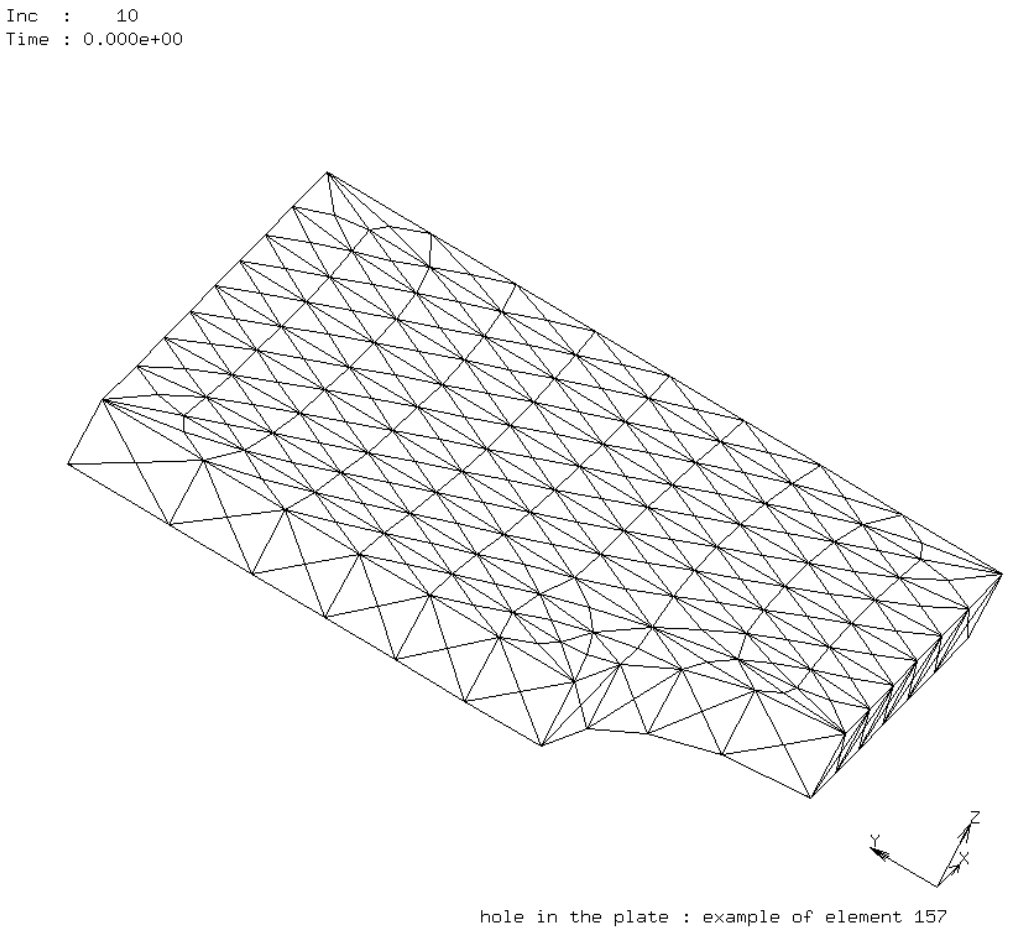


Figure 7.29-2 FE Mesh for e7x29c.dat

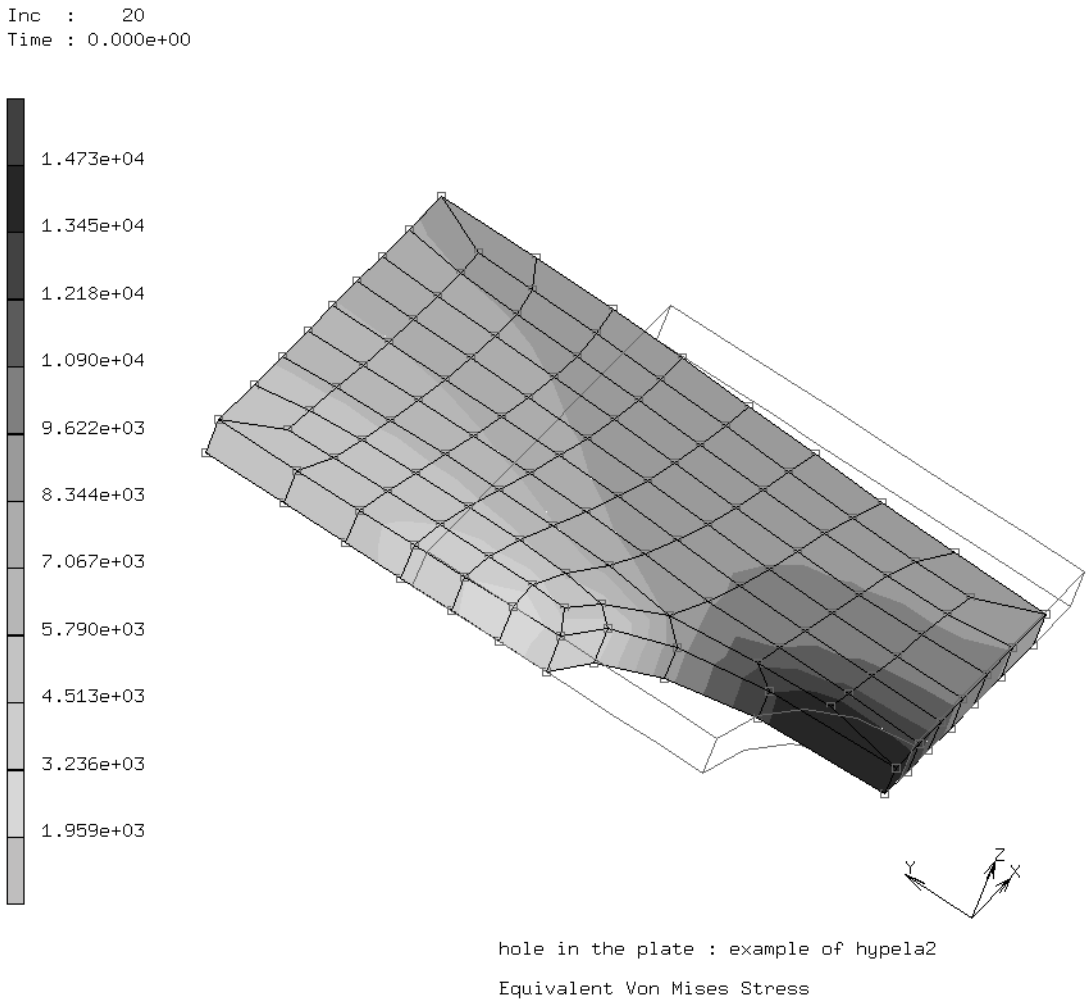


Figure 7.29-3 Deformed Model and Distribution of Equivalent von Mises Stress for e7x29a

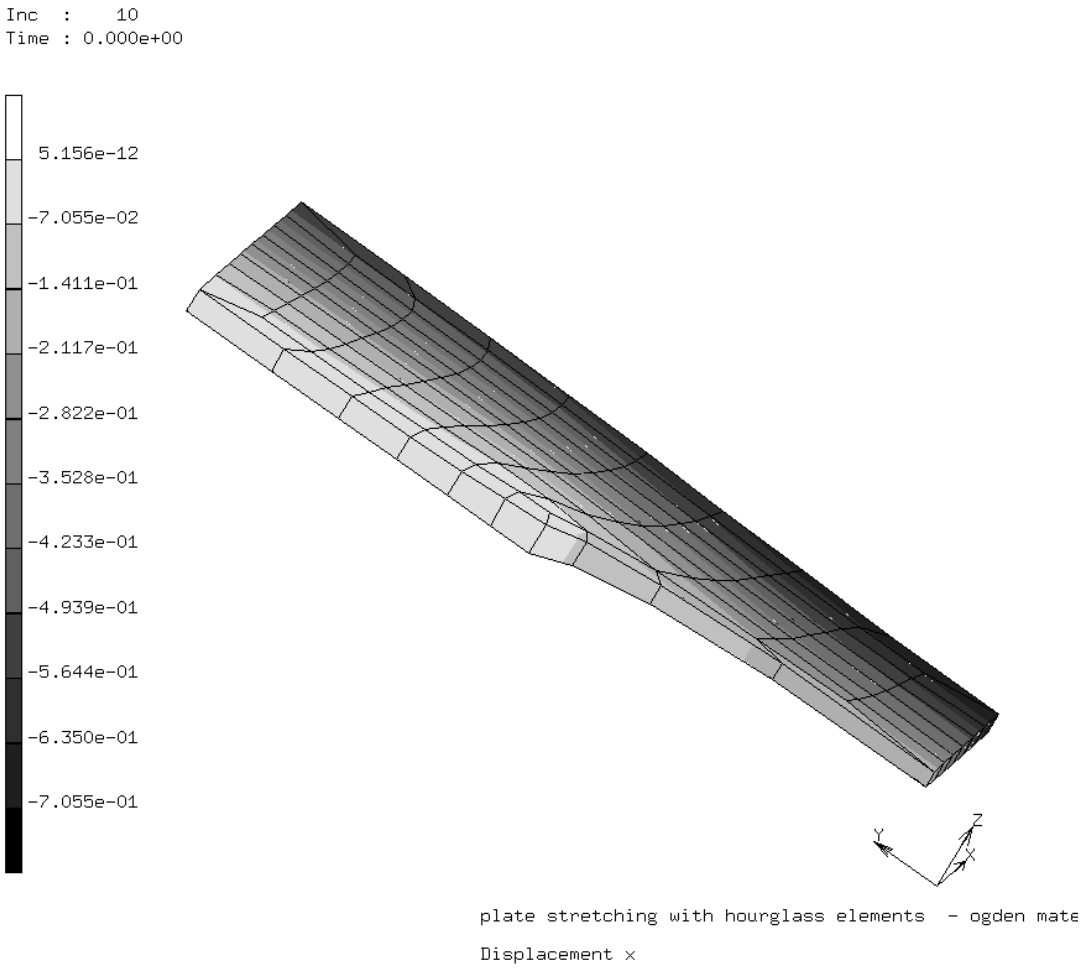


Figure 7.29-4 Deformed Model and Contour Plot of Displacement x for e7x29b.dat

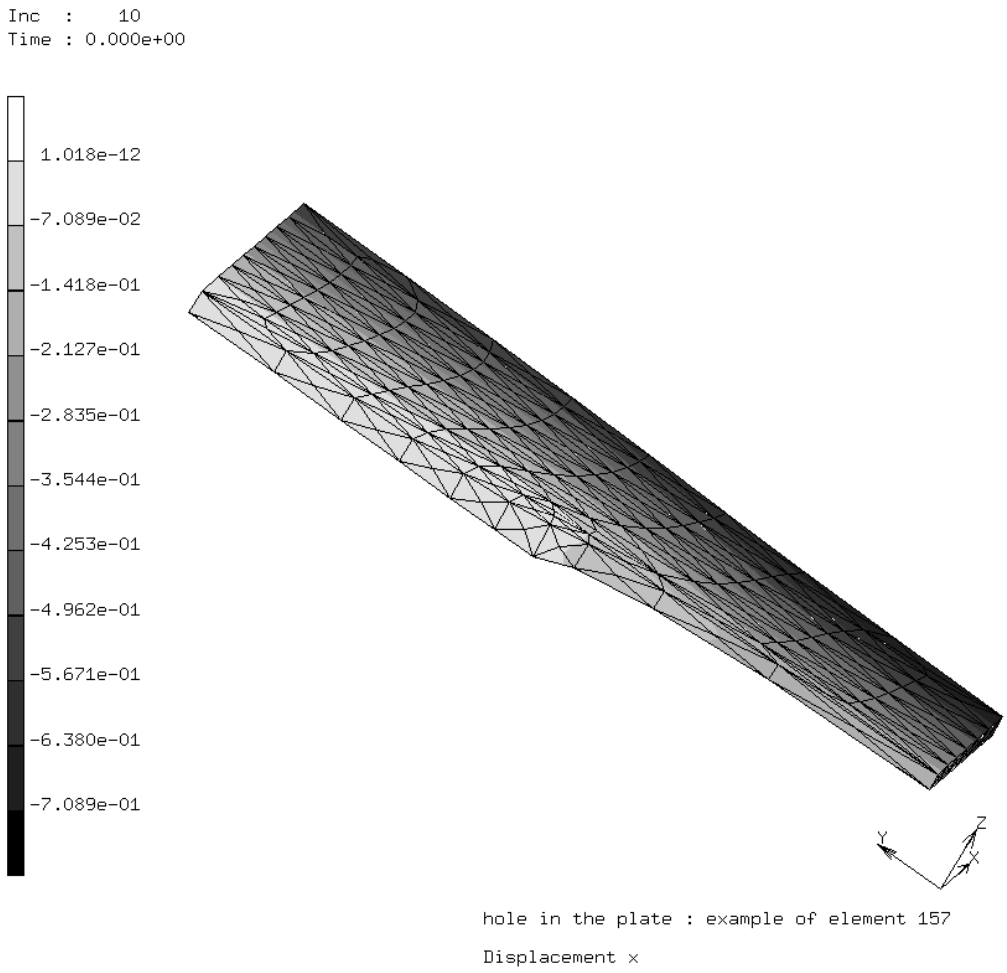


Figure 7.29-5 Deformed Model and Contour Plot of Displacement x for e7x29c.dat

7.32 Structural Relaxation of a Glass Cube

Free structural relaxation of a glass cube subjected a cyclic temperature history, is simulated using Narayanaswamy model. This problem is modeled by means of element type 7.

The annealing of flat glass requires that the residual stresses be of an acceptable magnitude, while the specification for optical glass components usually includes a homogenous refractive index. The design of heat treated processes (for example, annealing) can be accomplished using the Narayanaswamy model. This allows you to study the time dependence of physical properties (for example, volumes) of glass subjected to a change in temperature.

The glass transition is a region of temperature in which molecular rearrangements occur on a scale of minutes or hours, so that the properties of a liquid change at a rate that is easily observed. Below the glass transition temperature, T_g, the material is extremely viscous and a solidus state exists. Above T_g the equilibrium structure is arrived at easily and the material is in liquidus state. Hence, the glass transition is revealed by a change in the temperature dependence of some property of a liquid during cooling. If a mechanical stress is applied to a liquid in the transition region, a time-dependent change in dimensions results due to the phenomenon of visco-elasticity.

If a liquid in the transition region is subjected to a sudden change in temperature, a time-dependent change in volume occurs. The latter process is called structural relaxation. Hence, structural relaxation governs the time-dependent response of a liquid to a change of temperature.

Element

Library element 7 is a 8-node trilinear brick element with global displacements as degrees of freedom.

Model

The side length of the glass cube is 2 mm. Because of the symmetry, only one eighth of the cube is modeled with one brick element.

Material Properties

The instantaneous moduli are given via ISOTROPIC option as: Young's modulus is 5.58E4 N/mm^2; Poisson's ratio is 0.0814. The time dependent values are entered using VISCEPROP option as:

Term No.	Shear Constant	Relaxation Time
1	1.08876E4	9.97000E-2
2	1.09134E4	9.40000E-3
3	3.97320E3	3.00000E-4

The solid and the liquid coefficients of the thermal expansion are chosen as 5.50E-7 and 1.93E-6, respectively. The weights and the reference relaxation times, used to define the response function, for each term in the series are input through SHIFT FUNCTION as:

Term No.	Weight	Reference Relaxation Time
1	1.0800E-1	1.4780E+0
2	4.4300E-1	3.2970E-1
3	1.6600E-1	1.2130E-1
4	1.6100E-1	4.4600E-2
5	4.6000E-2	1,6400E-2
6	7.6000E-2	3.7000E-3

Loads

An initial temperature of 6.20E2 is applied to the glass cube at increment 0. A cyclic temperature history is then applied: At first, the cube is gradually cooled down to 0.20E2 in 100 equal increments; Afterwards, it is heated up to the initial temperature at the equal incremental size.

Boundary Conditions

Boundary conditions are applied to the glass cube according to the symmetry.

Results

Suppose a glass is equilibrated at temperature T_1, and suddenly cooled to T_2 at t_0. The instantaneous change in volume is $\alpha_g(T_2 - T_1)$, followed by relaxation towards the equilibrium value $V(\infty,T_2)$. The total change in volume due to the temperature change is $\alpha_l(T_2 - T_1)$ as shown in Figure 7.32-1b. The rate of volume change depends on a characteristic time called the relaxation time.

The slope of dV/dT changes from the high value characteristic of the fluid α_l to the low characteristic of the glass α_g as shown in Figure 7.32-2. The glass transition temperature T_g is a point in the center of the transition region. The low-temperature slope α_g represents the change in volume V caused by vibration of the atoms in their potential wells. In the (glassy) temperature range, the atoms are frozen into a particular configuration. As the temperature T increases, the atoms acquire enough energy to break bonds and rearrange into new structures. That allows the volume to increase more rapidly, so $\alpha_l > \alpha_g$. The difference $\alpha = \alpha_l - \alpha_g$ represents the structural contribution to the volume.

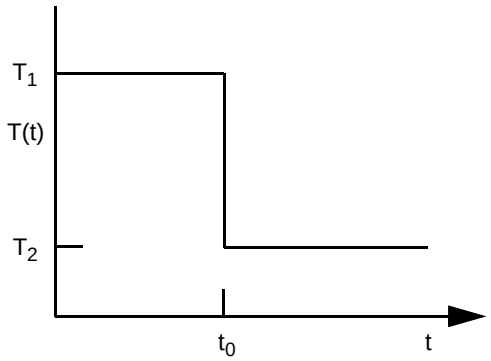
When a liquid is cooled and reheated, a hysteresis is observed.

The volume change of the glass cube with the change of the temperature 1 as calculated by Marc, is illustrated in Figure 7.32-3. The hysteresis shown in Figure 7.32-3 indicates the calculations are in a good qualitative agreement with experimental observations.

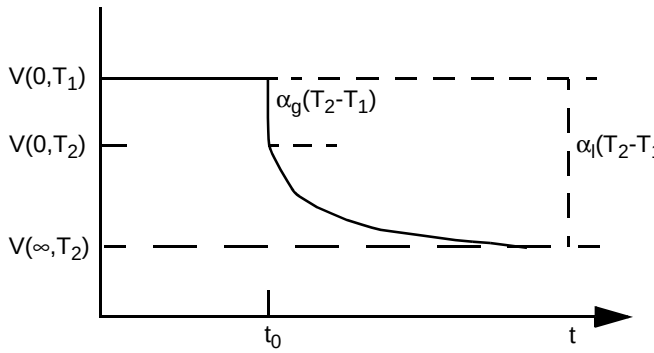
Parameters, Options, and Subroutines Summary

Example e7x32.dat:

Parameters	Model Definition Options	History Definition Options
ALL POINTS	CHANGE STATE	AUTO LOAD
ELEMENTS	CONNECTIVITY	CHANGE STATE
END	COORDINATES	CONTINUE
SETNAME	END OPTIONS	CONTROL
SIZING	FIXED DISP	TIME STEP
STATE VARS	ISOTROPIC	
TITLE	OPTIMIZE	
	PRINT CHOICE	
	POST	
	SHIFT FUNCTION	
	SOLVER	
	VISCEL EXP	
	VISCELPROP	
	$NO PRINT	

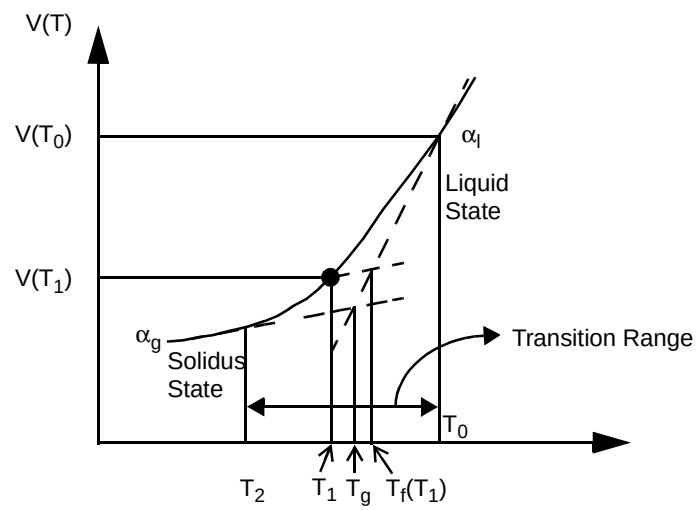


(a) Step Input for Temperature



(b) Volume Change as Function of Temperature

Figure 7.32-1 Structural Relaxation Phenomenon



T_f (T_1) : Fictive Temperature

Figure 7.32-2 Property (Volume) – Temperature Plot

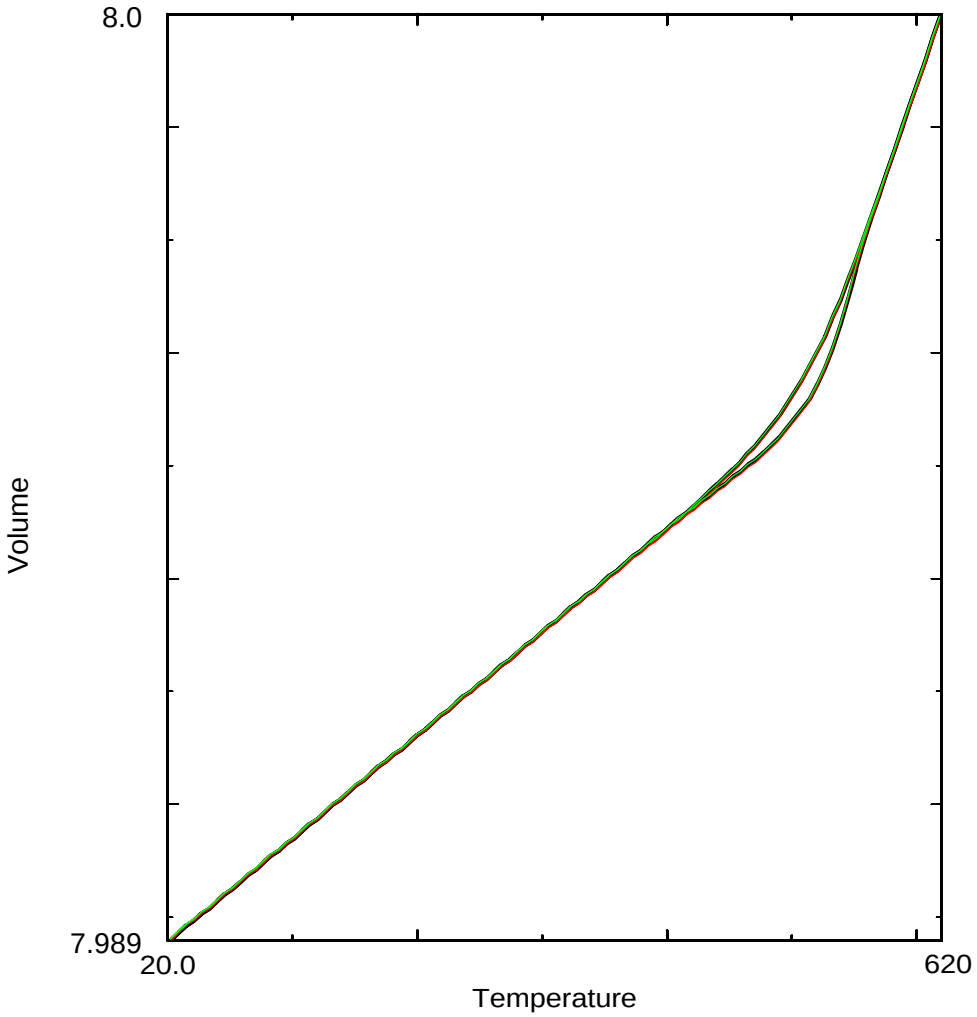


Figure 7.32-3 Volume Change during Cyclic Temperature History

Chapter 8
Advanced Topics

Table 8-1 Recent Analysis Capabilities in Marc

Problem Number	Element Type(s)	Parameters	Model Definition	History Definition	User Subroutines	Problem Description
8.1	26	ELASTIC SUBSTRUCT NEWDB SUPER	SUBSTRUCTURE DIST LOAD SUPERINPUT RESTART	BACK TO SUBS	SSTRAN	Hole in plate. Generate substructure (1-1) and 1-2). Combine substructures, perform analysis. Read RESTART tape, Go back to substructures to obtain results.
8.2	27	SUBSTRUCT NEWDB SUPER J-INT SCALE	SUBSTRUCTURE DIST LOADS SUPERINPUT J-INTEGRAL WORK HARD	AUTO LOAD PROPORTIONAL INC	WKSLP	Double-edge notch specimen using substructure. Elastic region away from the crack is treated as a substructure.
8.3	10 12	SUBSTRUCT NEWDB SUPER	SUBSTRUCTURE SUPERINPUT POINT LOADS GAP DATA	POINT LOADS AUTO LOAD BACK TO SUBS	—	End plate aperture breakaway problem. The rate is treated as a substructure leaving the contact elements to be at highest level.
8.4	26	—	ISOTROPIC CRACK DATA TYING	PROPORTIONAL INC AUTO LOAD	—	Collapse of a notched concrete beam.
8.5	75	PRINT	CONN GENER COMPOSITE ISOTROPIC ORTHOTROPIC	POINT LOADS AUTO INCREMENT	—	Cracking of a plate one-way reinforced using shell elements.
8.6	27 46	PRINT	CONN GENER NODE FILL CRACK DATA ISOTROPIC	POINT LOADS AUTO INCREMENT	REBAR	Cracking of a one-way reinforced plate using rebar elements.
8.7	11 12 39	FINITE UPDATE LARGE DISP COUPLE MESH PLOT	CONTROL FIXED DISP FIXED TEMP INITIAL TEMP ISOTROPIC GAP DATA CONVERT WORK HARD TEMP EFFECTS RESTART DIST FLUXES	TRANSIENT	—	Thermal-mechanically coupled analysis of the compression of a block.
8.8	21	—	ORTHOTROPIC DIST LOADS	—	HOOKLW ANELAS	Bending of a thick anisotropic plate.

Table 8-1 Recent Analysis Capabilities in Marc (Continued)

Problem Number	Element Type(s)	Parameters	Model Definition	History Definition	User Subroutines	Problem Description
8.9	3	—	DEFINE ORTHOTROPIC ORIENTATION FALL DATA PRINT ELEM	—	—	Failure criteria calculation of an orthotropic plate.
8.10	52	—	HYPOELASTIC	—	UBEAM	Nonlinear beam bending.
8.11	26	—	ERROR ESTIMATE	DEACTIVATE ACTIVATE	—	Example of Activate, Deactivate and error estimates.
8.12	10	PRINT,5 FINITE LARGE DISP UPDATE REZONING	WORK HARD CONTACT	AUTO LOAD TIME STEP REZONE CONTACT CHANGE ISOTROPIC CHANGE CONNECTIVITY CHANGE COORDINATE CHANGE END REZONE AUTO TIME	—	Forging of the head of a bolt.
8.13	10 116	PRINT,5 FINITE LARGE DISP UPDATE COUPLE	POST FIXED TEMP FIXED DISP TEMP EFFECTS WORK HARD DIST FLUXES CONTACT INITIAL TEMP CONVERT	TRANSIENT DISP CHANGE AUTO TIME	—	Coupled analysis of ring compression.
8.14	7	UPDATE FINITE LARGE DISP PRINT,5	—	AUTO LOAD TIME STEP	—	3D indentation problem demonstrating how rigid surfaces are defined.
8.15	11 27	UPDATE FINITE LARGE DISP PRINT,5 REZONING ADAPTIVE	CONTACT CONTACT TABLE DEFINE RESTART LAST	AUTO LOAD TIME STEP ADAPT GLOBAL	—	Double-sided contact between deformable bodies.
8.16	11	UPDATE FINITE LARGE DISP PRINT,5	SPRINGS CONTACT WORK HARD	AUTO LOAD TIME STEP RELEASE MOTION CHANGE	MOTION	Formation of a metal part and the examination of springback.

Table 8-1 Recent Analysis Capabilities in Marc (Continued)

Problem Number	Element Type(s)	Parameters	Model Definition	History Definition	User Subroutines	Problem Description
8.17	7	REZONING UPDATE FINITE LARGE DISP PRINT,8	CONTACT RESTART LAST	AUTO LOAD TIME STEP	—	Metal extrusion analysis using the CONTACT option. Coulomb friction.
8.18	75	SHELL SET,7 LARGE DISP UPDATE FINITE PRINT,8	CONTACT	AUTO LOAD TIME STEP MOTION CHANGE	WKSLP	Stretch forming of a circular sheet. Coulomb friction between sheet and punch.
8.19	7	UPDATE FINITE LARGE DISP PRINT,8 SIZING	CONTACT UMOTION	AUTO LOAD TIME STEP	MOTION	Three dimensional indentation rolling of elastic-perfectly plastic material.
8.20	39	SIZING ELECTRO	POINT CHARGE FIXED POTENTIAL	STEADY STATE	—	Point charge in a circular region.
8.21	43	SIZING ELECTRO	FIXED POTENTIAL POINT CHARGE	STEADY STATE	—	Point charge in a circular cylinder.
8.22	39	SIZING\ MAGNET	POINT CURRENT FIXED POTENTIAL	STEADY STATE	—	Point current in a circular region.
8.23	109	MAGNET	FIXED POTENTIAL POINT CURRENT	STEADY STATE	—	3D analysis of a magnetic field in a coil.
8.24	39	MAGNET	ISOTROPIC FIXED POTENTIAL POINT CURRENT B-H RELATION	STEADY STATE	—	2D nonlinear magnetostatic analysis.
8.25	39	ACOUSTIC PRINT,3	ISOTROPIC	DYNAMIC CHANGE	—	2D acoustic problem demonstrating the eigenvalue analysis in a circular cavity with barrier.
8.26	39	ACOUSTIC	ISOTROPIC FIXED PRESSURE	DYNAMIC CHANGE	FORCDT	2D acoustic problem demonstrating the eigenvalue analysis of a rectangular cavity.
8.27	26	INPUT TAPE	FIXED DISP ORTHOTROPIC	AUTO LOAD PROPORTIONAL CONTROL	—	Progressive failure of a plate with a hole.
8.28	41 103	ELECTRO	POINT CHARGE FIXED POTENTIAL	STEADY STATE	—	Linear distribution of dipoles.
8.29	41 103	MAGNET	POINT CHARGE FIXED POTENTIAL	STEADY STATE	—	Magnetic field around two wires carrying opposite currents.

Table 8-1 Recent Analysis Capabilities in Marc (Continued)

Problem Number	Element Type(s)	Parameters	Model Definition	History Definition	User Subroutines	Problem Description
8.30	111	EL-MA HARMONIC PRINT, 3	DIST CURRENT FIXED POTENTIAL	DIST CURRENT HARMONIC POINT CURRENT	—	Harmonic electromagnetic analysis of a waveguide.
8.31	112	EL-MA PRINT, 3	FIXED POTENTIAL	DYNAMIC CHANGE POTENTIAL CHANGE	—	Transient electromagnetic analysis around a conducting sphere.
8.32	113	EL-MA HARMONIC	FIXED POTENTIAL	DIST CURRENT HARMONIC	—	Calculate the resonance in a cavity.
8.33	111	EL-MA HARMONIC	FIXED POTENTIAL	POINT CURRENT CURRENT DYNAMIC CHANGE HARMONIC	—	Steady state analysis of an infinitely long wire using both harmonic and transient analysis.
8.34	28	PORE UPDATE ISTRESS	SOIL INITIAL PC INITIAL VOID INITIAL STRESS DIST LOADS	DIST LOADS TIME STEP DISP CHANGE AUTO LOAD	—	Drained triaxial test on normally consolidated clay.
8.35	32	PORE ISTRESS	SOIL SOLVER INITIAL PC INITIAL STRESS INITIAL VOID DIST LOADS DEFINE	DIST LOADS TIME STEP AUTO TIME CONTROL	—	Coupled pore-pressure calculation of stratified soil embankment.
8.36	116	PRINT, 5	SPRINGS CONTACT DEFINE	CONTACT TABLE AUTO LOAD TIME STEP	—	Interference fit of two cylinders.
8.37	11	PRINT, 8	CONTACT SPRINGS DEFINE	CONTACT TABLE AUTO LOAD TIME STEP	—	Interference fit between sectors of two cylinders. Demonstrates symmetry surfaces.
8.38	75	LARGE DISP UPDATE FINITE	CONTACT WORK HARD CONTACT TABLE	AUTO LOAD TIME STEP	—	Deep drawing of a box using rigid punch described as NURBS.
8.39	5	LARGE DISP	CONTACT POINT LOAD	AUTO INCREMENT POINT LOAD	—	Contact of two beams by a point load.
8.40	11	ADAPT ELASTIC	ADAPTIVE ATTACH NODES SURFACE POINT LOAD	—	—	Adaptive meshing of a disk subjected to point loads.
8.41	3	ADAPT ELASTIC	ADAPTIVE ERROR ESTIMATES	—	—	Adaptive meshing of a stress concentration.

Table 8-1 Recent Analysis Capabilities in Marc (Continued)

Problem Number	Element Type(s)	Parameters	Model Definition	History Definition	User Subroutines	Problem Description
8.42	11	ADAPT LARGE DISP FOLLOW FOR	CONTACT ATTACH NODE SURFACE DIST LOADS	MOTION CHANGE AUTO LOAD TIME STEP	—	Double-sided contact analysis with adaptive meshing.
8.43	119	LARGE DISP FOLLOW FOR ADAPT	ADAPTIVE MOONEY CONTACT	AUTO LOAD TIME STEP DISP CHANGE	—	Modeling a rubber seal with adaptive meshing.
8.44	11	UPDATE LARGE DISP FINITE ADAPT	WORK HARD ADAPTIVE CONTACT CONTACT TABLE	MOTION CHANGE TIME STEP AUTO LOAD	—	Rolling example with adaptive meshing.
8.45	11	EXTENDED	CHANGE STATE INITIAL STATE SPLINE CONTACT	AUTO LOAD TIME STEP MOTION CHANGE CHANGE STATE	—	Use of the SPLINE option for deformable-deformable contact.
8.46	3	EXTENDED DIST LOADS	CONTACT EXCLUDE FIXED DISP	AUTO LOAD DISP CHANGE DIST LOADS	—	Use of EXCLUDE option for contact analysis.
8.47	3	DIST LOADS	DIST LOADS FIXED DISP SPRINGS CONTACT	AUTO LOAD DIST LOADS TIME STEP	—	Simulation of contact with stick-slip friction.
8.48	3	LARGE DISP DIST LOADS	FIXED DISP SOLVER SPRINGS GEOMETRY	AUTO LOAD DISP CHANGE DIST LOADS TIME STEP	—	Simulation of deformable-deformable contact with stick-slip friction.
8.49	80	DIST LOADS LARGE DISP	CONTACT MOONEY ODGEN GENT ARRUDBOYCE	AUTO LOAD MOTION CHANGE TIME STEP	—	Rolling of a compressed rubber bushing with stick-slip friction (use various rubber models).
8.50	10	UPDATE FINITE LARGE DISP	GEOMETRY WORK HARD	AUTO LOAD MOTION CHANGE TIME STEP	—	Compression test of cylinder with stick-slip friction.
8.51	139 75	FINITE LARGE DISP UPDATE SHELL SECT	CONTACT CONTACT TABLE FIXED DISP WORK HARD GEOMETRY	AUTO LOAD MOTION CHANGE TIME STEP	—	Modeling of a spring.
8.52	75	FINITE LARGE DISP UPDATE SHELL SECT	GEOMETRY CONTACT CONTACT TABLE WORK HARD	AUTO LOAD DISP CHANGE TIME STEP	—	Deep drawing of a sheet.

