MARC

Analysis
Research
Corporation

PRIMER

INTRODUCTORY
TEXTBOOK

Disclaimer of Warranties and Limitation of Liabilities

MARC Analysis Research Corporation:

North America

Corporate Headquarters
MARC Analysis Research Corporation
260 Sheridan Avenue
Palo Alto, CA 94306, USA
Telephone: (415) 329-6800
FAX: (415) 323-5892

National Sales Office
MARC Analysis Research Corporation
#6 Venture Suite 202
Irvine, Ca 92718, USA
Telephone: (714) 453-0840
FAX: (714) 453-0141

Europe

European Headquarters
MARC-Europe
Bredewater 26
2715 CA Zoetermeer, The Netherlands
Telephone: 31-79-510411
FAX: 31-79-517560

German Office
MARC Software Deutschland GmbH
Ismaninger Strasse 9
85609 Aschheim, Germany
Telephone: 49-89-904-5033
FAX: 49-89-903-0676

Italian Office
Espri-MARC
Piazza Rossetti 5/16A
16129 Genova, Italy
Telephone: 39-10-585949
FAX: 39-10-585949

Pacific Rim

Far East Headquarters
Nippon MARC Co., Ltd.
P.O. Box 5056
Shinjuku Daiichi Seimei Bldg.
2-7-1 Nishi-Shinjuku
Shinjuku-ku, Tokyo 163, Japan
Telephone: 81-3-3345-0181
FAX: 81-3-3345-1529

Osaka Office
Nippon MARC Co., Ltd.
Dai 2 Kimi Bldg., 4F
2-11 Toyotsu-cho
Suita-city, Osaka 564, Japan
Telephone: 81-6-385-1101
FAX: 81-6-385-4343

TABLE OF CONTENTS

Preface vii

Chapter 1 Introduction and Overview 1-1

Sample Problem and Output 1-25

Chapter 2 Linear Static and Dynamic Problems 2-1

Example 1 Tensile Stress in a Sheet with Hole 2-5

Example 2A Thick Cylinder Under Internal Pressure–
Plane Strain Solution 2-31

Example 2B Thick Cylinder Under Internal Pressure–
Axisymmetric Solution 2-47

Example 3A Modal Analysis of a Cantilevered Beam 2-61

Example 3B Linear Dynamic Analysis Using
Direct Integration 2-79

Example 3C Damped Modal Superposition Response
Subjected to Initial Conditions 2-95

Example 4 Stiffened Composite Roof Under
Uniform Pressure 2-113

**Chapter 3 Nonlinear Structural Problems –
Plasticity, Large Deformation,
and Post-Buckling Analysis 3-1**

Example 5 Cantilevered Beam Loaded by Tip Load 3-3

Example 6 Large Displacement and Plasticity Analysis of a
Simply-Supported Square Plate 3-39

Example 7 Postbuckling Analysis of a Spherical Cap Under
Apex Load 3-71

**Chapter 4 Heat Transfer and
Thermal Stress Analyses 4-1**

Example 8 Transient Heat Conduction Analysis of a
Square Pipe with Circular Channel 4-3

Example 9 Thermal Stress Analysis of Square Pipe
with Circular Channel 4-25

TABLE OF CONTENTS *continued*

Chapter 5 Contact and Rubber Analyses 5-1

Example 10 Coupled Analysis of Ring Compression 5-3

Example 11 Side Pressing of a Solid Rubber Cylinder 5-49

Keyword Index A-1

Index IX-1

LIST OF FIGURES

Figure 0.1 Truss Under Tension 1-25
Figure 1.1 Square Plate with a Circular Hole 2-5
Figure 1.2 Second Component of Stress 2-29
Figure 2A.1 Thick Cylinder, Plane Strain Model 2-32
Figure 2A.2 Radial Displacement 2-46
Figure 2A.3 Radial Stress 2-46
Figure 2B.1 Thick Cylinder, Axisymmetric Model 2-47
Figure 2B.2 Radial Stress Through Radius 2-59
Figure 2B.3 Hoop Stress Through Radius 2-59
Figure 2B.4 Radial Displacement Through Radius 2-60
Figure 3A.1 Three-Dimensional Cantilevered Beam Model 2-61
Figure 3A.2 Eigenmodes 2-77
Figure 3B.1a Cantilevered Beam 2-79
Figure 3B.1b Time History of Load 2-79
Figure 3B.2 Deflected Shape 2-94
Figure 3B.3 Time History of Tip Deflection 2-94
Figure 3C.1 Cantilevered Beam with Initial Displacements 2-95
Figure 3C.2 Time History of Tip Deflection 2-110
Figure 4.1a Beam Cross Section 2-113
Figure 4.1b Beam and Material Orientation 2-113
Figure 4.2a Node Numbers 2-114
Figure 4.2b Element Numbers 2-114
Figure 4.3 Deformed geometry 2-136
Figure 4.4 Z-displacement contours 2-136
Figure 4.5 First component of stress in the preferred direction - Layer 1 2-137
Figure 5.1 Cantilevered Beam 3-3
Figure 6.1 Square Plate 3-39
Figure 6.2 Time History of Deformation 3-67
Figure 6.3 Contour Plot of Z-Displacement 3-67
Figure 6.4 Equivalent von Mises Stress - Layer I 3-68

LIST OF FIGURES *continued*

Figure 6.5 Equivalent Plastic Strain - Layer I 3-68
Figure 6.6 Equivalent von Mises Stress - Layer II 3-69
Figure 6.7 Equivalent Plastic Strain - Layer II 3-69
Figure 7.1 Spherical Cap 3-71
Figure 7.2 Cap Deformed Geometry From Increment 6 to 24 3-119
Figure 7.3 Node 5 Z-load History 3-120
Figure 7.4 Z-displacement History – Nodes 1, 3, 5, 7, 9, 11, 13 . . . 3-120
Figure 7.5 Load-deflection Curve for Spherical Cap 3-121
Figure 8.1 Square Plate with Circular Channel 4-3
Figure 8.2 Contour Plot of Temperatures 4-23
Figure 8.3 Time History of Nodal Temperatures 4-24
Figure 9.1 Square Plate with Circular Hole 4-25
Figure 9.2 Equivalent von Mises Stress 4-50
Figure 10.1 Ring Model 5-3
Figure 10.2 Deformed Geometry After 50 Increments 5-43
Figure 10.3 Deformed Geometry after 100 Increments 5-44
Figure 10.4 Equivalent Plastic Strains in Ring 5-45
Figure 10.5 Equivalent von Mises Stress Distribution in Ring 5-46
Figure 10.6a Temperature Distribution in Aluminum Ring 5-48
Figure 10.6b Temperature Distribution in Steel Disk 5-48
Figure 11.1 Rubber Cylinder and Contact Surface 5-49
Figure 11.2 Deformed Geometry 5-74
Figure 11.3 Equivalent Cauchy Stress Distribution 5-75
Figure 11.4 Strain Energy Density Distribution 5-76

Preface

This Primer is a tutorial for the first-time or occasional MARC user. MARC is a powerful, modern, general-purpose nonlinear finite element program for structural, thermal, and other types of engineering analysis; this Primer covers only some typical linear and nonlinear mechanical applications and does not describe all the capabilities in MARC. You are presumed to have had some exposure to linear finite element (FE) analysis, either through course-work or by having used other FE software. The Primer is written with the assumption that you have had little or no experience in nonlinear FE analysis.

In a typical FE analysis, you'll need to define the:

- mesh (which is an *approximate* model of the actual structure);
- material properties (Young's modulus, Poisson's ratio, etc.);
- applied loads (static, dynamic temperature, inertial, etc.);
- boundary conditions (geometric and kinematic constraints); and
- type of analysis (linear static, nonlinear, buckling, thermal, etc.).

These steps leading up to the actual FE analysis are generally termed *pre-processing*; currently, many users accomplish these steps by using an interactive color graphics pre- and post-processing program such as Mentat II. After an analysis, the results evaluation phase is called *post-processing*, where you check the adequacy of the design (and of the approximate FE model) in terms of critical stresses, deflection, temperatures, and so forth.

This Primer is written assuming you have Mentat II available for pre- and post-processing. Therefore, there will be no mention of MARC's built-in mesh generation and plotting capabilities, and little mention of such items as meshing considerations, load application details, and material properties. *The primary emphasis of the Primer is to explain MARC input and output for stress analysis problems.* (Example 8, however, introduces heat transfer analysis using MARC in preparation for the subsequent thermal stress analysis in Example 9). The eleven selected examples illustrate some simple but representative linear and nonlinear problems which can be solved using MARC. After you have read *Chapter 1: Introduction and Overview*, you may skip to any example you are interested in; each example is self-contained.

Once you have understood the modeling philosophy, input, output, and results discussion for any example, you are encouraged to experiment and re-analyze that example by changing input parameters – such as material properties, boundary conditions, or amount of damping (if it is a dynamic problem.) Then, you should examine the new MARC analysis results – and ask yourself whether the answers make sense! In this manner, you will develop a keen, critical view of the important role of numerical analysis in the design/analysis process. Building on this experience, you will gradually develop sound engineering judgement to enable you to properly evaluate the structural (or thermal) integrity of any product or design.

The version of MARC used is K5. The Mentat II version used for pre- and post-processing is Version 1.

CHAPTER 1: Introduction and Overview

This chapter provides a brief introduction to the MARC system. It serves as cursory background material for the eleven examples; more detailed tables and descriptions are found in the MARC User Manuals, Volumes A-E. You should read this chapter and be familiar with its contents before going on to the examples. Each example is self-contained and illustrates certain MARC features and input requirements. The first four examples are linear static/dynamic problems. Example 4 demonstrates the composites analysis capability in MARC. The remaining seven examples illustrate nonlinear analyses: plasticity, large deformations, buckling, heat transfer, thermal stress, contact/friction, and rubber materials.

The following topics are covered in this chapter: a guide to MARC and Mentat II documentation; MARC program features; element library (including a brief description of the 18 recommended elements); input description; output description; and a simple example of a rod loaded by an axial tensile load.

A Guide to MARC and Mentat II Documentation

In addition to this Primer, several other MARC manuals are available. These are referential in nature, and describe the features and applications of the MARC program in greater detail. These other manuals are:

MARC User Manuals

Volume A	User Information Manual (technical basis of program and capabilities)
Volume B	MARC Element Library
Volume C	Program Input
Volume D	User Subroutines and Special Routines

MARC Demonstration Manual (Volume E)

Volume E.1

Volume E.2

Volume E.3

Volume E.4

MARC Background Papers (Volume F)

Theoretical papers on MARC procedures

For reference purposes, Volumes B, C, and D are used most often. Volume A serves as an overview of the program capabilities, and contains some theoretical background material. **Volume E** demonstrates a variety of problems to illustrate MARC capabilities and correlation with theory or other published numerical solutions. After you have finished reading this Primer, you should refer to Volume E for additional demonstration problems. **Volume F** is a collection of journal papers which describe the algorithms and material characterizations in MARC.

Mentat II User's Guide

Tutorial sessions on Mentat II pre/postprocessing.

Program Features

MARC is a general-purpose finite element program designed for both linear and nonlinear analyses of structural, thermal, electric, and magnetic field problems. In addition, it can handle coupled thermal-mechanical, electro-thermal, and electromagnetic analyses. In nonlinear and transient problems, MARC makes your analysis easier by offering automatic load incrementation and time-stepping capabilities. You will be exposed to some of these concepts in this Primer.

Many types of analyses can be obtained by any combination of these basic MARC capabilities. The following is a cursory listing of MARC capabilities. Please refer to the appropriate MARC manual for more detailed descriptions.

Geometry

- 1-D: truss, beams (open or closed section)
- 2-D: plane stress, plane strain, generalized plane strain
- 2-D (axisymmetric): solid or shell (with nonaxisymmetric loading for linear problems)
- 3-D: solids, plates, shells, membranes

Behavior

- linear/nonlinear for geometry or material
- static/dynamic
- steady-state/transient

Material

- linear elastic
- isotropic/orthotropic/anisotropic composites
- elastic-plastic, work-hardening

- isotropic, kinematic, and combined hardening
- finite strain
- cyclic loading
- viscoplasticity
- rigid plastic flow
- nonlinear elastic, elastomers, rubber
- viscoelastic (Maxwell, Kelvin, combined)
- powder metallurgy
- metal and elastomer damage models

Boundary Conditions

Boundary conditions vary with:

- time/increment
- temperature
- displacements, velocities, accelerations
- open/close contact

Libraries

MARC has four comprehensive libraries:

- Procedure Library
- Element Library
- Material Library
- Function Library

You may combine almost any number of options from each of the four libraries, and, consequently, solve virtually any structural mechanics or thermal problem.

Procedure Library

This includes all of the analysis types available in MARC:

• **Linear elastic**	- standard linear finite element analysis - superposition of multiple load cases - Fourier (nonaxisymmetric) analysis of linear axisymmetric bodies
• **Substructuring**	- multilevel, quasi-static
• **Nonlinear**	- automatic load incrementation - elastoplastic scaling to first yield - large deformation/finite strain

  - total and updated Lagrangian approaches
  - buckling/collapse–linear/nonlinear
  - creep buckling
  - postbuckling–with adaptive load step
- rigid plastic flow–Eulerian, metal forming
- creep–with adaptive load step
- viscoelastic
  - state equations (Kelvin model)
  - hereditary integrals (generalized Maxwell or generalized Kelvin-Voigt model)
  - thermo-rheologically simple behavior
- viscoplastic–modified creep option to include plasticity effects
- contact/friction–automatic convergence

- **Fracture mechanics**
  - linear/nonlinear
  - brittle/ductile
  - J-integral evaluation
  - dynamic J-integral
  - brittle cracking concrete model
- **Dynamics**
  - modal analysis/eigenvalue extraction
    - inverse power sweep method
    - Lanczos method
  - transient response
    - modal superposition
    - direct integration:
    - Newmark-beta method
    - Houbolt method
    - central difference method
  - harmonic response
  - spectrum response
  - time-stepping–linear/nonlinear
  - adaptive time-stepping algorithm
- **Heat transfer**
  - steady-state and transient analyses
    - conduction–linear/nonlinear
    - convection/radiation boundary conditions
    - internal heat generation
    - latent heat phase changes
    - adaptive time steps
- **Hydrodynamic bearings**
  - lubrication problems
  - pressure distribution and mass flow
- **Joule heating**
  - coupled electric flow with heat transfer

- **Electromagnetics**
  - electrostatics
  - magnetostatics
  - coupled electromagnetic analysis
    - harmonics
    - transient
- **Fluid/structure interaction**
  - incompressible and inviscid fluid
- **Thermo-mechanical**
  - quasi-coupled thermally driven stress analysis
  - fully coupled thermo-mechanical analysis solved by staggered scheme
  - large displacement effects on thermal boundary conditions
  - automated contact/friction capability
- **Change of state**
  - transient thermal analysis with change of phase and volume
  - associated stress analysis with plasticity and residual stresses

Element Library

MARC has a library of approximately 130 elements. Only the most important subset (the "recommended elements") will be discussed in this Primer. A detailed description of the MARC Element Library is given later in this chapter.

Material Library

This includes more than 40 different material models:

- **Linear elastic**
  - isotropic, orthotropic, and anisotropic (properties may be temperature dependent)
- **Composites**
  - laminated plates and shells
  - isotropic, orthotropic, or anisotropic layers
  - elastic or elastic-plastic behavior
  - arbitrary material orientation definition
    - with respect to any element edge
    - with respect to global Cartesian axes
    - with respect to a user-defined axis or through user subroutines
  - relative ply angle for each layer
  - multiple failure criteria
    - maximum stress
    - maximum strain
    - Tsai-Wu
    - Hill
    - Hoffman, or
    - user-defined
- **Hypoelastic**
  - nonlinear elastic (reversible)

• **Elastomers**	- nonlinear elastic, incompressible - Mooney-Rivlin model (allows large strains) - Ogden model
• **Elastic-plastic**	- Prandtl-Reuss flow rule - user-defined non-associative flow law - von Mises yield criterion - Drucker-Prager yield criterion - isotropic, kinematic or combined hardening - strain hardening (or softening) as a function of strain rate and temperature - temperature dependence of yield stress and work hardening slopes - isotropic, orthotropic, and anisotropic - Hill's anisotropic model - Gurson damage model
• **Cyclic plasticity**	- isotropic, kinematic, combined hardening
• **Creep**	- deviatoric or volumetric (swelling) strains - piecewise linear or exponential forms for rate of equivalent creep strain - temperature dependence - Oak Ridge National Lab. (ORNL) model–combined creep, plasticity, and cyclic loadings
• **Viscoelasticity**	- Maxwell and Kelvin models - combined Kelvin-Voigt and Maxwell models - hereditary integrals of strain histories with both small and large strain formulations - thermo-rheologically simple behavior - isotropic or anisotropic material
• **Polymers**	- thermo-rheologically simple behavior - damage model
• **Viscoplasticity**	- combining plasticity and the Maxwell model of plasticity - general inelastic behavior - unified creep plasticity
• **Soils**	- yield surfaces as a function of hydrostatic stress - linear or parabolic Mohr-Coulomb law
• **Concrete**	- low-tension cracking - crushing surfaces - rebars

Function Library

This includes a variety of MARC utilities, mesh generation, and post-processing plotting options (these will not be discussed in any detail in this Primer since it is assumed you have Mentat II for these functions), kinematic constraints, loads, bandwidth optimization, rezoning, in-core and out-of-core solution, user subroutines, restart, output on post file, selective print, error analysis, etc. Only loads and constraints are summarized below; refer to the MARC manuals for descriptions of the others.

- **Loads and constraints**
  - mechanical loads–concentrated, distributed, centrifugal, volumetric forces
  - thermal loads–initial temperatures read from a post file produced from a thermal analysis, or from data files
  - initial stresses and initial plastic strains
  - kinematic constraints
    - transformation of degrees of freedom
    - elastic foundation
    - tying (multipoint constraints or MPC's)
    - boundary conditions in user-defined axes
    - springs and gaps–with and without friction
    - contact surfaces

Element Library

The heart of an FE program lies in its element library, which allows you to model a structure for analysis. MARC has a very comprehensive element library which lets you model virtually any conceivable 1-D, 2-D, or 3-D structure. This section gives some basic definitions, summarizes MARC element types, and describes the most commonly used elements of interest to the beginner.

Definitions

isoparametric	a single function is used to define both the element geometry and the deformation
numerical integration	a method used for evaluating integrals over an element. Element quantities–such as stresses, strains, and temperatures–are calculated at each integration point of the element
Gauss points	the optimal integration point locations for numerical accuracy
full integration (quadrature)	requires, for every element, 2^d integration points for linear interpolation, and 3^d points for quadratic interpolation. the scalar 'd' is the number of geometric dimensions of an element (i.e., d=2 for a quad, d=3 for a hexahedron). This results in exact integration of linear functions in linear elements, or quadratic functions in quadratic elements.

reduced integration	means using a lower number of integration than necessary to integrate exactly. For example, for an 8-node quadrilateral, the number of integration points is reduced from 9 to 4, and for a 20-node hexahedron, from 27 to 8 (this definition of "reduced integration" is included here for reference only; in this Primer we will not discuss or use reduced integration elements).

CAUTION

The use of reduced integration near singularities and in regions of high strain gradients can lead to oscillations in the displacement ("checkerboarding") and produce inaccurate results. This phenomenon is highly problem dependent.

interpolation (shape) function	an assumed function relating the displacements at points inside an element to the displacements at the nodes of an element. In MARC, four types of shape functions are used: linear, quadratic, cubic, and Hermitian.
degrees of freedom (DOF)	the number of unknowns at a node. In the general case, there are six DOFs at a node in structural analysis (three translations, three rotations), and one DOF in thermal analysis (nodal temperature). In special cases, the number of DOFs is: 2 (translations) for plane stress, plane strain, and axisymmetric elements; 3 (translations) for 3-D truss element; 6 (three translations, three rotations) for a 3-D beam element.
incompressible elements	MARC has a special class of elements which can be used to analyze incompressible (zero volume change) and nearly incompressible materials such as elastomers and rubber. They are based on a modified Herrmann variational principle, and are sometimes referred to as "Herrmann elements." Unlike the regular finite element formulations, they can handle the case of Poisson's ratio equal to one-half. They are used for elastic analysis, but are capable of analyzing large displacement effects as well as thermal and creep strains. The incompressibility constraint is imposed by using Lagrange multipliers.
assumed strain elements	a special class of elements which are enhanced such that they can accurately calculate the shear (bending) strain.

Element Types

MARC has an extensive element library numbering approximately 130 elements. They are basically of two categories: structural and thermal. They cover a wide variety of geometric domains and problems:

truss	3-D rod with axial stiffness only (no bending).
membrane	thin sheet with in-plane stiffness only (no bending resistance).
beam	3-D bar with axial, bending, and torsional stiffnesses.
plate	flat thin structure carrying in-plane and out-of-plane loads.
shell	curved thin or thick structure with membrane/ bending capabilities
plane stress	thin plate with in-plane stresses only. All normal and shear stresses associated with the out-of-plane direction are assumed to be zero. (In MARC, all plane stress elements lie in the global X-Y plane.)
plane strain	structure with in-plane strains only, with all normal and shear strains associated with the out-of-plane direction equal to zero. (In MARC, all plane strain elements lie in the global X-Y plane.)
generalized plane strain	same as plane strain except that the normal Z-strain can be a prescribed constant or function of x and y.
axisymmetric	2-D idealized structure with radial and circumferential degrees of freedom only. In MARC, all axisymmetric elements lie in the Z-R (X-Y) plane.

NOTE

For the purpose of this Primer only, the descriptions of MARC axisymmetric elements have been simplified. For a more complete description, see MARC *Volume A*, *Chapter 7*, and *Volume B*.

3-D solid	solid structure with only translational degrees of freedom for each node (linear or quadratic interpolation functions).
special	MARC's special elements include: a gap/friction element, a pipe-bend element, a shear panel element, rebar elements, and several "semi-infinite" elements (which are useful for modeling a domain unbounded in one direction).

Heat Transfer Elements

Heat transfer elements in MARC consist of 3-D links, planar and axisymmetric elements, 3-D solid elements, and shell elements. For each heat transfer element, there exists at least one corresponding stress element. Temperature is the only degree of freedom for each node in these elements (except in the case of Joule heating analysis, which is a coupled thermal-electrical analysis).

Element Usage Hints

The following general hints on element usage should be useful to most MARC users, especially the first-time user.

1. Element input data generally includes: element connectivity; thickness for 2-D beam, plate, and shell elements; cross section for 3-D beam elements; coordinates of nodal points; and face identifications for distributed loadings.
2. You may select different element types to represent various parts of a model. If they are incompatible (meaning conflicting degrees of freedom), you have to provide appropriate tying constraints.
3. You may use most MARC elements for both linear and nonlinear analyses; exceptions are noted in *MARC Volume B*.
4. In linear analysis, you should consider using higher-order elements, especially in problems involving bending action. In nonlinear analysis, lower-order elements are preferred.
5. When using lower-order elements (whether the analysis is linear or nonlinear), 4-node quadrilaterals are preferred over 3-node triangles in 2-D problems. Similarly, 8-node bricks perform significantly better than 4-node tetrahedra in 3-D problems.
6. Stresses and strains of all continuum elements are defined in the global coordinate system (X,Y,Z). For truss, beam, plate, and shell elements, stresses and strains are output in the local system for the element and the output must be interpreted accordingly. You should pay special attention to the use of these elements if the material properties have preferred orientations.
7. The coordinates and degrees of freedom of all continuum elements are defined in the global coordinate system. Truss, beam, plate, and shell elements may be defined in a local coordinate system–and you must interpret the output accordingly.
8. Distributed loads may be applied along element edges, over element surfaces, or over the volume of the element. MARC will automatically evaluate the consistent nodal forces using numerical integration. Concentrated forces are applied only at nodes.

9. For five bilinear elements (Types 7, 10, 11, 19, and 20), an optional integration scheme may be used which imposes a constant dilatational strain constraint on the element. This option is often useful in approximately incompressible, inelastic analyses such as large strain plasticity, because conventional elements give results which are too stiff for nearly incompressible behavior.
10. For four elements (Types 3, 7, 11, and 19), optional interpolation functions may be used which improve the behavior of these elements in bending. The reduced integration elements with hourglass control also use an assumed strain formulation.
11. Five Fourier shell and solid elements (Types 62, 63, 73, 74, and 90) exist for the analysis of linear axisymmetric structures with nonaxisymmetric loads. The circumferential load and displacement is represented by a Fourier series, but the geometry and material properties may not change in the circumferential direction. You can therefore uncouple a 3-D problem into a series of 2-D problems. These elements can only be used for linear elastic analysis because the principle of super-position applies only to this type of linear analysis.

In the abbreviated Element Summary Table on the next page, the most commonly used elements are indicated with an asterisk (*). The following element types are intentionally excluded from the table because they are unlikely to be used by the first-time user: generalized plane strain elements; axisymmetric shell/solid Fourier elements; axisymmetric solid elements with torsional and bending capabilities; incompressible elements which are of generalized plane strain and axisymmetric Fourier types; rebar elements; semi-infinite elements; and pipe bend elements.

Table 1-1: Element Summary Table (Abbreviated)

PROBLEM TYPE	ELEMENT TYPE	LINEAR		QUADRATIC OR CUBIC
		QUAD	TRIANGULAR	
PLANE	Plane stress	3*, 114	6	26
	Plane strain	11*, 115		27
AXISYMMETRIC	Solid	10*, 116	2	28, 67
	Shell	1, 15		89*
3-D SOLID	Continuum	7*, 117		21
SHELL	Shear panel	68		
	Membrane	18		30
	Thick shell	75*		22*
	Thin shell	72*		
BEAM	Truss	9*		64
	2-D beam	5		16
	3-D beam	52*, 98*		
	Open section	13		
	Closed section	14, 25*		
	With thick shell	78, 79		
	With thin shell	76, 77		
HEAT CONDUCTION	3-D link	36		64
	Planar	39*, 121	37	41
	Axisymmetric	40*, 122	38	42
	3-D solid	43*, 123		44
	Shell	85		86
	Axisymmetric shell	88		87
INCOMPRESSIBLE	Plane strain	80*, 118		32
	Axisymmetric	82*, 119		33, 66
	3-D solid	84*, 120		35
SPECIAL ELEMENT	Gap/Friction	12		

* = Most commonly used elements

Recommended Elements

The following 18 elements are recommended elements, which should serve the first-time user's (as well as the experienced user's) purposes for the bulk of structural and thermal analysis problems:

2-D 4-node quadrilaterals:	Elements 3, 10, 11, 39, 40, 80, 82
3-D 8-node hexahedra:	Elements 7, 43, 84
2-node truss and beams:	Elements 9, 25, 52, 98
3-node axisymmetric, curved thick shell:	Element 89
thin/thick shells:	Elements 22, 72, 75

These five classes of structural/thermal elements will be briefly described below. The use of most of these 18 recommended elements will be demonstrated in the eleven example problems in the Primer.

Element	Example(s)
3	1
9	rod example–end of chapter
10	2B, 10
11	2A, 9
25	5
39	8
52	3A, 3B, 3C
72	4
75	6
77	4
80	11
89	7

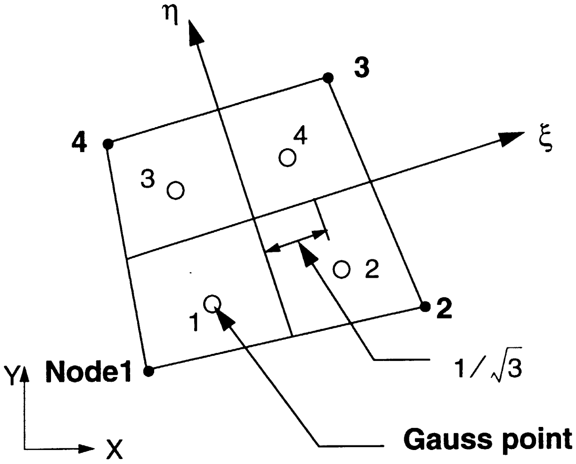
Notice the emphasis on quadrilateral elements–in preference to triangles– because of better overall performance. Likewise, hexahedral solid elements are preferred over tetrahedral/pentahedral elements.

In internal MARC calculations, stress-strain relationships are computed at the Gauss (integration) points. Stresses and strains may be printed at those integration points where the values are most accurate. To reduce computational costs for linear analysis, the user can optionally specify that calculations be performed only at the centroid. This is not recommended for nonlinear analysis. Nodal values of stresses and strains, produced by extrapolation of integration point values, can be printed out in both linear and nonlinear analyses.

2-D 4-node Quads:
(elements 3, 10, 11, 39, 40, 80, 82)

These are isoparametric quadrilateral 2-D continuum elements with straight edges and bilinear interpolation.

All these elements have four nodes with two DOFs per node, except for **Elements 80** and **82** (two of the so-called "Herrmann elements"), which have an extra node with a single DOF (pressure). The node numbering is counter-clockwise as shown. These elements use a four-point Gaussian integration scheme. **Elements 10** and **11** have an optional constant dilatation integration scheme, which is useful in plasticity problems.

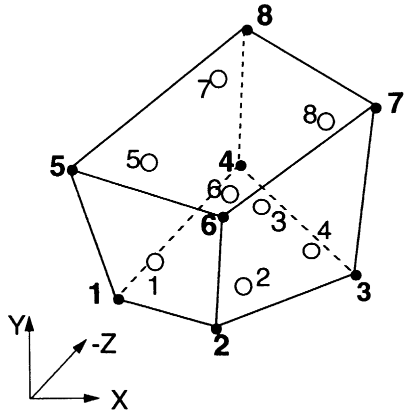


Conventional Element	Reduced Integration	Type
3	113	plane stress
10	116	axisymmetric solid
11	115	plane strain
39	121	thermal–planar
40	122	thermal–axisymmetric
80	118	incompressible–plane strain
82	119	incompressible–axisymmetric

8-node Hexahedra:
(elements 7, 43, 84)

These are 3-D isoparametric continuum elements with straight edges and trilinear interpolation.

These three elements are basically 8-noded elements with three DOFs per node, except that **Element 84** has an extra node with a single DOF (pressure). The node numbering is counterclockwise as shown, first for the bottom face and then for the top face. The elements have an eight-point Gaussian integration scheme. (**Element 7** also has an optional constant dilatation integration scheme.) These solid elements are arbitrarily distorted hexahedra.

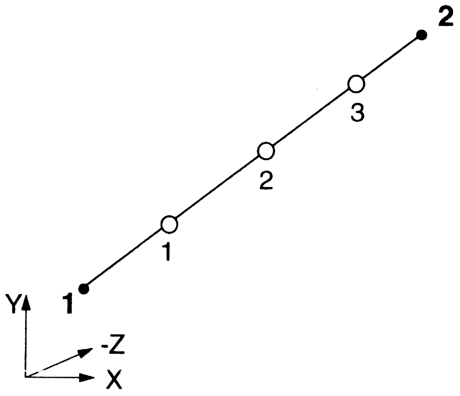


Conventional Element	Reduced Integration	Type
7	117	solid cube–stress analysis
43	123	solid cube–thermal
84	120	solid cube–incompressible

2-node Truss and Beams:
(elements 9, 25, 52, 98)

These are straight truss or beam elements with constant cross sections.