Table 8-1 Recent Analysis Capabilities in Marc (Continued)

Problem Number	Element Type(s)	Parameters	Model Definition	History Definition	User Subroutines	Problem Description
8.53	75 49	DIST LOADS LARGE DISP SHELL SECT	DIST LOADS GEOMETRY FIXED DISP	AUTO LOAD DIST LOADS DISP CHANGE TIME STEP	—	Shell-shell contact and separation.
8.54	75	LARGE DISP SHELL SECT UPDATE	CONTACT FIXED DISP	AUTO LOAD DISP CHANGE TIME STEP	—	Self contact of a shell structure.
8.55	75	FINITE LARGE DISP UPDATE SHELL SECT	CONTACT POINT LOAD WORK HARD	AUTO LEAD POINT LOAD TIME STEP	—	Deep drawing of copper sheet (velocity and load controlled dies.
8.56	10	FINITE LARGE DISP UPDATE	CONTACT WORK HARD	AUTO LOAD DISP CHANGE TIME STEP	—	2D contact problem (load and velocity controlled dies).
8.57	75 138 139 140	ADAPTIVE ELASTIC SHELL SECT	ADAPTIVE POINT LOAD	—	—	The adaptive capability with shell elements.
8.58	75	ADAPTIVE ELASTIC SHELL SECT	GEOMETRY DIST LOADS ADAPTIVE	—	—	Adaptive meshing in multiple connected shell structures.
8.59	10 40	CONSTANT D PLASTICITY	FIXED TEMPERATURES WORK HARD	TEMP CHANGE TRANSIENT NON AUTO	—	Thermal-mechanical coupling capability.
8.60	11	PLASTICITY	CONTACT WORD HARD FIXED DISP	AUTO LOAD MOTION CHANGE TIME STEP	—	Simulation of sheet bending.
8.61	10	ELASTICITY	FIXED DISP MOONEY AXITO3D	AUTO LOAD DISP CHANGE TIME STEP	—	Simulation of rubber bushing (axisymmetric to 3-D analysis).
8.62	7	PLASTICITY	ISOTROPIC WORK HARD FIXED DISP CONTACT POINT LOAD	AUTO LOAD TIME STEP POINT LOAD	—	Torsion of a bar with square cross section (load controlled dies).
8.63	40 82	HARMONIC ACOUSTIC	ACOUSTIC REGION FIXED DISP CONTACT EXCLUDE	HARMONIC PRESS CHANGE AUTO LOAD TIME STEP DISP CHANGE	—	Coupled structural-acoustic analysis.
8.64	11 80	REZONING ADAPTIVE PROCESSOR	MOONEY CONTACT TABLE ADAPT GLOBAL CONTROL	ATO LOAD MOTION CHANGE TIME STEP CONTACT TABLE	—	Simulation of rubber cushion with metal fastener.

Table 8-1 Recent Analysis Capabilities in Marc (Continued)

Problem Number	Element Type(s)	Parameters	Model Definition	History Definition	User Subroutines	Problem Description
8.65	10	ELASTICITY	CONTACT TABLE MOONEY SPLINE NO PRINT	AUTO STEP DISP CHANGE	—	Pipe moved on a nozzle with rubber seal between pipe and nozzle.
8.66	7	DYNAMIC	CONNECTIVITY CONTACT DAMPING	DYNAMIC CHANGE MOTION CHANGE PARAMETERS	—	Block sliding along a rigid, flat surface.
8.67	10 144 7 146	ELASTICITY	AXITO3D CONTACT ISOTROPIC MOONEY REBAR	AUTO LOAD AUTO STEP MOTION CHANGE	—	Axisymmetric to 3-D data transfer capability for rebar elements - analysis of automobile tire.
8.68	7	ADAPTIVE LARGE DISP	CONTACT CONTACT TABLE ISOTROPIC NO PRINT	AUTO LOAD CONTROL MOTION CHANGE TIME STEP	—	Use of the CONTACT TABLE option

8.33 Coupled Analysis of Ring Compression

This example demonstrates Marc's ability to perform a large deformation problem, incorporating thermal mechanical coupling and automated contact.

A ring of aluminum is deformed by a block of steel. Both have the capacity to deform, and possibly, slide between each other.

Coupling

There are several sources of coupling in this analysis.

1. As the temperature changes, thermal stresses are developed due to nonzero coefficient of thermal expansion.
2. As the temperature changes, the mechanical properties change. It happens in this case because of the temperature-dependent flow stress.
3. As the geometry changes, the heat transfer problem changes. This includes changes in the contacting interface.
4. As plastic work is performed, internal heat is generated.
5. As the bodies slide, friction generates heat.

Parameters

The UPDATE, FINITE, and LARGE DISP parameters indicate this is a finite deformation analysis.The COUPLE option is used to indicate that a coupled thermal-mechanical analysis is being performed. A four-node bilinear axisymmetric element is used. The PRINT,5 option requests additional information in the output regarding nodes acquiring or losing contact.

Mesh Definition

Mentat was used to create the mesh. There are separate nodes along both sides of the contact interface so that sliding is possible. Due to symmetry, only one quarter of the region is modeled. The mesh is shown (with the units in mm) in Figure 8.13-1. In a coupled analysis, a displacement element automatically produces the coupled (displacement-temperature) formulation to be used.

This analysis is performed using both element type 10 and element type 116. Both elements are four-node axisymmetric elements. Element type 116 uses a single integration point and an hourglass stiffness stabilization procedure.

The standard CONTACT option is used. Free surfaces can have convection heat transfer to the environment. As soon as contact is detected, a contact thermal barrier, defined by means of a film coefficient, starts operating.

Geometry

A ‘1’ is placed in the second field to indicate that the constant dilatation formulation is used. This is not necessary for the analysis using element type 116.

Boundary Conditions

Symmetry displacement boundary conditions are imposed on the ring meridian plane and on the block axis. The block is moved down by application of displacement boundary conditions to the face opposite to the contact face. The displacement boundary conditions are entered in the FIXED DISPLACEMENT option. On the outside surface of the block, the temperature is constrained to 20°C, to simulate a much larger size block. This is done with the FIXED TEMPERATURE option.

Control Options

A formatted post file containing stress components and effective plastic strain is written at the end of 50 increments. The NO PRINT option limits the amount of output to a minimum. Displacement control is used in the deformation part of the analysis with a relative error of 15%. As far as the heat transfer part of the analysis is concerned, a 10°C maximum error in temperature estimate is entered. Even if thermal material properties are not temperature-dependent, this provides a means of forcing recycling when heat transfer between two bodies produces large variations of temperature per increment.

Initial Conditions

The ring is given an initial temperature of 427°C, and the block is given an initial temperature of 20°C.

Material Properties

The ring is treated as an elastic-plastic material with a Young’s modulus of 10,000 MPa, a Poisson’s ratio of 0.33, a coefficient of thermal expansion of 1.3×10^{-5} mm/mm°C, and an initial yield stress of 3.4 MPa, corresponding to a reference temperature of 200°C. The material workhardens from the initial yield stress to a yield stress of 5.78 MPa for strains above 70%, according to a piecewise linear function entered via

WORK HARD DATA. The flow stress and work hardening slope decrease with temperature increases at a rate of 0.007 MPa per degree. The thermal properties are conductivity of 242.0 N/s°C and specific heat of 2.4255 Nmm/g°C.

The block is treated as an elastic material with a Young's modulus of 100,000 MPa, a Poisson's ratio of 0.33[3]. The thermal properties are conductivity of 19.0 N/s°C and specific heat of 3.77 Nmm/g°C.

Distributed Flux

This distributed flux block is used to indicate that internal heat is generated due to plastic deformation.

Convert

The option is used to give the conversion factor between the mechanical energy and the thermal energy.

The internal volumetric flux per unit volume becomes:

$$\phi = cw^p$$

where w^p is the plastic strain energy density.

Contact

The CONTACT option declares that there are two bodies with adhesive friction between them. Marc calculates the contact tolerance.

The first body is deformable and is made of the elements of the ring. There is no need to specify any motion. The ring's free surfaces have convection heat transfer defined by a film coefficient of 0.01, and a sink temperature of 20°C. The second body is also deformable and made out of the elements of the block. A reference point and an axial velocity are given, although they are not used in the calculations; this is done as a reminder of what the imposed boundary conditions are simulating. A friction coefficient of 0.5 defines the interface friction conditions, based on the cohesive model. The block's free surfaces have convection heat transfer defined by a film coefficient of 0.01 N/s-mm°C, and a sink temperature of 20°C. The contact surfaces have a thermal barrier defined by a film coefficient of 35.

This ordering of the two bodies results in imposing the constraint so that the nodes of the ring do not penetrate the surface of the block. Friction and thermal barrier at the interface use data taken from the body defining the block.

Load Control

This problem is performed with a fixed time step and fixed increment size. It is specified with a time step of 0.0003 seconds with a total of 0.03 seconds requested. Each increment imposes a displacement of 0.045mm to the nodes of the block in the plane opposed to the contact surface. This displacement increment is declared in DISP CHANGE and not in the original boundary conditions because the CONTACT option always bypasses increment zero.

In e8x13b, an adaptive time stepping is used with the AUTO STEP option. The time step here is limited such that the increase on plastic strain in each step cannot exceed 0.002 up to 10% in total plastic strain and 0.005 for total plastic strains above 10%. The total time period and the initial time step is the same as in e8x13.

The third variant, e3x13c, is identical to e8x13 except that the reduced integration element 116 is used.

Results

Figure 8.13-2 shows the deformed body at the end of 100 increments compounding to 50% reduction in height of the ring for e8x13. Due to the high friction, the ring folds several times into the block on both sides, and there is an increase of the outer diameter as well as a decrease of the inner diameter. It can be seen that the amount of interface sliding is very small, also due to the high friction. Elastic deformations on the block are not visible, therefore it looks like the block had a rigid body translation.

Figure 8.13-3 shows equivalent plastic strain contours produced on the ring. They range from small amounts in the middle of the contact area (neutral zone) and in the free surface, to very large amounts at the corners where folding took place, and in the center of the middle plane.

In Figure 8.13-4, the equivalent von Mises stresses give an idea of the stresses produced in the block, which are higher than in the ring. They increase from low values in the free standing areas towards the center. Local peaks in the friction shearing zones also appear.

The thermal part of the analysis produces the temperatures of Figure 8.13-5. The total time for the deformation is only 0.03 seconds. Therefore, all the effects are confined to the contact region. Aluminum’s high temperature, low flow stress produces no noticeable heating due to plastic deformation. On the ring side, the temperature decreases about 75°C at the interface, while the block heats around 50°C. Steel’s lower conductivity produces steeper temperature gradients.

Figure 8.13-6 shows the balance between total strain energy of the deformed body and the total work done by external forces.

Parameters, Options, and Subroutines Summary

Example e8x13.dat:

Parameters	Model Definition Options	History Definition Options
COUPLE	CONNECTIVITY	CONTINUE
ELEMENT	CONTACT	DISP CHANGE
END	CONVERT	TRANSIENT NON AUTO
FINITE	COORDINATE	TEMP CHANGE
LARGE DISP	DIST FLUXES	CONTROL
PRINT	END OPTION	PARAMETERS
SIZING	FIXED DISP	DIST FLUXES
TITLE	FIXED TEMPERATURE	TITLE
UPDATE	GEOMETRY	
	INITIAL TEMPERATURE	
	ISOTROPIC	
	NO PRINT	
	POST	
	TEMPERATURE EFFECTS	
	WORK HARD	

Example e8x13b.dat:

Parameters	Model Definition Options
COUPLE	CONNECTIVITY
ELEMENT	CONTACT
END	CONTROL
FINITE	CONVERT
LARGE DISP	COORDINATE
PRINT	DIST FLUXES
SIZING	END OPTION
TITLE	FIXED DISP
UPDATE	FIXED TEMPERATURE
	GEOMETRY
	INITIAL TEMPERATURE
	ISOTROPIC
	NO PRINT
	POST
	TEMPERATURE EFFECTS
	WORK HARD

Example e8x13c.dat:

Parameters	Model Definition Options	History Definition Options
ALIAS	CONNECTIVITY	CONTINUE
COUPLE	CONTACT	DISP CHANGE
ELEMENT	CONTROL	TRANSIENT
END	CONVERT	
FINITE	COORDINATE	
LARGE DISP	DIST FLUXES	
PRINT	END OPTION	
SIZING	FIXED DISP	
TITLE	FIXED TEMPERATURE	
UPDATE	GEOMETRY	
	INITIAL TEMPERATURE	
	ISOTROPIC	
	NO PRINT	
	POST	
	TEMPERATURE EFFECTS	
	WORK HARD	

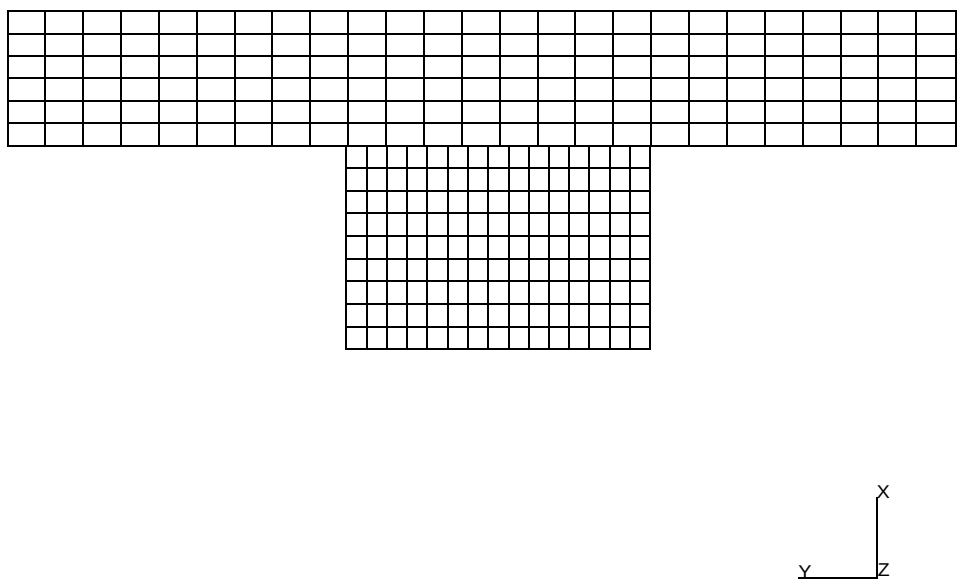


Figure 8.13-1 Original Mesh

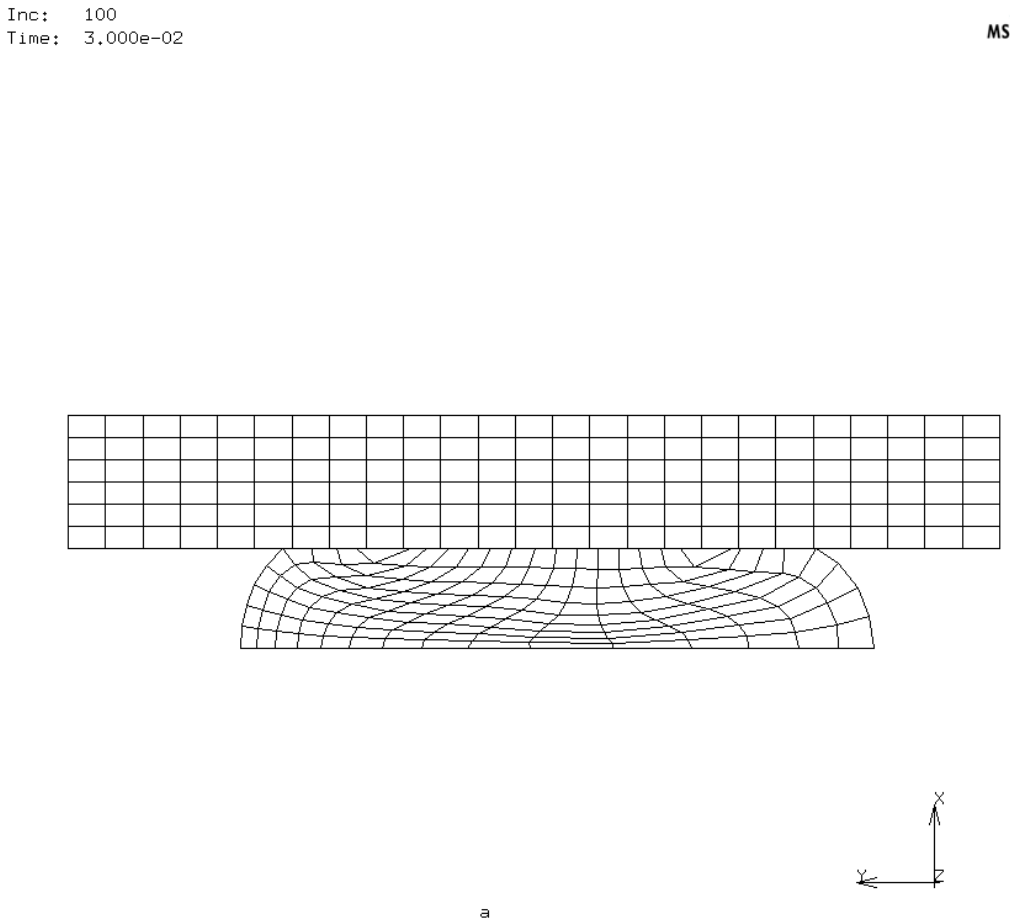


Figure 8.13-2 Deformed Mesh (50% Height Reduction)

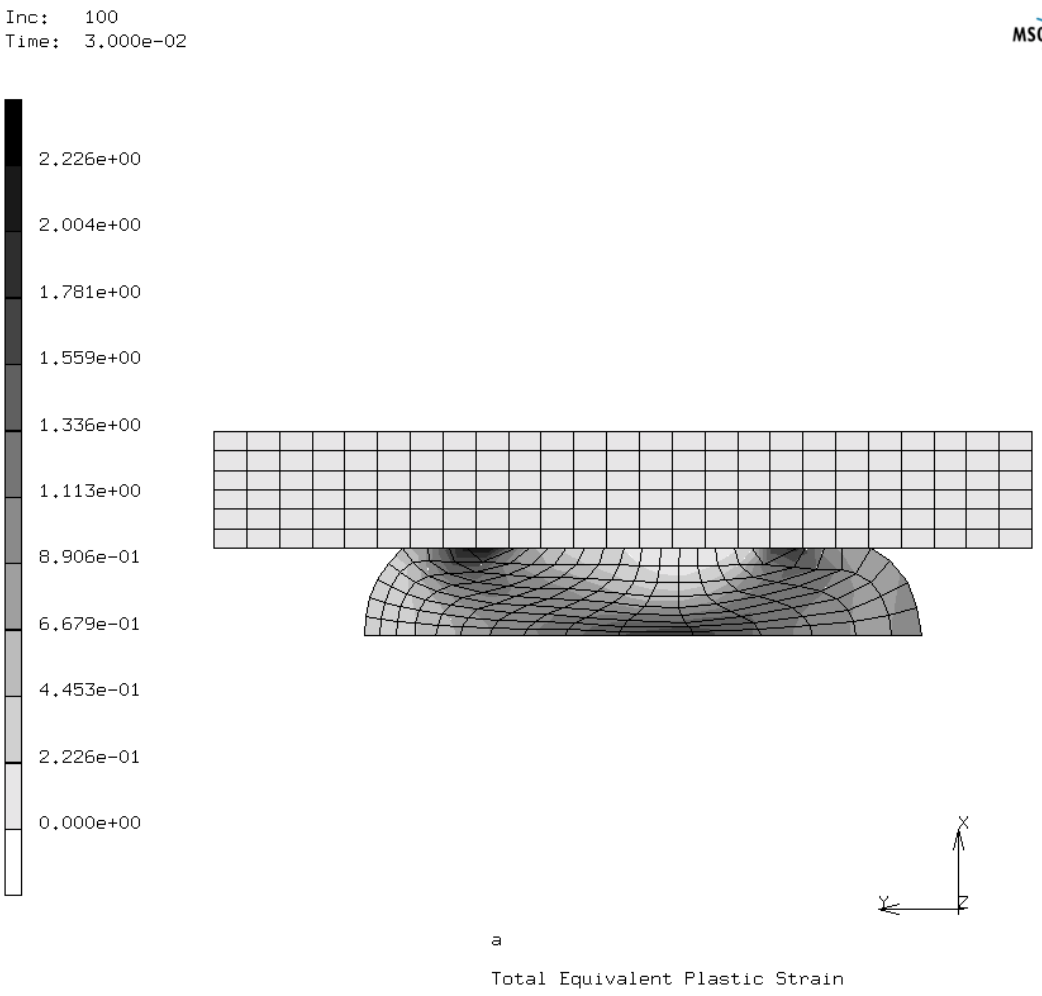


Figure 8.13-3 Equivalent Plastic Strain

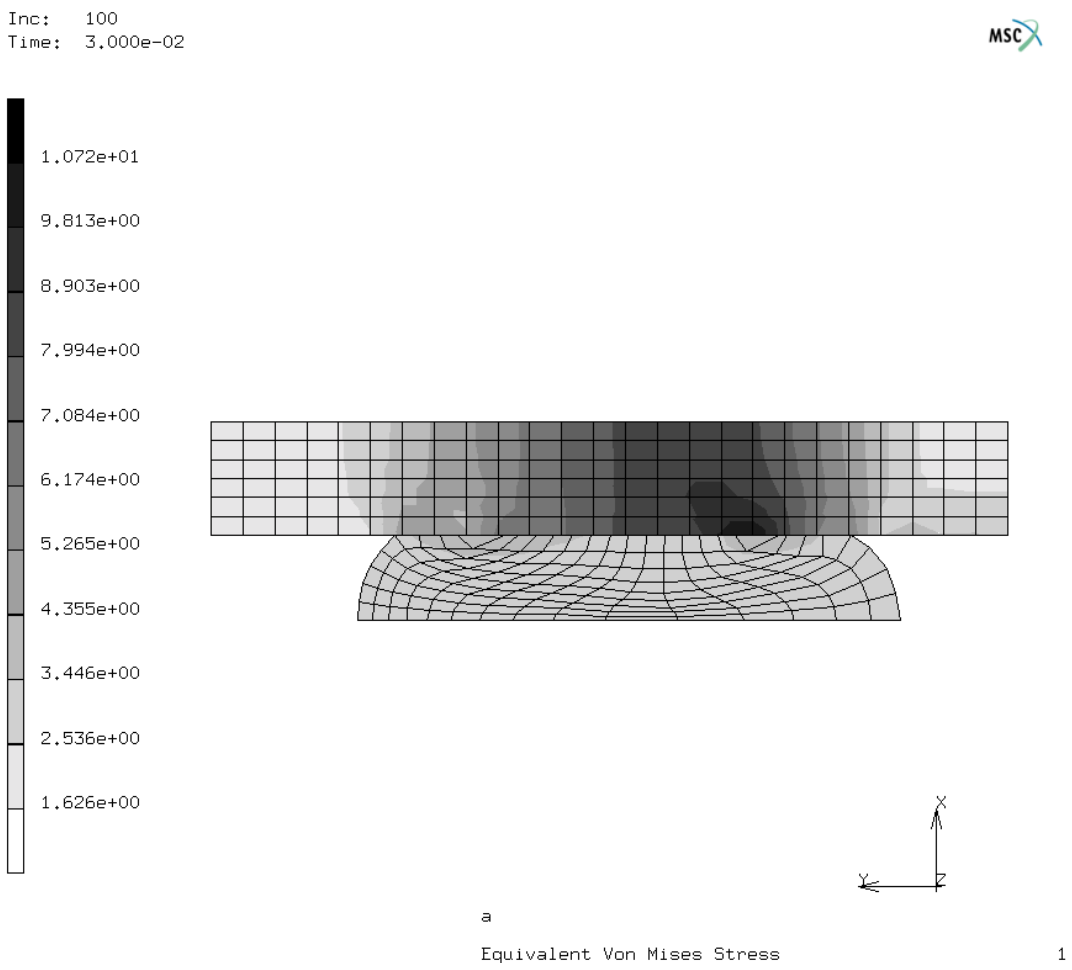


Figure 8.13-4 Equivalent von Mises Tensile Stress

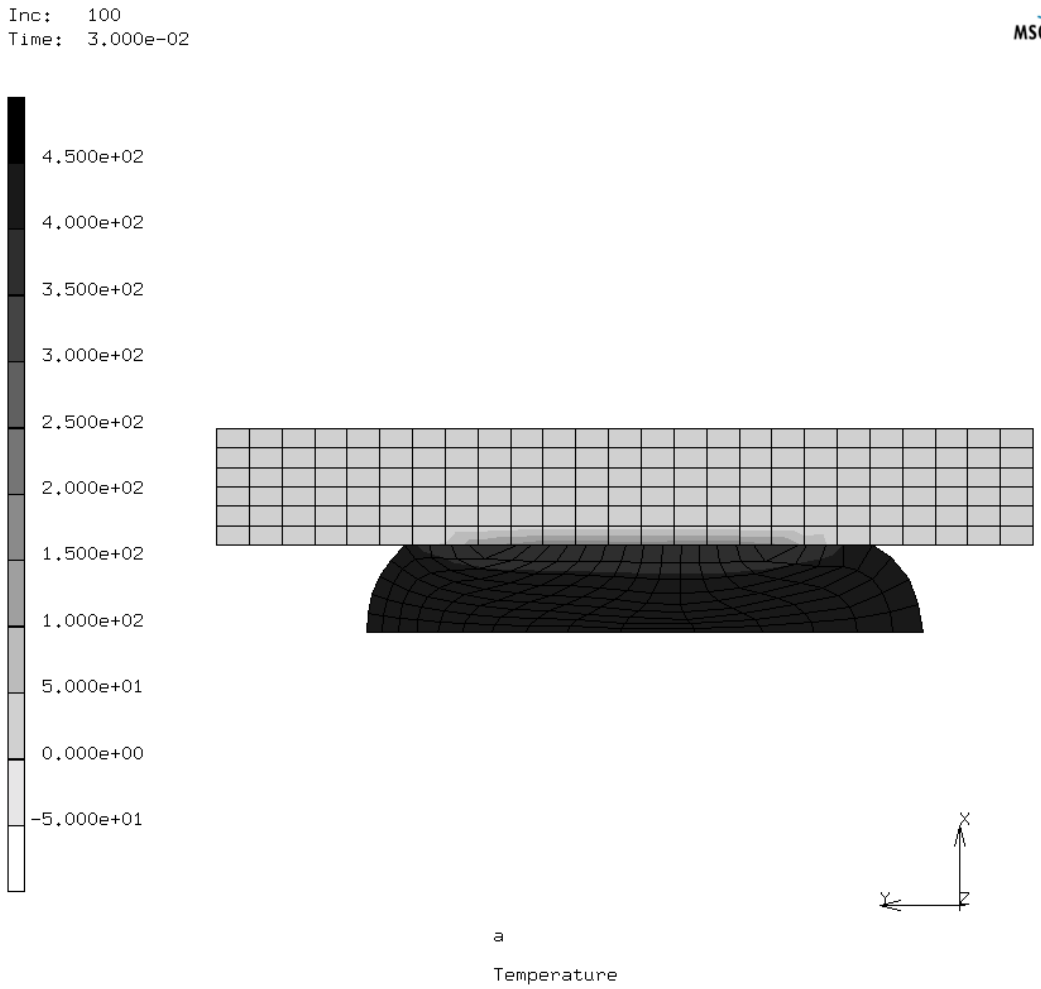


Figure 8.13-5 Total Nodal Temperature

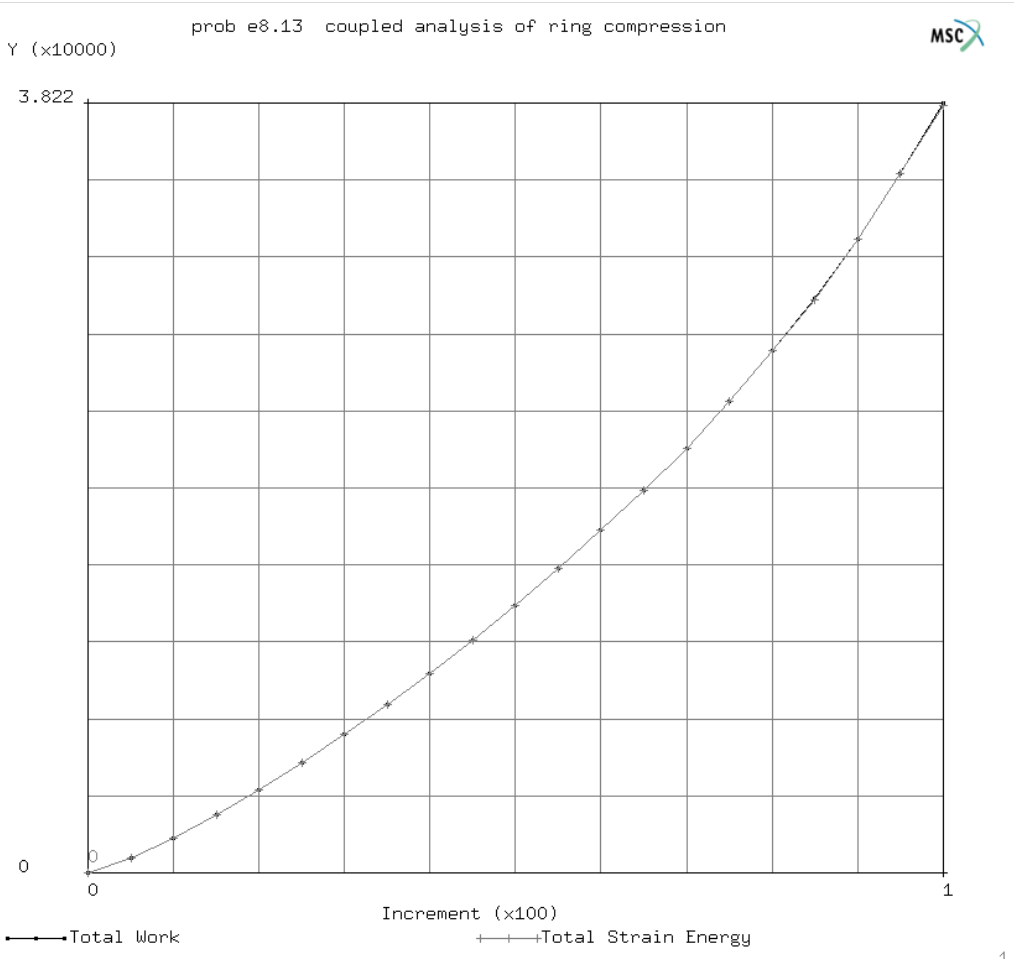


Figure 8.13-6 Energy Balance Between Total Strain Energy and Total Work by External Forces

8.23 3-D Magnetostatic Analysis of a Coil

This problem shows magnetostatic analysis capability using a 3-D element formulation in Marc. Since the potential to be solved is a new vector potential, the normal “heat transfer” approach cannot be used. Instead, an eight-noded magnetostatic element (type 109) is used for this analysis.

Parameters

The MAGNETO parameter is included to indicate a magnetostatic analysis.

Mesh Definition

One quarter of the coil is modeled using element type 109. The outside radius is 3.0 cm. Figure 8.23-1 shows the mesh and applied current.

Boundary Conditions

Along the y = 0 edge $A_1 = 0$. Along the x = 0 edge $A_2 = 0$. Along the outside radius $A_1 = A_2 = 0$. $A_3 = 0$ everywhere to simulate a two-dimensional problem.

Material Properties

The magnetic permeability of all elements is set to 1000.0 Henry/cm.

Currents

A current running in the circumferential direction at a radius of 1 cm is applied. The point currents ranging in value from -0.951 amps to +0.951 amps are applied at nodes 111 to 120.

POST

The following variables are written to both a binary and a formatted post file:

141-143 } Components of magnetic flux

144-146 } Components of magnetic intensity

Results

The third component of the magnetic induction is shown as a function of the radius in Figure 8.23-2. You can observe that a steep gradient occurs about the ring of nodes to which the current is applied. In addition, the magnetization inside the coil cancels out with the magnetization outside the coil, as:

$$\pi r_c^2 \, (28650) \, / \, \pi(r_o^2 - r_c^2) \, (-3560) = -1.006$$

where $r_c = 1$, and $r_o = 3$. Using the so called Biot-Savart equation we find for the magnetic induction inside the coil B = 31830, while from Figure 8.23-2 B = 28650. The difference is due to forcing $A_1 = A_2 = 0$ at the outside radius, while with the Biot-Savart equation the coil is in infinite space.

The vector potential A is shown in Figure 8.23-3.