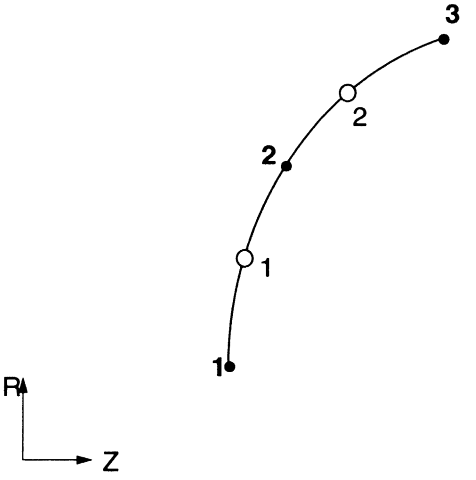
These four elements are two-noded straight elements with linear interpolation (constant axial force) along the axis. In addition, **element 52** features cubic interpolation (constant beam curvature) normal to the axis and also has linear interpolation for twist. **Element 25** is a thin-walled closed-section beam for which material nonlinearity is allowed in the cross section. **Element 9** has three DOFs at each node, which are the three translations. It can be used for large strain, large displacement analysis. **Element 52** has six DOFs at each node: three translations and three rotations. It can be used only for elastic materials. Large curvature changes are neglected in the large displacement formulation. **Element 98** is a straight elastic beam including transverse shear effects.

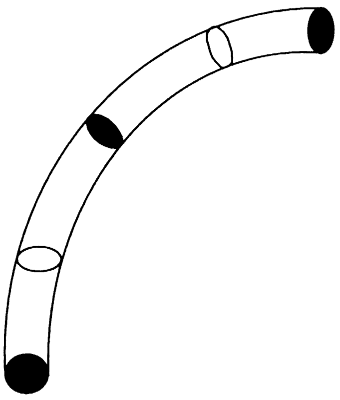


Element	Type
9	truss
25	3-D closed-section thin-walled beam with twist
52	3-D solid-section elastic beam
98	3-D elastic beam with transverse shear

Axisymmetric, Curved Thick Shell:
(element 89)

This 3-noded curved element has better performance than the 2-noded curved **Element 1**. In addition, it includes transverse shear effects and hence is recommended for axisymmetric thick shell analysis. It is suitable for large displacement analysis with small strains.





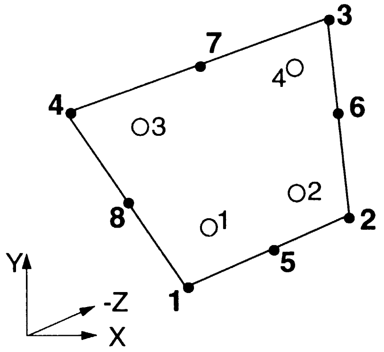
Thin/Thick Shells:
(elements 22, 72, 75)

These thin/thick shell elements will be appropriate for the bulk of plate and shell analyses.

Element 22 is a quadratic thick-shell element with global displacements and rotations as DOFs. Second order interpolation is used for coordinates, displacements, and rotations. The membrane strains are obtained from the displacement field, and the curvatures are obtained 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. It has eight nodes (4 corners, 4 midsides) and 6 DOFs per node (3 displacements, 3 rotations). Bilinear thickness variation is allowed in the plane of the element. There are four Gaussian integration points.

Element 72 is an 8-noded thin shell element. It has straight edges. A bilinear variation in thickness is allowed. Bilinear interpolation is used for global displacement and coordinates. Global rotations are interpolated quadratically from the rotation vectors at the centroid and at the mid-side nodes. The element has three DOFs (global Cartesian displacements) at the four corner nodes, and one rotation DOF (of the edge about itself) at each of the mid-side nodes. It has four Gaussian integration points. The element is efficient for the analysis of curved shells and plate structures, and is fairly insensitive to distortion. Because of the relatively few degrees of freedom per element, a large number of elements may be required.

Element 75 is a 4-noded bilinear thick shell element. A bilinear variation in thickness is allowed. Each node has six DOFs: three global displacements, three global rotations. Bilinear interpolation is used for the coordinates, displacements, and rotations. The membrane strains are obtained from the displacement field, and the curvatures are obtained from the rotation field. The transverse shear strains are calculated at the middle of the edges and interpolated to the integration points. The element has four Gaussian integrations points. It is very efficient for analyzing curved shells, plate structures, and nonlinear problems, and is not very sensitive to distortion.



Element	Type
22	8-noded curved quadrilateral thick-shell element
72	8-noded bilinear constrained thin-shell element
75	4-noded bilinear thick shell

Input

This section highlights MARC input concepts. Concepts such as PARAMETER, MODEL DEFINITION, and LOAD INCREMENTATION are briefly described, as are input formats (fixed versus free field input of numerical data, lists) and input of loads and constraints. For details, please refer to *MARC User Information Manual Volume C.*

Input Units

Many of the examples in this Primer use English units; an exception is Example 10, which uses SI units. No units are actually entered in the input file by the user. MARC simply assumes that *all* input is being provided in a consistent manner.

Input Sections

MARC is a batch program. This means that the user defines the input, and this input is not changed during the program execution. This input may be created using Mentat II or a text editor. The input may be modified upon restart for nonlinear or transient analysis.

MARC input consists of three major sections:

PARAMETER options	define the title of the analysis, the storage allocation, analysis type, element type(s), etc. (This section terminates with an END option.)
MODEL DEFINITION options	define coordinates, connectivity, materials, boundary conditions, initial loads, initial stresses, nonlinear analysis controls, output options, etc. (This section terminates with END OPTION.)

Nonlinear and/or transient analyses are performed by increments (steps). The information required to define the load history requires the additional section:

LOAD INCREMENTATION or **HISTORY DEFINITION** options	define the increments in terms of load increments and/or boundary condition changes occurring during the **HISTORY DEFINITION** increment. (This section ends with a CONTINUE option.) *(At this stage, one or more increments are analyzed.)*

The first two sections (PARAMETER, MODEL DEFINITION) are *always* present. You may stack as many load incrementation options as you want; they are analyzed by MARC in sequence until the last CONTINUE option is encountered. At the end of this chapter, you will see the input of a simple linear statics example, which only shows the first two sections.

Input Format

A MARC input file consists of many blocks of lines of input, each headed by a keyword. A keyword describes some property of the FE model of the structure (coordinates, materials, boundary conditions, etc.). A keyword can also describe a control function for the analysis (generation of printout, writing of a post file, numerical tolerances, etc.).

A block may contain three different types of input:

alphabetic keyword	describes the contents of the block; placed on a single line.
numerical data	quantifies the properties of the model; floating point or integer; placed on one or more lines.
lists	denotes the nodes, elements, and DOFs to which the properties apply. Free format.

The numerical data may be in free or fixed format. Lines in free and fixed formats may both exist in the input, although a particular line may use only one format.

free field	is easier, safer, and recommended for hand-generated input (Mentat II casts input data in fixed field format). It is flagged by at least one comma existing in the input line. Data items on a line are separated by commas, which may be preceded or followed by an arbitrary number of blanks. No imbedded blanks may appear within the data item itself. Each line must contain no more data items than would have appeared if fixed format was used. If fewer items appear, the remaining entries are assumed to be zero. If only one item is given, a comma should follow the entry. Floating point numbers may be given with or without an exponent. The mantissa must contain a decimal point. If an exponent is given, it must be preceded by the letter E or D and must immediately follow the mantissa (no embedded blanks). EXAMPLE: 5.4E6,0.3,11.,0.,18.
fixed field	is described in detail in MARC *Volume C*. Standard FORTRAN conventions are observed. Integers must be right-justified in field. Floating point numbers may be given with or without exponent. The mantissa must contain a decimal point. If an exponent is given, it must be preceded by the letter E or D and must be right-justified.

A list is a convenient way to identify a set of elements, nodes, DOFs, integration points, shell layers, etc. Lists come in three forms:

sequence (n1 n2 n3)	the list includes n numbers placed on one or more lines, separated by blanks or comas. If a sequence continues onto another line, a C must be the last item on the line.
range (m TO n BY p)	the list includes all numbers from m to n with interval p. (Default p=1)
set name (STEEL)	the list includes the numbers in the set named STEEL previously specified by the DEFINE command of the MODEL DEFINITION options.

Furthermore, lists can be operated upon by the logical operations AND, EXCEPT, and INTERSECT. For example:

2 TO 38 BY 3 AND STEEL

PARAMETER Section

PARAMETER options control the scope and type of the analysis. Typically, the first option, TITLE, is the name of the problem. The SIZING option defines the problem size in words of the core buffer used by MARC. ELEMENTS indicates what MARC element types are used in the analysis. Other optional PARAMETER options include: ALL POINTS (asking for stress output at all the integration points of the elements); BEAM SECT (defining the cross-sectional properties of a beam, i.e., prismatic or thin-walled); CENTROID (asking for stress output only at the centroids of the elements); ELASTIC (flags linear elastic static analysis); SHELL SECT (defines the number of integration points through the shell thickness: ranging from 3 to 99); STOP (telling MARC not to do the analysis–a check run of input only); and THERMAL (flags initial temperatures being input for stress analysis).

In this set of options, only the TITLE, SIZING, and END options are mandatory; the ELEMENTS option may, however, be used instead of (or in conjunction with) the SIZING option. All other PARAMETER options are optional.

The PARAMETER options may appear in any order. The only requirement is that they must terminate with an END option.

MODEL DEFINITION Section

The MODEL DEFINITION options describe the complete FE model for analysis.

- mesh
- materials
- applied loads
- constraints
- controls

The following paragraphs describe those options which you will encounter most frequently. In a nonlinear analysis, you may alter most of this data during the later stages of the analysis. For a linear elastic analysis, the model is defined once in the MODEL DEFINITION options. The MODEL DEFINITION options also control the output. The selective output feature will be described later in the OUTPUT section of this chapter.

Mesh

The shape and geometry of the FE mesh are specified using the following MODEL DEFINITION options:

COORDINATES	of the nodes in the mesh
CONNECTIVITY	of the elements connecting the nodes
GEOMETRY	geometric properties of beam and shell elements (e.g., beam cross section, shell thickness, etc.)
PROPERTY	material properties, e.g.: ISOTROPIC ORTHOTROPIC GAP DATA MOONEY OGDEN WORK HARD TEMPERATURE EFFECTS STRAIN RATE RATE EFFECTS CREEP

The DOFs (loads, displacements) at a node depend on the element type connected to the node, unless a triad of local axes is defined for a set of nodes using:

TRANSFORMATIONS	establishes the directions of the local nodal axes with respect to the global axes.

Mechanical Loads

Mechanical loads are of two types: concentrated and distributed.

POINT LOAD	concentrated load vector acting on a node.
DIST LOADS	volumetric (body forces such as gravity) or pressure loads (acting on surfaces or edges). The type is specified by defining the variable IBODY. Can be uniform or non-uniform.

Thermal Loads

The INITIAL STATE option can be used to define a nonhomogeneous initial temperature field in a stress analysis. This temperature does not produce any thermal strains. The temperatures can then be modified using the CHANGE STATE option.

The change in temperature causes thermal strains, and possibly changes in the material properties if TEMPERATURE EFFECTS are included.

Kinematic Constraints

You can prescribe values to individual DOFs using:

FIXED DISP	prescribed values for specified DOFs, on a set of nodes

The input displacements refer to the directions associated with the element, generally global Cartesian, unless a TRANSFORMATIONS option is provided to refer them to user-defined local axes. These prescribed displacements can be subsequently modified using the DISP CHANGE load incrementation option.

Support Springs

Elastic springs may be defined between any two DOFs at any two nodes:

SPRINGS	assigned spring constant between two DOFs for two nodes.

CONTROL Option

Another important MODEL DEFINITION option is the CONTROL option, which lets you select input parameters governing convergence and accuracy in nonlinear analysis. Items in CONTROL are mostly integers (except for tolerances–which are in floating point). The first two items arc thc most important. Note that the number of cycles includes the first cycle, and the number of increments likewise includes the first increment.

ITEM	MEANING	DEFAULT
step	maximum number of increments (loads) in this analysis	4
cycl	maximum number of iterations per increment	3

There are other items on the CONTROL option, but they are usually not needed by the first-time user. These items flag such options as convergence tests, iteration schemes, non-positive definiteness checks, etc. (See *Volume C*).

The first increment in an analysis is considered increment zero and should be linear elastic. Thus, four increments imply increments 0, 1, 2, and 3. Similarly, three cycles imply the first cycle and two iterations.

OPTIMIZE Option

Finally, you need to be aware of the OPTIMIZE option in the MODEL DEFINITION section. This option lets you choose a bandwidth optimization algorithm. The default algorithm is Cuthill-McKee, which is widely used in many FE codes and suffices for most cases. Minimizing the bandwidth in your problem reduces computer costs in medium to large-sized problems. Therefore, you should make it a habit to invoke the OPTIMIZE option before performing an analysis. For a description of other available bandwidth optimization algorithms, see MARC *Volume C.* Note that it is not necessary to use the OPTIMIZE option when the element-by-element solver is used.

Output

This section summarizes MARC output and post-processing options. MARC output can be obtained in five forms:

- standard printed output
- selective printed output
- post file for Mentat II post-processing
- restart file (for continuation of analysis)
- plotted output from MARC (not covered in this PRIMER).

Printed Output

A standard printed output from a MARC run contains three different parts:

- input echo and interpretation
- analysis messages
- output of analysis results.

Input Echo and Interpretation

This portion repeats the input to allow you to verify its correctness. It includes various items such as: position of the line columns; a line count for the blocks; set up of parameters for the run; and interpretation of the input (e.g., connectivity, coordinates, properties, geometry, boundary conditions, loads, etc.).

Analysis Messages

During the analysis, MARC produces several diagnostic messages. Those of interest include:

- algebraic sum of the distributed and point loads over the whole model.
- singularity ratio of the matrix. This is a measure of the conditioning number (hence the accuracy) in the solution of the linear equations. The ratio and its meanings are:

between 10^{-4} and 1	acceptable
between 10^{-8} and 10^{-4}	possible numerical problems (...watch out)
on order of machine accuracy (10^{-14} to 10^{-8})	singular equations (unreliable solution)

During the analysis, MARC will print out the elapsed CPU time at the following points:

- start of increment
- start of assembly
- start of matrix solution
- end of matrix solution
- end of increment.

Output of Analysis Results

At the end of the analysis, MARC will print out (for each increment) element data (stresses, strains, etc.), and nodal data (displacements, equivalent nodal forces, and reaction forces at fixed boundary conditions).

Element Output

At every Gaussian integration point, stresses (or forces) and strains are printed out, depending on the element type. (If you include a CENTROID PARAMETER option, only the centroidal results will be reported.)

continuum elements	physical components (in global axes); principal values; mean normal values (hydrostatic); Tresca and von Mises equivalent values.
shell elements	generalized total stress and strain resultants (stretch, curvature) at midplane; total physical stresses at integration points through the thickness.
beam elements	resultant forces at Gauss points: axial force; bending moment (referred to local axes of beam element); and torque.

Nodal Output

For every node, the vectors of these nodal quantities are printed out, depending on the analysis:

static	incremental and total displacements; equivalent nodal loads; reaction forces (at boundary nodes); residual loads (at nodes without boundary conditions).

(If convergence has occurred during the increment, the residual loads should be small compared with the reaction forces.)

dynamic	eigenvectors (for modal analysis)
	for transient analysis: total displacements, velocities, and accelerations equivalent nodal loads reaction forces

residual loads

for heat transfer:
total temperatures and optional fluxes

Selective Output

You may selectively print out data for elements or nodes using these MODEL DEFINITION options:

PRINT ELEMENT	selects elements, integration points, and layers (for plate and shell elements) to be printed in the output.

NOTE

All the stress components are printed out. The selected layers and integration points apply to all the selected elements in the model.

PRINT NODE	selects nodes and nodal quantities to be printed (e.g., displacements, input load vectors, output reactions/residuals).

Post File

You may use the POST command to flag the writing of a MARC post file, which can be processed later by Mentat II. The post file can be either binary or formatted. A binary file is machine-dependent, but is usually quite a bit smaller than a formatted file. A formatted file is portable across different types of computers, but is usually larger than a binary file. It can also be edited.

The file output includes:

- complete mesh data (nodal coordinates, element connectivities)
- all nodal variables (displacements, forces, etc.)
- element variables (strains, stresses, etc.) as selected in the POST option. You may select which stress components to write out for which layer; the output will be produced for all integration points of all elements.

A restart file can be made using the RESTART model definition option (see MARC Volume C). This option is very convenient in nonlinear analysis. Its use will be illustrated later in Example 11.

Sample Problem and Output

A very simple linear statics problem is analyzed. The entire MARC output follows for the purpose of illustrating typical input echo and interpretation, analysis messages, and output of analysis results.

The FE model is a one-element, 2-node rod held at the left end and loaded by an axial tensile load (P) of 10,000 lbs at the right end. The rod length (L) is 10 in., and the cross-section area (A) is 1 in^2. Young's modulus (E) is 30E6 (30×10^6 or 30,000,000) psi.

The theoretical axial displacement at the right end is easily calculated to be:

$$u_x = PL/AE = (1000)\,(10)/(1)\,(30E6) = 3.3333E\text{-}3 \text{ in.}$$

$$(30E6 = 30 \times 10^6 \text{ or } 30{,}000{,}000)$$
$$(3.3333E\text{-}3 = 3.333 \times 10^{-3} \text{ or } .003333)$$

The MARC calculated displacement agrees exactly with the theoretical value.

The axial stress is of course merely (P/A), or 10,000 psi–the same as that obtained by MARC.

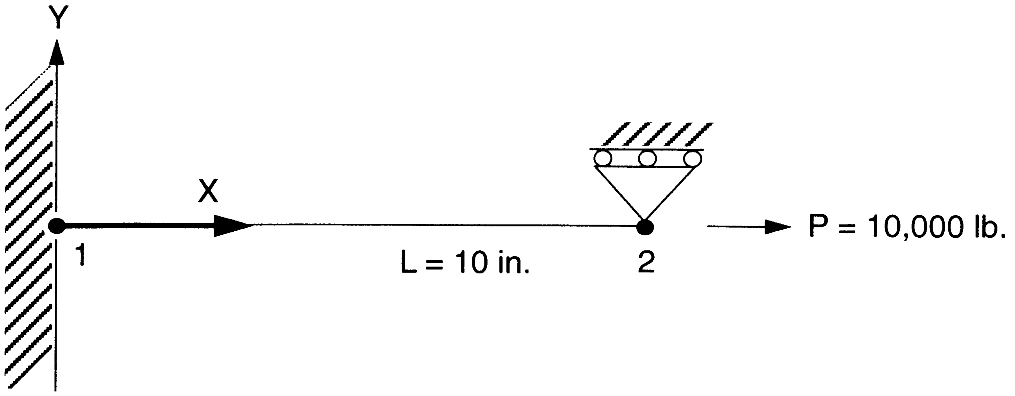
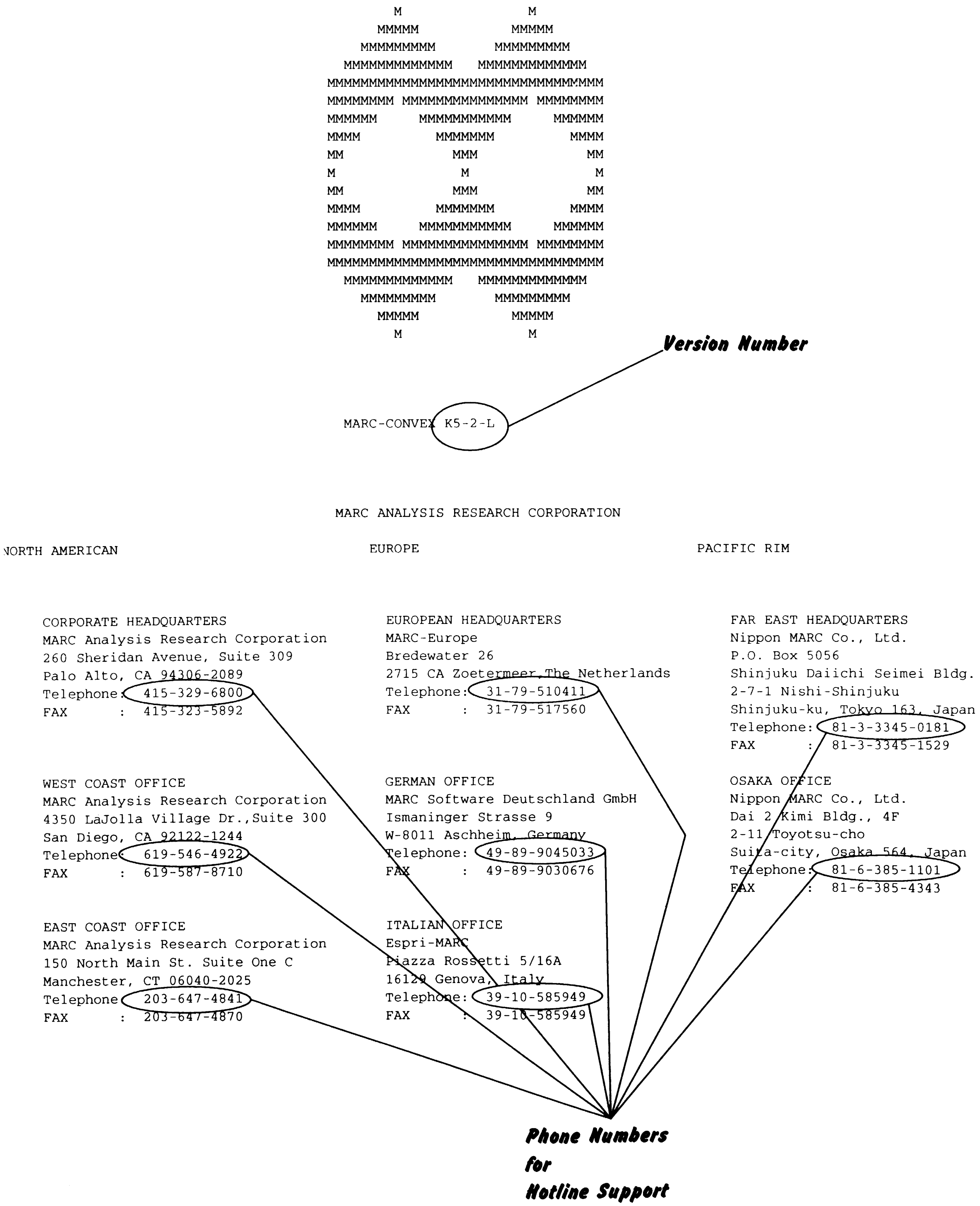


Figure 0.1 Truss Under Tension



```
                                               M               M
                                             MMMMM           MMMMM
                                           MMMMMMMMM       MMMMMMMMM
                                         MMMMMMMMMMMMM   MMMMMMMMMMMMM
                                       MMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMM
                                       MMMMMMMM MMMMMMMMMMMMMMM MMMMMMMM
                                       MMMMMM     MMMMMMMMMMM     MMMMMM
                                       MMMM         MMMMMMM         MMMM
                                       MM             MMM             MM
                                       M               M               M
                                       MM             MMM             MM
                                       MMMM         MMMMMMM         MMMM
                                       MMMMMM     MMMMMMMMMMM     MMMMMM
                                       MMMMMMMM MMMMMMMMMMMMMMM MMMMMMMM
                                       MMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMM
                                         MMMMMMMMMMMMM   MMMMMMMMMMMMM
                                           MMMMMMMMM       MMMMMMMMM
                                             MMMMM           MMMMM
                                               M               M


                                         MARC-CONVEX K5-2-L


                                        MARC ANALYSIS RESEARCH CORPORATION

NORTH AMERICAN                              EUROPE                                 PACIFIC RIM


     CORPORATE HEADQUARTERS                   EUROPEAN HEADQUARTERS                    FAR EAST HEADQUARTERS
     MARC Analysis Research Corporation       MARC-Europe                              Nippon MARC Co., Ltd.
     260 Sheridan Avenue, Suite 309           Bredewater 26                            P.O. Box 5056
     Palo Alto, CA 94306-2089                 2715 CA Zoetermeer,The Netherlands       Shinjuku Daiichi Seimei Bldg.
     Telephone: 415-329-6800                  Telephone: 31-79-510411                  2-7-1 Nishi-Shinjuku
     FAX      : 415-323-5892                  FAX      : 31-79-517560                  Shinjuku-ku, Tokyo 163, Japan
                                                                                       Telephone: 81-3-3345-0181
                                                                                       FAX      : 81-3-3345-1529

     WEST COAST OFFICE                        GERMAN OFFICE                            OSAKA OFFICE
     MARC Analysis Research Corporation       MARC Software Deutschland GmbH           Nippon MARC Co., Ltd.
     4350 LaJolla Village Dr.,Suite 300       Ismaninger Strasse 9                     Dai 2 Kimi Bldg., 4F
     San Diego, CA 92122-1244                 W-8011 Aschheim, Germany                 2-11 Toyotsu-cho
     Telephone: 619-546-4922                  Telephone: 49-89-9045033                 Suita-city, Osaka 564, Japan
     FAX      : 619-587-8710                  FAX      : 49-89-9030676                 Telephone: 81-6-385-1101
                                                                                       FAX      : 81-6-385-4343

     EAST COAST OFFICE                        ITALIAN OFFICE
     MARC Analysis Research Corporation       Espri-MARC
     150 North Main St. Suite One C           Piazza Rossetti 5/16A
     Manchester, CT 06040-2025                16129 Genova, Italy
     Telephone: 203-647-4841                  Telephone: 39-10-585949
     FAX      : 203-647-4870                  FAX      : 39-10-585949
```

Input Echo
(First of 3 passes)

```
                              m a r c
                          i n p u t   d a t a


              5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
           --------------------------------------------------------------------------------
           TITLE, UNIAXIAL TENSION TEST OF A TRUSS
           COMMENT, MARC BASIC LINEAR ELASTIC INPUT DECK
           COMMENT, U DISPLACEMENT AT NODE 2 = P*L/(E*A) = 3.33E-3
           SIZING,30000
card     5 ELEMENTS,9
           END
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -

                   CONNECTIVITY

                   1,9,1,2,
     card     10   COORDINATES

                   1,0.,0.,
                   2,10.,0.,
                   PROPERTY
     card     15
                   30.E6,
                   1
                   GEOMETRY

     card     20   1.0,
                   1
                   FIXED DISP

                   0.,
     card     25   1 TO 3
                   1
                   0.,
                   2 AND 3
                   2
     card     30   POINT LOAD

                   10000.,
                   2
                   END OPTION
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -

           --------------------------------------------------------------------------------
              5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
           --------------------------------------------------------------------------------
           --------------------------------------------------------------------------------
```

TITLE

PARAMETER Options
(terminates with END option)

Theoretical Value

Element Type

MODEL DEFINITION Options
(terminates with END OPTION option)

NOTE

This example does NOT have a LOAD INCREMENTATION or HISTORY DEFINITION section.

(Second of 3 Passes)

Summary of pre-reader

```
 ************************************************
 ************************************************

  program sizing and options requested as follows
```

This pre-reader is a very important visible part of the program.

It scans the input file to determine maximum core allocation parameters such as:

- ***max no. of elements***
- ***max no. of nodes***
- ***max no. of boundary conditions, etc.***

```
element type requested*************************       9
number of elements in mesh*********************       1
number of nodes in mesh************************       2
max number of elements in any dist load list***       0
maximum number of boundary conditions**********       5
load correction flagged or set*****************
number of lists of distributed loads***********       3
stresses stored at all integration points******
tape no.for input of coordinates + connectivity       5
no.of different materials     1 max.no of slopes      5
maximum elements variables per point on post tp      33
number of points on shell section *************      11
option for terminal debug**********************
new style input format will be used************
maximum number of set names is*****************      10
number of processors used *********************       1
vector length used ****************************       1
```

Always glance at table to verify that the numbers are correct

```
     end of parameters and sizing
    ************************************************
    ************************************************



key to stress, strain and displacement output
```

Element type data

```
          element type    9


2-node, 3-d truss

stress and strain are uniaxial

displacements in global directions
  1=u global x direction
  2=v global y direction
  3=w global z direction


workspace needed for input and stiffness assembly      3893


internal core allocation parameters
degrees of freedom per node (ndeg)  3
coords per node (ncrd)  3
strains per integration point (ngens)  1
max. nodes per element (nnodmx)  2
max.stress components per int. point (nstrmx)   1
max. invariants per int. points (neqst)  1
```

```
      flag for element storage (ielsto) 0
      elements in core, words per element (nelsto)       198
                        total space required             198
      vectors in core, total space required        89


      words per track on disk set to 4096


      internal element variables


      internal element number  1  library code type  9
      number of nodes=  2
      stresses stored per integration point =  1
      direct continuum components stored =  1
      shear continuum components stored =  0
      shell/beam flag = 0
      curvilinear  coord. flag = 0
      int.points for elem. stiffness  1
      number of local inertia directions  3
      int.point for print if all points not flagged  1
      int. points for dist. surface loads (pressure)  2
      library code type =  9
      no local rotation flag = 1
      generalized displ. flag = 0
      large disp. row counts     6
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
```

ELSTO flag:
Caution - program can turn this on automatically. This flag indicates element storage "in-core" [0] or "out-of-core" [1].

```
            residual load correction is invoked


 connectivity
 ----------

 meshr1,iprnt
     5     0
 elem no., type,        nodes
     1       9        1    2

 coordinates
 ----------

 ncrd1 ,meshr1,iprnt
     3      5     0
  node       coordinates
     1      0.           0.          0.
     2  10.000           0.          0.

 property
 ----------


 youngs mod.,poisson r.,density, alpha   ,tot.temp.,   yielp,   yielp2,  mat


 *** warning - material id unspecified.  matid  =     1 assumed.
```

(Third of 3 passes)
Actual reading and interpretation of input file by MARC

```
 0.300E+08 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.100E+21 0.000E+00     1
a list of elements given below
     1

geometry
----------


  egeom1     egeom2     egeom3     egeom4     egeom5     egeom6
  0.100E+01  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
a list of elements given below
     1

fixed disp
----------


fixed displacement =  0.000E+00  0.000E+00  0.000E+00
from degrees of freedom    1 to degrees of freedom    3 by     1
a list of nodes given below
     1
fixed displacement =  0.000E+00  0.000E+00  0.000E+00
a list of degrees of freedom given below
     2
       and
a list of degrees of freedom given below
     3
a list of nodes given below
     2




fixed boundary condition summary.
total fixed degrees of freedom read so far =     5

 b.c.    node   degree of  magnitude                 b.c.    node   degree of  magnitude
number            freedom                           number            freedom


    1       1         1    0.000E+00                   2       1         2    0.000E+00
    3       1         3    0.000E+00                   4       2         2    0.000E+00
    5       2         3    0.000E+00

point load
----------

read from unit     5
 0.100E+05 0.000E+00 0.000E+00
a list of nodes given below
     2

end option
----------
-----------------------------------------------------------------------------------------

   maximum connectivity is     2   at node     2


   maximum half-bandwidth is     2   between nodes     1   and     2

    number of profile entries including fill-in is               3

    number of profile entries excluding fill-in is               3
   total workspace needed with in-core matrix storage =        3971
```

Determination of required workspace for stiffness matrix.