Parameters, Options, and Subroutines Summary

Example e8x23.dat:

Parameters	Model Definition Options	History Definition Options
ELEMENT	CONNECTIVITY	CONTINUE
END	COORDINATE	STEADY STATE
MAGNETOSTATIC	END OPTION	
SIZING	FIXED POTENTIAL	
TITLE	ISOTROPIC	
	POINT CURRENT	
	POST	

Example e8x23b.dat:

Parameters	Model Definition Options	History Definition Options
ELEMENT	CONNECTIVITY	CONTINUE
END	COORDINATE	STEADY STATE
MAGNETOSTATIC	DEFINE	
SIZING	END OPTION	
TITLE	FIXED POTENTIAL	
	ISOTROPIC	
	POINT CURRENT	
	POST	

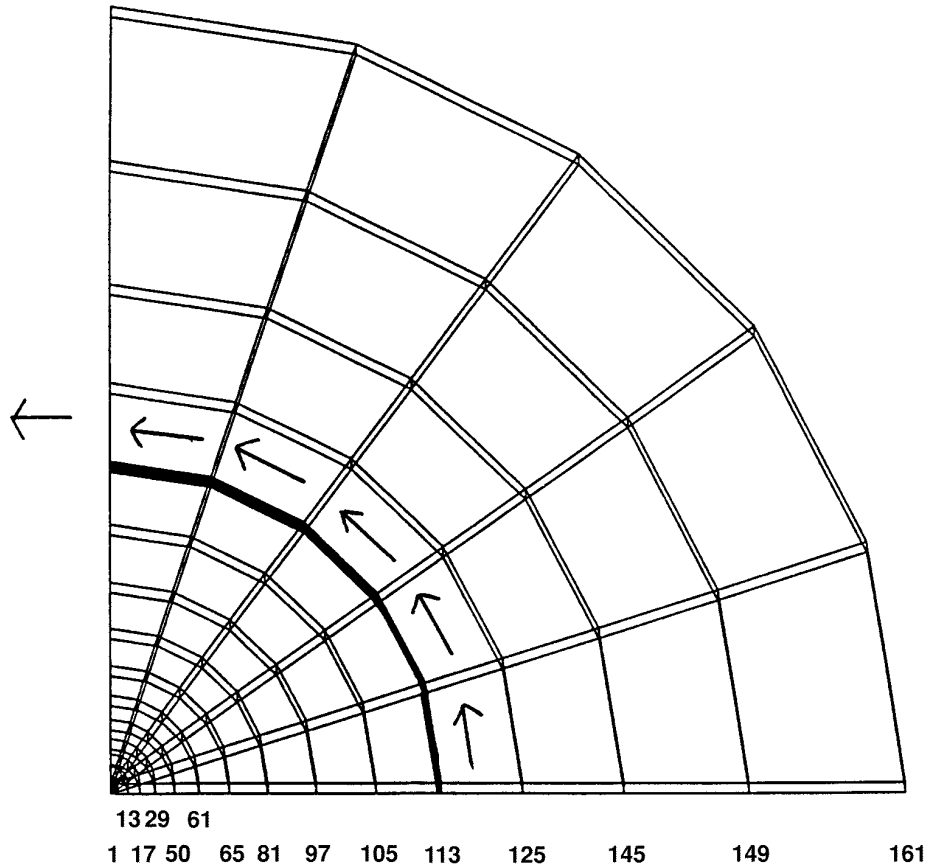


Figure 8.23-1 Mesh and Applied Current

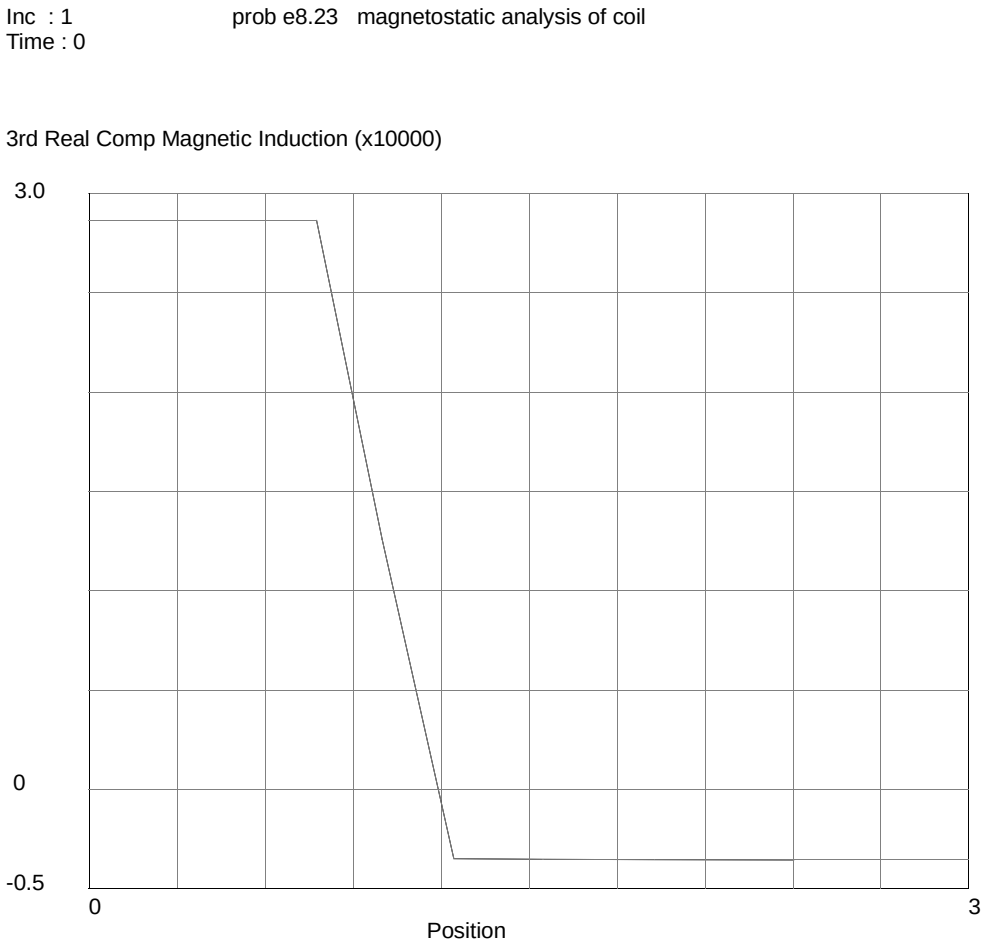


Figure 8.23-2 Third Component of Magnetic Induction along Radial Line

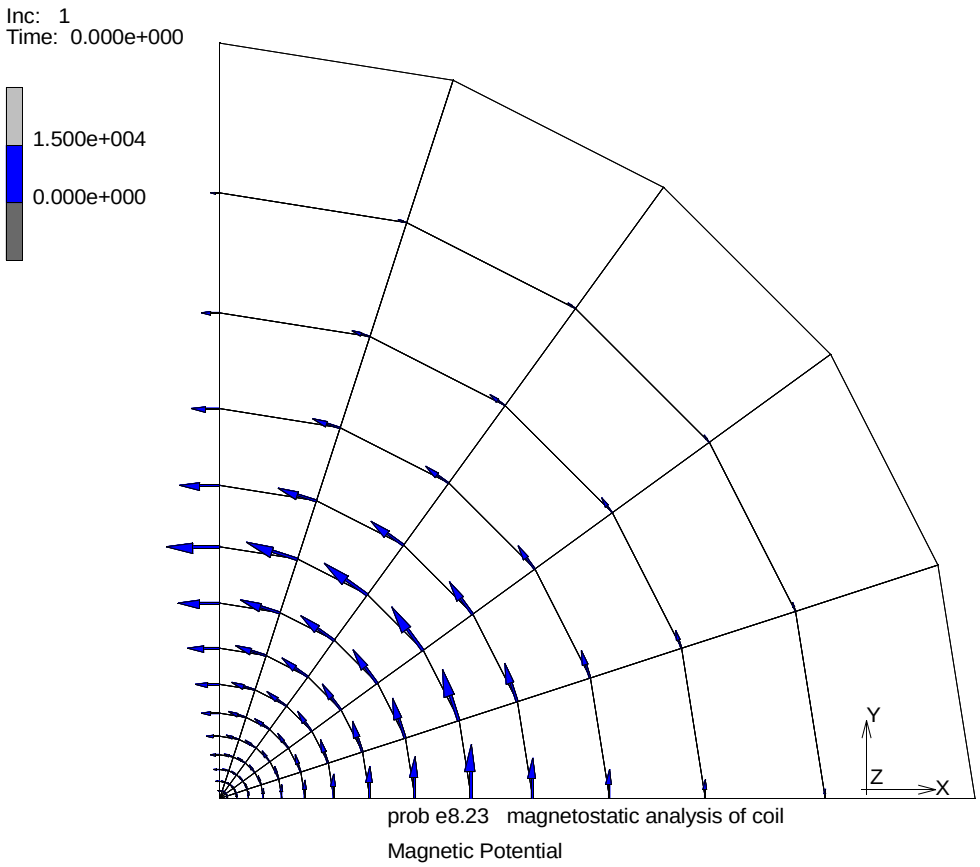


Figure 8.23-3 Magnetic Potential Vector

8.37 Interference Fit Analysis

This example demonstrates the interference fit capabilities in Marc and the use of symmetry planes.

Element

Element type 11, a four node plane strain element, is used in this analysis. The model, as shown in Figure 8.37-1, consists of a 90° segment of two rings rotated by 45°. Ten elements (9° each) are used in the circumferential direction. The inner cylinder, with r_i = 1 inch and r_0 = 2 inches, has five elements through the radius. The outer cylinder, with r_i = 2 inches and r_o = 3 inches, has six elements. Two symmetry surfaces at 45° and 135° are used. To prevent any rigid body motion, a spring was placed between the two bodies. While this was not necessary in this problem, it is often a good idea.

Loading

The kinematic boundaries are specified using the symmetry surfaces. This problem is driven by the overclosure fit of 0.01 inch specified through the CONTACT TABLE option.

Material Properties

The material is a high strength steel with a Young's modulus of 30 x 10^6 psi, a Poisson's ratio of 0.3 and a yield stress of 50,000 psi. The material remains elastic in this analysis.

Contact

There are four bodies defined: the inner cylinder, outer cylinder, symmetry surface at θ = 135°, and symmetry surface at θ = 45°.

No friction exists on any surface. Note the flag set on the fourth data block to indicate that surface 3 and 4 are symmetry surfaces. The CONTACT TABLE option is used to indicate which bodies can potentially contact others and to specify the closure distance and the overclosure amount. The overclosure was set to 0.01 inch.

The SPLINE option is used to obtain a more accurate calculation of the surface normals than would have been otherwise.

Control

Displacement control was used with a convergence tolerance of 1%. A post file was created using POST and the output was suppressed using NO PRINT. A single increment with a time step of 0.03 second was performed. In this rate independent problem, the time step is arbitrary. The OPTIMIZE option is used to reduce the bandwidth. This is very important in deformable-deformable contact problems.

The PRINT,8 option was used to obtain additional information regarding the contact conditions, such as when a node comes into contact and the displacements relative to rigid surfaces.

Results

The reaction and contact normal forces are shown in Figure 8.37-2. You can observe a nice uniform pattern along the contact surfaces.

Parameters, Options, and Subroutines Summary

Example e8x37.dat:

Parameters	Model Definition Options	History Definition Options
ELEMENT	CONNECTIVITY	AUTO LOAD
END	CONTACT	CONTACT TABLE
PRINT	CONTROL	CONTINUE
SIZING	COORDINATES	TIME STEP
TITLE	DEFINE	
	END OPTION	
	ISOTROPIC	
	NO PRINT	
	OPTIMIZE	
	POST	
	SPLINE	

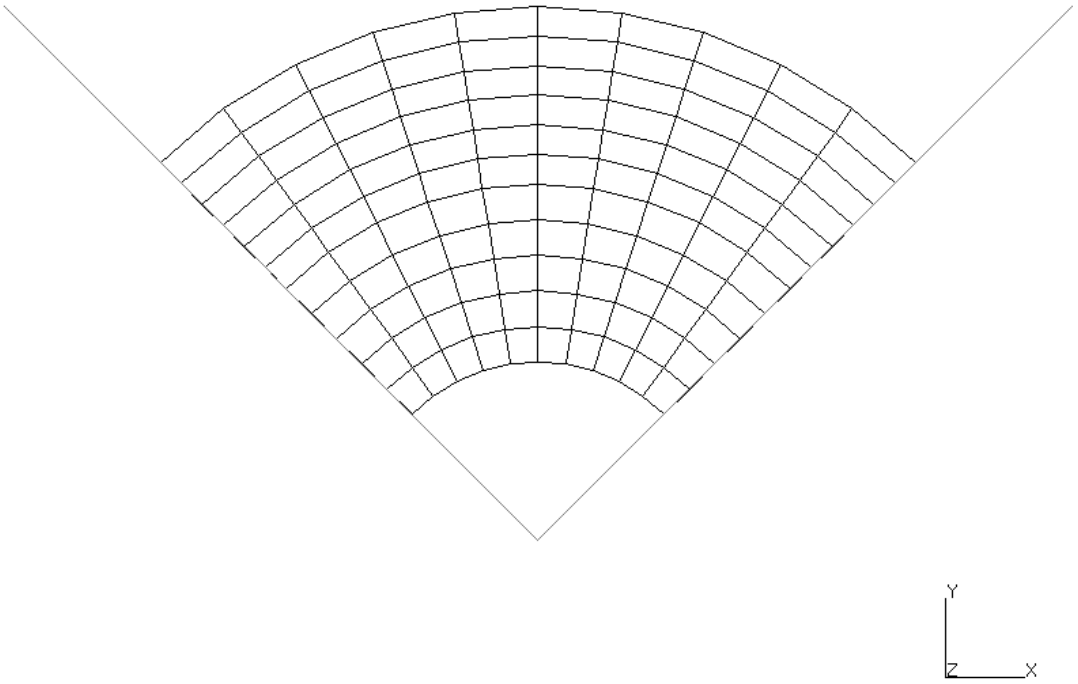


Figure 8.37-1 Finite Element Mesh with Symmetry Surfaces

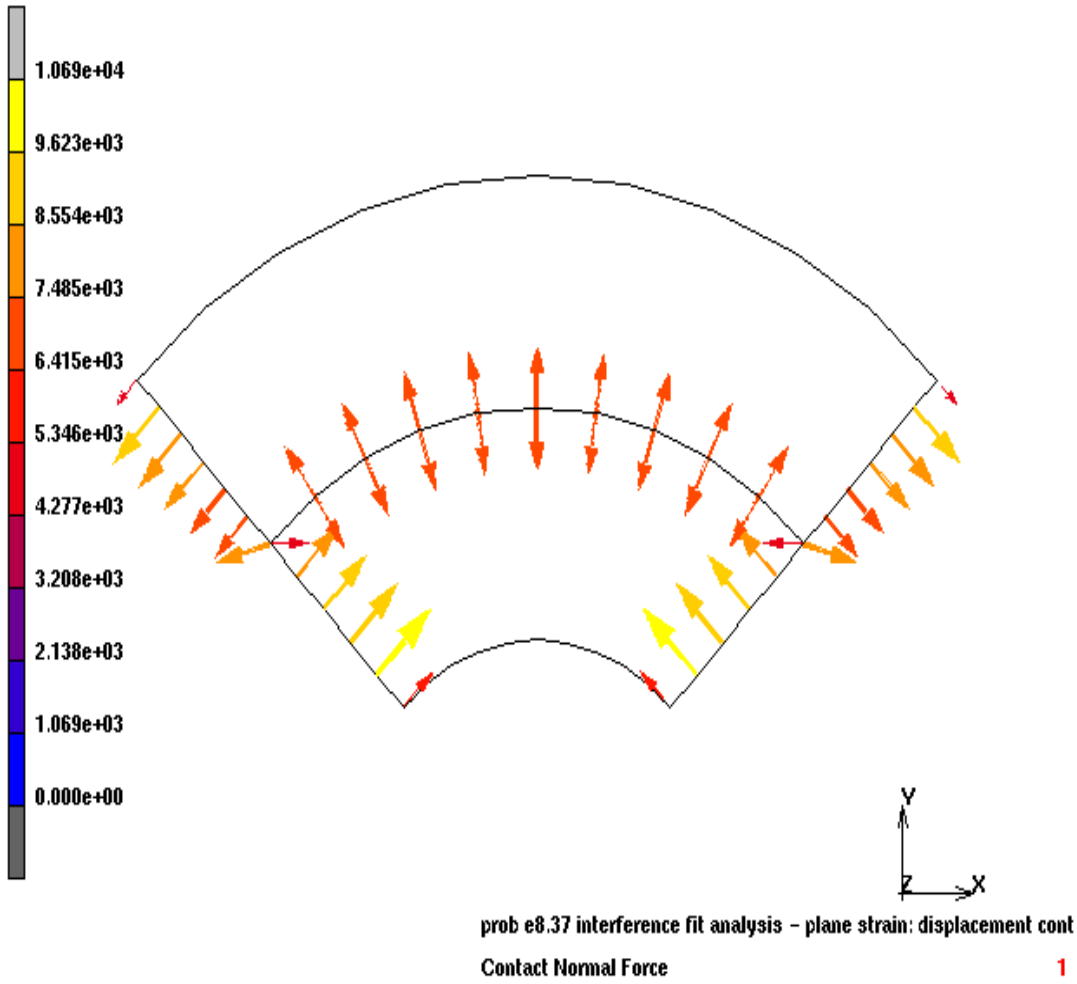


Figure 8.37-2 Reaction and Contact (Normal) Forces

8.55 Deep Drawing of Copper Sheet

This example shows the deep drawing of a copper sheet. Shell elements are used to model the sheet. There are two data sets in this example. Data set e8x55a uses a velocity controlled punch. Data set e8x55b uses a load controlled punch. The effect of a blankholder is modeled using a fixed rigid surface at a small distance above the sheet.

Model

The initial model is shown in Figure 8.55-1. A 15° sector of the punch is used. The model consists of 79 elements and 121 nodes to model the sheet. There are five rigid bodies in the model, a punch, a lower die, two symmetry bodies, and a blankholder. In the case of data set e8x55b, an extra detached node (number 122) is associated with the rigid punch and a point load is applied to this node.

Element

The 4 noded thick shell element number 75 is used in this example to model the copper sheet.

Material Properties

The sheet is assumed to be isotropic with an elastic modulus of 17×10^6 psi and a Poisson's ratio of 0.33. The initial yield stress is assumed to be 1.08×10^4 psi. The workhardening characteristics of the sheet are given by the WORK HARD model definition option.

Boundary Conditions

The boundary conditions for this problem are imposed using the CONTACT option.

Contact

There are 6 contact bodies in this problem. Contact body 1 is the deformable body representing the sheet. Body 2 is a rigid body representing the punch. In the case of data set e8x55a, the punch is a velocity controlled body moving at a speed of 3.2 in/s along the +x direction. For data set e8x55b, the punch is load controlled. Contact body 3 represents the lower die. Contact bodies 4 and 5 represent the symmetry surfaces in the problem and contact body 6 is the blankholder.

The stick-slip Coulomb friction model is used with a friction coefficient of 0.04. Default contact settings are used except that the iterative increment splitting and stress-based separation options are used.

Control

The convergence control is displacement based. The maximum allowed relative change in displacement increment is 0.05.

History Definition

For data set e8x55a, the punch is moved along the +x direction at a speed of 3.2 in/s for a further 99 increments. For data set e8x55b, a load of 3.08 lbs per increment is applied along the x direction for 79 increments.

Results

The final deformed geometry is shown in Figure 8.55-2 for data set e8x55a. The contours of total effective plastic strain are superimposed. The final deformed geometry is shown in Figure 8.55-3 for data set e8x55b. The contours of total effective plastic strain are superimposed. Figure 8.55-4 shows the total strain energy, the total work by external forces, and the contribution from friction.

Parameters, Options, and Subroutines Summary

Example e8x55a.dat and e8x55b.dat:

Parameters	Model Definition Options	History Definition Options
ALL POINTS	CONNECTIVITY	AUTO LOAD
ELEMENTS	CONTACT	CONTINUE
END	CONTROL	POINT LOAD
FINITE	COORDINATES	TIME STEP
LARGE DISP	ISOTROPIC	
SHELL SECT	OPTIMIZE	
SIZING	POINT LOAD	
TITLE	WORK HARD	
UPDATE		

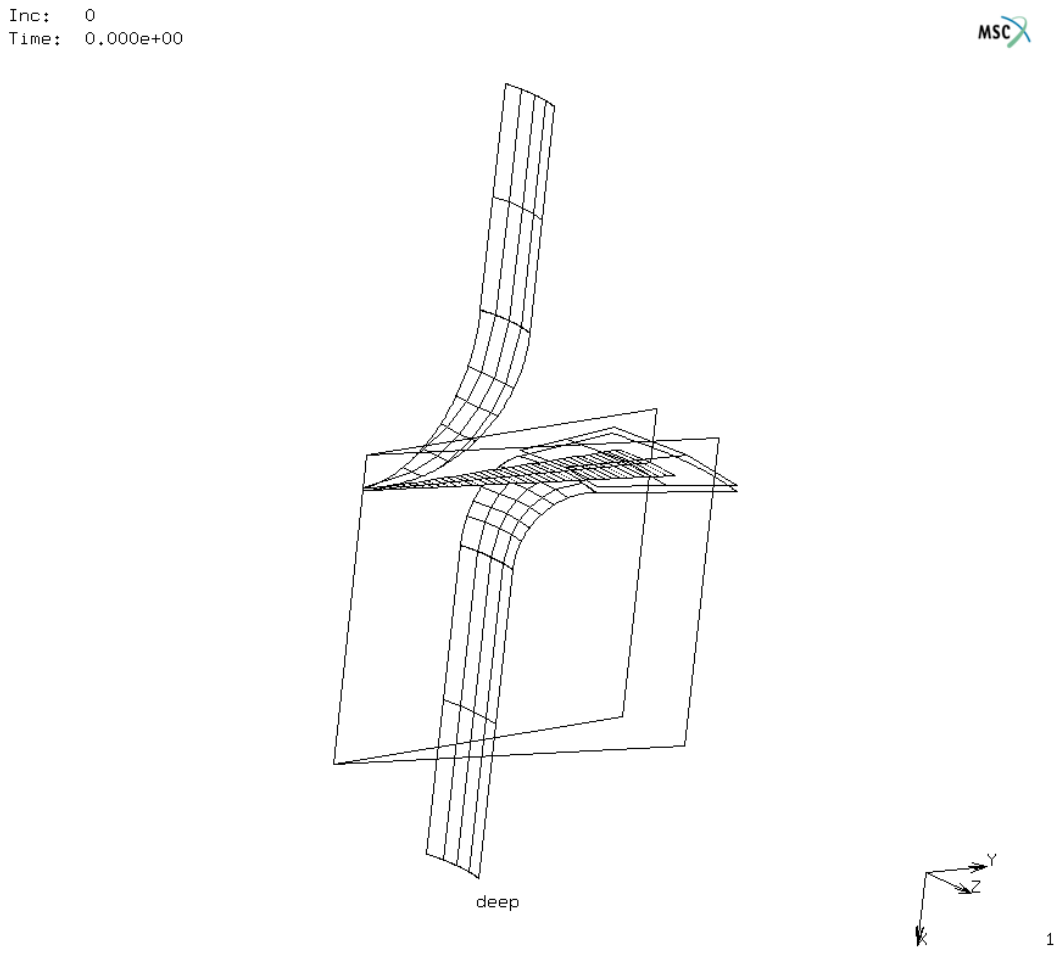


Figure 8.55-1 Initial Model

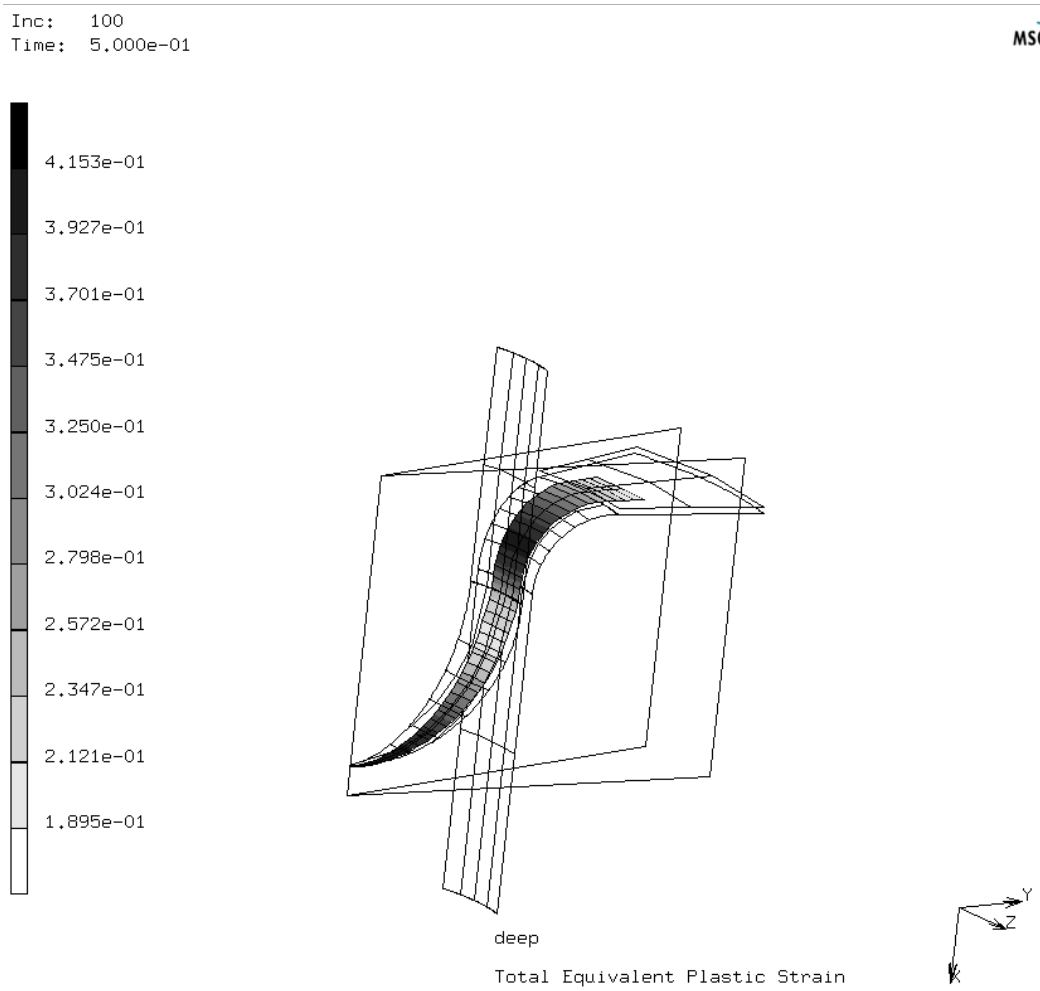


Figure 8.55-2 The Final Deformed Geometry for Data Set e8x55a

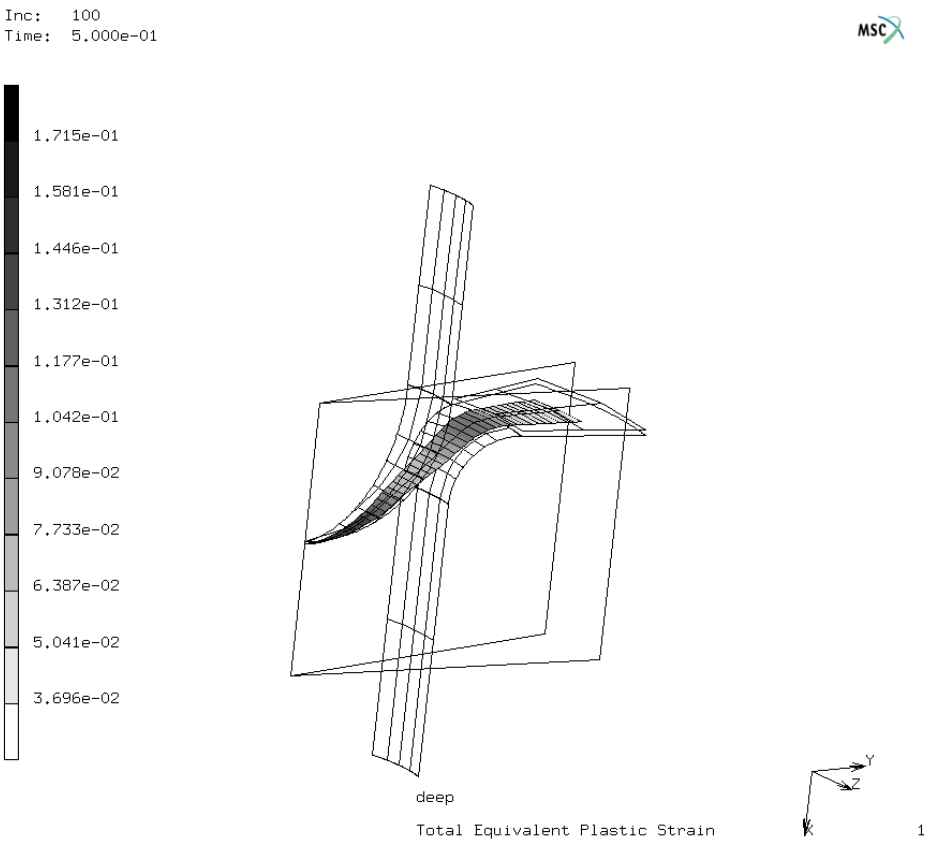


Figure 8.55-3 The Final Deformed Geometry for Data Set e8x55b

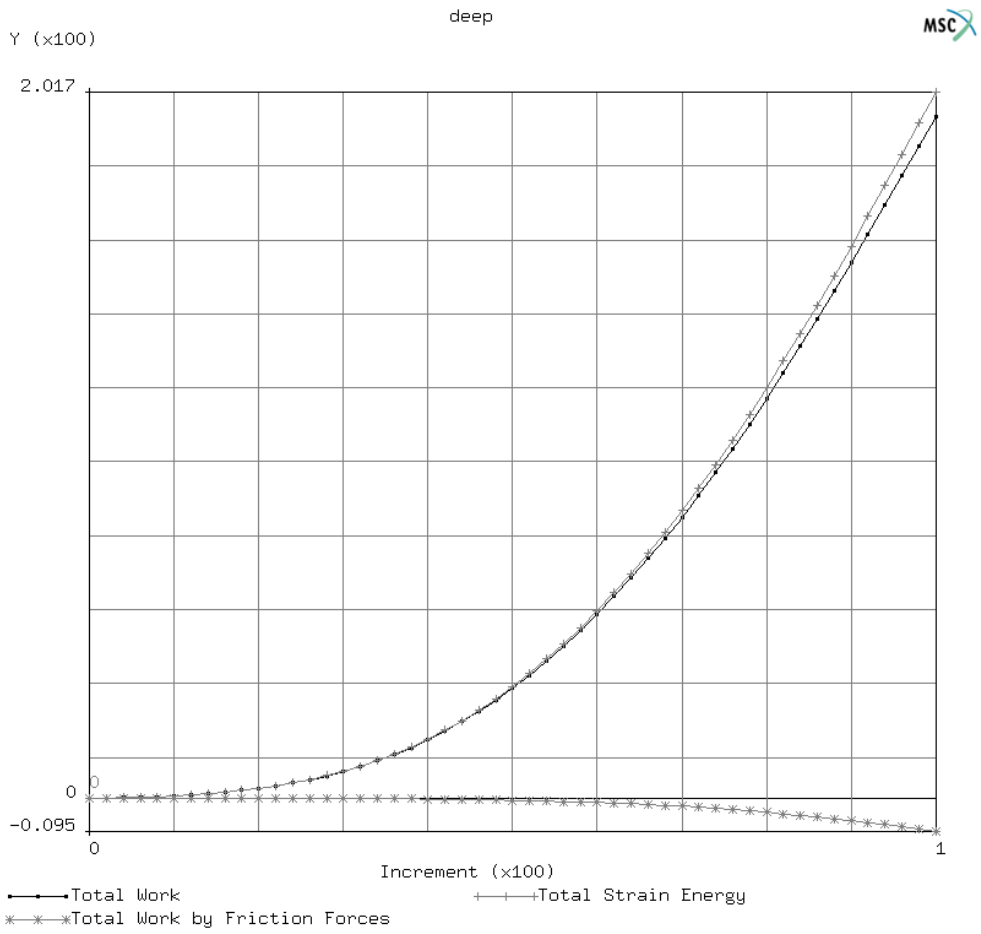


Figure 8.55-4 Total Strain Energy and Total Working External Forces

8.59 Thermal-Mechanical Coupled Simulation of Cylinder Upsetting with Plastic and Friction Heat Generation

This example demonstrates the thermal-mechanical coupling capability in Marc. It simulates a cylinder upsetting process under a non-isothermal condition. The mechanical and heat transfer analysis are handled in a stagger manner. While the mechanical analysis works on the deformation, the heat generation due to plastic deformation and friction on the contact surfaces is analyzed in the heat transfer analysis. The model is created based on the literature [Ref. 1] and the results are compared with the experiments.

Model

The model is set up as an axi-symmetric, thermal-mechanical coupled problem. One quarter of the cylinder is used with the symmetric plane and axis, and with a punch (shown in Figure 8.59-1). The punch is assumed to be a rigid heat transferring body first and is later defined as a purely rigid body for comparison. The model will be analyzed without friction first to show heat generation due to plastic deformation and later with friction to show the combined effect. The conversion factor from plastic work and friction work to the heat source and flux is 0.9. Some heat loss is due to the release of dislocations or to the lubricant.

Element

In the model, the 4-node iso-parametric quadrilateral axi-symmetric element type 10 is used for the cylinder. Thermal type element 40 is used for the punch.

Material Properties

The material property for the cylinder is assumed to be isotropic and elastoplastic. The Young's modulus is 200000 N/mm^2 and the Poisson's ratio is 0.30. According to the literature [Ref. 1], the flow stress is assumed to be plastic strain dependent only. This is correct as the upsetting will be simulated at the room temperature. The flow stress function takes the following form:

$$\sigma = 275\ N/mm^2,\ \varepsilon = 0$$

$$\sigma = 722\varepsilon^{0.262}\ N/mm^2,\ \varepsilon > 0$$

The heat transfer properties are the thermal conductivity and the heat capacity:

For the cylinder:

$$k = 36\ N/sK$$

$$\rho c = 3.77\ N/mm^2K$$

For the punch:

$$k = 19\ N/sK$$

$$\rho c = 3.77\ N/mm^2K$$

Initial Conditions

Initial temperature is set at the room temperature 293K for both the cylinder and the punch.

Boundary Conditions

A fixed temperature at 293K is applied to the top surface of the punch (see Figure 8.59-2).

Contact

Contact bodies are:

Cylinder as the deformable body
Punch as the rigid thermal body

The contact boundary conditions are:

1. Friction coefficient between the cylinder and the punch with shear friction law: 0.65
2. Heat transfer coefficient between cylinder and punch: 4 N/s/mm/K
3. Film coefficient to environment: 0.00295 N/s/mm/K

The motion of the punch represents a type of mechanical press and is defined as

$$v = 12*\sqrt{(H-20)}mm/s$$

where H is the current height of the cylinder. This motion is simulated through the use of the user subroutine u8x59.f.

Control

The convergence is checked with the relative residual criterion with 0.1 as tolerance. Maximum 20 iterations are allowed.

History Definition

Constant time increment of 0.01 is used with maximum 50 increments. This reaches 1/3 of total reduction in height, comparable to the literature [Ref. 1].

Results

The results are presented through following three examples:

1. Example 1: e8x59a.dat

   Upsetting without friction. All the heat generation is due to the plastic deformation. See Figure 8.59-3. The temperature is changed from 293K to maximum 337.2K in the cylinder and 302.8K in the punch (see Figure 8.59-4).

2. Example 2: e8x59b.dat

   Upsetting with friction. See the temperature distribution in Figure 8.59-5. And see Figure 8.59-6 for the temperature distribution in the punch. The results are closed to the results presented in the literature (Ref.1). In Figure 8.59-7, we show temperature history of some selected nodes (see Figure 8.59-1) and comparisons with the experiment data [Ref. 1].

3. Example 3: e8x59c.dat

   If we replace the punch with a rigid body at fixed room temperature, the temperature distribution is shown in Figure 8.59-8.