If SIZING is greater than this number, then in-core is used. If not, out-of-core solution will be used.

```
                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
 0.000E+00 0.000E+00 0.000E+00


                    point loads
 1.000E+04 0.000E+00 0.000E+00

                    start of assembly
                     time =        0.25

                    start of matrix solution
                     time =        0.27

                    singularity ratio    1.0000E+00

                    end of matrix solution
                     time =        0.28
```

MARC analysis messages

(Beginning of analysis output)

```
 MARC            output for increment    0.     uniaxial tension test of a truss



 element with highest stress relative to yield is    1  where equivalent stress is 0.100E-15  of yield



 tresca     mises    mean   p r i n c i p a l   v a l u e s         p h y s i c a l   c o m p o n e n t s
       intensity intensity  normal   minimum intermediate maximum    1        2        3        4        5        6
                 intensity


 element     1  point   1        integration pt. coordinate=      0.500E+01     0.000E+00     0.000E+00
 section thickness = 0.100E+01

 stress  1.000E+04 1.000E+04 3.333E+03 0.000E+00 0.000E+00 1.000E+04 1.000E+04

 strain  3.333E-04 2.722E-04 0.000E+00 0.000E+00 0.000E+00 3.333E-04 3.333E-04
```

σ_{xx}

ε_{xx}

Physical components

```
                n o d a l   p o i n t   d a t a


                i n c r e m e n t a l   d i s p l a c e m e n t s

      1    0.         0.         0.              2  3.33333E-03     0.         0.


                t o t a l   d i s p l a c e m e n t s

      1    0.         0.         0.              2  3.33333E-03     0.         0.
```

Calculated X-displacement at node 2

```
          total equivalent nodal forces (distributed plus point loads)

      1    0.         0.         0.              2   10000.         0.         0.


            reaction forces at fixed boundary conditions, residual load correction elsewhere

         1  -10000.         0.         0.              2     0.         0.         0.
                          summary of externally applied loads
```

Reaction Force

Residual Load

```
         0.10000E+05    0.00000E+00    0.00000E+00

                          summary of reaction/residual forces

        -0.10000E+05    0.00000E+00    0.00000E+00

                          e n d   o f   i n c r e m e n t   0
                           time =       0.34


                          *** end of input deck - job ends




                           marc exit number 3004
```

Standard exit number for a normal MARC run. (...other exit numbers may indicate trouble!)

CHAPTER 2: Linear Static and Dynamic Problems

This chapter contains four linear static/dynamic problems. Each illustrates particular MARC features; the first three use isotropic material properties, while the fourth example demonstrates the input of composite properties. You should already be familiar with all the fundamental MARC concepts covered in Chapter 1 before you attempt any problem. The four example problems are:

Example 1: Tensile Stress in a Sheet with Hole.

Example 2: Thick Cylinder under Internal Pressure.

Example 3: Modal and Dynamic Analyses of a Cantilevered Beam.

Example 4: Stiffened Composite Roof under Uniform Pressure.

Again, remember that the intent of each example is tutorial. The descriptions accompanying each example explain the rationale behind the choice of the FE model, MARC input, special MARC features being demonstrated, and output highlights. The FE meshes are generated using Mentat II and are intentionally kept simple; they are not meant to demonstrate mesh refinement techniques and MARC correlation with theory or other published results.

The typical format used in the description of each example in this Primer is as follows:

Title	short problem description, purpose of analysis.
Sketch	engineering sketch of problem, dimensions, loads, boundary conditions, idealized FE model, etc.
Model	modeling considerations, choice of finite element and rationale, assumptions, Mentat II FE undeformed plot, geometry parameters (plate/shell thickness, layer specification, beam cross-section), etc.
Properties	material properties of model, assumptions.
Loads	thermal, mechanical, or body loads; use of certain MARC options to apply loads.
Boundary Conditions	symmetry boundary conditions (if any), constraints, tying, etc.
Special Features	user subroutines, bandwidth minimization, restart, rezoning, convergence controls, hints and avoidances.

Input	input echo pointing out special input parameters and format, descriptions of specific lines and data.
Output	(for selective examples) to illustrate format, output parameters for different analysis types.
Results	comparison with theory or other solutions, comments on accuracy and convergence, alternate solution methods, Mentat II plots of deformed geometry or stress/strain/temperature contours, references (if any), suggested exercises.

In order to avoid redundancy, references will often be made to MARC manuals and other examples in the Primer. At the end of the Primer, Appendix A will help you correlate MARC keywords with a particular example.

USING THE PRIMER

The following general comments apply when you are using this Primer:

1. For reference purposes, the input listing is always included for each example.
2. A complete output listing is included only for a few selected examples (e.g., Example 1). Selected portions from the printed output of each example are usually shown for explanation and annotation purposes, so that you can verify your run if you are trying to duplicate an example. (Notice the use of four dots to indicate that certain parts of the printout have been omitted.)
3. MARC K.5 offers the convenient optional use of a blank line–right after the header–to indicate that you are not furnishing a count (as previously required) of the number of data lines which follow for this option.
4. The three common "terminators" which end major input sections will not be explained in detail for each example:

END	terminates the PARAMETER section.
END OPTION	terminates the MODEL DEFINITION section.
CONTINUE	terminates the HISTORY DEFINITION or LOAD INCREMENTATION section (if any).

5. Ellipses or ovals are usually used in the printed output to denote maximum/minimum nodal or element quantities, or other items of interest.
6. For your convenience, Appendix A summarizes some commonly used MARC input options in the PARAMETER, MODEL DEFINITION, and HISTORY DEFINITION sections. It references sections in

MARC Volume C where you can find detailed input descriptions of each option (keyword), and cites examples in the Primer which illustrate the options. Please refer to *MARC Volume C* for those options you cannot find.

Example 1

Tensile Stress in a Sheet with Hole

This stress concentration example is a classical problem in elasticity[1]. A thin square sheet with a circular hole is loaded in tension uniformly along two opposite edges. The "hole radius" to "sheet width" ratio is chosen to approximate an infinite sheet in order to compare the results with the analytical solution. This plane-stress example illustrates basic input/output concepts of a simple linear static analysis, stress concentration effects, symmetry considerations, and mesh optimization.

Sketch

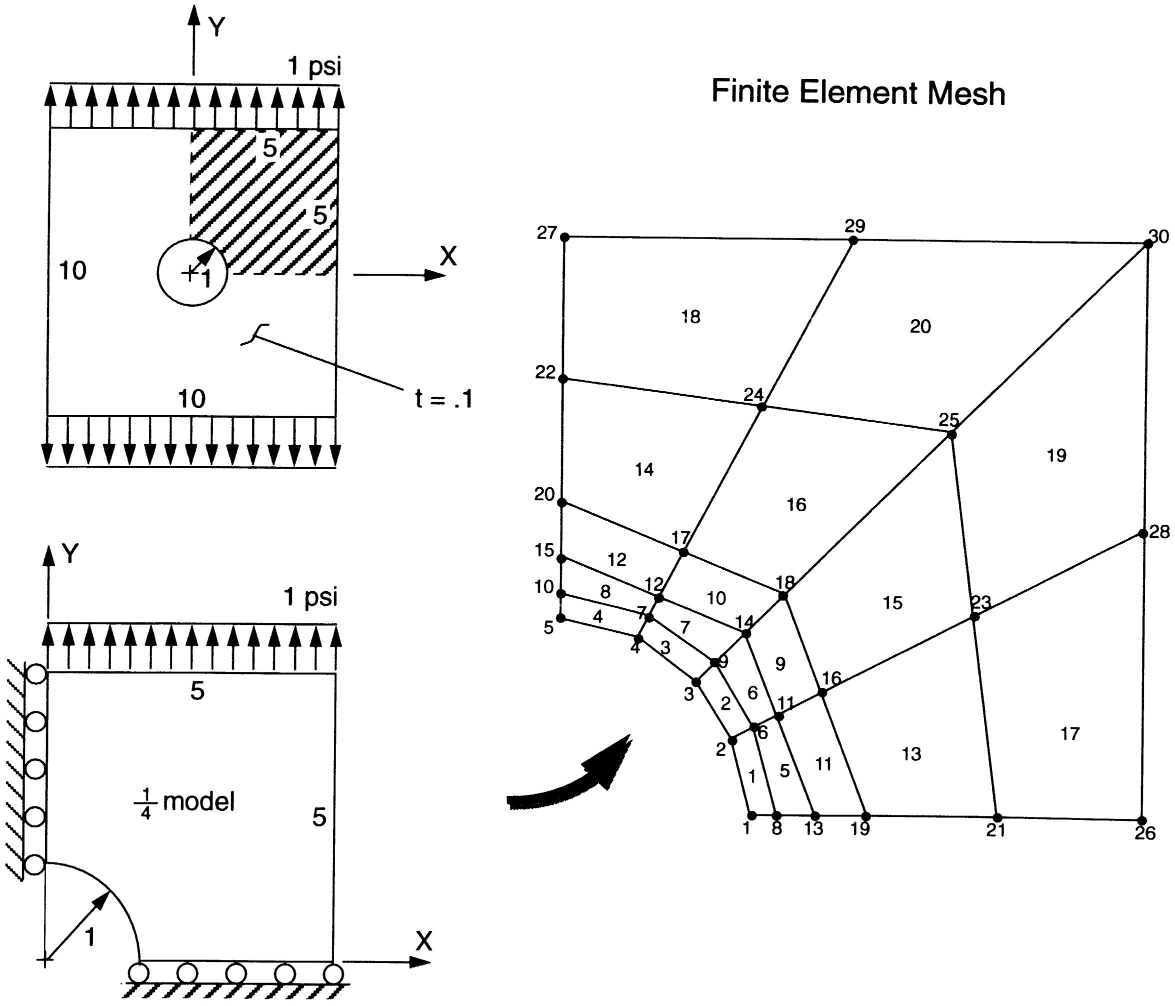


Figure 1.1 **Square Plate with a Circular Hole**

---

[1] Timoshenko, S.P. and Goodier, J.N., *Theory of Elasticity* (3rd ed.), McGraw-Hill, 1970, pp. 90-97.

Model

The finite element model should be the simplest idealized form of the structure and take advantage of symmetry (which exists in this case). The sheet has dimensions of 10 in., 0.1 in. thick, with a central hole of 1 in. radius. The material properties are isotropic and linear elastic, with a Young's modulus of 30E6 psi and Poisson's ratio of 0.3.

Because of two planes of symmetry, only one quarter of the sheet needs to be modeled. The necessary boundary conditions are shown in the bottom left sketch on the previous page. Along the left and bottom edges of the model, symmetry stipulates that there shall be no displacements across the plane of symmetry. A coarse 20-element model using Element 3 is used. Element 3 is a four-noded linear plane stress element, with two translational DOFs at each node. A complete description of Element 3 is given in *MARC Volume B*. The origin is chosen to be at the center of the hole, that is, at the bottom left corner of the model. Since stress concentration is expected to be greatest in the vicinity of the hole, the FE mesh is intentionally refined near the hole to account for the anticipated steep stress gradient. (Alternatively, you could have selected Element 26, an 8-noded quadratic plane stress element–see *MARC Volume C and Volume E.*)

Properties

For an isotropic linear elastic material, only two independent quantities are needed to completely define the stress-strain behavior of the material: Young's modulus and Poisson's ratio. We'll show you how to input these later when the MODEL DEFINITION options are described. (Note that this is an isothermal example. If thermal loads were present, you would also have to specify the coefficient of thermal expansion so that MARC can calculate the thermal strain and the resulting thermal stresses.)

Loads

To simulate a tension load acting at infinity, a 1-psi edge pressure is applied to the top edge of the model. This pressure is applied using the DIST LOADS block of the MODEL DEFINITION sections. For MARC Element 3, distributed loads are considered positive into the element. Therefore, applying a tension load at the edge means you need to specify a value of -1.00 psi here.

Special Features

Use of the OPTIMIZE MODEL DEFINITION option allows us to optimize the node numbering of an FE mesh and minimize the bandwidth and solution time. In this example, you will use by default the Cuthill-McKee algorithm, which is extensively used in FE codes and will result in nearly optimal node numbering in most cases. (There are six other bandwidth minimization choices, including the Grooms algorithm and several wavefront schemes–see *MARC Volume C* for details.)

Input

A complete input echo from the printout is included for this first example. We'll discuss significant items about the input of the PARAMETER and MODEL DEFINITION options. (For this example, the HISTORY DEFINITION section is not necessary since this is a linear elastic problem and no MARC mesh or post plotting were requested.)

PARAMETER Section

Only four PARAMETER options are needed for this linear elastic analysis: TITLE, SIZING, ELEMENTS, and END. The formats and options of these lines are described in *MARC Volume C*. For this example, the title "Elastic Analysis of a Thin Sheet with Hole" is chosen for the "TITLE" line. This title will appear throughout the output listing.

The "SIZING" line specifies the size of the workspace buffer in number of words. A value of 100,000 words in the "SIZING" line is usually sufficient for most problems. For a larger problem, try a size of 300,000.

NOTE

Check this value with your in-house MARC expert. You may refer to Volume C, Tables c2.1-1 and c2.1-2 after the SIZING line description in order to establish an estimate of the work space required for your problem. Remember, this estimate is computer-dependent and is only an approximation, since MARC will adjust the variables to use some out-of-core storage if necessary.

The "ELEMENTS" line names Element 3 to be the selected element type.

The last line which completes the PARAMETER section is the "END" line.

MODEL DEFINITION Section

The MODEL DEFINITION options contain the FE model data for the analysis. In this case, they make up the remainder of the input file. In this example, the data represents:

1. the FE mesh topology (element connectivity, nodal coordinates, and sheet thickness);
2. material properties (Young's modulus, Poisson's ratio);
3. pressure loading and prescribed boundary conditions; and
4. bandwidth optimization and output controls.

An index of MODEL DEFINITION keywords may be found in *MARC Volume C*.

FE Mesh Topology

The FE model in this example is defined by three blocks: CONNECTIVITY, COORDINATES, and GEOMETRY. (These blocks are described in *MARC Volume C*.) All the topology data was generated using Mentat II.

This mesh consists of 20 elements and 30 nodes. A typical "CONNECTIVITY" line is illustrated by the first line which defines the element connectivity for element 1, i.e.:

```
1    3    2    1    8    6
```

where the first "1" indicates element number 1, "3" means element type 3, and "2 1 8 6" are the four nodes (specified counterclockwise) defining the element.

A typical "COORDINATES" line is the first one:

```
1      1.00000      0.00000
```

which says node 1 has an X-coordinate of 1.0 and a Y- coordinate of 0.0 in the global coordinate system

Finally, the sheet thickness of 0.1 in. is entered through the GEOMETRY block. The first field of the third line in this block (after the blank line) is set to be 0.1. The next line "1 to 20" means the 0.1 thickness value applies to all twenty elements in the mesh.

Material Properties

In this linear elastic example, the only data required are Young's modulus and Poisson's ratio. The same material is used for the whole mesh (elements 1 to 20). For input of material properties, you use the ISOTROPIC MODEL DEFINITION block (see *MARC Volume C*). The blank line means you do not need to count how many data lines follow in this block. The "1" on the next line means element material identifier 1. Young's modulus (30×10^6 psi), Poisson's ratio (0.3) and a proportional limit strength of 50,000 psi are entered on the next line. The last line in the block ("1 to 20") designates these properties to be applicable to all twenty elements in the mesh.

Pressure Loading and Prescribed Displacement Boundary Conditions

The uniform distributed pressure loading acts on two elements (18 and 20) at the top of the model, along a line defined by nodes 27-29-30. If you refer to the detailed description of Element 3 in *MARC Volume B*, you will see that this line represents the 3-4 face of elements 18 and 20, and the appropriate Load Type for this face is 8. Also, a positive value means that the pressure is acting toward the element (compression). Accordingly, after the DIST LOADS header (and a blank line), the next line indicates the Load Type is 8 and the uniform distributed pressure is -1.0 psi (with the negative sign meaning the pressure is tensile or acting away from the elements). The last line shows that the pressure acts only on elements 18 and 20 of the mesh.

The next block named FIXED DISP denotes nodes which have kinematic constraints due to either physical constraints or symmetry conditions. After the blank line, the "0.0" line followed by the "1" line means that a zero value is prescribed for the first DOF (or X-translation) of the nodes to be specified. The next line names the six nodes lying on the Y-axis to which this applies: 5, 10, 15, 20, 22, and 27. In other words, these six nodes act as if they have been placed on rollers. Only translations along the vertical (Y) axis are permitted; there can be no displacements in the horizontal (X) direction. In the same manner, the second DOF (Y-translation) is suppressed for the six nodes lying on the X-axis: 1, 8, 13, 19, 21, and 26. With these boundary conditions, we have now completed the specification of the "symmetry" boundary conditions for this problem. Remember, in static analysis, the specified boundary conditions must remove all rigid-body modes from the analysis.

Post-Processing

The POST block (see *MARC Volume C*) tells MARC that a post-processor file is to be written for later post-processing by Mentat II. The "3," line means three element variables are to be written to a file. Then, the next three lines ("11," "12," and "13,") denote the post-code numbers assigned for the first (SIGMA XX), second (SIGMA YY), and third (SIGMA XY) components of stress. (SIGMA XX and YY are the normal stresses in the X- and Y- directions; SIGMA XY is the shear stress.)

Bandwidth Minimization

The OPTIMIZE block (see *MARC Volume C*) switches on the bandwidth optimization procedures in MARC. This option will reduce computing costs in large problems, although optimization is not critical in this small problem. OPTIMIZE creates an internal numbering scheme different from your node numbering, but all data input and output will still be in your node numbering scheme. We recommend that you use OPTIMIZE in every analysis. In this example, since we did not explicitly select one of the seven available optimization options, the default Cuthill-McKee scheme will be used. The "3," line shown after OPTIMIZE means we want MARC to try a maximum of three different node numbering schemes, then choose the one which results in the lowest bandwidth. (A number between 10 and 20 is appropriate for most analyses.)

Output Controls

In order to reduce the amount of printout, the PRINT ELEM option (see *MARC Volume C*) indicates that we want to selectively print element quantities for a number of elements and integration points. The "STRESS STRAIN" line means we want the total stress and total strain printed. Other element quantities we could print include:

- PLASTIC strain
- CREEP strain
- THERMAL strain
- CRACKing strain

- strain ENERGY
- CAUCHY stress
- STATE variables
- ALL of the above

(See *MARC Volume C* for more information.)

The first "1 TO 4" line that follows selects elements 1 to 4 (the row closest to the hole) as the elements for which we want printed output. The second "1 TO 4" line denotes the list of integration points for which we want results, which happens to be all the integration points for Element type 3.

The "END OPTION" line terminates the MODEL DEFINITION section, and in this example also ends the input file.

Output

The complete printout for this example follows this section. After the page giving the MARC logo, MARC version number, and office addresses and telephone numbers, an echo of the input appears. These are followed by a table which summarizes the parameters and sizing options. You should always look over the contents of this table to make sure the data and flags are correct.

At this point, MARC allocates core for input of the MODEL DEFINITION data and assembly of the element stiffness matrix. It prints out a heading line

> KEY TO STRESS, STRAIN, AND DISPLACEMENT OUTPUT

for each element type chosen. Column numbers which identify output quantities are referenced to the appropriate components of stress, strain, or displacement. Then, the required number of words in the workspace (23021 words in this case) for input and stiffness assembly is printed out. This message is followed by a list of the internal core allocation parameters, which reflect the maximum requirements imposed by the element type. (These element variables are different for each element type, and are repeated for each element type used in a given analysis.)

The next information message says

> RESIDUAL LOAD CORRECTION IS INVOKED

This is done automatically in the current MARC version, and is important only in nonlinear or dynamic problems. (The residual load is applied in MARC as a correcting force to ensure that equilibrium is satisfied, so that an accurate solution is obtained for nonlinear problems–see *MARC Volume A*.)

The next output segment shows how MARC interprets the MODEL DEFINITION data. The output displays the following groups of data sequence (as read from the input file):

- CONNECTIVITY

- COORDINATES
- GEOMETRY (geometric property–in this case, only the sheet thickness)
- ISOTROPIC material properties
- DIST LOADS
- FIXED DISP (giving a fixed boundary condition summary)
- POST variables (to be stored in a file after the analysis)
- OPTIMIZE algorithm (Cuthill-McKee, 3 iterations to be attempted)
- PRINT ELEM option
- END OPTION

Next comes the bandwidth minimization results from using the OPTIMIZE option. The program first informs you that the

MAXIMUM CONNECTIVITY IS 6 AT NODE 9

and that 21954 words are needed in the workspace for optimizing. Since the Cuthill-McKee algorithm minimized the mean bandwidth, the messages printed relate to the number of entries in the profile. The correspondence table between user nodes and internal nodes is printed.

After the bandwidth calculation and optimization, the program assigns the necessary workspace for the in-core solution of this stiffness matrix (in this case, 24709 words). If the workspace allocated in the "SIZING" line was insufficient, it will attempt to allocate workspace for an out-of-core solution.

NOTE

Workspace numbers will be different for different computers; the numbers shown above are typical.

MARC then calculates the loading and sums the load applied to each DOF for distributed loads and point loads. This load summary information is useful for checking the total loads in the different DOFs of the model.

The program prints out the time (0.47 sec.) at the start of matrix assembly, measured from the start of the job. It then shows the time at the start of the matrix solution (0.55). (If the out-of-core solver is used, a graphic figure representing the profile of the global stiffness matrix is shown.) Next, the singularity ratio gives you an estimate of the condition number of the matrix. The value for this example is 2.9588E-01, which is acceptable. (As discussed in Chapter 1, if the singularity ratio is on the order of the accuracy of the machine–10^{-14} for 64 bits–the equations may be considered singular and the solution unreliable.) For nonlinear problems, changes in the singularity ratio from increment to increment will reflect approaching instabilities.

The program then prints out the time at the end of the matrix solution (0.56). This corresponds to the time at the end of the matrix triangularization.

At this stage, the program calculates the displacements at the nodes by performing back substitution, followed by computation of element strains and stresses. (If you do not input a value, a default yield stress of 1×10^{20} will be set by the program for a linear elastic analysis.) Since this example is linear elastic, you will only get results for increment 0. The element stresses and strains are preceded by a heading. This heading contains:

- TRESCA INTENSITY (used for ASME code applications)
- MISES INTENSITY (equivalent yield stress/strain)
- MEAN NORMAL INTENSITY
- PRINCIPAL VALUES (MINIMUM, INTERMEDIATE, and MAXIMUM)
- PHYSICAL COMPONENTS (columns 1 to 6–with only the first three used in this example, representing the normal stress/strain values in the X- and Y- directions and the shear value)

The stress and strain results are followed by the nodal incremental and total displacements, which in this linear elastic example are, of course, identical. A printout of the reaction forces follows. The final item is an indication of the magnitude of the distributed load and the computational time.

The message

```
END OF INCREMENT 0
```

signifies the end of the analysis for increment 0. MARC informs you that binary post data has been written to Unit 16. The job has now ended. MARC exit number 3004 is the normal exit for a successful run.

NOTE

The times printed are dependent on both the problem size and computer type. Using bandwidth optimization and the in-core options reduces both computational time and wall time.

input data

```
              5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
             ---------------------------------------------------------------------------------
             TITLE          ELASTIC ANALYSIS OF A THIN PLATE WITH HOLE
             SIZING        100000                                                  PARAMETER options
             ELEMENTS       3
             END
card     5   COMMENT, USE CONNECTIVITY OPTION TO DEFINE ELEMENT CONNECTIVITY.
                 CONNECTIVITY

                    1    3    2    1    8    6
                    2    3    3    2    6    9
    card   10       3    3    4    3    9    7
                    4    3    5    4    7   10
                    5    3    6    8   13   11
                    6    3    9    6   11   14
                    7    3    7    9   14   12
    card   15       8    3   10    7   12   15                                  MODEL DEFINITION options
                    9    3   14   11   16   18
                   10    3   12   14   18   17
                   11    3   11   13   19   16
                   12    3   15   12   17   20
    card   20      13    3   16   19   21   23
                   14    3   20   17   24   22
                   15    3   18   16   23   25
                   16    3   17   18   25   24
                   17    3   23   21   26   28
    card   25      18    3   22   24   29   27
                   19    3   25   23   28   30
                   20    3   24   25   30   29
                COMMENT, USE COORDINATES OPTION TO DEFINE NODAL COORDINATES.
                COORDINATES
    card   30
                   1    1.00000    0.00000
                   2    0.92381    0.38247
                   3    0.70700    0.70700
                   4    0.38247    0.92381
    card   35      5   0.000000    1.00000
                   6    1.10190    0.45623
                   7    0.45623    1.10190
                   8    1.25000    0.00000
                   9    0.88350    0.88350
                ---------------------------------------------------------------------------------
                   5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
   10  0.000000   1.25000
   11   1.28000   0.53000
   12   0.53000   1.28000
   13   1.50000   0.00000
   14   1.06000   1.06000
   15  0.000000   1.50000
   16   1.70000   0.70000
   17   0.70000   1.70000
   18   1.40000   1.40000
   19   2.00000   0.00000
   20   0.00000   2.00000
   21   3.50000   0.00000
   22  0.000000   3.50000
   23   3.35000   1.60000
   24   1.60000   3.35000
   25   3.20000   3.20000
   26   5.00000   0.00000
   27  0.000000   5.00000
   28   5.00000   2.50000
   29   2.50000   5.00000
   30   5.00000   5.00000
COMMENT, GEOMETRY OPTION SPECIFIES THICKNESS OF PLANE STRESS ELEMENT 3.
GEOMETRY

   0.10000   0.00000   0.00000   0.00000   0.00000   0.00000
    1 TO      20
COMMENT, ISOTROPIC OPTION SPECIFIES MATERIAL DATA FOR ALL ELEMENTS.
ISOTROPIC

    1
 0.30000+8   0.30000 0.0       0.0       50000.
    1 TO     20
COMMENT, DIST LOADS OPTION SPECIFIES UNIT TENSILE STRESS ON 3-4 FACE
COMMENT,  OF ELEMENTS AT Y=5.0
DIST LOADS

    8  -1.00000
   18   20
COMMENT, BOUNDARY CONDITIONS ARE ALL DUE TO SYMMETRIC MODEL OF ONE-QUARTER
COMMENT,   OF THE PLATE.
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
FIXED DISP

  0.0
    1
    5   10   15   20   22   27
  0.0
    2
    1    8   13   19   21   26
COMMENT,  POST CODE 11 = 1ST COMPONENT OF STRESS (SIGMA XX)
COMMENT,  POST CODE 12 = 2ND COMPONENT OF STRESS (SIGMA YY)
COMMENT,  POST CODE 13 = 3RD COMPONENT OF STRESS (SIGMA XY)
POST
3,
11,
12,
13,

COMMENT,  ALWAYS USE OPTIMIZE TO REDUCE MEMORY USAGE AND SOLUTION TIME!
OPTIMIZE
3,
COMMENT, PRINT ELEM IS USED TO LIMIT OUTPUT OF ELEMENT INTEGRATION POINT
COMMENT,  QUANTITIES.  ABSENCE OF PRINT NODE MEANS WE GET ALL NODAL OUTPUT.
PRINT ELEM

STRESS STRAIN
1 TO 4
1 TO 4                          End of MODEL DEFINITION section
END OPTION
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
--------------------------------------------------------------------------------
```

Interpretation of Input Data

```
 ************************************************
 ************************************************

  program sizing and options requested as follows



  element type requested************************      3
  number of elements in mesh********************     20
  number of nodes in mesh***********************     30
  max number of elements in any dist load list***     2
  maximum number of boundary conditions**********    12
  load correction flagged or set*****************
  number of lists of distributed loads***********     3
  stresses stored at all integration points******
  tape no.for input of coordinates + connectivity     5
  no.of different materials     1 max.no of slopes    5
  maximum elements variables per point on post tp    33
  number of points on shell section *************    11
  option for terminal debug**********************
  new style input format will be used************
  maximum number of set names is*****************    10
  number of processors used *********************     1
  vector length used ****************************     1



  end of parameters and sizing
 ************************************************
 ************************************************


key to stress, strain and displacement output



         element type    3


4-node isoparametric plane stress


stresses and strains in global directions
  1=xx
  2=yy
  3=xy


displacements in global directions
  1=u global x direction
  2=v global y direction
```

```
          workspace needed for input and stiffness assembly     23021


          internal core allocation parameters
          degrees of freedom per node (ndeg)  2
          coords per node (ncrd)  2
          strains per integration point (ngens)  3
          max. nodes per element (nnodmx)  4
          max.stress components per int. point (nstrmx)   3
          max. invariants per int. points (neqst)  1


          flag for element storage (ielsto)  0
          elements in core, words per element (nelsto)       848
                           total space required          16960
          vectors in core, total space required       893


          words per track on disk set to 4096


          internal element variables


          internal element number  1  library code type  3
          number of nodes=  4
          stresses stored per integration point =  3
          direct continuum components stored =  2
          shear continuum components stored =  1
          shell/beam flag = 0
          curvilinear  coord. flag = 0
          int.points for elem. stiffness  4
          number of local inertia directions  2
          int.point for print if all points not flagged  5
          int. points for dist. surface loads (pressure)  2
          library code type =  3
          no local rotation flag = 1
          generalized displ. flag = 0
          large disp. row counts     4    4    7



             residual load correction is invoked


 comment, use connectivity option to define element connectivity.
```