Parameters, Options, and Subroutines Summary

Example e8x59a.dat, e8x59b.dat and e8x59c.dat:

Parameters	Model Definition Options	History Definition Options
ADAPTIVE	CONNECTIVITY	CONTINUE
ALL POINTS	CONTACT	DIST FLUXES
CONSTANT D	CONTROL	MOTION CHANGE
COUPLE	CONVERT	TEMP CHANGE
ELASTITICY	COORDINATES	TRANSIENT NON AUTO
ELEMENTS	DIST FLUXES	
END	END OPTION	
FLUXES	FIXED TEMPERATURE	
LARGE DISP	GEOMETRY	
LUMP	ISOTROPIC	
PLASTICITY	OPTIMIZE	
PROCESSOR	PARAMETERS	
REZONING	POST	
SETNAME	SOLVER	
SIZING	UMOTION	
UPDATE	WORK HARD	

References

1. N.Rebelo and S.Kobayashi: “A Coupled Analysis of Viscoplastic Deformation and Heat Transfer – II”, *Int.J.Mech.,Sci*. Vol.22, pp.707-718

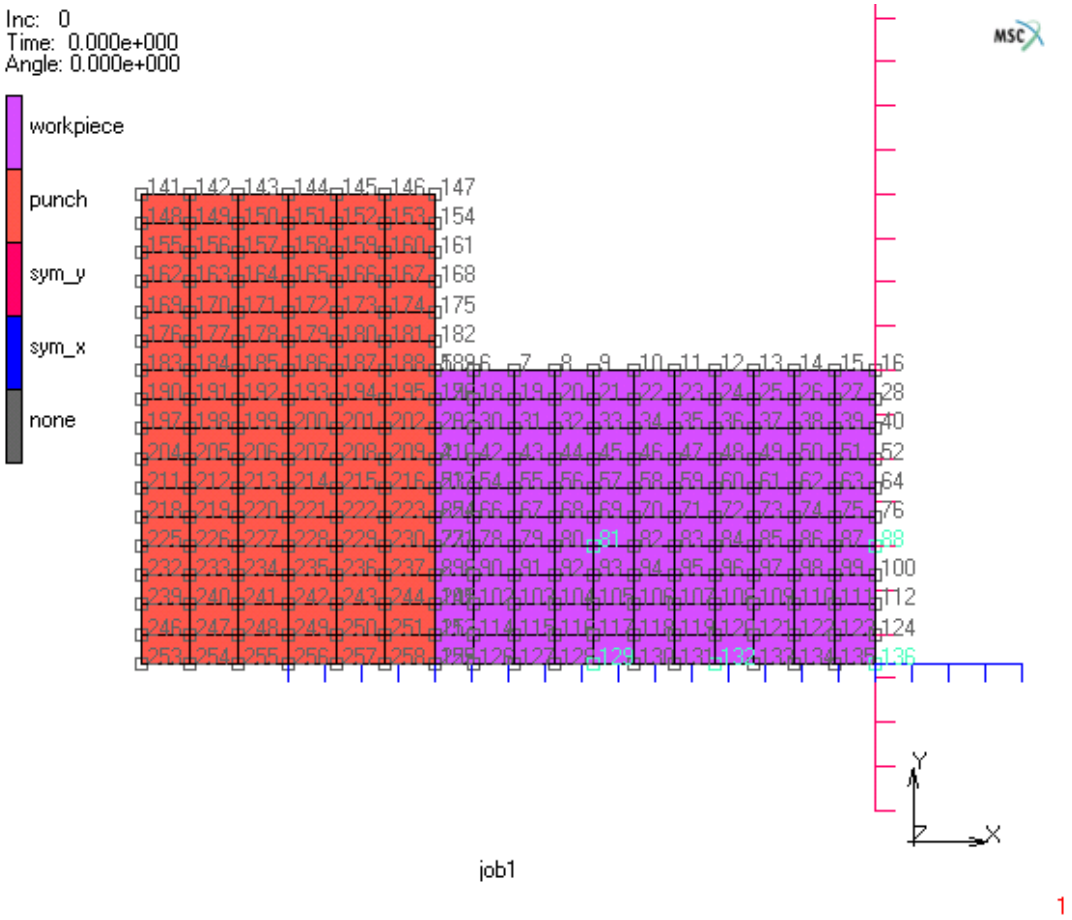


Figure 8.59-1 Cylinder Upsetting Simulation

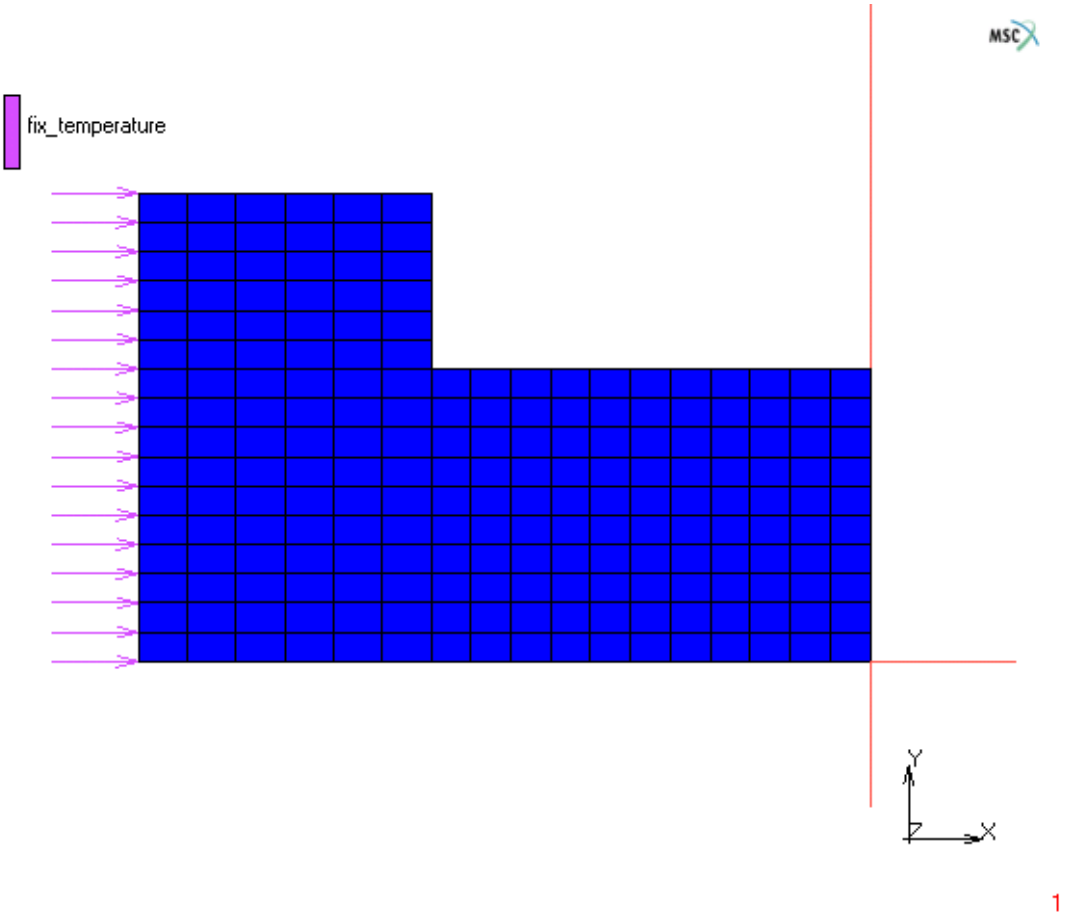


Figure 8.59-2 Prescribed Temperature on the Punch

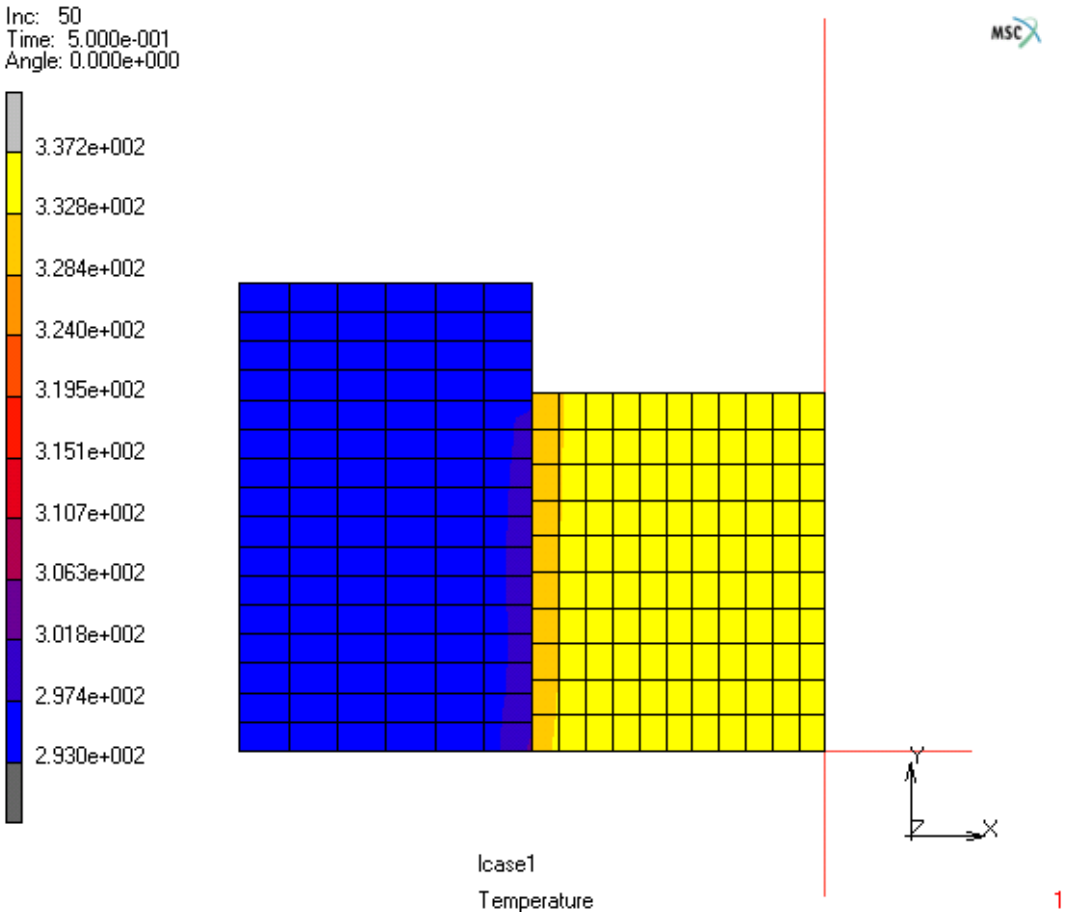


Figure 8.59-3 Temperature Distributuion of the Frictionless Model

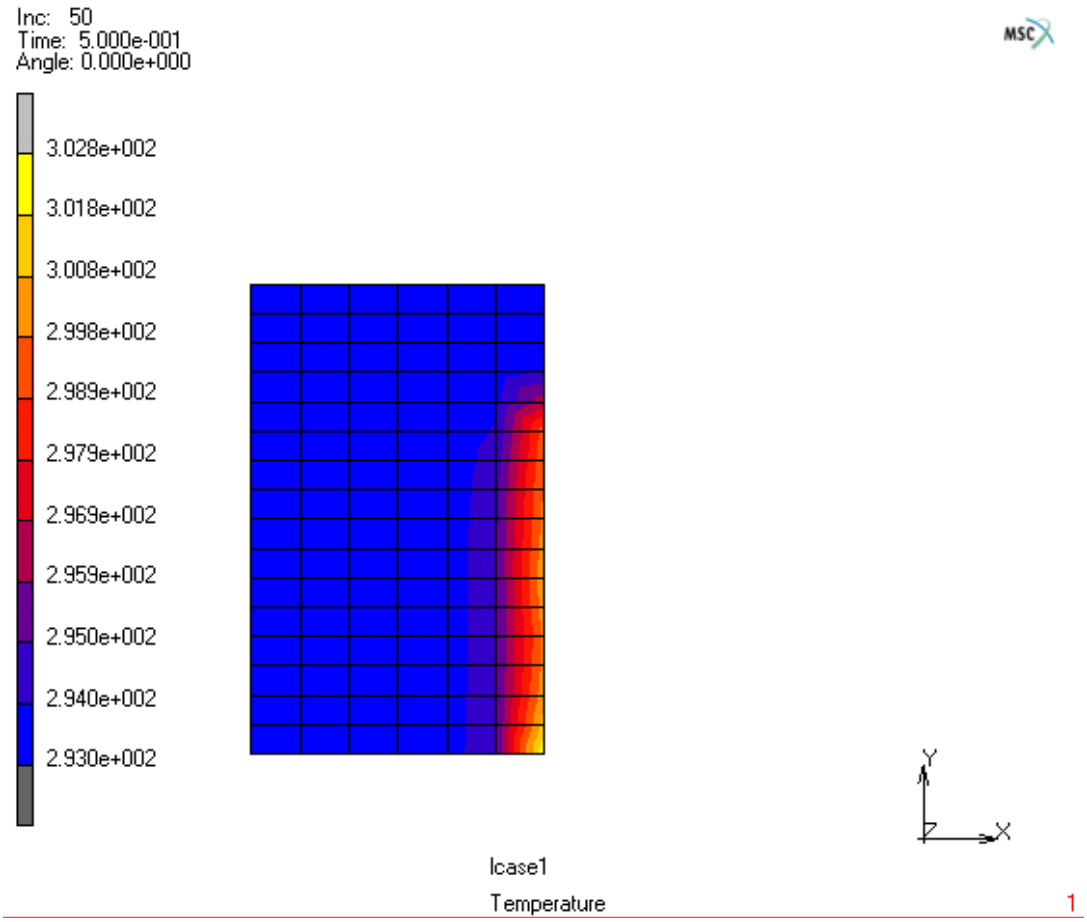


Figure 8.59-4 Temperature Distribution in the Punch of the Frictionless Model

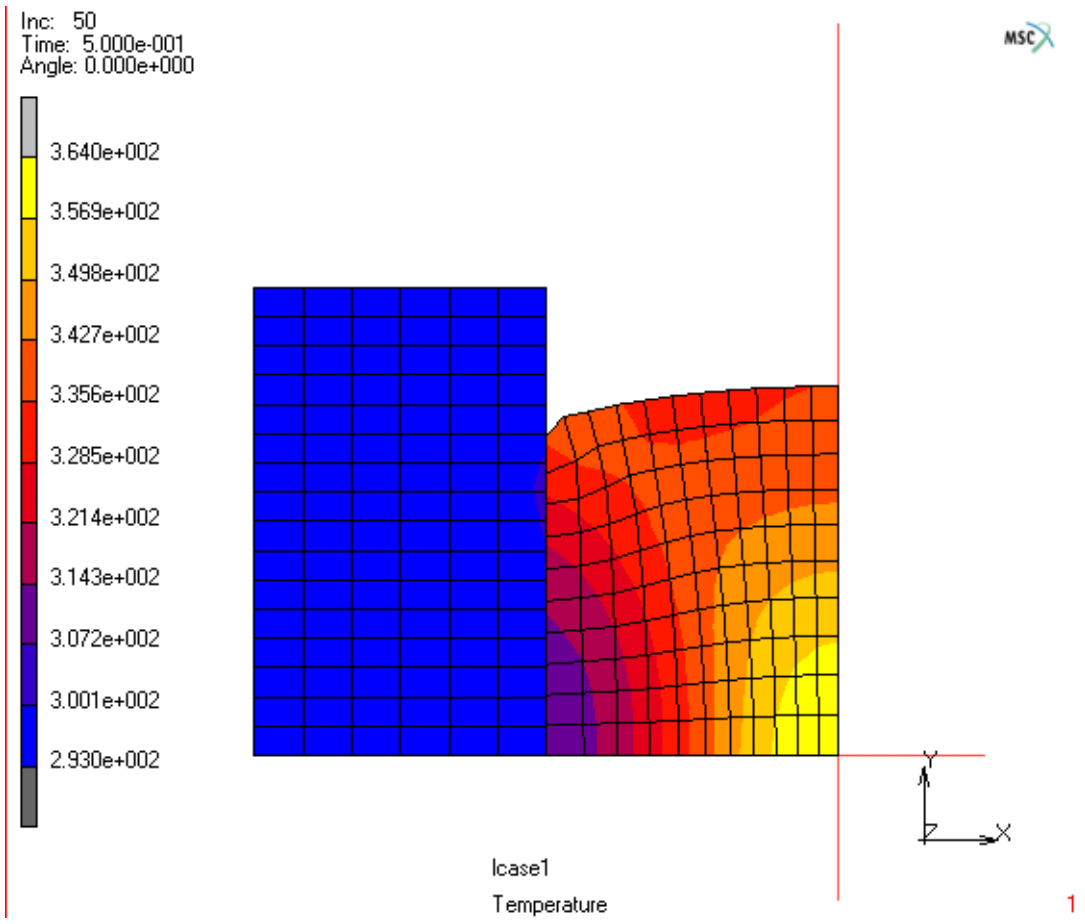


Figure 8.59-5 Temperature Distribution of the Model with Friction

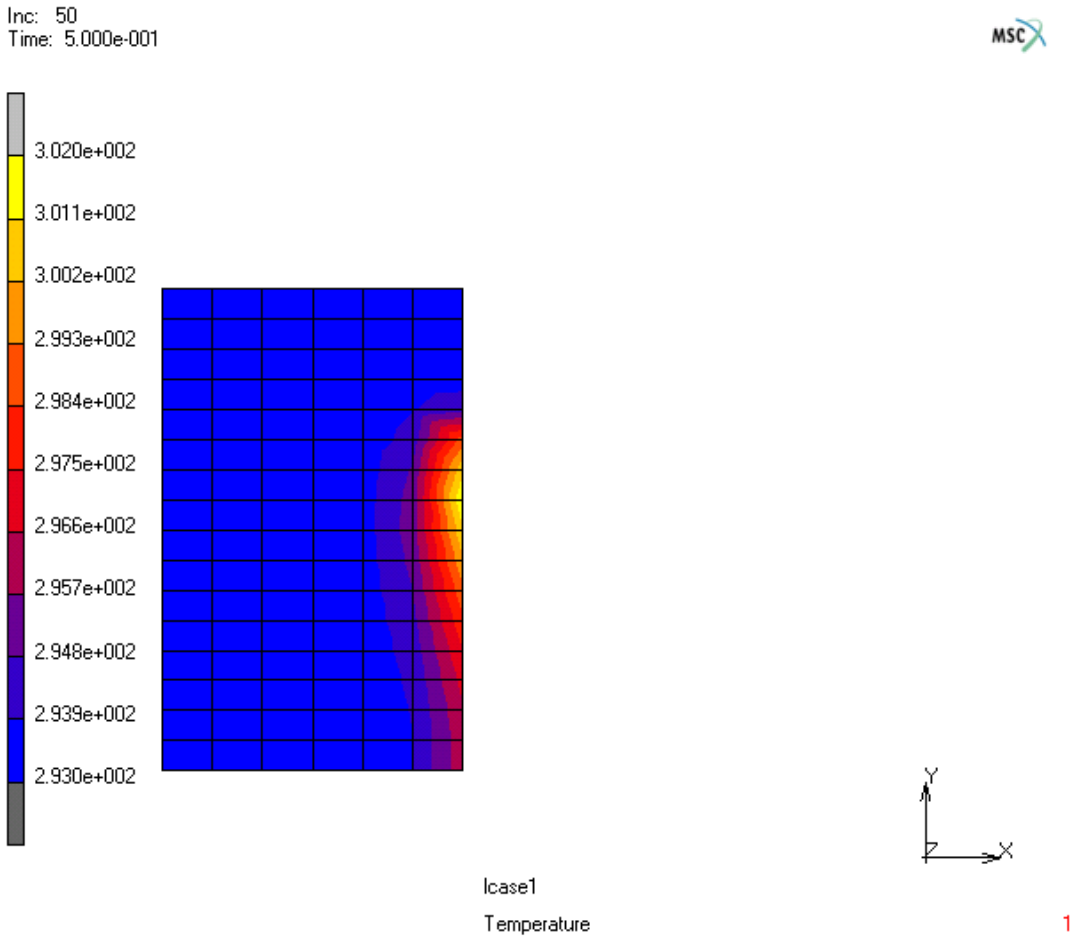


Figure 8.59-6 Temperature Distribution in the Punch of the Model with Friction

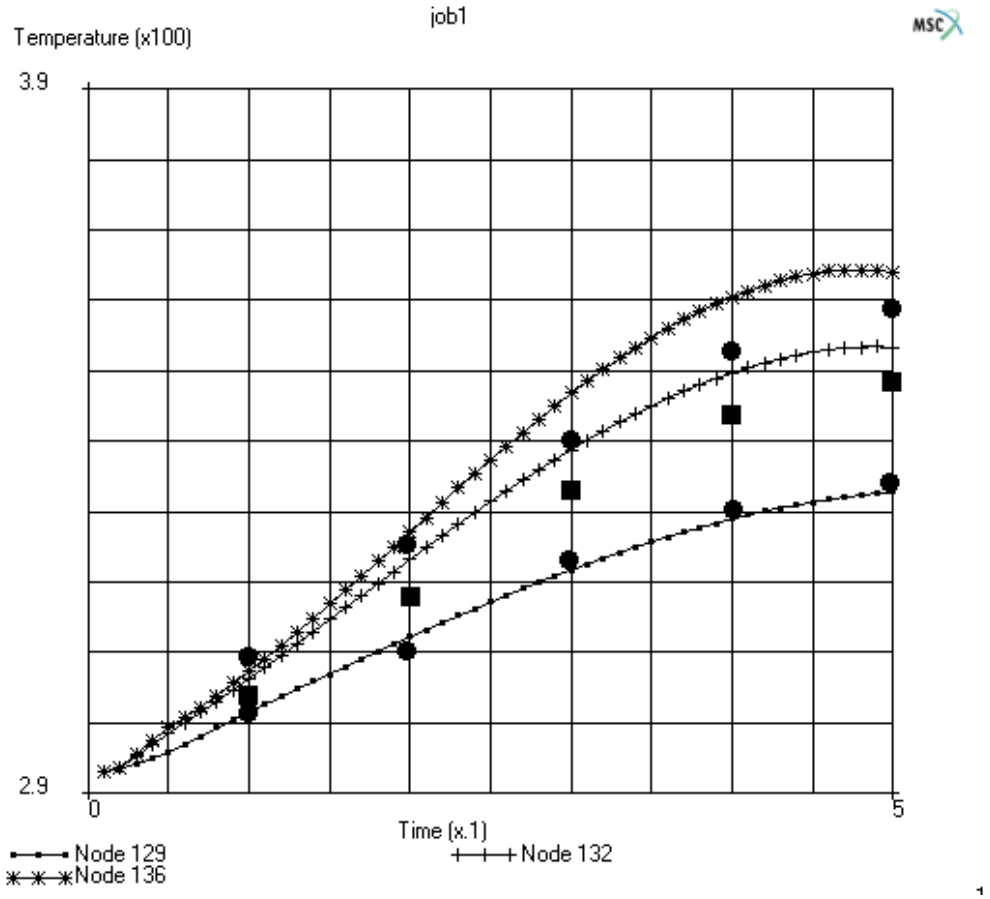


Figure 8.59-7 (A) Temperature Comparison with Experiment at Node 129, 132, and 136

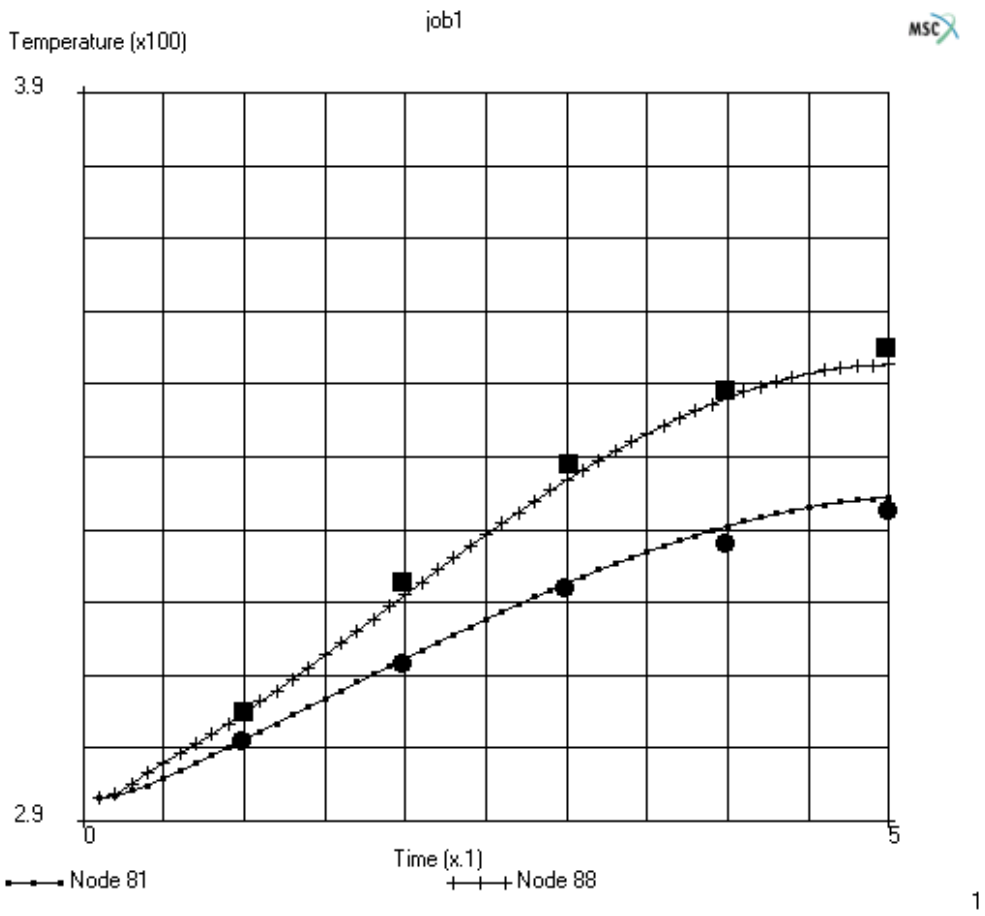


Figure 8.59-7 (B) Temperature Comparison with Experiment at Node 81 and 82

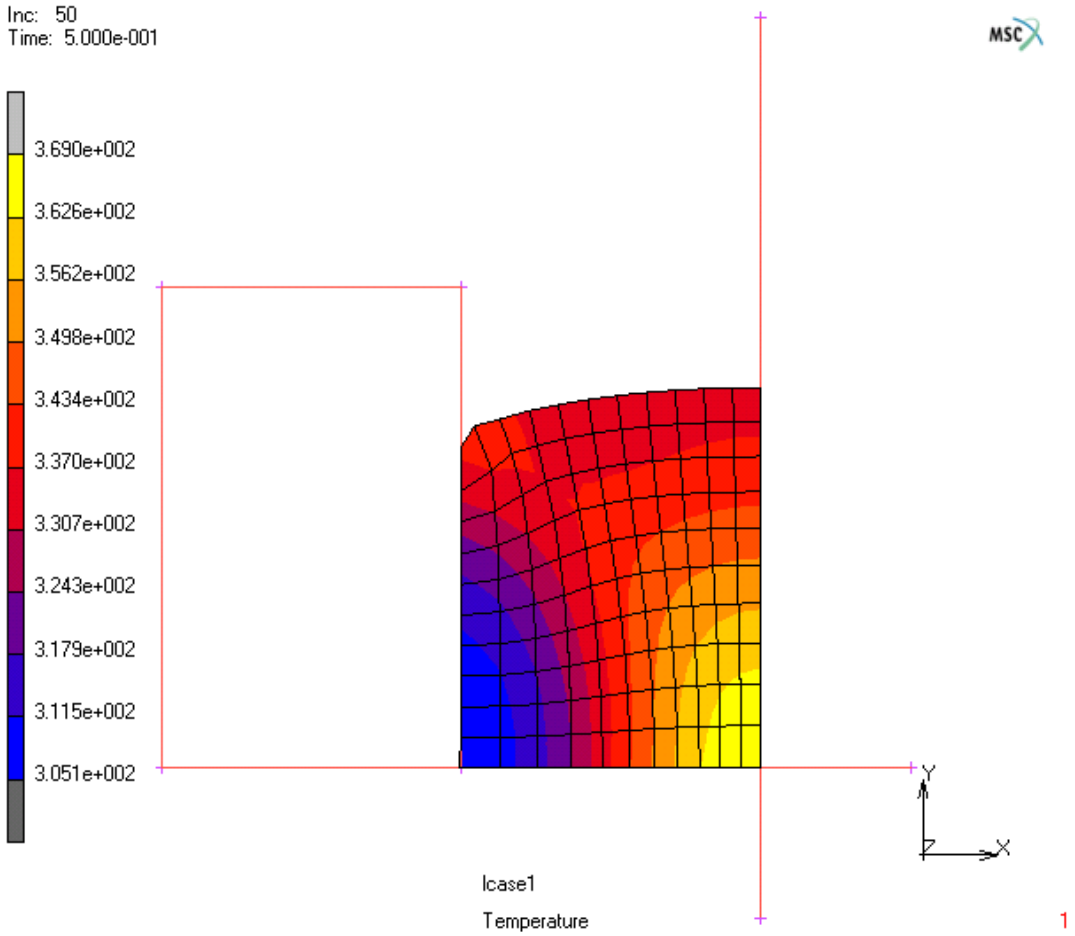


Figure 8.59-8 Temperature Distribution of the Model with Rigid Punch

8.60 Simulation of Sheet Bending

This example shows a simulation of the sheet bending process. A sheet is bent by deforming it with a punch into a die. In the sheet forming terminology, it is also called air bending or v-bending. This example demonstrates the forming and springback of the sheet.

Model

The sheet is modeled with 300 elements and 366 nodes. The punch and die are modeled as rigid bodies. The initial model is shown in Figure 8.60-1.

In e8x60.dat, the sheet is made of pure metal. In e8x60b.dat, the sheet is made of composite materials; therefore, element type 151 is used for the analysis.

Element

In e8x60.dat, the 4-node isoparametric quadrilateral plane strain element number 11 is used with the constant dilatation option. In e8x60b.dat, the element type 151 is used. This is a 4-node, plane strain, composite element.

Material Properties

In e8x60.dat, the sheet is assumed to be isotropic. The Young's modulus is 3×10^7 psi and the Poisson's ratio is 0.30. The initial yield stress is 5×10^4 psi. The workhardening behavior is input using the WORK HARD model definition option. In e8x60b.dat, two types of materials are used for different layers of composite. One is the same as the material used in e8x60.dat. The other is also isotropic, but with a Young's modulus of 100 psi and a Poisson's ratio of 0.49.

Boundary Conditions

The boundary conditions along the x direction are enforced by setting a zero x displacement boundary condition on all nodes at the center line of the sheet. The boundary conditions along the y direction are enforced through the contact option.

Contact

There are a total of three contact bodies in the problem. Contact body 1 is the deformable sheet. Body 2 is a velocity controlled rigid surface and models the punch. Body 3 is also a rigid surface and represents the lower die. Shear friction is assumed

with a coefficient of 0.10. The relative velocity below which a node is assumed to be sticking to a contact surface is set to be 1×10^{-3} in/s. The nodal reaction force required to separate a contacting node from its contacted surface is assumed to be 1×10^{-2} lb.

Control

The convergence control is governed by a relative displacement increment norm. The maximum allowed relative change in displacement increment is set to 0.10.

History Definition

The loading is done by moving the punch (contact body 2) along the negative y direction with a speed of -5×10^{-3} inches per second for 120 increments.

The motion direction of the punch is reversed at the end of 120 increments by prescribing a speed of 1×10^{-2} inches per second along the positive y direction for an additional 50 increments. The AUTO SWITCH option is used in this springback process.

Results

For e8x60.dat, the deformed shape is shown for increment 25 in Figure 8.60-2. The deformed shape is shown for increment 50 in Figure 8.60-3.The deformed shape is shown for increment 100 in Figure 8.60-4. At increment 118, the sheet contacts the flat portion of the lower die. For the next 2 increments, the sheet is driven into the lower die by the downward motion of the punch. The deformed shape is shown for increment 120 in Figure 8.60-5. For the next 50 increments, the punch moves upward and the sheet springs back. The final deformed configuration after springback is shown in Figure 8.60-6. A magnified view of contours of total effective plastic strain for increment 170 is shown in Figure 8.60-7.

For e8x60b.dat, the deformed shapes for increments 100, 120, and 170 are shown in Figures 8.60-8, 8.60-9, and 8.60-10, respectively.

Parameters, Options, and Subroutines Summary

Example e8x60.dat:

Parameters	Model Definition Options	History Definition Options
ALL POINTS	CONNECTIVITY	AUTO LOAD
ELEMENTS	CONTACT	CONTINUE
END	CONTROL	MOTION CHANGE
PLASTICITY	COORDINATES	TIME STEP
SETNAME	END OPTION	
SIZING	FIXED DISP	
TITLE	ISOTROPIC	
	SOLVER	
	WORK HARD	

Example e8x60b.dat:

Parameters	Model Definition Options	History Definition Options
ALL POINTS	COMPOSITE	AUTO LOAD
ELEMENTS	CONNECTIVITY	CONTINUE
END	CONTACT	MOTION CHANGE
PLASTICITY	CONTROL	TIME STEP
SETNAME	COORDINATES	
SIZING	END OPTION	
TITLE	FIXED DISP	
	ISOTROPIC	
	SOLVER	
	WORK HARD	

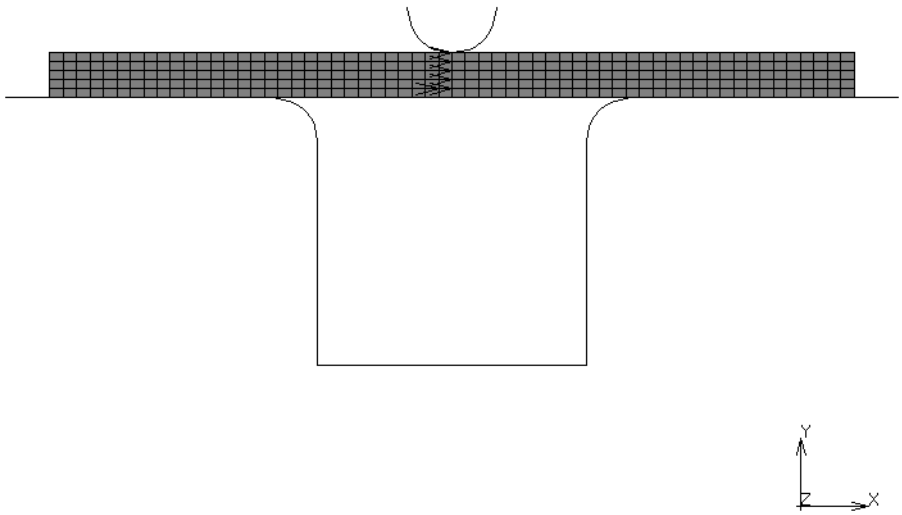


Figure 8.60-1 Initial Model

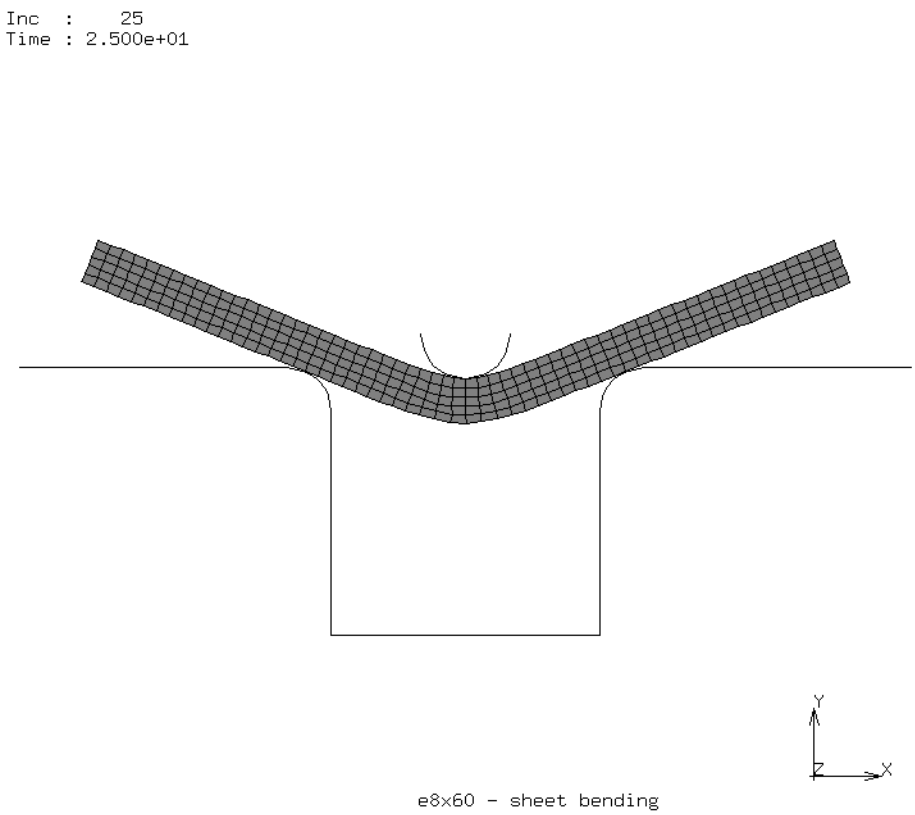


Figure 8.60-2 Deformed Geometry for Increment 25

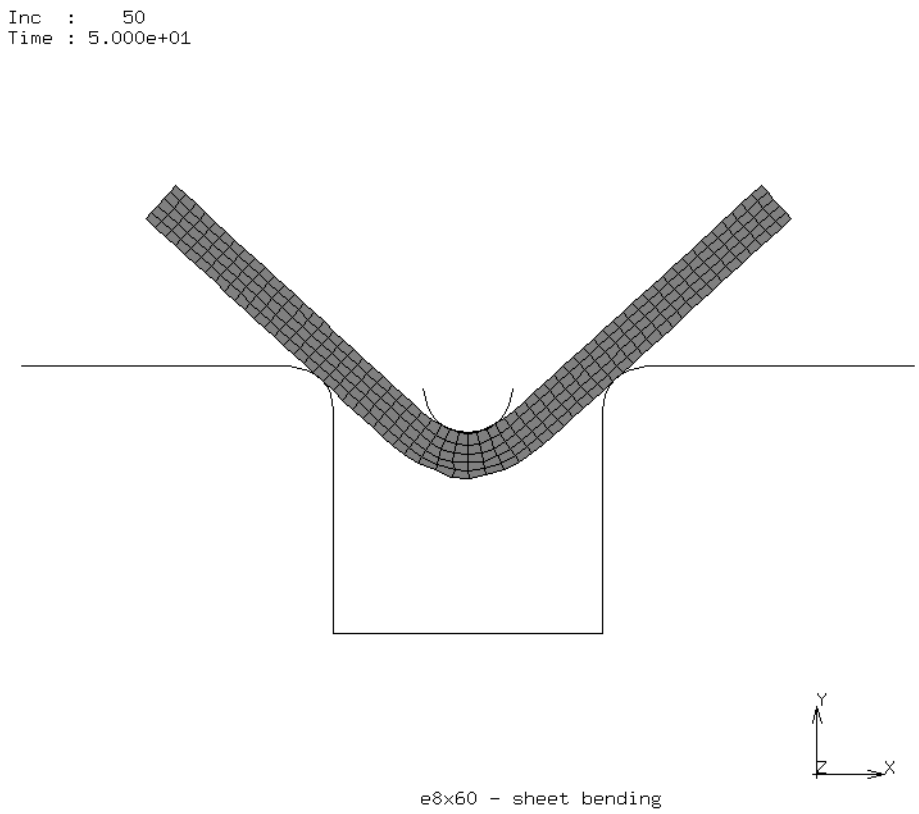


Figure 8.60-3 Deformed Geometry for Increment 50

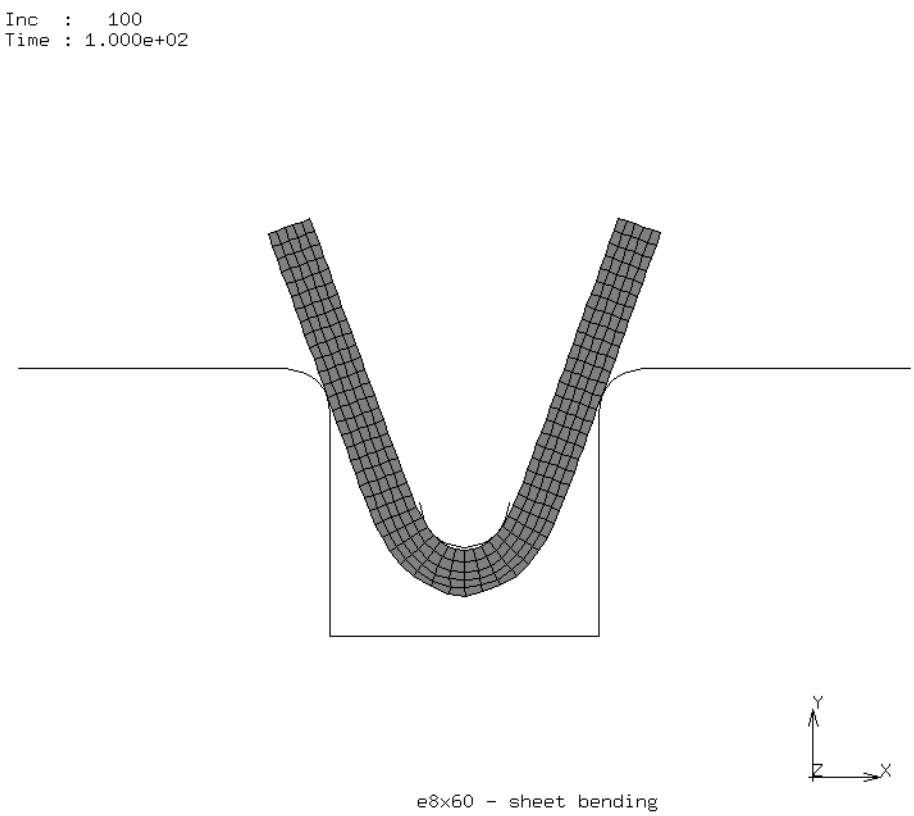


Figure 8.60-4 Deformed Geometry for Increment 100

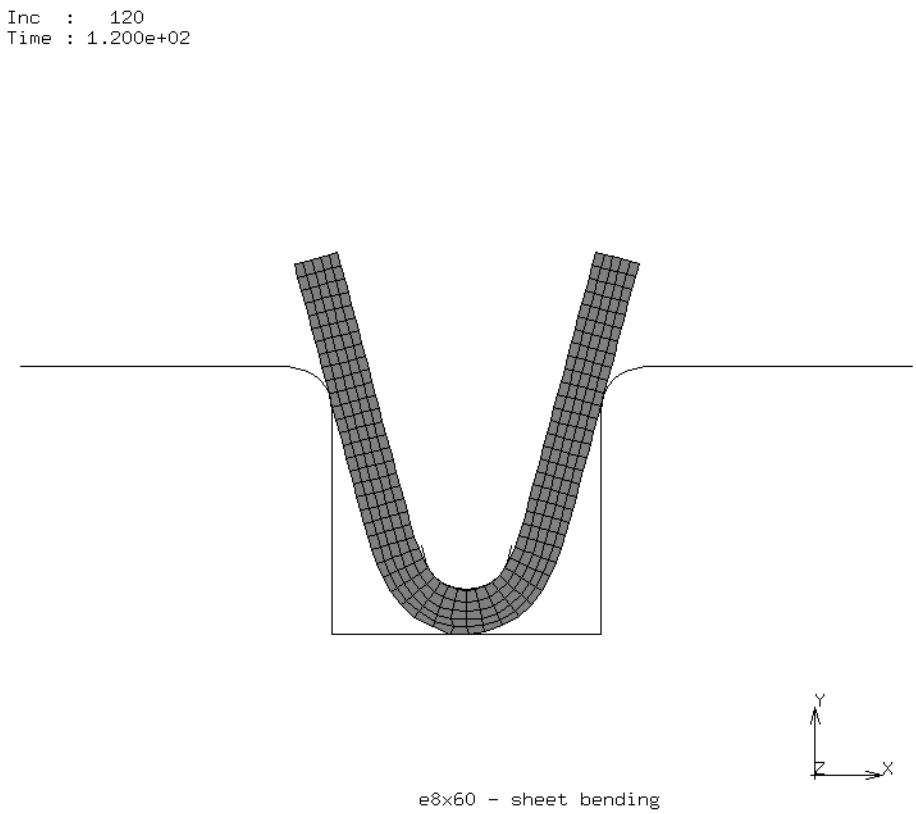


Figure 8.60-5 Deformed Geometry for Increment 120

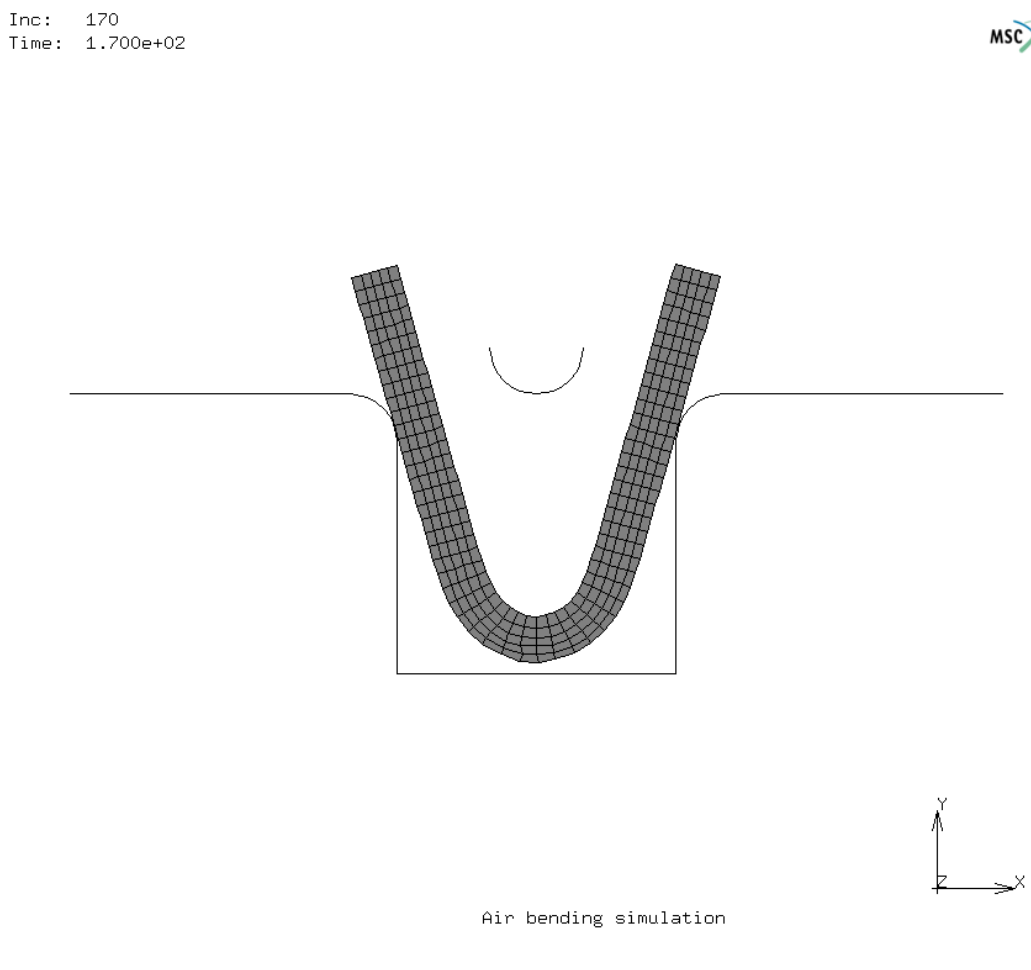


Figure 8.60-6 Deformed Geometry for Increment 170

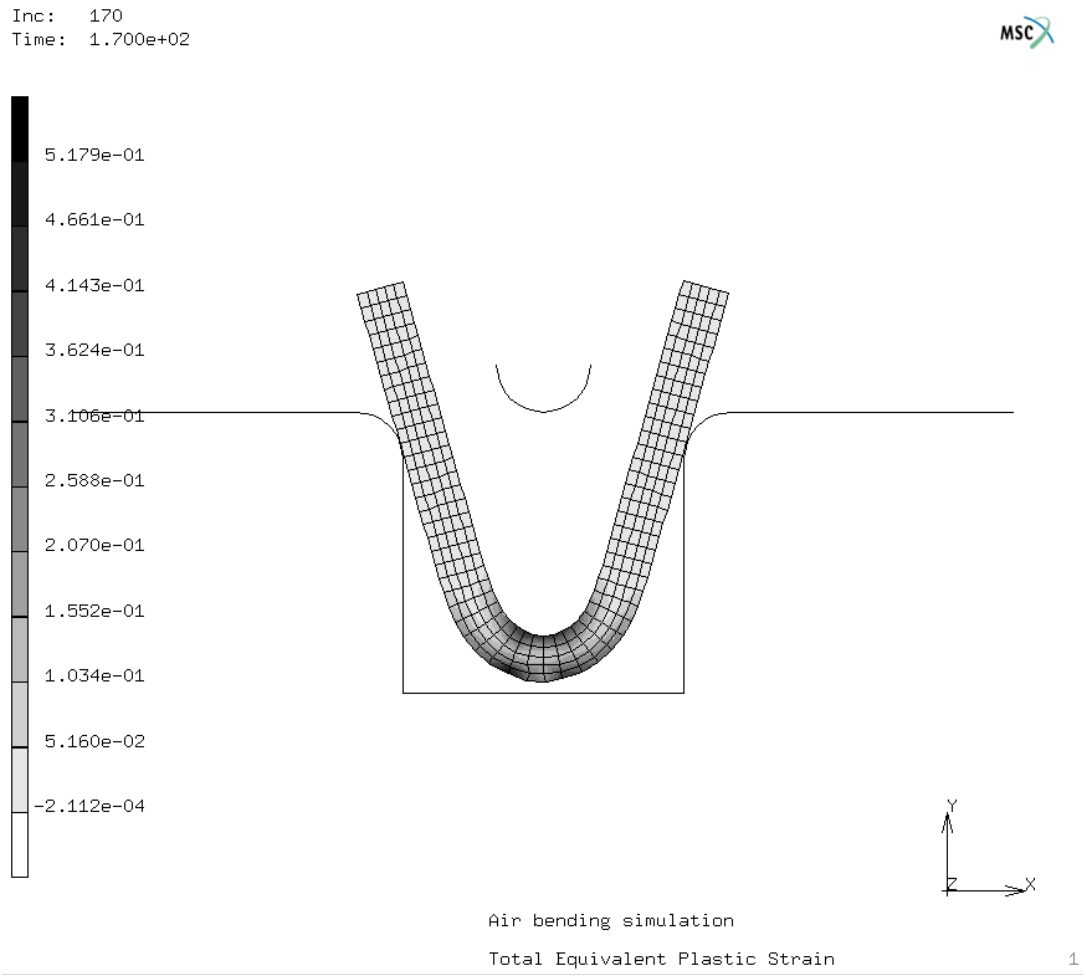


Figure 8.60-7 Magnified View of Contours of Total Effective Plastic Strain for Increment 170

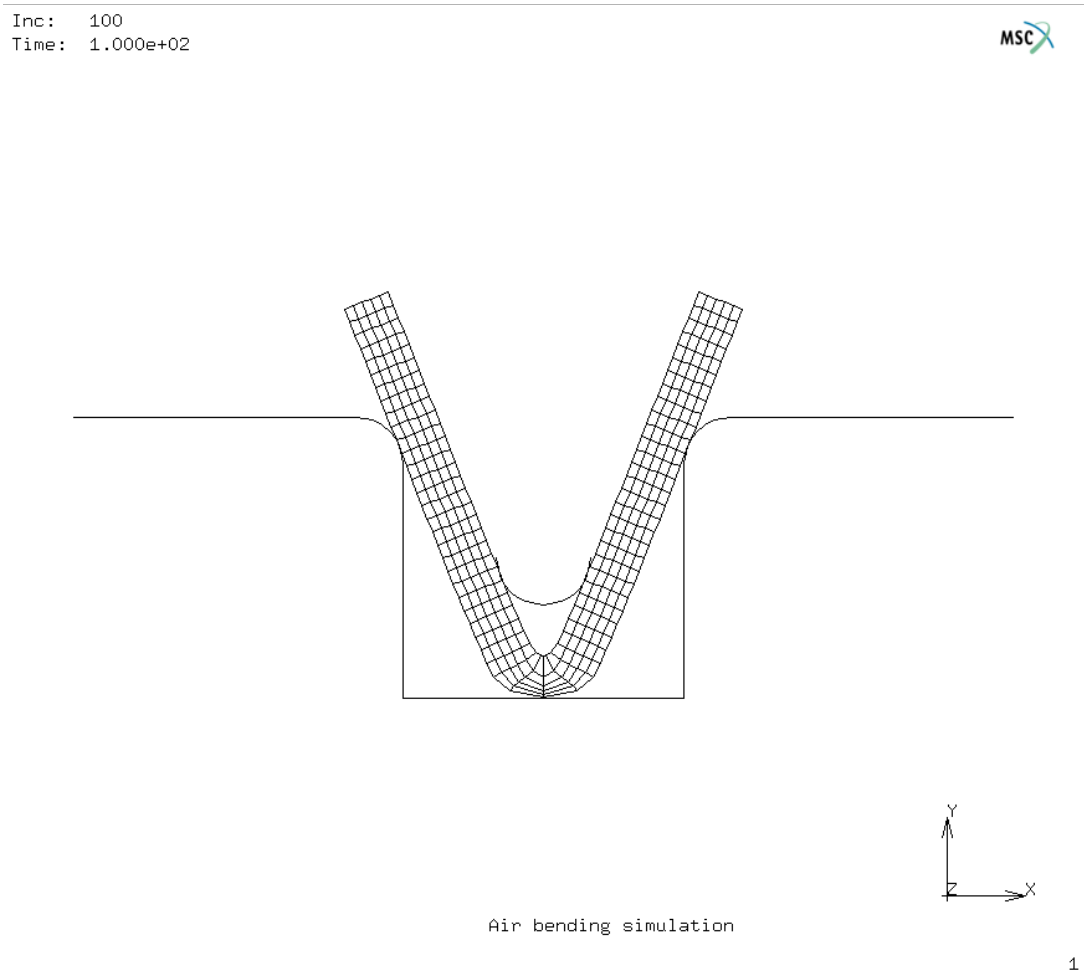


Figure 8.60-8 Deformed Mesh at Increment 100 for e8x60b.dat

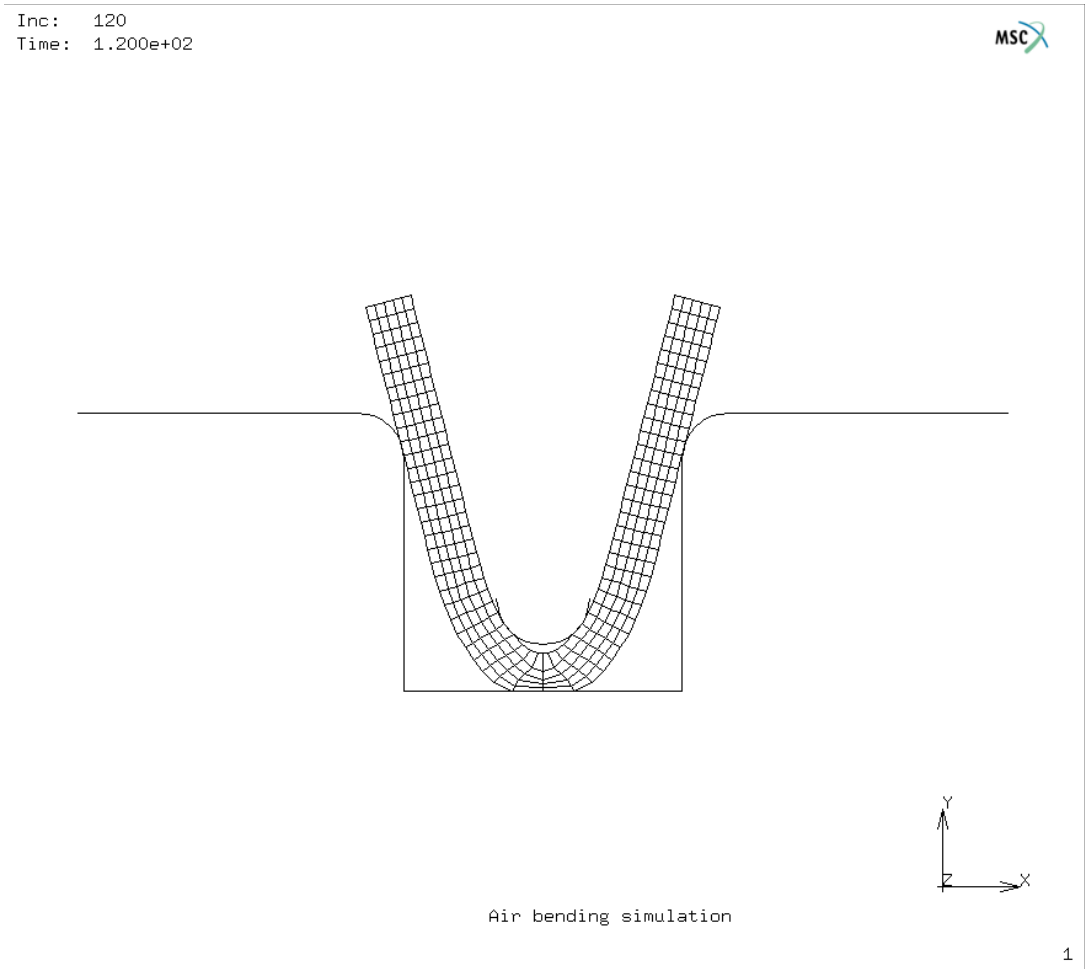


Figure 8.60-9 Deformed Mesh at Increment 120 for e8x60b.dat

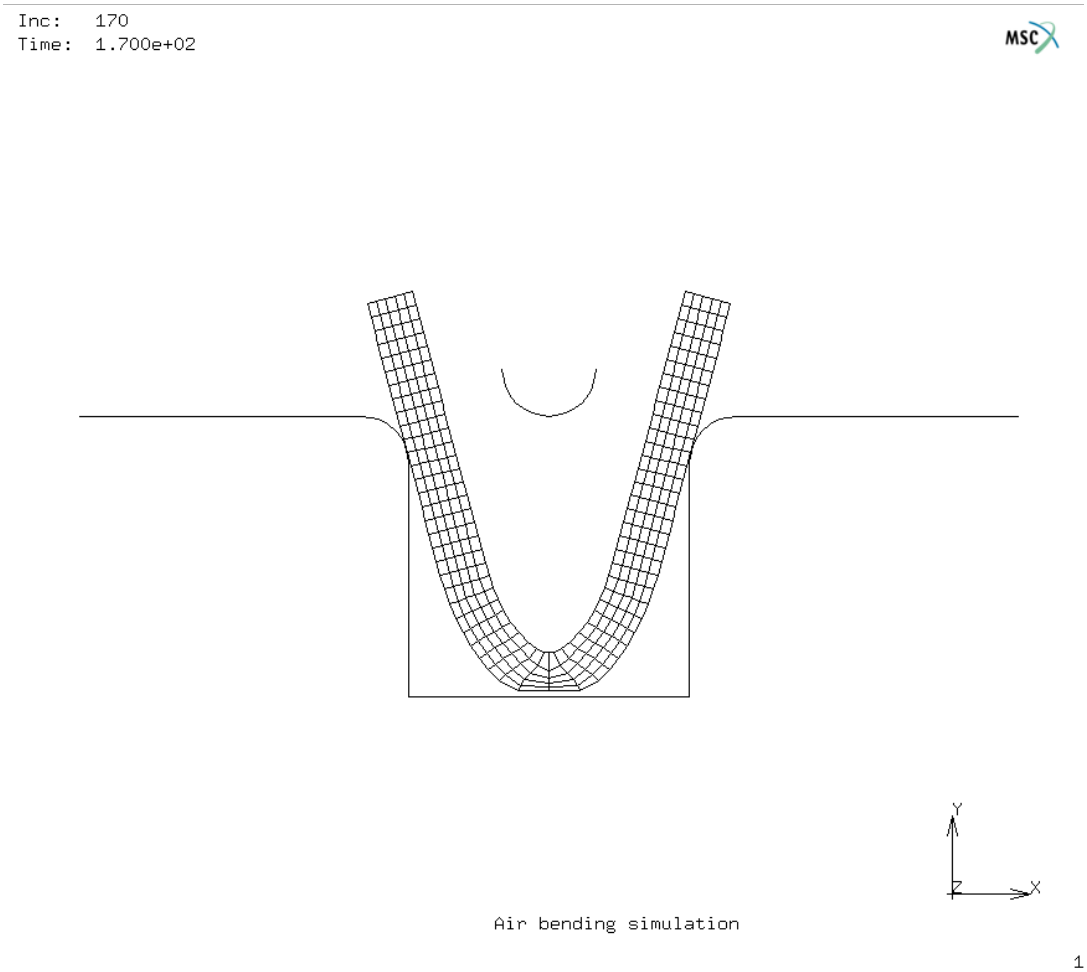


Figure 8.60-10 Deformed Mesh at Increment 170 for e8x60b.dat

8.61 Simulation of Rubber Bushing

A rubber bushing with an outer diameter of 10 cm and an inner diameter of 2 cm is considered. The length of the rubber bushing is 8 cm. Both outside and inside surfaces are glued to two steel tubes with corresponding diameters so that shape of the surfaces keeps unchanged during deformation. Two load sequences are applied. In the first step, a displacement of 2 cm along the symmetric axis is applied to the outside steel tube. During this load step, the deformation is purely axisymmetric. Afterwards, the outside steel tube moves 1 cm in the radial direction. In the second step, the problem becomes fully three-dimensional.

This example demonstrates the use of the data transfer capabilities of Marc from an axisymmetric analysis to an fully three-dimensional analysis.

Model

In axisymmetric analysis, the rubber bushing is modeled with 320 element and 371 nodes. The finite element mesh is shown in Figure 8.61-1. The model used in 3-D analysis is shown in Figure 8.61-2, which contains 3840 elements and 4823 nodes. Because of symmetry, only half of the rubber bushing is considered. The 3-D mesh in Figure 8.61-2 is an expansion of the deformed axisymmetric mesh in Figure 8.61-1.

Elements

The 4-node isoparametric quadrilateral axisymmetric element 10 is used in the axisymmetric run. The corresponding element type in 3-D run is 7 which is the 8-node isoparametric hexahedral element. In the analysis, both element types are based on mixed formulations and formulated on the deformed (updated) configuration. This is activated using ELASTICITY,2 parameter.

Material Properties

The rubber bushing is modeled using Mooney constitutive model. The material parameters are given as C_1 = 8 N/cm and C_2 = 2 N/cm.

Boundary Conditions and Load Definitions

Both outside and inside surfaces of the rubber bushing are glued to two steel tubes with corresponding diameters so that shape of the surfaces keeps unchanged during deformation. Two load sequences are applied. In the first step, a displacement of 2 cm along the symmetric axis is applied to the outside steel tube within 10 equal increments. During this load step, the deformation is purely axisymmetric and therefore an axisymmetric analysis is performed. Afterwards, the outside steel tube

moves 1 cm in the radial (Y) direction within 5 equal increments. In the second step, the problem becomes fully three-dimensional and therefore a 3-D analysis is performed.

Results

The deformed mesh and the distribution of equivalent von Mises stress at the end of axisymmetric analysis are shown in Figure 8.61-3. The corresponding results at increment 0 of the 3-D analysis is shown in Figure 8.61-4 which demonstrates the correctness of the axisymmetric to 3-D data transfer. Figure 8.61-5 contains the final deformed shape and the distribution of the equivalent von Mises stress.

Parameters, Options, and Subroutines Summary

Example e8x61a.dat:

Parameters	Model Definition Options	History Definition Options
ALL POINTS	CONNECTIVITY	AUTO LOAD
ELASTICITY	COORDINATES	CONTINUE
ELEMENTS	END OPTION	CONTROL
END	FIXED DISP	DISP CHANGE
PROCESSOR	MOONEY	TIME STEP
SETNAME	NO PRINT	TITLE
SIZING	OPTIMIZE	
TITLE	POST	
	SOLVER	

Example e8x61b.dat:

Parameters	Model Definition Options	History Definition Options
ALL POINTS	AXITO3D	AUTO LOAD
ELASTICITY	CONNECTIVITY	CONTINUE
ELEMENTS	COORDINATES	CONTROL
END	END OPTION	DISP CHANGE
PROCESSOR	FIXED DISP	TIME STEP
SETNAME	MOONEY	TITLE
SIZING	NO PRINT	
TITLE	OPTIMIZE	
	POST	
	SOLVER	

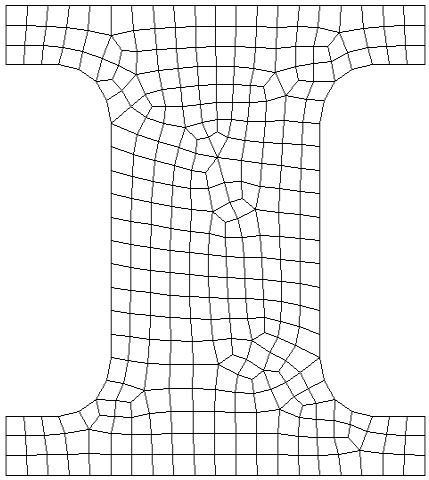


Figure 8.61-1 FE-Mesh for Axisymmetric Analysis

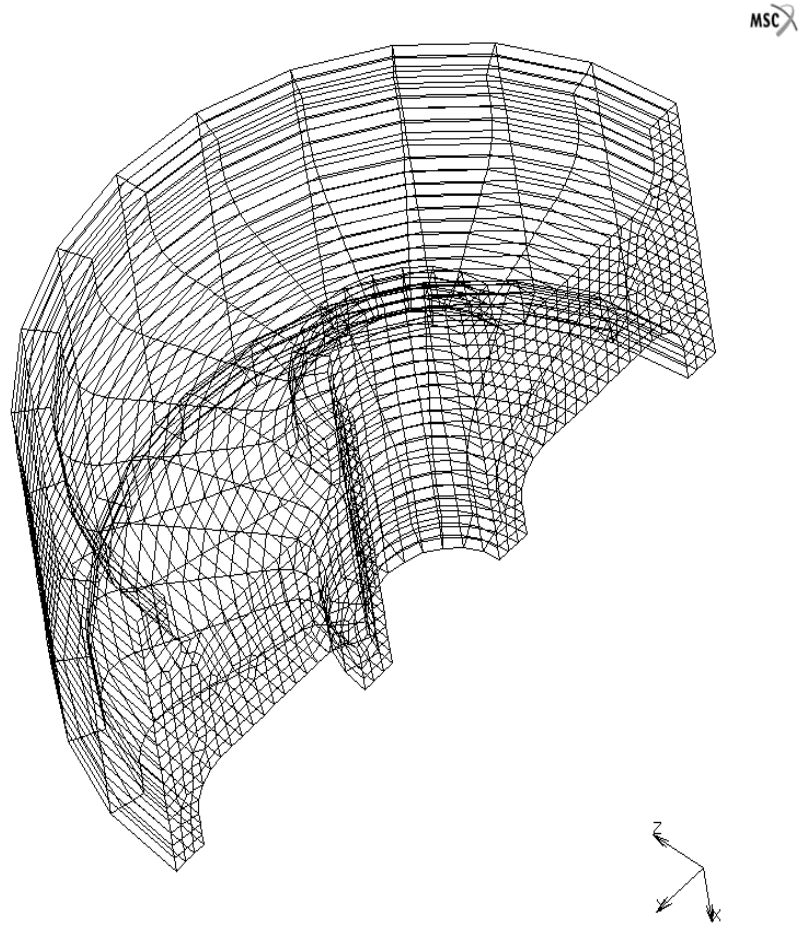


Figure 8.61-2 FE-Mesh for 3-D Analysis

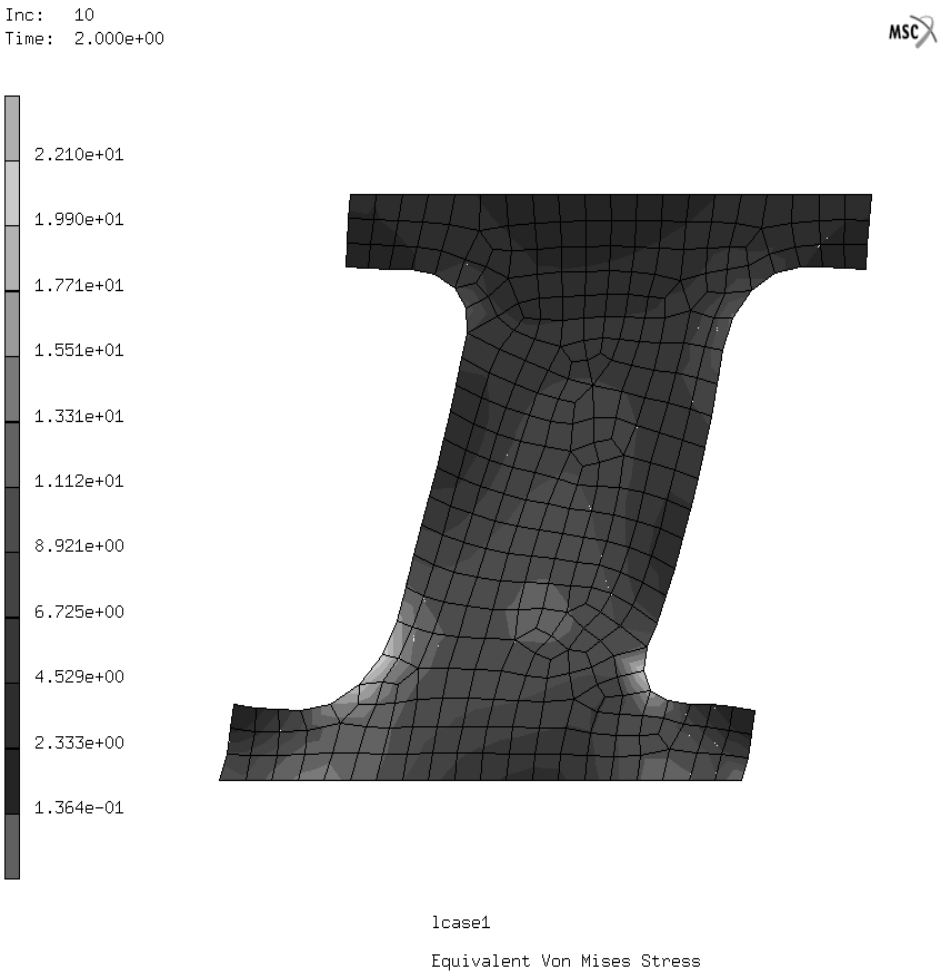


Figure 8.61-3 Deformed Mesh and Distribution of Equivalent von Misses Stress at End of First Load Step

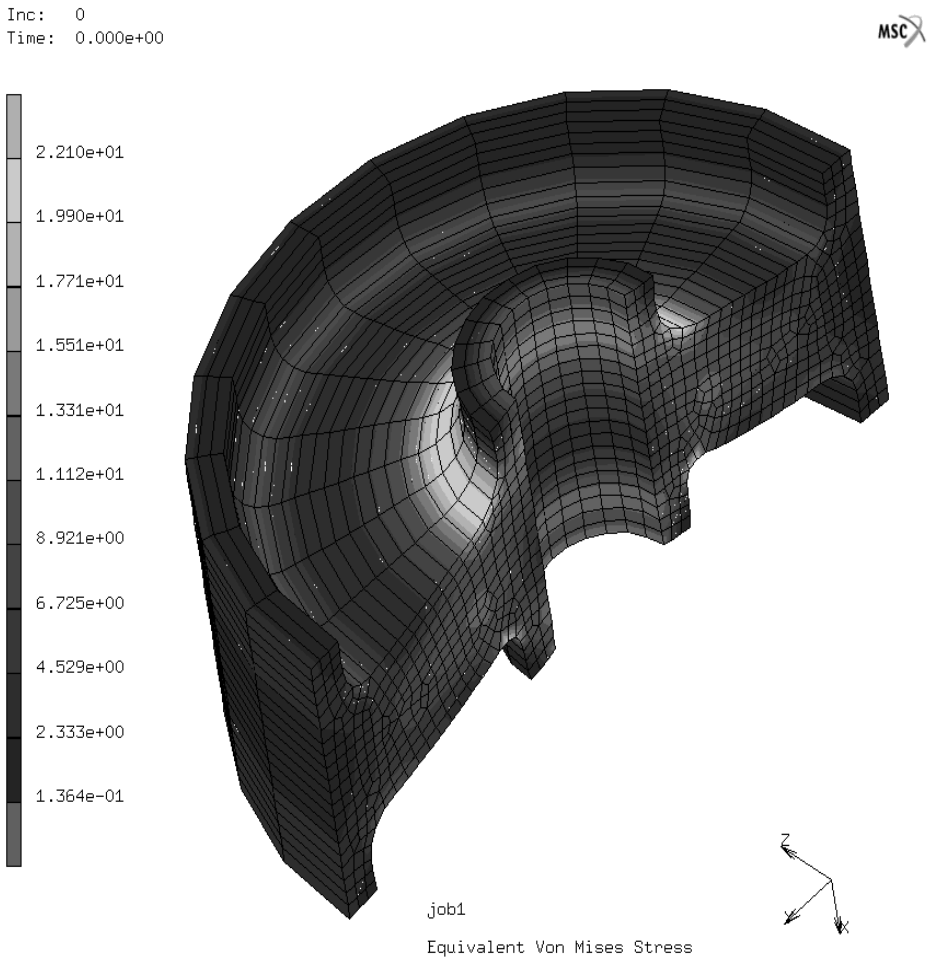


Figure 8.61-4 Deformed Mesh and Distribution of Equivalent Stress at Beginning of 3-D Analysis

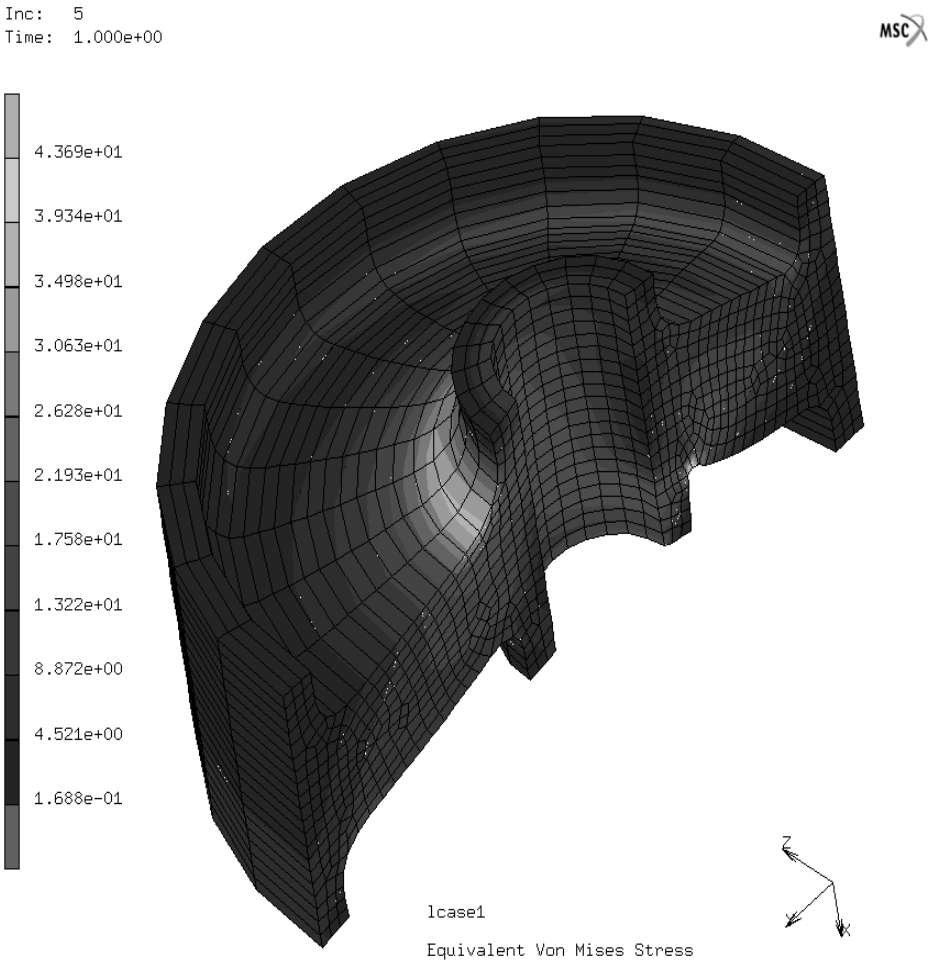


Figure 8.61-5 Deformed Mesh and Distribution of the Equivalent Stress at End of Second Load Step

8.63 Coupled Structural-acoustic Analysis

In this example, a harmonic structural-acoustic analysis is performed on two spherical rooms, filled with air and separated by a rubber membrane (see Figure 8.63-1).