Useful Information for User Subroutines Used

Element Data Stored In-Core

```
 connectivity
----------


 meshr1,iprnt
     5       0
 elem no., type,        nodes
     1        3         2    1    8    6
     2        3         3    2    6    9
     3        3         4    3    9    7
     4        3         5    4    7   10
     5        3         6    8   13   11
     6        3         9    6   11   14
     7        3         7    9   14   12
     8        3        10    7   12   15
     9        3        14   11   16   18
    10        3        12   14   18   17
    11        3        11   13   19   16
    12        3        15   12   17   20
    13        3        16   19   21   23
    14        3        20   17   24   22
    15        3        18   16   23   25
    16        3        17   18   25   24
    17        3        23   21   26   28
    18        3        22   24   29   27
    19        3        25   23   28   30
    20        3        24   25   30   29
 comment, use coordinates option to define nodal coordinates.

 coordinates
----------


 ncrd1 ,meshr1,iprnt
     2       5       0
  node       coordinates
     1  1.0000           0.
     2 0.92381      0.38247
     3 0.70700      0.70700
     4 0.38247      0.92381
     5      0.       1.0000
     6  1.1019      0.45623
     7 0.45623       1.1019
     8  1.2500           0.
     9 0.88350      0.88350
```

```
     10      0.      1.2500
     11  1.2800     0.53000
     12 0.53000      1.2800
     13  1.5000          0.
     14  1.0600      1.0600
     15      0.      1.5000
     16  1.7000     0.70000
     17 0.70000      1.7000
     18  1.4000      1.4000
     19  2.0000          0.
     20      0.      2.0000
     21  3.5000          0.
     22      0.      3.5000
     23  3.3500      1.6000
     24  1.6000      3.3500
     25  3.2000      3.2000
     26  5.0000          0.
     27      0.      5.0000
     28  5.0000      2.5000
     29  2.5000      5.0000
     30  5.0000      5.0000
 comment, geometry option specifies thickness of plane stress element 3.

 geometry
----------


   egeom1     egeom2     egeom3     egeom4     egeom5     egeom6
   0.100E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
 from element     1 to element    20 by     1
 comment, isotropic option specifies material data for all elements.

 isotropic
----------


 isotropic material  material id =     1
  von mises  yield criteria
  isotropic  hardening rule
    e          nu         rho        alpha       yield      yield2
   0.300E+08  0.300E+00  0.000E+00  0.000E+00  0.500E+05  0.500E+05
 from element     1 to element    20 by     1
 comment, dist loads option specifies unit tensile stress on 3-4 face
 comment,  of elements at y=5.0
```

```
dist loads
----------


read from unit     5
type index distributed load
    8    0 -0.1000000E+01  0.0000000E+00  0.0000000E+00
a list of elements given below
    18     20
comment, boundary conditions are all due to symmetric model of one-quarter
comment,   of the plate.


fixed disp
----------


fixed displacement =  0.000E+00  0.000E+00
a list of degrees of freedom given below
     1
a list of nodes given below
     5     10     15     20     22     27
fixed displacement =  0.000E+00  0.000E+00
a list of degrees of freedom given below
     2
a list of nodes given below
     1      8     13     19     21     26
comment,  post code 11 = 1st component of stress (sigma xx)
comment,  post code 12 = 2nd component of stress (sigma yy)
comment,  post code 13 = 3rd component of stress (sigma xy)




fixed boundary condition summary.
total fixed degrees of freedom read so far =   12

 b.c.     node    degree of  magnitude             b.c.     node    degree of  magnitude
number              freedom                       number              freedom



    1       5          1    0.000E+00                 2       10          1    0.000E+00
    3      15          1    0.000E+00                 4       20          1    0.000E+00
    5      22          1    0.000E+00                 6       27          1    0.000E+00
    7       1          2    0.000E+00                 8        8          2    0.000E+00
    9      13          2    0.000E+00                10       19          2    0.000E+00
   11      21          2    0.000E+00                12       26          2    0.000E+00
```

```
post
----------



*** note - format of post code cards has changed.
           in k4, enter code in first field and  layer number in second field


elem vars,post tape,prev tape,  type  , conn fl ,post tape, prev tape,  repost ,frequency, k2post
        3       16       17         0         1         0         0         0         1         0
element variables appear on post-processor tape 16 in following order
post variable   1 is post code   11 =
post variable   2 is post code   12 =
post variable   3 is post code   13 =


***maximum record length on binary post file=   30     approximate no. of words per increment on
file=     256
comment,  always use optimize to reduce memory usage and solution time!


optimize
----------



cuthill-mckee algorithm
comment, print elem is used to limit output of element integration point
comment,  quantities.  absence of print node means we get all nodal output.


print elem
----------




values will be printed at integration points


element quantities printed every    1 increments


stress strain
from element     1 to element     4 by     1
from integration point     1 to integration point     4 by     1


end option
----------
```

```
                              maximum connectivity is    6  at node    9
```

Information Regarding Non-Zero Values in Global Stiffness Matrix

```
                         workspace needed for optimizing =     21954
                         maximum sky-line including fill-in is      193 at try   0 (forward numbering)
                         maximum sky-line including fill-in is      194 at try   0 (backward numbering)
                         maximum sky-line including fill-in is      205 at try   1 (forward numbering)
                         maximum sky-line including fill-in is      181 at try   1 (backward numbering)
                         maximum sky-line including fill-in is      205 at try   2 (forward numbering)
                         maximum sky-line including fill-in is      181 at try   2 (backward numbering)
                         maximum sky-line including fill-in is      225 at try   3 (forward numbering)
                         maximum sky-line including fill-in is      193 at try   3 (backward numbering)


                        c o r r e s p o n d e n c e   t a b l e   f o r   n o d e s

  user, internal,   user, internal, etc

   1...   30    2...   29    3...   26    4...   21    5...   14    6...   27    7...   20    8...   28
   9...   25   10...   13   11...   23   12...   19   13...   24   14...   22   15...   12   16...   17
  17...   15   18...   16   19...   18   20...   11   21...   10   22...    6   23...    9   24...    7
  25...    8   26...    5   27...    1   28...    4   29...    2   30...    3


            maximum connectivity is    6  at node   15


            maximum half-bandwidth is   10  between nodes    9  and   18

            number of profile entries including fill-in is       181

            number of profile entries excluding fill-in is       119
```

Workspace required to keep matrix in core

```
          total workspace needed with in-core matrix storage =    24709


            load increments associated with each degree of freedom
            summed over the whole model

            distributed loads

  0.000E+00 5.000E-01
```

Sum of Distributed Load Magnitude

```
                   point loads
 0.000E+00 0.000E+00


                   start of assembly
                    time =       0.47




                   start of matrix solution
                    time =       0.55



                   singularity ratio    2.9588E-01



                   end of matrix solution
                    time =       0.56
```

```
MARC          output for increment    0.         elastic analysis of a thin plate with hole

element with highest stress relative to yield is    1  where equivalent stress is 0.538E-04  of yield
```

Yielding would occur here first

```
                tresca     mises    mean   p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
            intensity intensity  normal   minimum intermediate maximum      1         2         3         4         5         6
                                 intensity




element     1  point    1      integration pt. coordinate=     0.981E+00    0.314E+00
section thickness = 0.100E+00
stress  2.739E+00 2.584E+00 1.028E+00 0.000E+00 3.456E-01 2.739E+00 4.836E-01 2.601E+00-5.579E-01
strain  1.037E-07 7.289E-08 0.000E+00-1.587E-08 0.000E+00 8.785E-08-9.893E-09 8.187E-08-4.835E-08
```

Maximum equivalent stress

Maximum σ_{yy}

```
element     1  point    2      integration pt. coordinate=     0.103E+01    0.841E-01
section thickness = 0.100E+00
stress  2.827E+00 2.692E+00 1.040E+00 0.000E+00 2.933E-01 2.827E+00 2.989E-01 2.822E+00-1.182E-01
strain  1.098E-07 7.606E-08 0.000E+00-1.849E-08 0.000E+00 9.130E-08-1.825E-08 9.106E-08-1.024E-08

element     1  point    3      integration pt. coordinate=     0.109E+01    0.348E+00
section thickness = 0.100E+00
stress  2.043E+00 1.977E+00 7.268E-01 0.000E+00 1.378E-01 2.043E+00 2.461E-01 1.934E+00-4.411E-01
strain  8.255E-08 5.598E-08 0.000E+00-1.583E-08 0.000E+00 6.671E-08-1.114E-08 6.202E-08-3.823E-08

element     1  point    4      integration pt. coordinate=     0.117E+01    0.931E-01
section thickness = 0.100E+00
stress  2.214E+00 2.163E+00 7.736E-01 0.000E+00 1.068E-01 2.214E+00 1.085E-01 2.212E+00-5.976E-02
strain  9.131E-08 6.129E-08 0.000E+00-1.858E-08 0.000E+00 7.273E-08-1.851E-08 7.266E-08-5.179E-09

element     2  point    1      integration pt. coordinate=     0.790E+00    0.671E+00
section thickness = 0.100E+00
stress  1.749E+00 1.687E+00 6.268E-01 0.000E+00 1.317E-01 1.749E+00 3.997E-01 1.481E+00-6.013E-01
strain  7.007E-08 4.773E-08 0.000E+00-1.310E-08 0.000E+00 5.698E-08-1.486E-09 4.536E-08-5.211E-08

element     2  point    2      integration pt. coordinate=     0.916E+00    0.471E+00
section thickness = 0.100E+00
stress  1.832E+00 1.804E+00 6.304E-01 0.000E+00 5.892E-02 1.832E+00 2.188E-01 1.673E+00-5.079E-01
strain  7.685E-08 5.117E-08 0.000E+00-1.636E-08 0.000E+00 6.049E-08-9.432E-09 5.356E-08-4.402E-08

element     2  point    3      integration pt. coordinate=     0.892E+00    0.760E+00
section thickness = 0.100E+00
stress  1.494E+00 1.472E+00 5.129E-01 0.000E+00 4.479E-02 1.494E+00 2.568E-01 1.282E+00-5.121E-01
strain  6.279E-08 4.176E-08 0.000E+00-1.344E-08 0.000E+00 4.934E-08-4.257E-09 4.016E-08-4.438E-08

element     2  point    4      integration pt. coordinate=     0.102E+01    0.526E+00
section thickness = 0.100E+00
stress  1.585E+00 1.565E+00 5.006E-01-4.154E-02 0.000E+00 1.543E+00 7.731E-02 1.425E+00-4.174E-01
strain  6.868E-08 4.452E-08 0.000E+00-1.682E-08 0.000E+00 5.186E-08-1.167E-08 4.671E-08-3.618E-08
```

```
element     3  point   1        integration pt. coordinate=      0.471E+00     0.916E+00
section thickness = 0.100E+00
stress  8.027E-01 7.108E-01 9.867E-02-2.534E-01 0.000E+00 5.494E-01-1.616E-01 4.576E-01-2.554E-01
strain  3.478E-08 2.047E-08 0.000E+00-1.394E-08 0.000E+00 2.085E-08-9.962E-09 1.687E-08-2.214E-08

element     3  point   2        integration pt. coordinate=      0.671E+00     0.790E+00
section thickness = 0.100E+00
stress  7.717E-01 6.926E-01 1.211E-01-2.041E-01 0.000E+00 5.675E-01-1.251E-01 4.885E-01-2.339E-01
strain  3.344E-08 1.992E-08 0.000E+00-1.248E-08 0.000E+00 2.096E-08-9.056E-09 1.754E-08-2.027E-08

element     3  point   3        integration pt. coordinate=      0.526E+00     0.102E+01
section thickness = 0.100E+00
stress  7.869E-01 7.000E-01 1.068E-01-2.332E-01 0.000E+00 5.536E-01-1.385E-01 4.589E-01-2.561E-01
strain  3.410E-08 2.015E-08 0.000E+00-1.331E-08 0.000E+00 2.079E-08-9.205E-09 1.668E-08-2.219E-08

element     3  point   4        integration pt. coordinate=      0.760E+00     0.892E+00
section thickness = 0.100E+00
stress  7.603E-01 6.851E-01 1.260E-01-1.911E-01 0.000E+00 5.692E-01-1.085E-01 4.866E-01-2.367E-01
strain  3.295E-08 1.969E-08 0.000E+00-1.206E-08 0.000E+00 2.089E-08-8.481E-09 1.730E-08-2.051E-08

element     4  point   1        integration pt. coordinate=      0.841E-01     0.103E+01
section thickness = 0.100E+00
stress  8.859E-01 8.836E-01-2.922E-01-8.813E-01 0.000E+00 4.602E-03-8.657E-01-1.101E-02-1.166E-01
strain  3.839E-08 2.511E-08 0.000E+00-2.942E-08 0.000E+00 8.966E-09-2.875E-08 8.290E-09-1.010E-08

element     4  point   2        integration pt. coordinate=      0.314E+00     0.981E+00
section thickness = 0.100E+00
stress  7.771E-01 7.240E-01-2.996E-01-7.771E-01-1.216E-01 0.000E+00-7.454E-01-1.533E-01 1.406E-01
strain  2.841E-08 2.039E-08 0.000E+00-2.469E-08 0.000E+00 3.719E-09-2.332E-08 2.346E-09 1.218E-08

element     4  point   3        integration pt. coordinate=      0.931E-01     0.117E+01
section thickness = 0.100E+00
stress  6.837E-01 6.254E-01-1.343E-01-5.433E-01 0.000E+00 1.404E-01-5.054E-01 1.025E-01-1.564E-01
strain  2.963E-08 1.795E-08 0.000E+00-1.951E-08 0.000E+00 1.011E-08-1.787E-08 8.471E-09-1.356E-08

element     4  point   4        integration pt. coordinate=      0.348E+00     0.109E+01
section thickness = 0.100E+00
stress  3.664E-01 3.643E-01-1.193E-01-3.622E-01 0.000E+00 4.238E-03-3.500E-01-8.002E-03 6.584E-02
strain  1.588E-08 1.036E-08 0.000E+00-1.212E-08 0.000E+00 3.763E-09-1.159E-08 3.233E-09 5.706E-09
```

```
                       n o d a l   p o i n t   d a t a


              i n c r e m e n t a l   d i s p l a c e m e n t s


    1 -3.51778E-08       0.             2 -3.37059E-08  3.87785E-08          3 -2.48754E-08  7.20814E-08
    4 -1.33285E-08  9.52174E-08         5       0.      1.02000E-07          6 -3.44590E-08  3.20395E-08
    7 -7.49801E-09  9.37428E-08         8 -4.03733E-08       0.              9 -1.94431E-08  6.47509E-08
   10       0.      1.04522E-07        11 -3.46280E-08  2.99026E-08         12 -5.51577E-09  9.34625E-08
   13 -4.25317E-08       0.            14 -1.83576E-08  6.37780E-08         15       0.      1.06861E-07
   16 -3.61221E-08  3.13129E-08        17 -5.38301E-09  9.77375E-08         18 -2.04133E-08  6.91158E-08
   19 -4.53935E-08       0.            20       0.      1.14470E-07         21 -5.68953E-08       0.
   22       0.      1.53978E-07        23 -4.63700E-08  5.71663E-08         24 -1.37371E-08  1.37475E-07
   25 -3.22167E-08  1.15603E-07        26 -7.04721E-08       0.             27       0.      2.00172E-07
   28 -5.72335E-08  7.78845E-08        29 -1.24705E-08  1.83980E-07         30 -3.69232E-08  1.61221E-07


              t o t a l   d i s p l a c e m e n t s


    1 -3.51778E-08       0.             2 -3.37059E-08  3.87785E-08          3 -2.48754E-08  7.20814E-08
    4 -1.33285E-08  9.52174E-08         5       0.      1.02000E-07          6 -3.44590E-08  3.20395E-08
    7 -7.49801E-09  9.37428E-08         8 -4.03733E-08       0.              9 -1.94431E-08  6.47509E-08
   10       0.      1.04522E-07        11 -3.46280E-08  2.99026E-08         12 -5.51577E-09  9.34625E-08
   13 -4.25317E-08       0.            14 -1.83576E-08  6.37780E-08         15       0.      1.06861E-07
   16 -3.61221E-08  3.13129E-08        17 -5.38301E-09  9.77375E-08         18 -2.04133E-08  6.91158E-08
   19 -4.53935E-08       0.            20       0.      1.14470E-07         21 -5.68953E-08       0.
   22       0.      1.53978E-07        23 -4.63700E-08  5.71663E-08         24 -1.37371E-08  1.37475E-07
   25 -3.22167E-08  1.15603E-07        26 -7.04721E-08       0.             27       0.      2.00172E-07
   28 -5.72335E-08  7.78845E-08        29 -1.24705E-08  1.83980E-07         30 -3.69232E-08  1.61221E-07


       total equivalent nodal forces (distributed plus point loads)


    1      0.          0.              2      0.          0.               3      0.          0.
    4      0.          0.              5      0.          0.               6      0.          0.
    7      0.          0.              8      0.          0.               9      0.          0.
   10      0.          0.             11      0.          0.              12      0.          0.
   13      0.          0.             14      0.          0.              15      0.          0.
   16      0.          0.             17      0.          0.              18      0.          0.
   19      0.          0.             20      0.          0.              21      0.          0.
   22      0.          0.             23      0.          0.              24      0.          0.
   25      0.          0.             26      0.          0.              27      0.      0.12500
   28      0.          0.             29      0.      0.25000             30      0.      0.12500


       reaction forces at fixed boundary conditions, residual load correction elsewhere
```

```
         1 -3.07100E-17 -3.83602E-02           2  5.20417E-17       0.                  3 -1.56125E-17  1.14492E-16
         4 -8.67362E-19 -1.02891E-16           5  1.32805E-02  1.16687E-16            6  8.71699E-17  1.38778E-17
         7 -2.68882E-17 -2.66280E-16           8  4.51028E-17 -5.26883E-02            9 -9.19403E-17  1.24900E-16
        10  1.08043E-02 -1.81658E-16          11 -8.23994E-17  1.04083E-17           12 -1.99493E-17 -2.60209E-16
        13  3.46945E-17 -5.88250E-02          14 -1.73472E-18  4.16334E-17           15  4.42958E-03 -1.45717E-16
        16 -1.25767E-17 -8.32667E-17          17 -2.08167E-17 -3.12250E-17           18 -5.20417E-17  2.77556E-17
        19 -1.38778E-17 -0.12232              20 -2.85062E-03  3.46945E-18           21 -1.21431E-17 -0.15979
        22 -9.43531E-03  2.77556E-17          23  8.28330E-17 -1.24900E-16           24 -1.19696E-16  1.80411E-16
        25  7.11237E-17  1.38778E-16          26 -4.33681E-17 -6.80118E-02           27 -1.62284E-02  5.55112E-17
        28  1.17094E-17  4.16334E-17          29  3.98986E-17       0.                30 -1.25767E-17       0.

                   summary of externally applied loads

 0.00000E+00    0.50000E+00

                   summary of reaction/residual forces

-0.98933E-17   -0.50000E+00



              distributed load    type    current
              list number                 magnitude


                      1             8     -1.000          0.          0.

                   e n d   o f   i n c r e m e n t   0

                   binary post data at increment   0.   0   on tape 16
                    time =       0.80



                   *** end of input deck - job ends



                            marc exit number 3004
```