First, a harmonic analysis is performed on the stress-free structure. Then, a pre-stress is applied on the membrane, followed by a second harmonic analysis.

Elements

Element type 40, a 4-node axisymmetric isoparametric heat transfer element, is used to model the air. Element type 82, a 4-node axisymmetric isoparametric element for incompressible material, is used to model the membrane. Both element types use full integration.

Harmonic

The HARMONIC parameter is used since a frequency response analysis will be performed.

Acoustic

A harmonic acoustic analysis will be performed, which must be set by the ACOUSTIC parameter.

Mooney

The material properties of the elements corresponding to the membrane are given by a Mooney constant $C_{01} = 80\times10^5 N/m^2$ and a density of $1000 kg/m^3$.

Acoustic

The air in the spherical rooms is characterized by a bulk modulus of $1.5\times10^5 N/m^2$ and a density of $1 kg/m^3$.

Region

The REGION model definition option is used to indicate which elements correspond to the acoustic and which elements correspond to the solid part of the model.

Fixed Displacement

The nodes at the outer radius of the membrane are fixed in both x- and y-direction.

Contact

Three contact bodies are defined. The first two bodies are acoustic bodies and contain the air in the spherical rooms. The third body is a deformable body and contains the elements of the rubber membrane (see Figure 8.63-2).

Exclude

The EXCLUDE option is used to avoid that nodes of the acoustic contact bodies will touch segments of the deformable contact body which have a normal vector being parallel to the y-axis.

No Print

The NO PRINT model definition option is used to suppress print out.

Post

The default nodal variables are put on the post file; no element variables are selected.

Harmonic

The external load is applied at a frequency range from 60 to 90 Hz, with a step size of 0.3 Hz.

Press Change

Node 63 is loaded by a nodal pressure with magnitude 10.

Displacement Change

A y-displacement of 0.001 is applied to the nodes at the outer radius of the membrane in order to introduce a pre-stressed state in the membrane, prior to a subsequent harmonic analysis.

Auto Load

The total displacement to get the pre-stress is applied in one step.

Time Step

Defining a time step during the pre-stressing of the membrane is necessary, because the CONTACT option is used.

Results

The pressure at node 168, located in the right room at the membrane as a function of the frequency is given in Figures 8.63-3 and 8.63-4, corresponding to the stress-free and the pre-stressed membrane, respectively. Due to the pre-stress, the peak value shifts to a higher frequency.

Parameters, Options, and Subroutines Summary

Example e8x63a.dat:

Parameters	Model Definition Options	History Definition Options
TITLE	SOLVER	TITLE
SIZING	OPTIMIZE	HARMONIC
ELEMENTS	CONNECTIVITY	PRESS CHANGE
PROCESSOR	COORDINATES	CONTROL
$NO LIST	MOONEY	AUTO LOAD
HARMONIC	ACOUSTIC	TIME STEP
ACOUSTIC	REGION	DISP CHANGE
END	FIXED DISP	CONTINUE
	CONTACT	
	EXCLUDE	
	NO PRINT	
	POST	
	END OPTION	

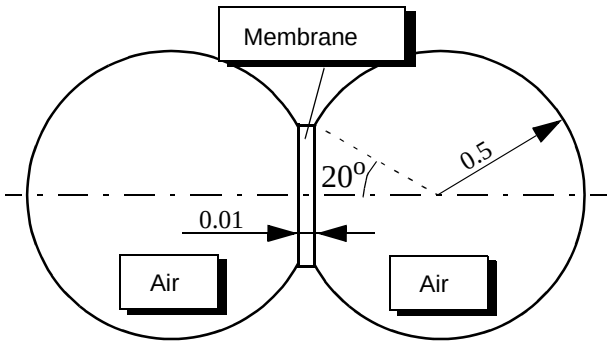


Figure 8.63-1 Coupled Structural-acoustic Analysis: Problem Description

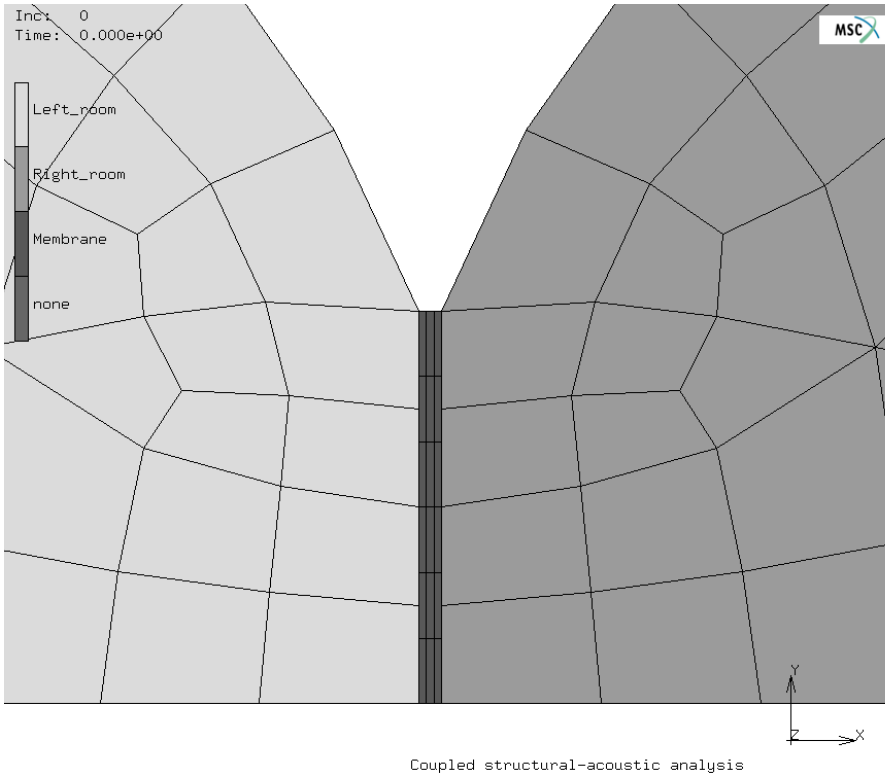


Figure 8.63-2 Contact Bodies Used

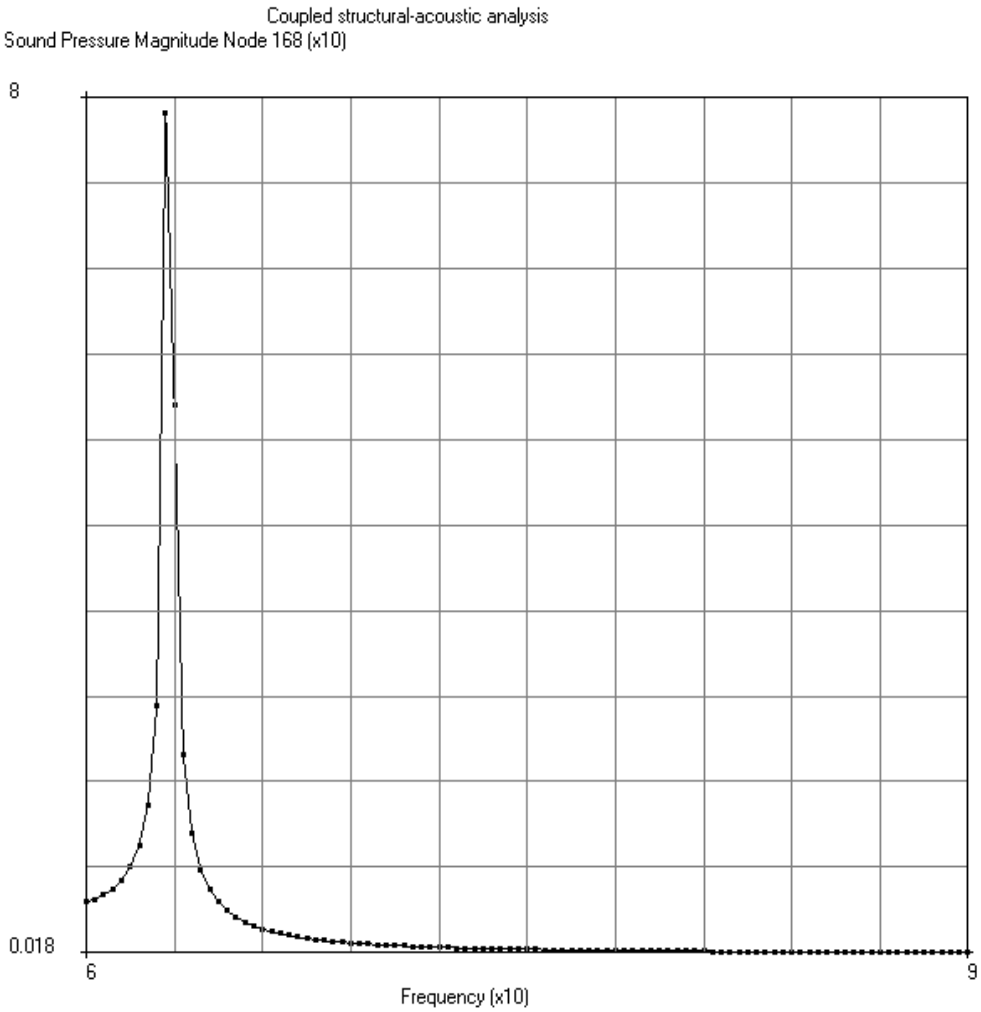


Figure 8.63-3 Sound Pressure Magnitude as a Function of the Frequency (Stress-free Membrane)

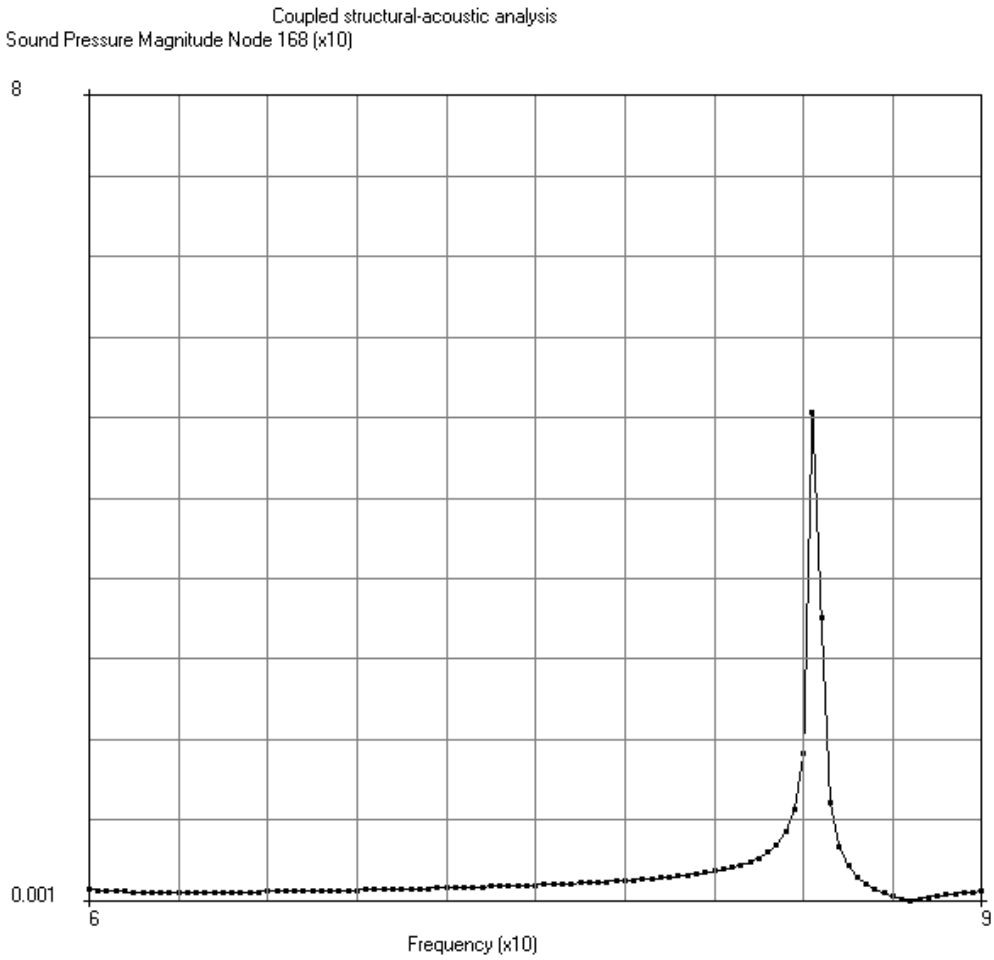


Figure 8.63-4 Sound Pressure Magnitude as a Function of the Frequency (Pre-stressed Membrane)

8.64 Simulation of Rubber and Metal Contact with Remeshing

This example shows a simulation of a rubber cushion with a metal fastener. The weight on the rubber cushion forces rubber to deform and then settles into a metal fastener. This example shows the necessity of the global remeshing and deformable-deformable contact between rubber and metal.

Model

The model is set up as a plane strain problem. Immediate remeshing on the rubber is used to create the initial mesh for the analysis (shown in Figure 8.64-1). The metal is steel with elastic material property and the rubber is of a mooney type. The global remeshing is controlled by penetration check and increment frequency check. The data file is named e8x64.dat.

Element

In e8x64.dat, the 4-node isoparametric quadrilateral plane strain element type 11 is used for the steel and the 4-node Herrmann element type 80 is used for the rubber cushion.

Material Properties

The steel is assumed to be isotropic. The Young's modulus is 21000 N/mm^2 and the Poisson's ratio is 0.30. The rubber cushion is using the Mooney constitutive model. The material properties are given as C^{10} = 0.8 N/mm^2, C^{01} = 0.2 N/mm^2 and K = 2000 N/mm^2 with mass density = 1.

Boundary Conditions

No boundary conditions are needed in the model.

Contact

A total of six contacting bodies is defined. Body 1 is the rubber cushion. Body 2 is the steel and Body 3 is the ground base. Body 4 is the tool that carries the weight and Body 5 is the ground base for the steel. Symmetric body is defined as body 6. A contact table is used, which defines rubber cushion in contact with all other bodies except the steel base. The steel is fixed to the steel base. The contact is controlled with a contact tolerance of 0.5mm and a 0.99 bias towards the rubber. This is to prevent penetration during the contact computation. No friction is applied to the boundary. A non-incremental splitting – the iterative penetration check and splitting is activated.

Global Remeshing Control

Because the deformation in the rubber is large, the global remeshing is required from time to time. This control is instructed though the ADAPT GLOBAL option. Advancing front mesher is selected. The remeshing is performed according to the penetration checking and increment frequency. The immediate remeshing flag is also turned on for the initial meshing. The penetration limit is set to 0.05mm. The new element size is set to 8.0mm. The minimum element size is limited to 2.0mm. Curvature of the boundary is used for an adaptive element size on the boundary.

Control

The convergence is controlled by either the relative residual criterion with 0.01 as tolerance or the relative displacement criterion with 0.001 as the tolerance. During the iteration loops, if analysis satisfies either criterion, the convergence is assumed reached. A maximum of 20 iterations is allowed.

History Definition

Constant displacement loading is used to move tool (Body 4) in the -Y direction with a velocity of 1mm/s. The loadcase uses 30 increments with time step 1.0s.

Results

In Figure 8.64-2 (A through C), the mesh and von Mises stress at various increments are shown.

In Figure 8.64-3, the X displacement of a node on the tip of the steel fastener is displayed. Finally, in Figure 8.64-4, the load applied to the rubber cushion and the load applied to the steel in the X direction are shown. All the results here demonstrate the capability of Marc to simulate the interaction of metal and rubber with large deformation and nonlinear material properties. The global remeshing capability allows analysis to avoid element distortion and penetration. The simulation allows you to design rubber cushion and steel fastener with the required weight.

Parameters, Options, and Subroutines Summary

Example e8x64.dat:

Parameters	Model Definition Options	History Definition Options
ALL POINTS	CONNECTIVITY	AUTO LOAD
ELEMENTS	CONTACT	CONTINUE
END	CONTROL	MOTION CHANGE
PLASTICITY	COORDINATES	TIME STEP
SETNAME	END OPTION	CONTACT TABLE
SIZING	MOONEY	
ELASTICITY	ISOTROPIC	
REZONING	CONTACT TABLE	
ADAPTIVE	GEOMETRY	
PROCESSOR	SOLVER	
	OPTIMIZE	
	PARAMETERS	
	ADAPT GLOBAL	

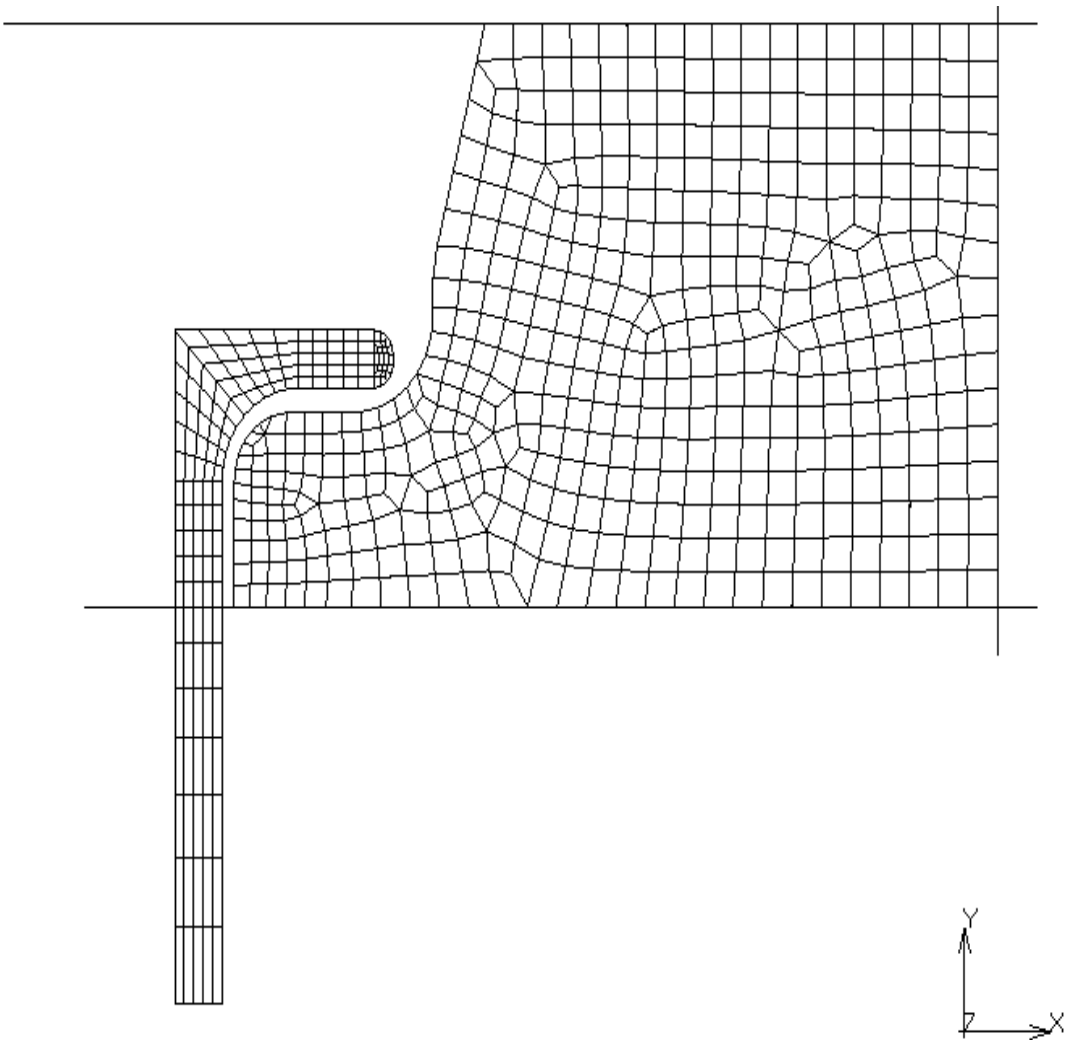


Figure 8.64-1 Initial Mesh of Rubber Cushion and Steel Fastener

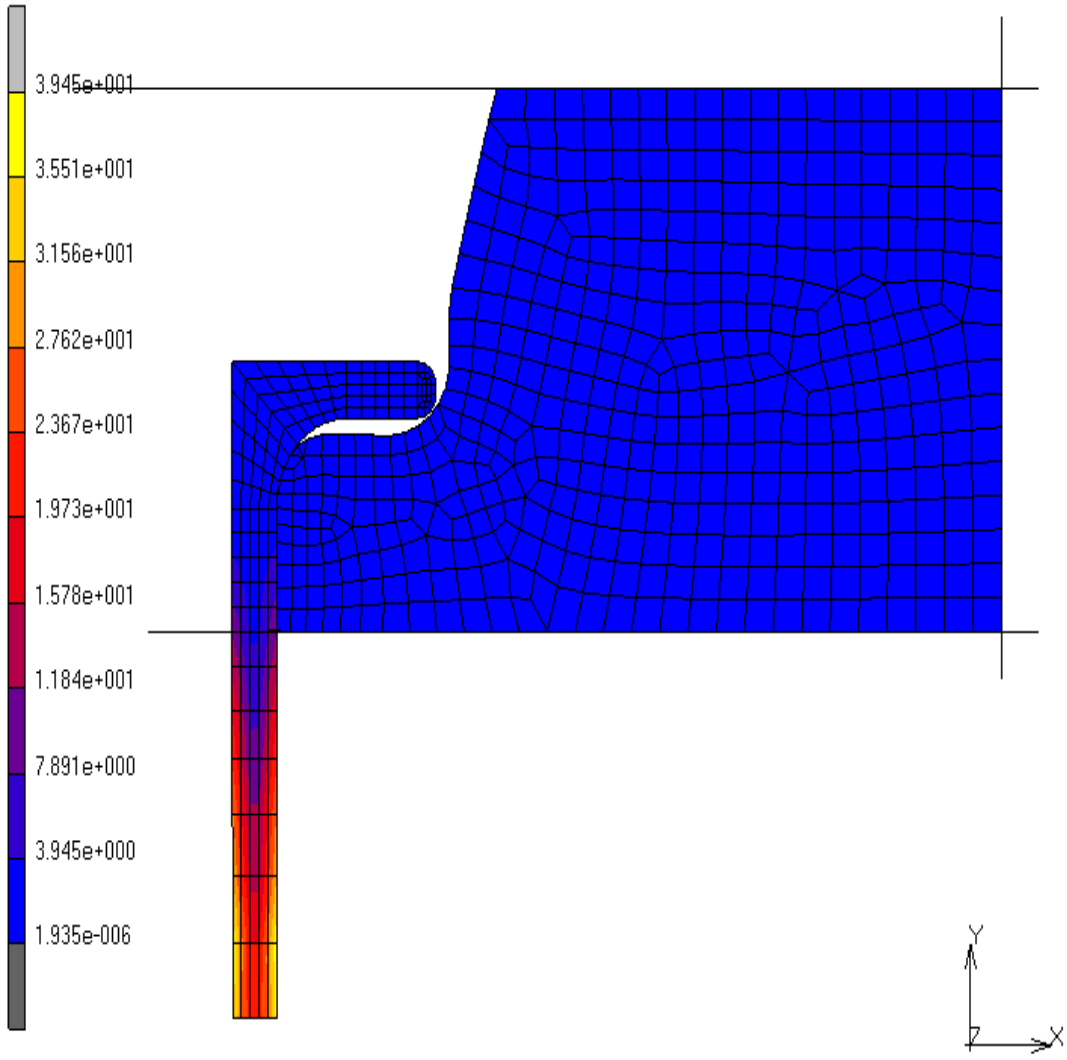


Figure 8.64-2 (A) von Mises Stress at Increment 5

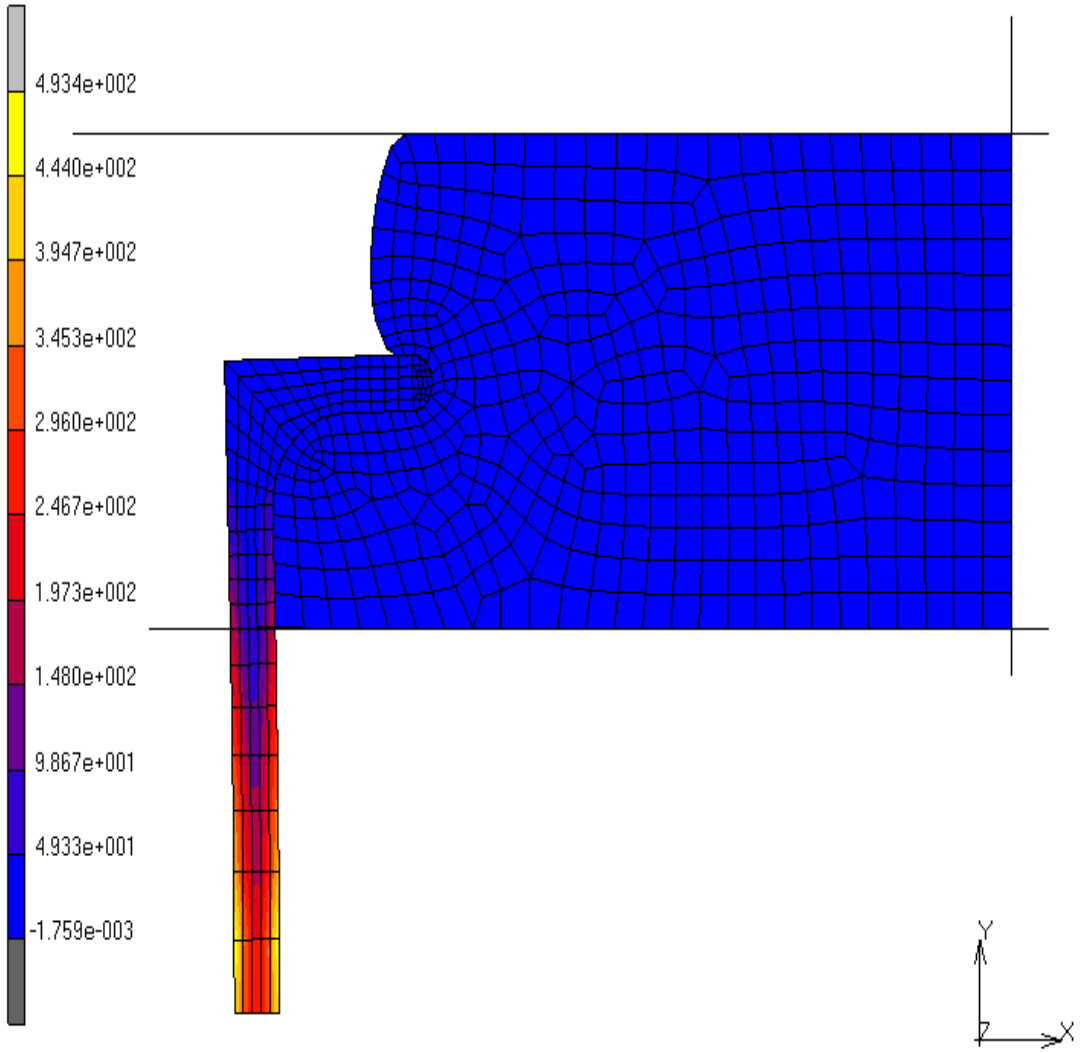


Figure 8.64-2 (B) von Mises Stress at Increment 15

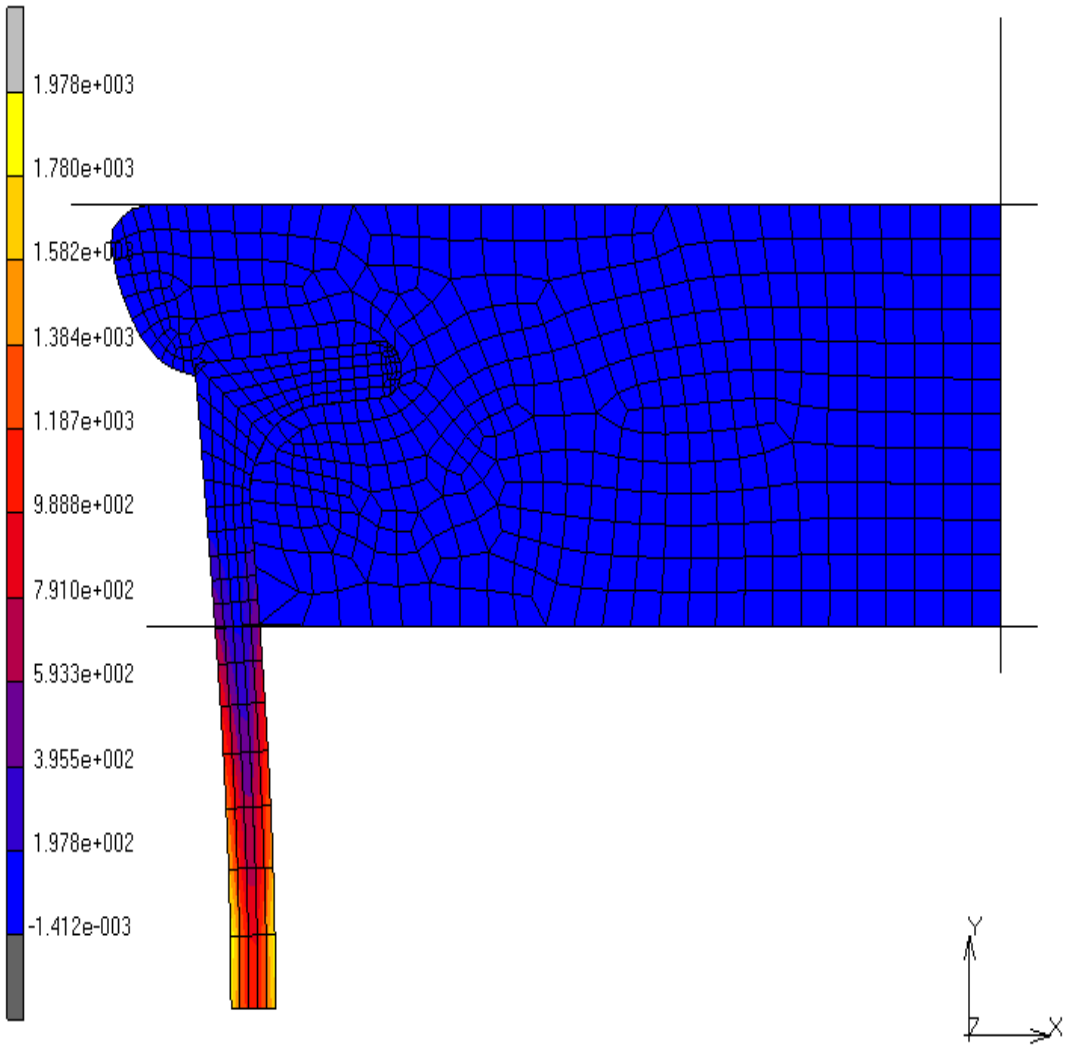


Figure 8.64-2 (C) von Mises Stress at Increment 30

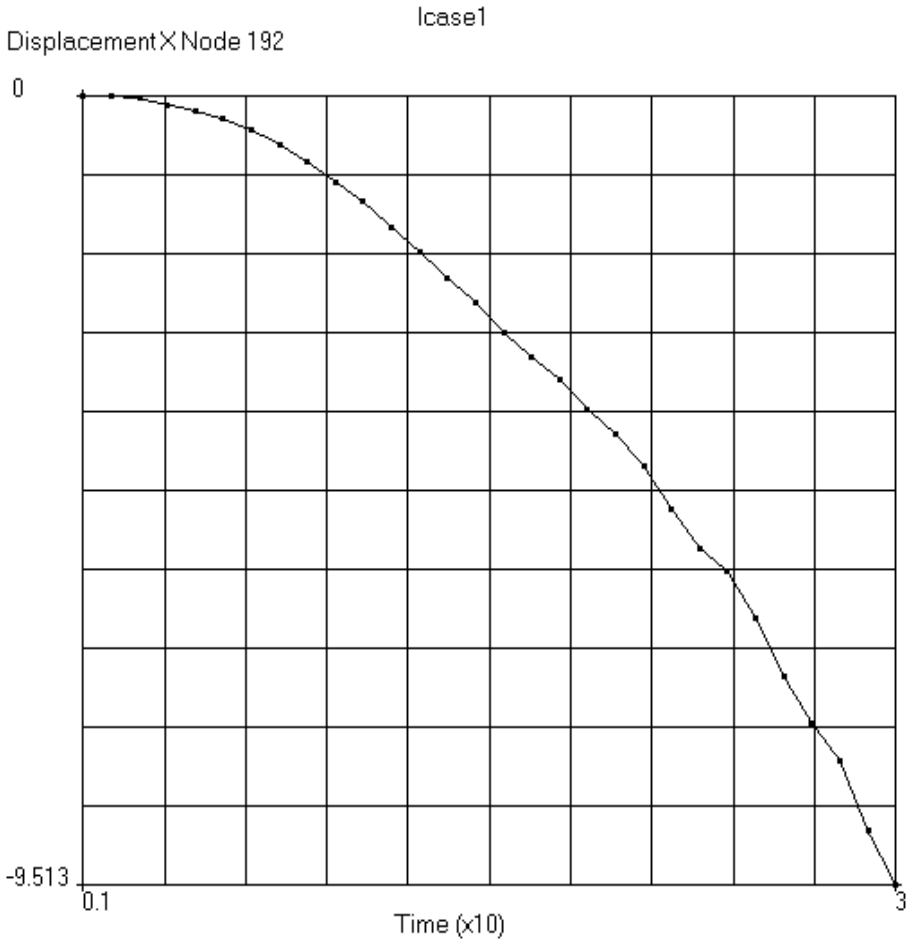


Figure 8.64-3 Displacement of the Steel at the Tip in the X Direction with Time

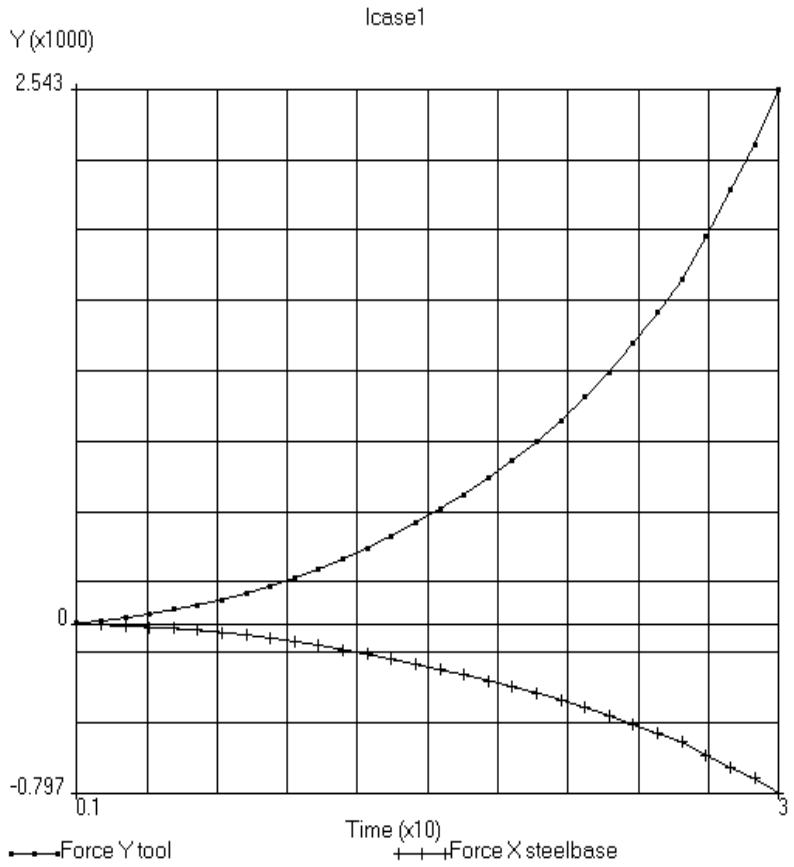


Figure 8.64-4 Load Curves of Tool and X Load on the Steel with Time

8.65 Pipe-nozzle Connection with a Rubber Seal

In this example, a pipe is moved on a nozzle and a rubber seal between the pipe and the nozzle is used to avoid leakage (see Figure 8.65-1). The CONTACT TABLE option is used to automatically determine the proper order in which the search for contact will be performed

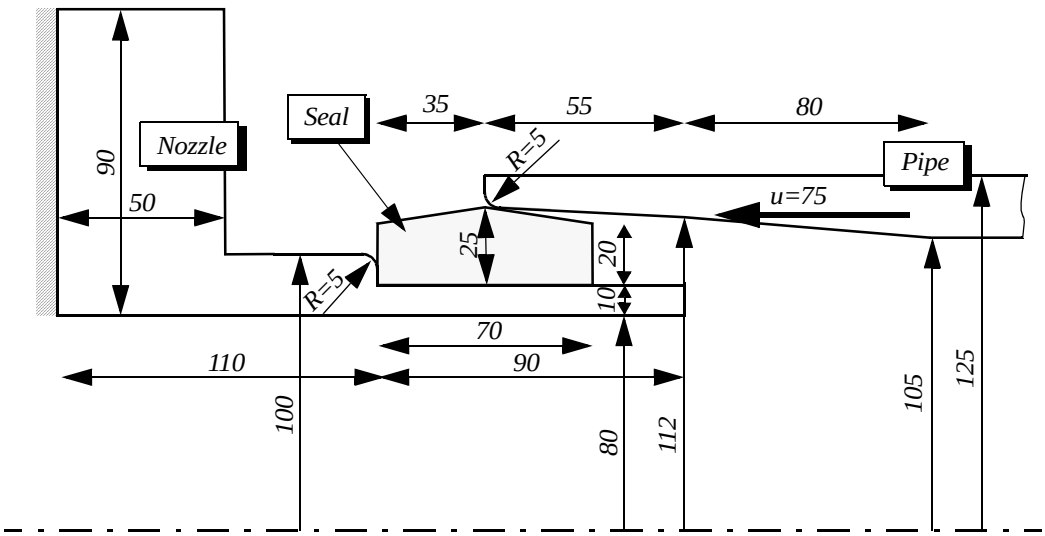


Figure 8.65-1 Pipe-nozzle connection with a rubber seal: problem description (units: mm)

The pipe is moved in the left-hand side direction over a distance $u = 75\ mm$. The nozzle is assumed to be fully clamped at the left edge.

Elements

Element type 10, a 4-node axisymmetric isoparametric element with full integration, is used to model both the steel nozzle and pipe as well as the rubber seal. The element mesh is shown in Figure 8.65-2. Notice that the nozzle and the pipe have been meshed in two separate regions, which will be glued together using the CONTACT TABLE option.

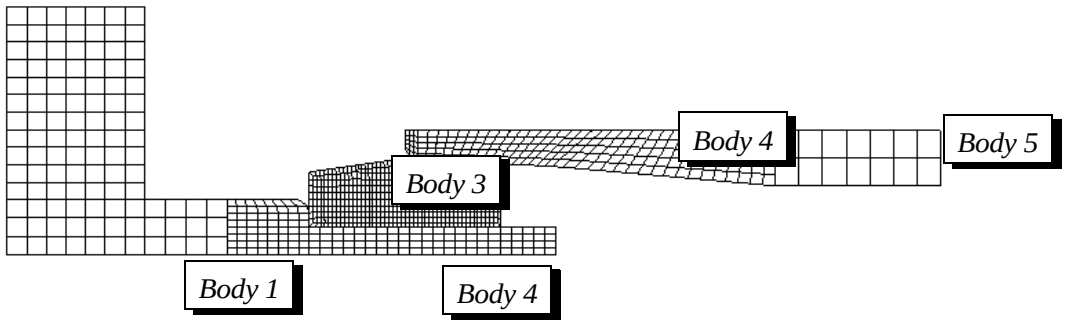


Figure 8.65-2 Finite element mesh and contact bodies

Elasticity

The ELASTICITY parameter option is used to indicate that the elasticity procedure used will be based on the Updated Lagrange approach.

Mooney

The material properties of the elements corresponding to the seal are given by Mooney constants $C_{01} = 0.8\ N/mm^2$ and $C_{10} = 0.2\ N/mm^2$.

Isotropic

Both the nozzle and pipe are modeled using an isotropic material with Young's modulus $E = 2.0 \times 10^5\ N/mm^2$ and Poisson's ratio $\upsilon = 0.3$

Fixed disp

The nodes on the left-hand side of the nozzle are fixed in both x- and y-direction, where the nodes in the right-hand side of the modeled part of the pipe will be moved in the left-hand side direction.

Contact

Five contact bodies are defined. The first two bodies build up the nozzle, the third body corresponds to the seal, the fourth and fifth body build up the pipe (see also Figure 8.65-2). Friction will be neglected.

Spline

The SPLINE option is used to get a smooth boundary description of the curved part of the nozzle. So it is only invoked for the second contact body. The nodes where the normal vector to the boundary of this body is discontinuous are given as input to the program.

Contact table

The CONTACT TABLE option is used here for various purposes. First, only a limited number of contact body pairs need to be considered; this reduces the computational time. Second, glued contact is activated between the two bodies defining the nozzle (bodies 1 and 2) and the two bodies defining the pipe (bodies 4 and 5). Third, the order in which the search for contact will be performed has to be determined by the program and will be based on the rule that for a particular contact body pair, nodes of the body with the smallest element edge length at the boundary will be checked with respect to the other body.

No print

The NO PRINT model definition option is used to suppress print out.

Post

The default nodal variables will be put on the post file; the element variable selected is the Cauchy stress tensor.

Control

The default control parameters are used. An incremental solution is accepted if the ratio of the maximum residual force component and the maximum reaction force component is less than 0.1. A full Newton-Raphson procedure is applied and the maximum number of iterations per increment is 10.

Auto step

Automatic load incrementation based on the AUTO STEP option is selected. The initial time increment is 0.025 times the total time. The desired number of recycles per increment is set to 3 and the load incrementation factor is set to 1.2.

Disp change

The nodes on the left-hand side of the nozzle are completely fixed, where the nodes on the right-hand side of the pipe are moved in the negative x-direction over a distance of 75.

Results

The equivalent Cauchy stress is the final deformed configuration is given in Figure 8.65-3. Figure 8.65-4 shows the relation between the increment number and the time. An increasing time step resulting from the AUTO STEP option can be observed.

Summary of Options Used

Parameter Options	Model Definition Options	History Definition Options
TITLE	SOLVER	TITLE
SIZING	OPTIMIZE	CONTROL
ELEMENTS	CONNECTIVITY	AUTO STEP
PROCESSOR	COORDINATES	DISP CHANGE
$NO LIST	MOONEY	CONTINUE
ELASTICITY	ISOTROPIC	
END	FIXED DISP	
	CONTACT	
	SPLINE	
	CONTACT TABLE	
	NO PRINT	
	POST	
	END OPTION	

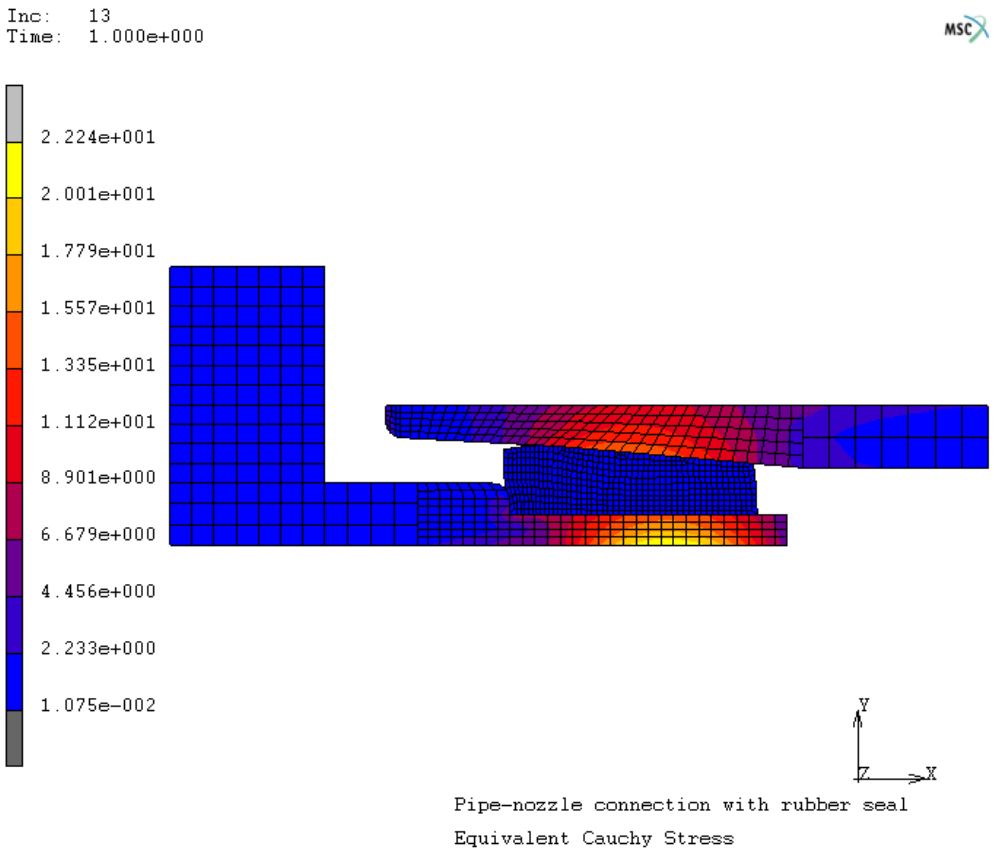


Figure 8.65-3 Equivalent Cauchy stress in final deformed configuration

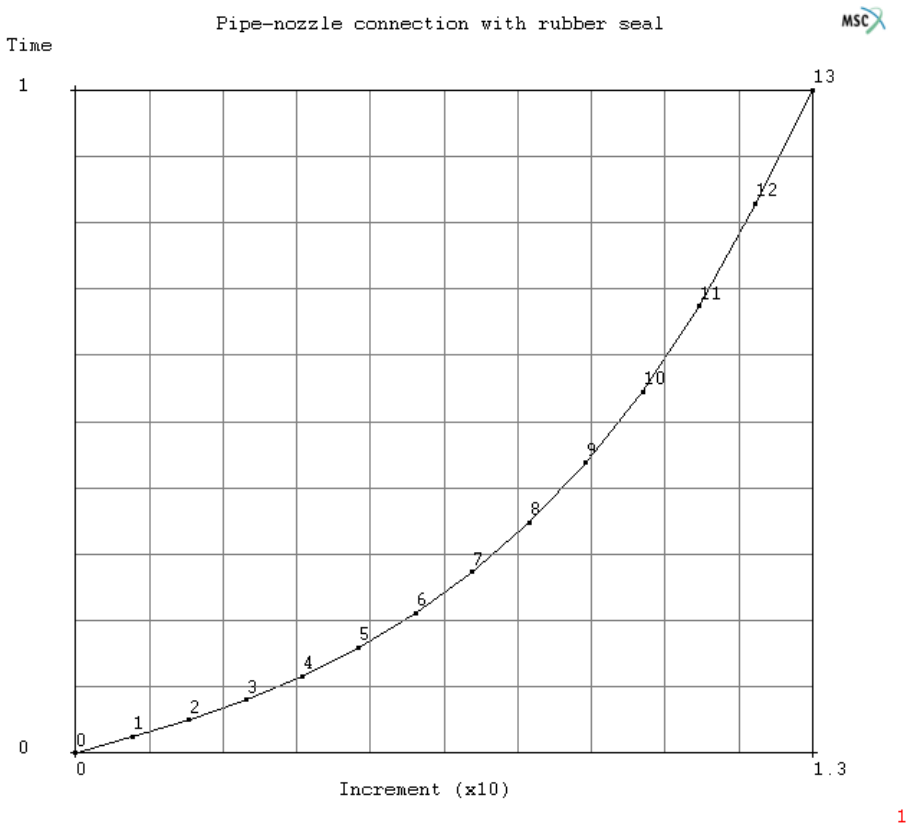


Figure 8.65-4 Time versus increment number

8.66 A Block Sliding over a Flat Surface

This example shows how a block, with an initial velocity of 4.905, slides on a rigid surface. This block sliding process requires dynamic analysis with considerations of contact, friction, heat transfer, dampings, and the conversion of friction energy into thermal energy. The energy changes with the sliding of the block are demonstrated.

Model

The block is modeled with 8 brick elements. In e8x66.dat, the block is pure metal. The sliding surface is modeled as rigid body. The initial model is shown in Figure 8.66-1.

Element

Element type 7 is used for the analysis.

Material Properties

In e8x66.dat, the block is assumed to be isotropic for both mechanical and thermal analysis. The Young's modulus is 2.1×10^{11} and the Poisson's ratio is 0.3. Mass density is given as 7854 for both dynamic and heat transfer analysis. The conductivity is 60.5 and the specific heat is set as 434. Only proportional mass damping is applied with a ratio of 0.3. Lumped mass matrix is used in the example. The conversion rate for friction work into thermal energy is given as 1.0.

Boundary Conditions

The boundary conditions along the x-direction are enforced by setting a y-displacement boundary condition of zero on all nodes. In order to keep the block sliding on the surface, a body force of -9.81 is applied to each element along the z-direction. The block is given an initial velocity of 4.905 along the x-direction and an initial temperature of 0 at the beginning of the sliding process.

Contact

There is a total of two contact bodies in the problem. Contact Body 1 is the deformable block. Body 2 is a velocity controlled rigid surface, but not moving in space. Coulomb friction is applied with a friction co-efficient of 0.5 based on nodal forces. The relative sliding velocity for friction below which a node is assumed to be sticking to a contact surface is set at 0.1. The nodal reaction force required to separate a contacting node from its contacted surface is assumed to be 1×10^{11}.

Control

The convergence control is governed by a relative displacement increment norm. The maximum allowed relative change in displacement is set to 0.10. For the heat transfer part, the maximum temperature change is set to be 20.

History Definition

The process is analyzed by coupled dynamics with SSH method (DYNAMIC,6). The total time of the process is 2 which takes 50 increments.

Results

At the end of increment 50, the equivalent von Mises stress and temperature are shown in Figures 8.66-2 and 8.66-3, respectively. Figure 8.66-4 shows the energy changes from increment 0 to increment 50. It shows that the kinetic energy is eventually transferred into damping energy and friction energy. The strain energy is very small. Figure 8.66-5 shows the conversion of friction work into thermal energy. It also demonstrates that the friction forces contribute, to a high degree, part of the work done by external force in this example.

Parameters, Options, and Subroutines Summary

Example e8x66.dat:

Parameter	Model Definition Option	History Definition Options
ALL POINTS	CONNECTIVITY	CONTINUE
COUPLED	CONTACT	CONTROL
DIST LOADS	CONTROL	DIST LOADS
DYNAMIC	CONVERT	DYNAMIC CHANGE
ELEMENTS	COORDINATES	MOTION CHANGE
END	DAMPING	PARAMETERS
EXTENDED	DIST LOADS	
LARGE DISP	END OPTION	
LUMP	FIXED DISP	
PROCESSOR	INITIAL TEMP	
SETNAME	INITIAL VEL	
SIZING	NO PRINT	
TITLE	OPTIMIZE	
	PARAMETERS	
	POST	
	SOLVER	

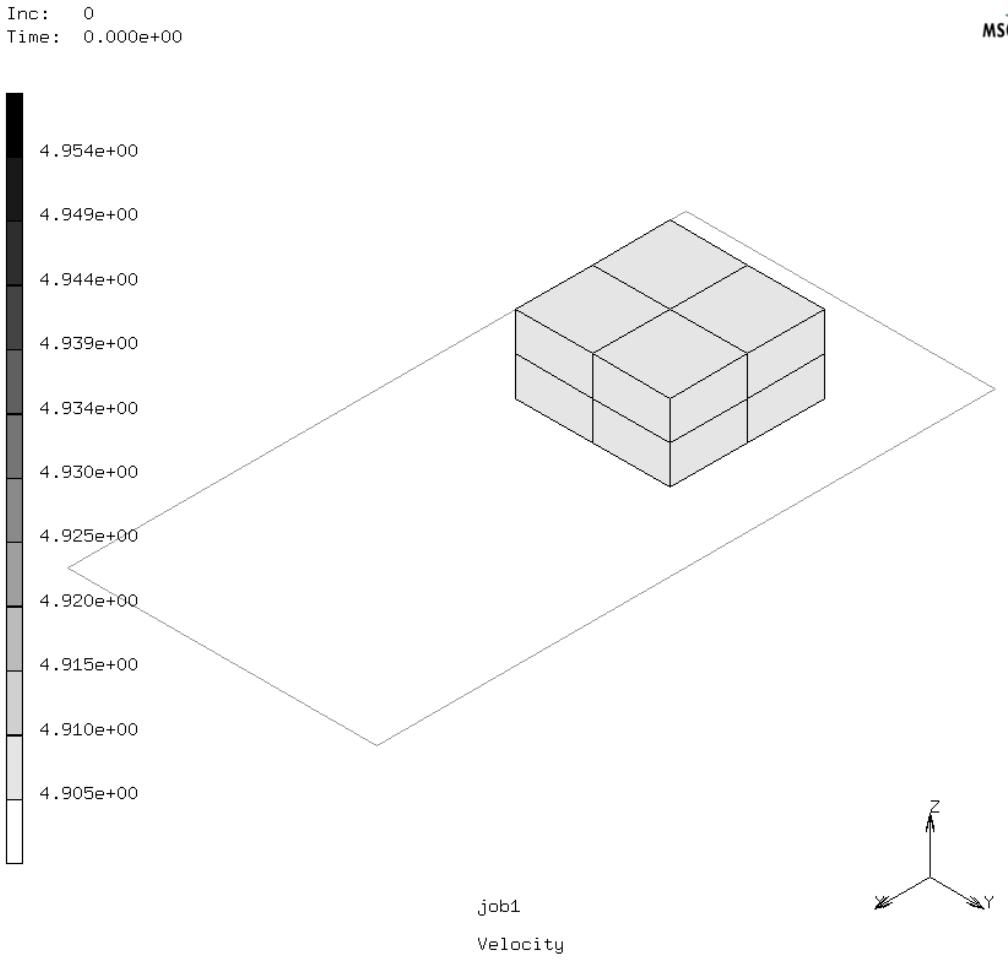


Figure 8.66-1 Initial Geometry and Velocity of the Sliding Block

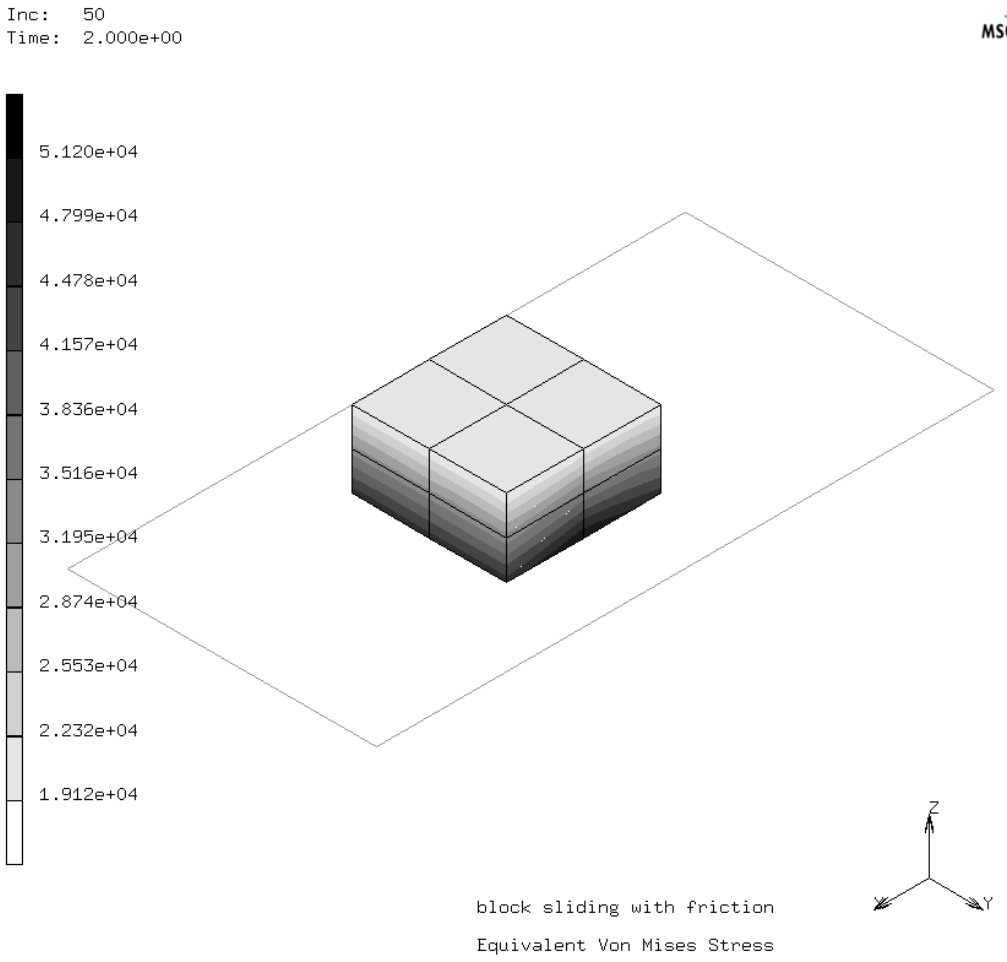


Figure 8.66-2 The Equivalent von Mises Stress and Increment 50

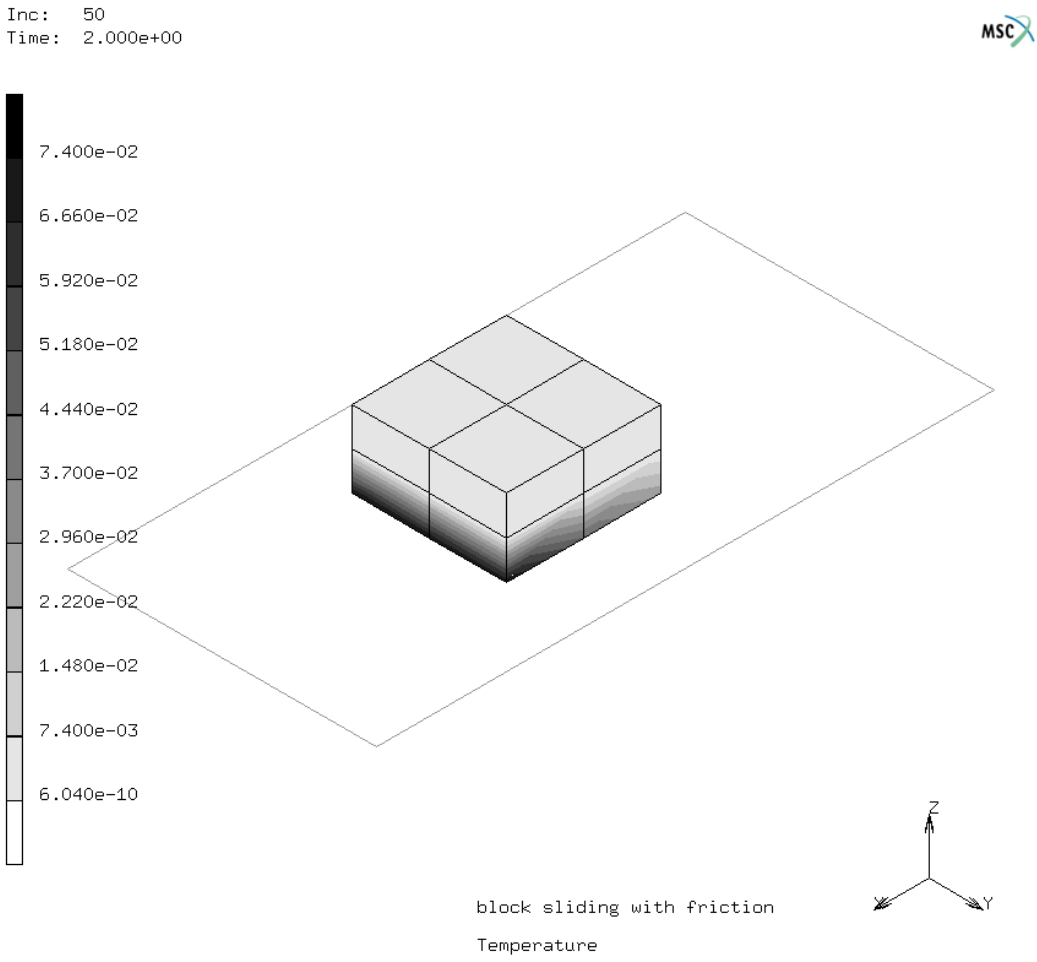


Figure 8.66-3 The Temperature Distribution in the Block at Increment 50

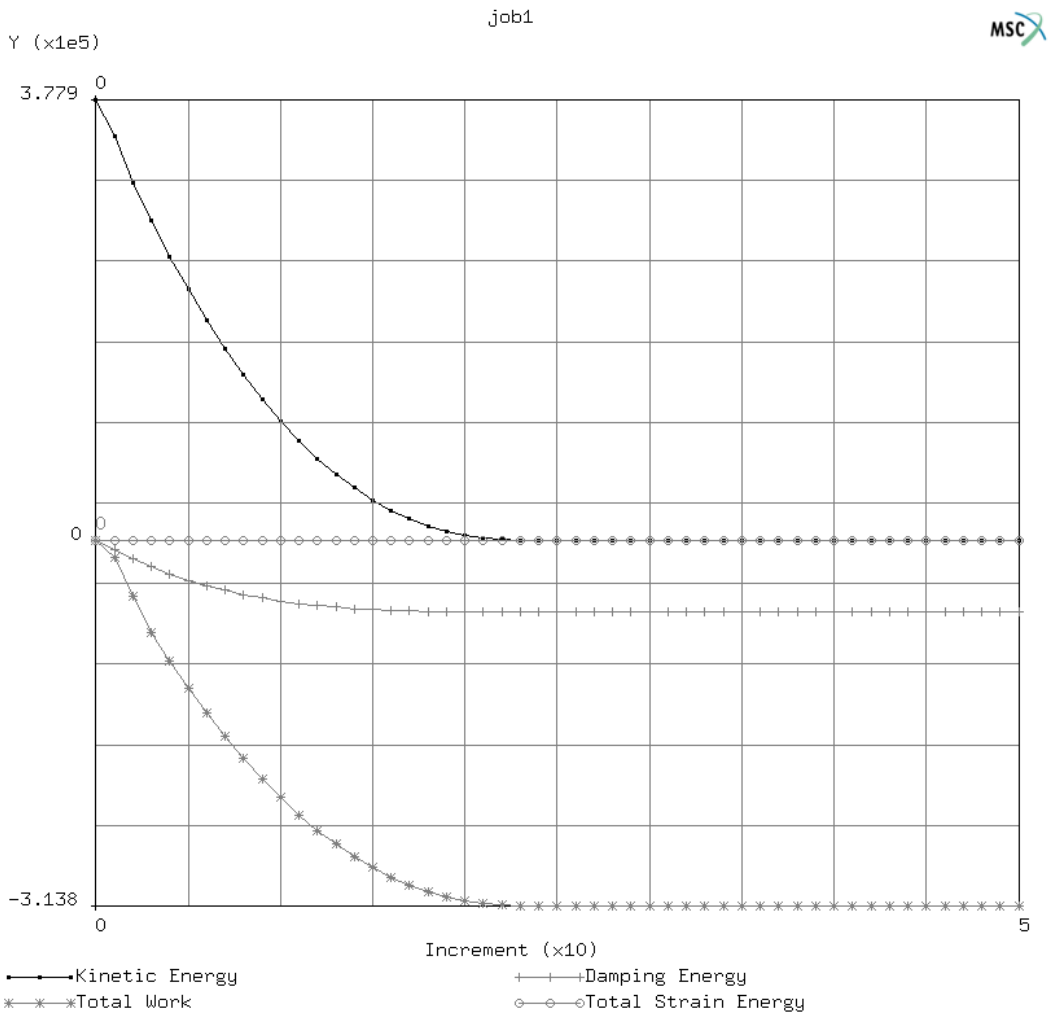


Figure 8.66-4 The Energy Changes from Increment 0 to Increment 50

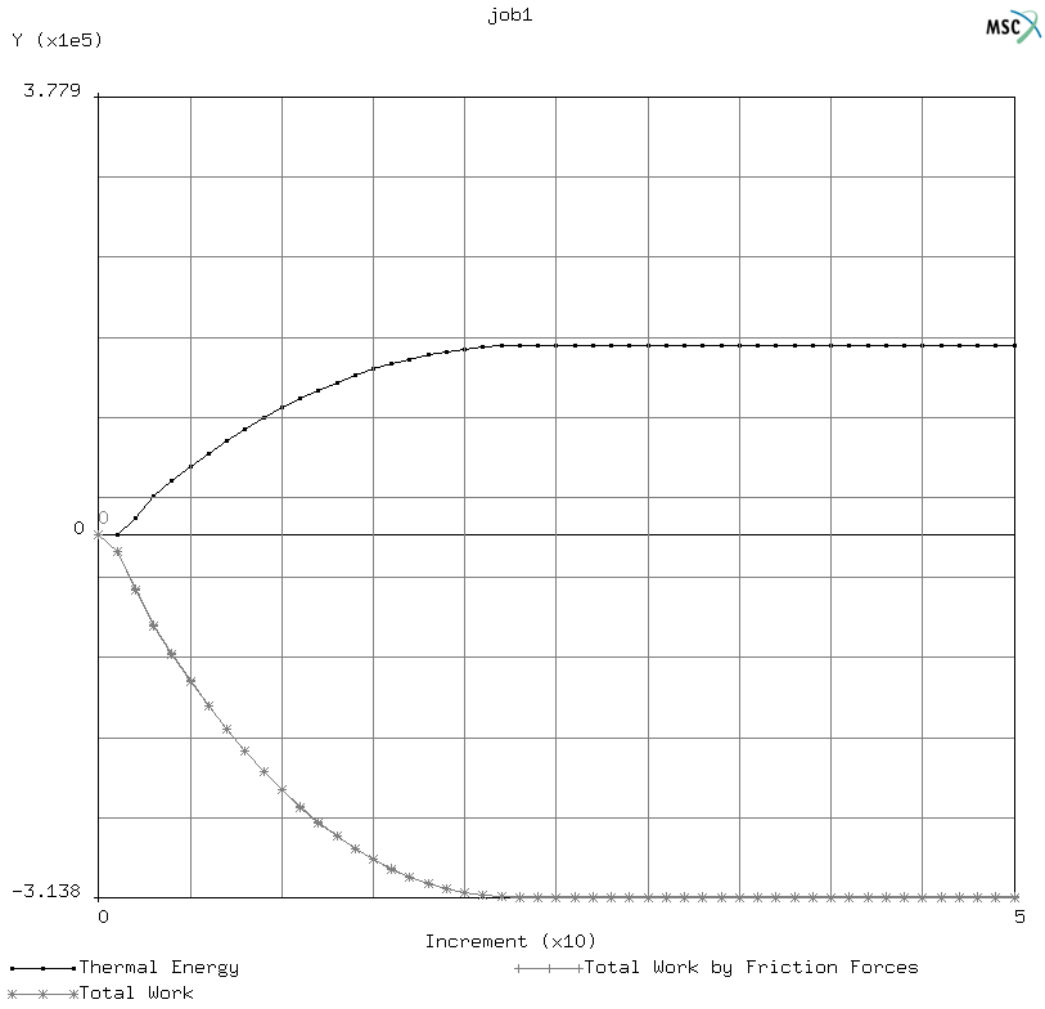


Figure 8.66-5 The Conversion of Friction Work into Thermal Energy

8.67 Analysis of an Automobile Tire

3-D finite element analysis of automobile tires is complicated because of the complex structure of tires which are made of several types of rubber, reinforced with cord layers and a steal core. This problem demonstrates the use of Marc's axisymmetric to 3-D data transfer capability for rebar elements. An automobile tire with a smooth tread, denoted as 195/65/R15, is analyzed. See [Helnwein et al., "A new 3D finite element model for cord-reinforced rubber composites - Application to analysis of automobile tires", *Finite Elements in Analysis and Design*, Vol. 14, 1-16(1993)] for detailed description of the model. The analysis includes the numerical simulation of three steps: (1) mounting the tire on the wheel, (2) inflating the tire up to 2.0 bar, and (3) pressing it against a road surface.

During the first two steps, the deformation is purely axisymmetric and, therefore, an axisymmetric analysis is performed. See e8x67a.dat. The simulation of tire mounting on the wheel is carried out using 10 equal increments. Afterwards, the inflation pressure is applied with 10 more equal increments. In the third step, the tire contacts with the road surface. The problem becomes fully three dimensional. See e8x67b.dat. A total vertical movement of 25 mm of the tire against the road surface is applied using the AUTO STEP option and the analysis is completed after 13 increments. In e8x67b.dat, the AXITO3D option is used to transfer data from axisymmetric mesh (e8x67a.dat) to 3-D mesh (e8x67b.dat).

Model

In axisymmetric analysis (e8x67a.dat), the tire is modeled with 180 4-node isoparametric quadrilateral axisymmetric elements. Among them, 120 elements (element type 10) are used to model the rubber materials and 60 rebar elements (element type 144) are used to model the reinforcing cords. The axisymmetric mesh is shown in Figure 8.67-1. There are 162 nodes in the mesh. The tire wheel is modeled using an analytical rigid curve.

The 3-D model (see Figure 8.67-2), containing 3120 rubber elements (element type 7) and 1560 rebar elements (element type 146), is obtained by expanding the axisymmetric model in Figure 8.67-1, using EXPAND =>AXISYMMETRIC MODEL TO 3D option in Mentat. There are totally 4680 8-node isoparametric hexahedral elements and 4212 nodes in the 3-D model. The curve representing rigid wheel is also expanded automatically with the mesh into a 3-D rigid surface. The road is modeled as a flat rigid surface.

Elements

The element types 10 and 144 are used in the axisymmetric run. The corresponding 3-D element types are 7 and 146. ELASTICITY,2 activates the updated Lagrangian formulation. However, internally rebar elements always use total Lagrangian framework.

Material Properties

The rubber is modeled using Mooney constitutive model. The material properties are:

Tread: $C_1 = 0.35$ N/mm^2 and $C_2 = 0.16$ N/mm^2;

Base: $C_1 = 0.58$ N/mm^2 and $C_2 = 0.26$ N/mm^2;

Other part: $C_1 = 0.45$ N/mm^2 and $C_2 = 0.36$ N/mm^2.

The bead core and reinforcing cords are modeled with isotropic materials. The material properties are:

Bead core and steel belts: Young's modulus 198700 N/mm^2 and Poisson's ratio 0.3;

Carcasses: Young's modulus 6800 N/mm^2 and Poisson's ratio 0.3;

Cap ply: Young's modulus 1950 N/mm^2 and Poisson's ratio 0.3.

Rebar layer properties are defined using REBAR option. See e8x67a.dat and e8x67b.dat for details.

Boundary Conditions and Load Definitions

In axisymmetric analysis (e8x67a.dat), there are three sets of boundary and load definitions. The mounting of the tire into the wheel is simulated using a set of point loads. The set of loads is then released after the mounting is completed. A set of symmetric condition is applied to remove the rigid body motion. The inflation pressure is applied to the inner surface of the tire within 10 equal increments.

In 3-D analysis (e8x67b.dat), the symmetric condition and the point loads are no longer needed and inflation pressure is unchanged. To simulate the tire-road contact, the flat, rigid surface moves up 25 mm against the tire. The AUTO STEP option is used.

Results

The deformed and the undeformed mesh after the tire inflation is in Figure 8.67-3. The final deformed shape of the 3-D tire model is shown in Figure 8.67-4. The contact pressure after 25 mm contact displacements is shown in Figure 8.67-5.

Parameters, Options, and Subroutines Summary

e8x67a.dat

Parameters	Model Definition Options	History Definition Options
ALL POINTS	CONNECTIVITY	AUTO LOAD
DIST LOADS	CONTACT	CONTINUE
ELASTICITY	COORDINATES	CONTROL
ELEMENTS	DEFINE	DIST LOADS
END	DIST LOADS	MOTION CHANGE
PROCESSOR	END OPTION	POINT LOAD
SETNAME	FIXED DISP	TIME STEP
SIZING	ISOTROPIC	TITLE
TITLE	MOONEY	
	NO PRINT	
	OPTIMIZE	
	POINT LOAD	
	POST	
	REBAR	
	SOLVER	

e8x67b.dat

Parameters	Model Definition Options	History Definition Options
ALL POINTS	AXITO3D	AUTO STEP
DIST LOADS	CONNECTIVITY	CONTINUE
ELASTICITY	CONTACT	CONTROL
ELEMENTS	COORDINATES	DIST LOADS
END	DEFINE	MOTION CHANGE
PROCESSOR	DIST LOADS	TITLE
SETNAME	END OPTION	
SIZING	ISOTROPIC	
TITLE	MOONEY	
	NO PRINT	
	OPTIMIZE	
	POST	
	REBAR	
	SOLVER	

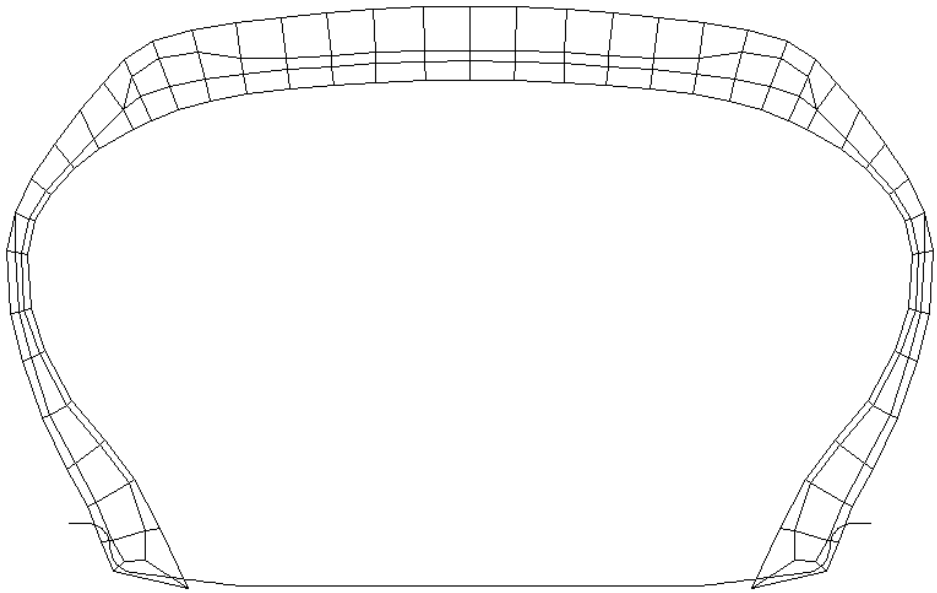


Figure 8.67-1 Finite Element Mesh of Axisymmetric Analysis

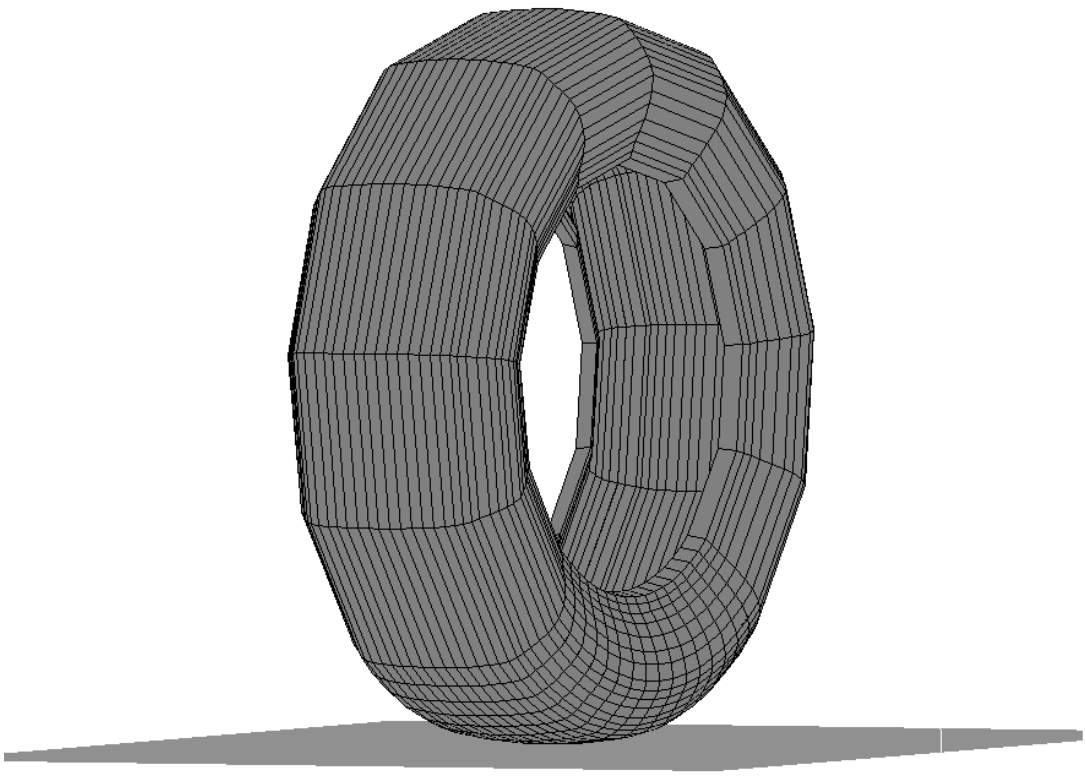


Figure 8.67-2 Finite Element Mesh of 3-D Analysis

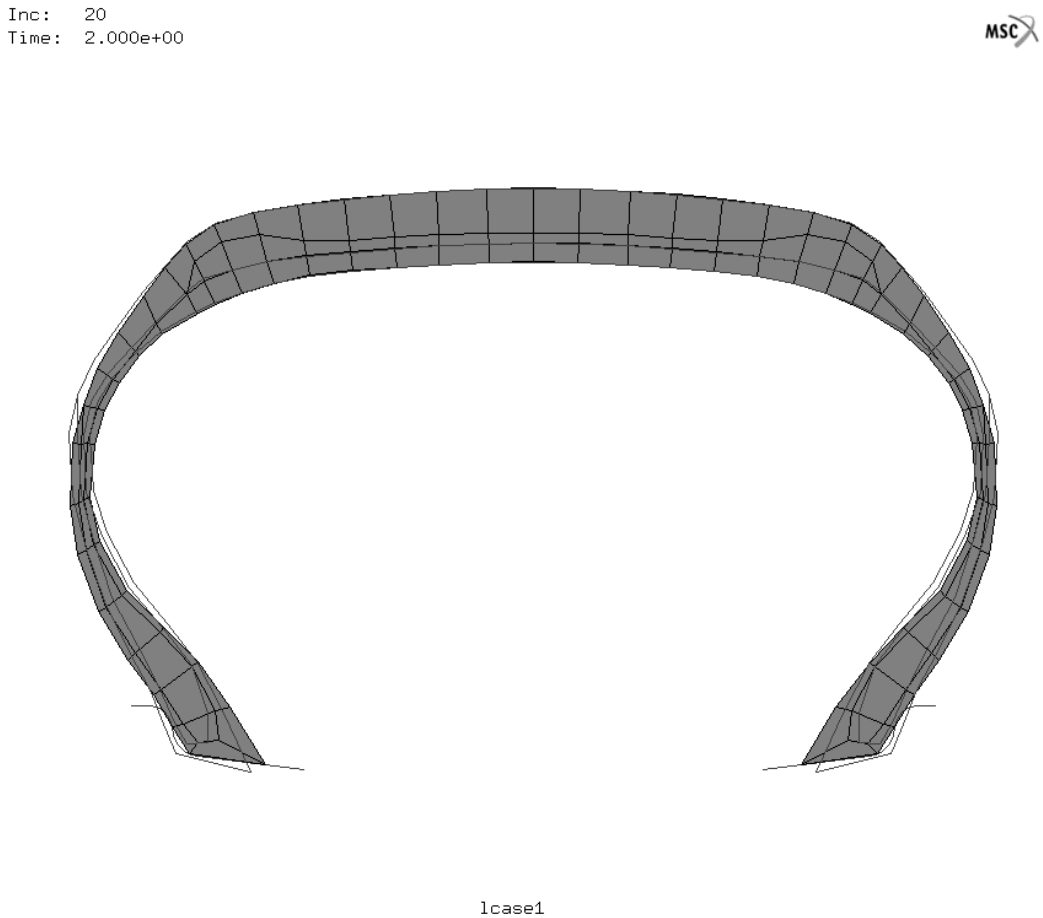


Figure 8.67-3 Deformed and Undeformed Mesh After Tire Inflation

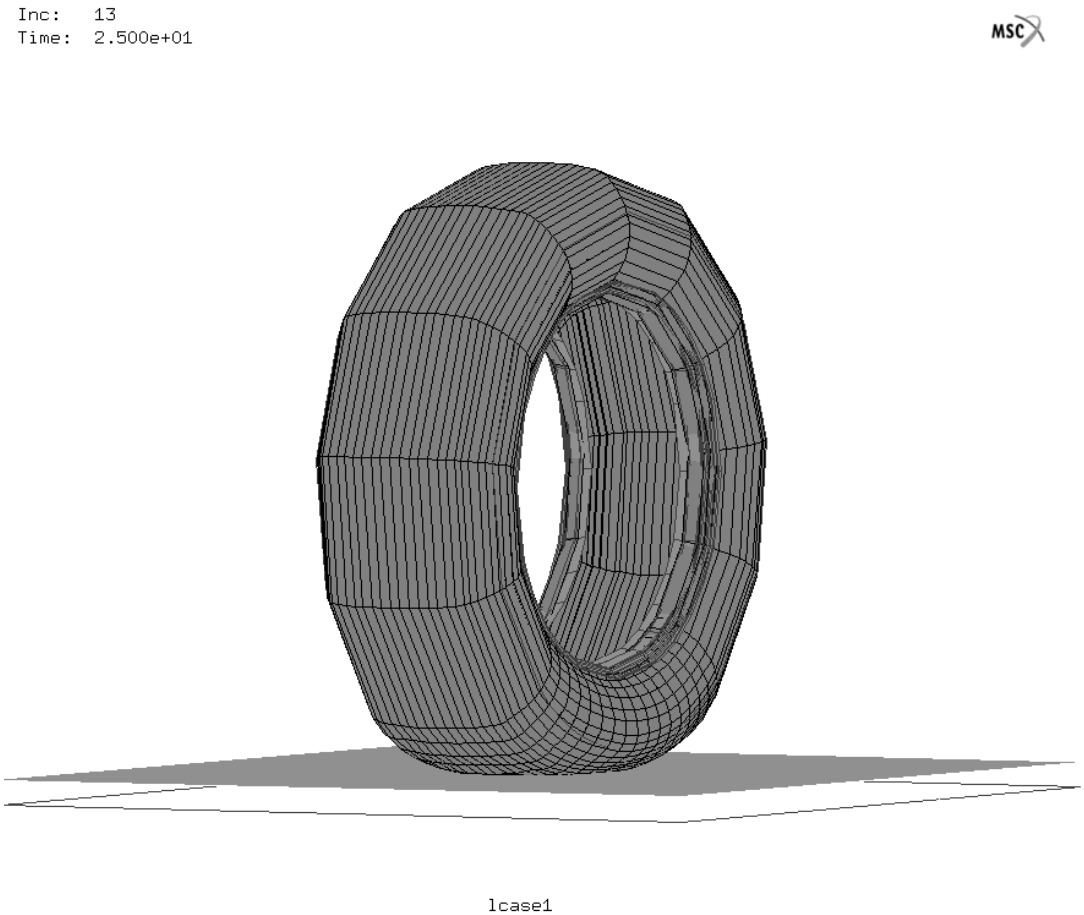


Figure 8.67-4 Final Deformed Shape of the 3-D Tire Model

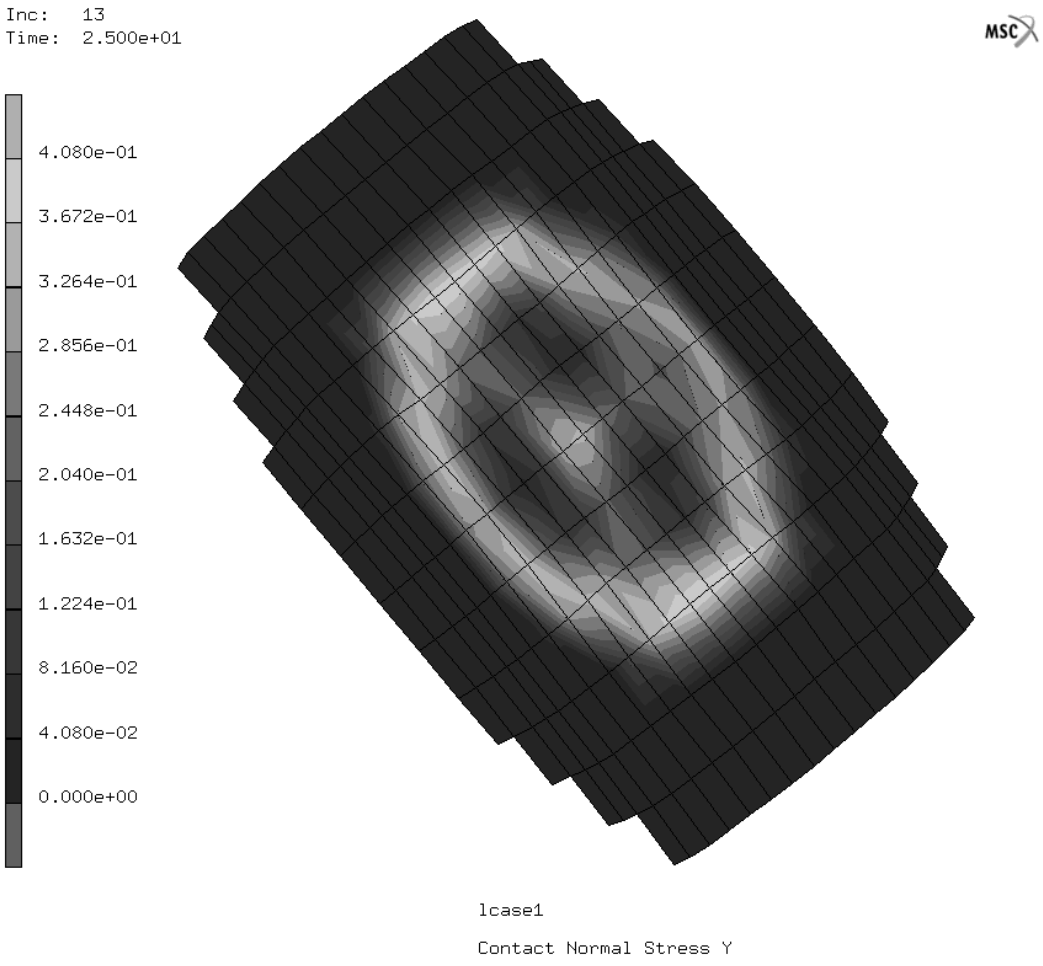


Figure 8.67-5 Contact Pressure at 25 mm Contact Displacement

8.68 Squeezing of two blocks

This problem demonstrates the use of the CONTACT TABLE option for the following purposes: stress-free projection at initial contact, delayed sliding off a contacted segment, and automatic detection of the proper order to search for contact between two deformable bodies.

The model used consists of two deformable bodies positioned between rigid plates, as shown in Figure 8.68-1. The upper rigid plate is moved downwards over a distance of 0.15. Between the deformable and the rigid bodies, glued contact is assumed. The coordinates of node 26 of the lower body have been adjusted to simulate a geometric imperfection in the model. As the node will be within the contact tolerance zone with respect to the upper body, the stress-free projection forces its coordinates to be changed such that the node is positioned exactly on the contacted segment without introducing stresses.

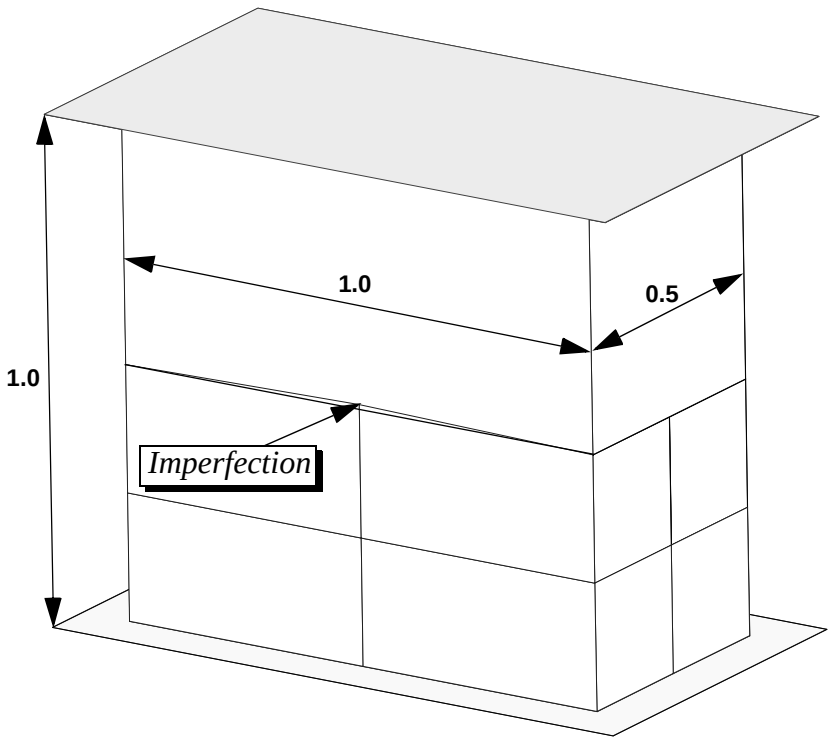


Figure 8.68-1 Squeezing of two blocks: finite element model

During the analysis, adaptive mesh refinement is applied. As a result, in increment 1, the mesh density of the upper body is equal to that of the lower body and from increment 2 onwards, the mesh density of the upper body is larger than that of the lower body. This is accounted for by automatically adapting the search order for contact.

The different material properties of the deformable bodies cause a difference between the lateral displacements of the nodes in the contact area between the deformable bodies. The delayed sliding off option forces nodes not to slide off a contacted segment at a sharp corner, but to remain in contact until it has moved over 10 times the contact tolerance beyond the edge of the contacted segment.

Elements

Element 7, an eight-node brick element with full integration, is used in this example. The initial finite element mesh contains nine elements.

Large disp

The LARGE DISP parameter option is used to perform a geometrically nonlinear analysis.

Adaptive

Due to adaptive mesh refinement there will be increase in the number of nodes and elements. With the ADAPTIVE parameter option the upper bound to the number of nodes and elements is set to 500.

Isotropic

The material properties are given by Young's modulus $E = 1\times10^4$ and Poisson's ratio $\nu = 0.3$ for the lower body and $E = 9\times10^3$ and $\nu = 0.34$ for the upper body.

These properties are entered via the ISOTROPIC model definition option.

Contact

Two deformable and two rigid bodies are defined. The lower deformable body consists of eight elements and the upper deformable body of one element. Each of the two rigid bodies consists of a flat surface.

Contact Table

The CONTACT TABLE model definition option is used to activate the following:

- Glued contact between the rigid and deformable bodies;
- Automatic determination of the optimal order to search for contact, based on the element edge length at the boundary of the contact bodies (a check on contact will be performed only for nodes corresponding to the body with the smallest element edge length);
- Stress-free projection at initial contact;
- Delayed sliding off a contacted segment at a sharp corner.

Adaptive

The “node in contact” criterion for adaptive mesh refinement is activated for element 1. The maximum number of refinement levels is set to 2.

No print

The NO PRINT model definition option is used to suppress print out.

Post

The default nodal variables are put on the post file. The stress tensor is selected as an element variable.

Control

The default control settings are used: convergence checking is done based on residual forces with a tolerance of 0.1.

Motion Change

The velocity of the upper rigid body in y-direction is set to -0.15.

Results

Figure 8.68-2 shows the model in increment 0. Clearly, due to the stress-free projection, the geometric imperfection has been removed by adjusting the coordinates of node 26. The contour band plot of the contact status shows that nodes of the lower deformable body are touching the upper deformable body. In Figure 8.68-3, the mesh density of the upper body has changed due to adaptive remeshing and nodes of the upper deformable body are now touching the lower deformable body. Finally,

Figure 8.68-4 shows the effect of the delayed sliding off option: nodes at the outer edge of the upper deformable body passed the edge of the contacted segment of the lower body, but still remain in contact.

Parameters, Options, and Subroutines Summary

Example e8x68.dat:

Parameters	Model Definition Options	History Definition Options
ADAPTIVE	CONNECTIVITY	AUTO LOAD
ELEMENTS	CONTACT	CONTINUE
END	CONTACT TABLE	CONTROL
EXTENDED	COORDINATES	MOTION CHANGE
LARGE DISP	DEFINE	TIME STEP
SETNAME	END OPTION	
SIZING	ISOTROPIC	
TITLE	NO PRINT	
	OPTIMIZE	
	POST	

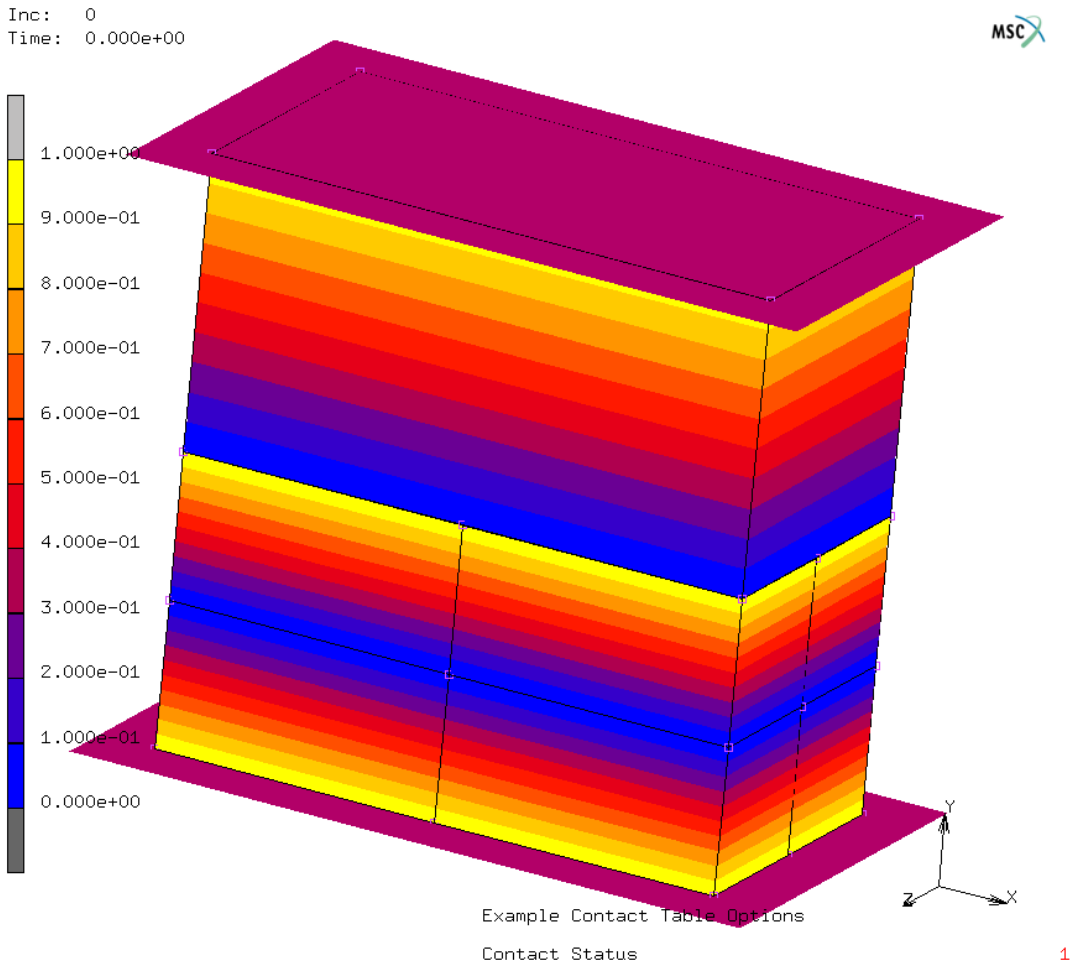


Figure 8.68-2 Increment 0: Node projected on contacted segment; contour band plot of contact status

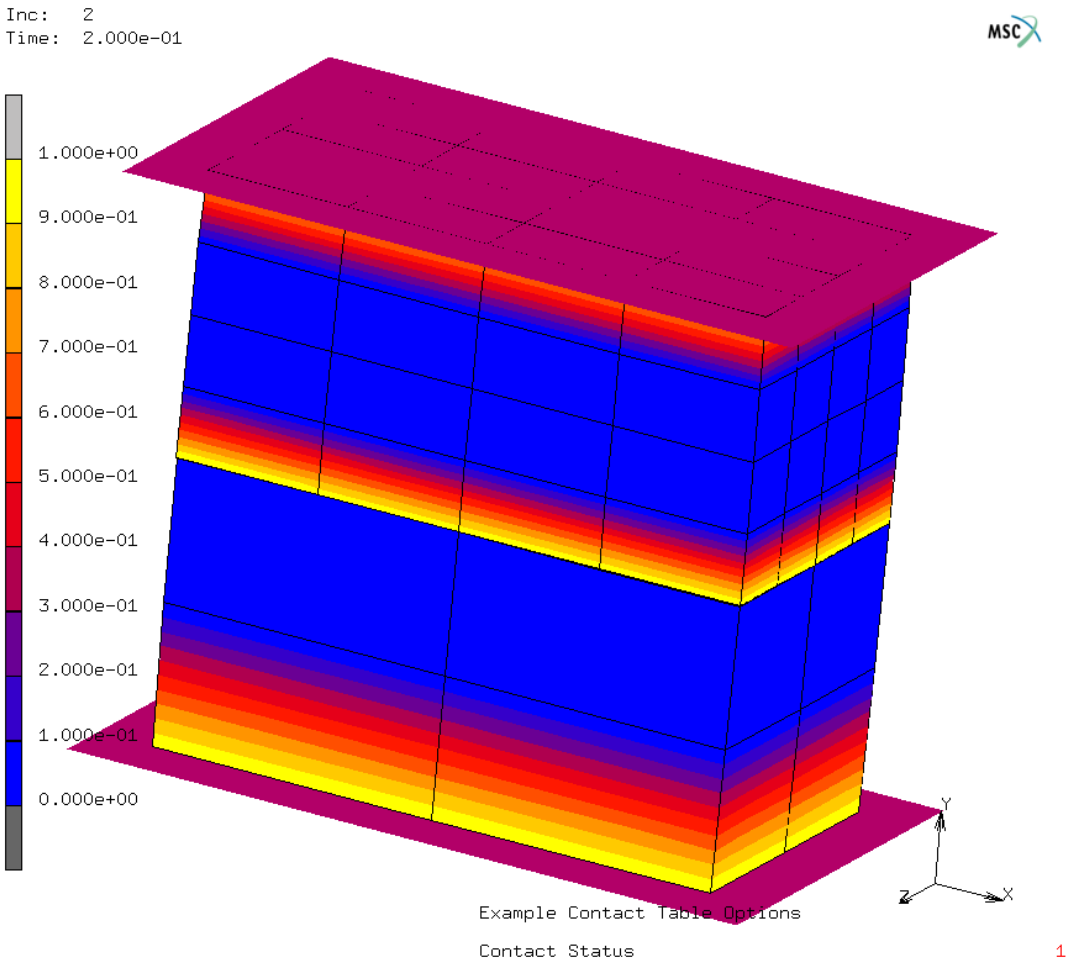


Figure 8.68-3 Increment 2: Change of contact status; nodes of the upper deformable body are touching the lower deformable body

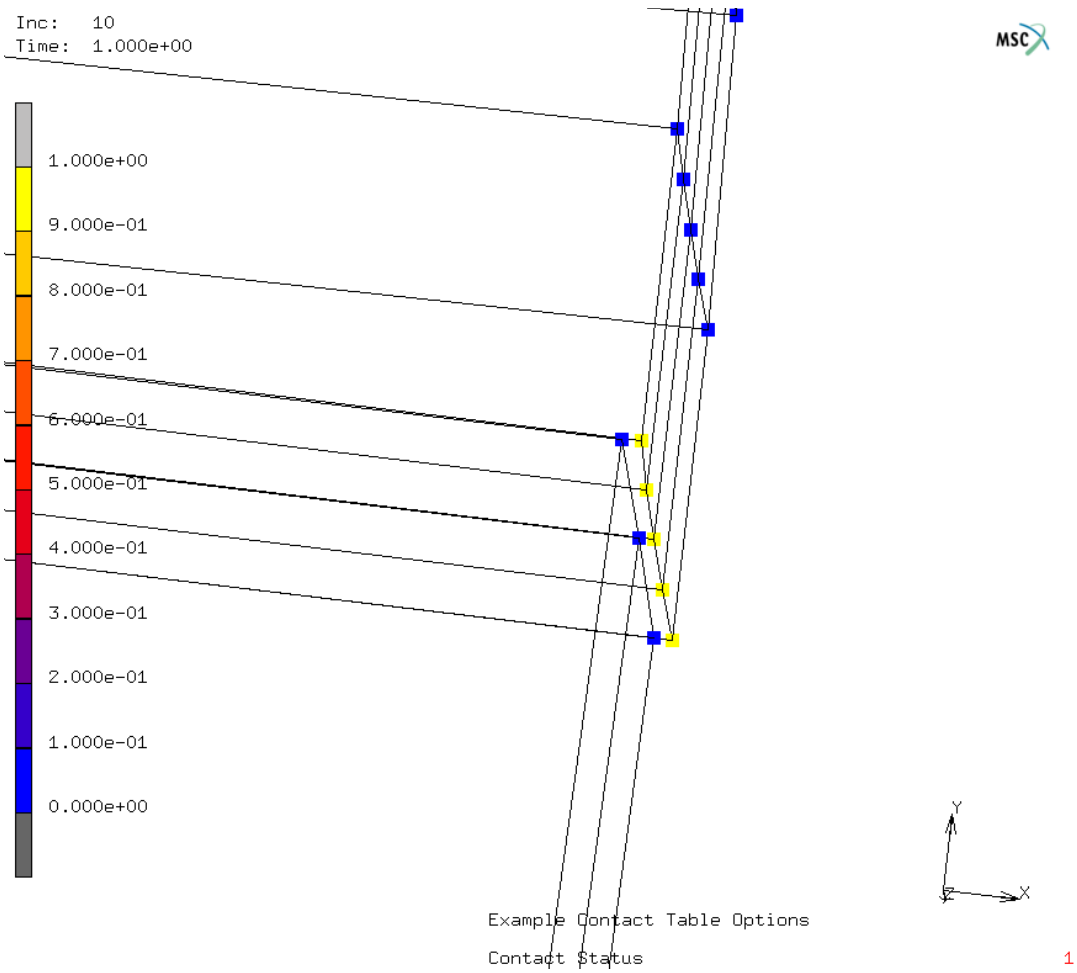


Figure 8.68-4 Increment 10: Effect of delayed sliding off; nodes have passed the edge of the contacted segment, but still remain in contact