Results

R. E. Peterson's book *Stress Concentration Factors in Design* (Wiley-Interscience, 1953) gives an analytical value for the stress concentration factor of 3.14 in this problem.

At Gauss point 2 of element 1, the maximum stress raising factor calculated using this model of 20 four-noded elements is:

$$K = (\sigma_{YY}) / (\sigma_{YY} \text{ without hole}) = 2.822/1.0 = \mathbf{2.822}$$

Mentat II extrapolates the Gauss point values to nodal stresses. At node 1, the point of maximum stress concentration according to theory, such an extrapolation produces an approximate stress concentration factor of:

$$K = (\sigma_{yy} \text{ at node 1}) = (\sigma_{yy} \text{ without hole}) = 3.135/1.0 = \mathbf{3.135}$$

as shown in the following banded contour plot of σ_{yy}. Thus, the extrapolated nodal value for K is almost exactly the same as the analytical value.

Use of a finer mesh and/or a higher-order finite element will, of course, give a more accurate answer. For instance, a more refined mesh using 20 eight-noded elements (Element 26) gives:

K = 3.137 (-0.96% off analytical value)

at the same nodal location (see *MARC Volume C*).

In summary, Example 1 illustrated how to set up a simple, linear static, 2-D plane stress problem for analysis and how to interpret the MARC printed output. In addition, mesh, optimization, stress concentration, and symmetry considerations have been discussed.

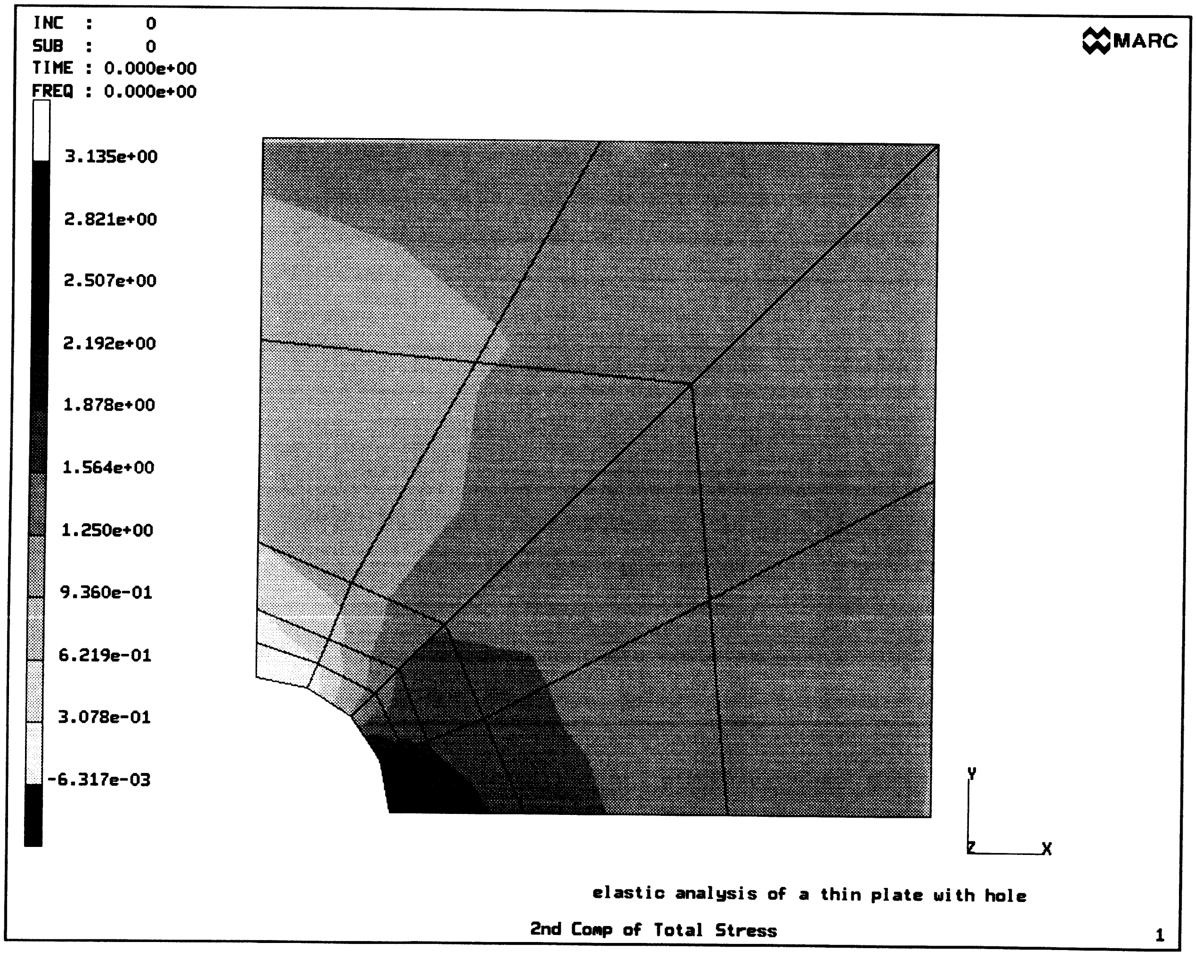


Figure 1.2 Second Component of Stress

Example 2A

Thick Cylinder Under Internal Pressure–Plane Strain Solution

A thick-walled cylinder under internal pressure is a common problem in strength of materials as well as engineering applications. This example will demonstrate the elastic analysis of such a thick cylinder using two alternative models:

Example 2A: Plane Strain Solution
Example 2B: Axisymmetric Solution

This example illustrates the following:

- input preparation for a plane strain and an axisymmetric problem
- symmetry considerations
- the input concepts of element sets and node sets
- transformations for the application of boundary conditions at skewed edges
- the use of ORIENTATION for defining a material coordinate system with respect to the global coordinate system
- selective printout options such as PRINT NODE, NO PRINT, SUMMARY, ELEM SORT and NODE SORT.

Sketch

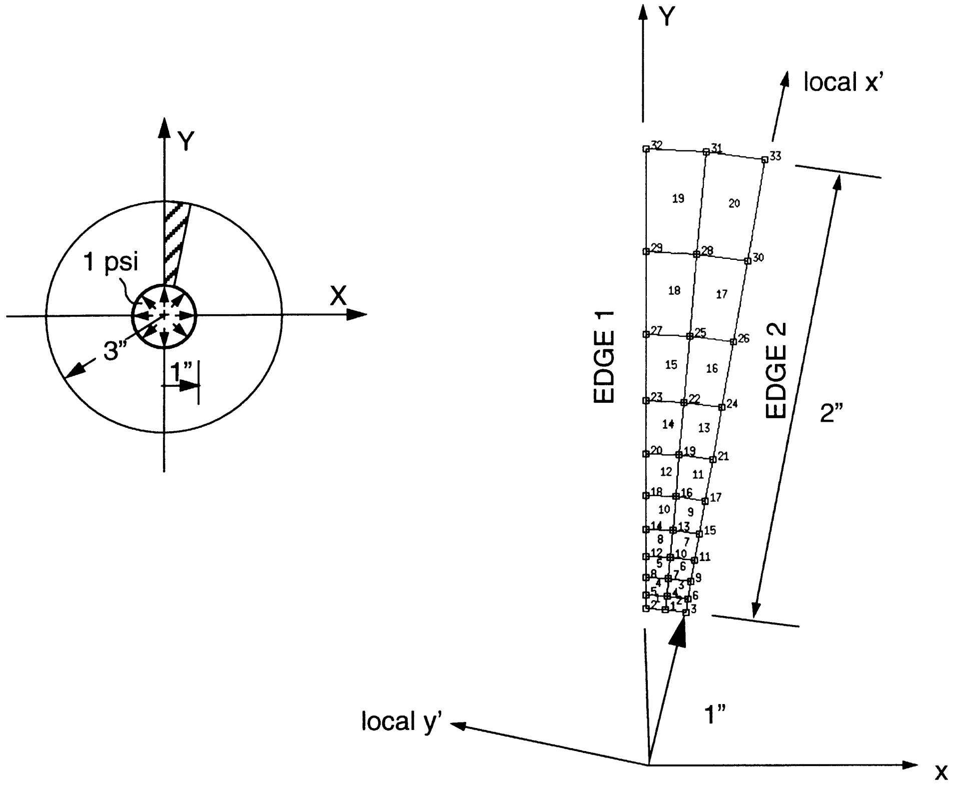


Figure 2A.1 **Thick Cylinder, Plane Strain Model**

Model

The idealized finite element model is a narrow wedge (10°) made of quadrilateral plane-strain elements (MARC Element 11). The cylinder has an inner radius of 1.00 in. and an outer radius of 3.00 in. The material properties are isotropic and linear elastic as in Example 1, with a Young's modulus of $30\text{x}10^6$ psi and Poisson's ratio of 0.3.

Because of axial symmetry, both the left and right edges of the wedge model are planes of symmetry, across which no displacements are permitted. A coarse 20-element model using Element 11 is used, with a total of 33 nodes. Element 11 is a four-noded linear plane strain element, with two translational DOFs at each node and four Gaussian integration points. A complete description of Element 11 is given in *MARC Volume B*. The origin of the global coordinate system (X,Y) is at the center of the cylinder. Since both radial and tangential stresses are expected to be greatest near the inner edge where the pressure is applied, the mesh is made more refined there. (Alternatively, we could have selected Element 6, a 3-noded triangular plane strain element with 2 DOFs at each node and one Gauss point, or Element 27, an 8-noded quadrilateral plane strain element.)

Properties

As in Example 1, for an isotropic linear elastic material, only two material properties are necessary to define the material completely: Young's modulus and Poisson's ratio. Also illustrated is the ORIENTATION option to transform the material coordinate system from the default element system to a user-defined system. This option is used here so that we can interpret the stresses in the radial and tangential directions, rather than in the global X- and Y-directions.

Loads

The uniform 1-psi pressure is applied to the inner edge of the model using the DIST LOADS block of the MODEL DEFINITION options. However, instead of naming the element numbers in describing the location of the pressure load, we have chosen to identify the inner two elements (numbered 1 and 2) as an element set named INNER.

Boundary Conditions

Both the left edge (node set EDGE1) and right edge (node set EDGE2) of the model are planes of symmetry. No displacements are allowed perpendicular to these two edges. The eleven nodes (3, 6, 9, 11, 15, 17, 21, 24, 26, 30, 33) along EDGE2 will be transformed into a local coordinate system x', y'.

Input

A complete input echo from the printout is included. The following paragraphs discuss items of interest in the PARAMETER and MODEL DEFINITION options. Note the use of both fixed and free-field input formats. (As in Example 1, no HISTORY DEFINITION options are necessary, since this is a linear elastic problem and no MARC mesh or post plotting was requested.)

PARAMETER Section

The “TITLE” line is self-explanatory.

The “SIZING” line sets 100,000 words as the workspace.

The “ELEMENTS” line says MARC element type 11 will be used in the model. The “END” line terminates the input of the PARAMETER section.

MODEL DEFINITION Section

The MODEL DEFINITION options constitute the remainder of the input. The bulk data in this example consist of:

- the FE mesh topology (element connectivity, nodal coordinates, definition of element sets and node sets);
- material properties;
- pressure loading and prescribed boundary conditions (in this case, making use of the TRANSFORMATION option); and
- output controls.

FE Mesh Topology

The FE mesh in this example is defined by three blocks:

- CONNECTIVITY
- COORDINATES
- DEFINE

The CONNECTIVITY and COORDINATES data (shown in fixed-field input format) define the element connectivities and nodal coordinates as in Example 1. Again, all the topology data were generated using Mentat II. The first line after the CONNECTIVITY option shows a "20", which is an optional value representing the number of elements to be read in and defaults to the total number of elements in the mesh. (A blank line could also have been used as in Example 1. *See MARC Volume C for a complete description of the CONNECTIVITY option.* Likewise, the line following the COORDINATES option shows a "2", which represents the number of coordinates per node, and a "33", which is the number of nodes read in (optional, defaults to the number of nodes in the mesh). Again, a blank line could have been used here as well, as in Example 1.

The DEFINE option (*See MARC Volume C*) is a powerful and convenient MODEL DEFINITION option which allows you to define a set name and to associate members to the set. Each set consists of one type of these entities:

- ELEMENT
- NODE
- INT (integration points)
- LAYER (beam or shell layers)
- DOF (degrees of freedom)
- INCS (increment numbers)

A set name may have up to 12 characters in length. Once certain members are named to a set, the set name can be conveniently used over and over again, such as in prescribing loads, boundary conditions, material properties, and selective printout options.

In this example, we used DEFINE ELEMENT SET and DEFINE NODE SET twice. First, the "DEFINE ELEMENT SET ALLE" line and the "1 TO 20" line placed all twenty elements into an element set name ALLE (which we will use later in applying material properties and defining orientations). Next, the "DEFINE ELEMENT SET INNER" line and the following line "1 TO 2" placed elements 1 and 21 into an element set named INNER (which is later used in prescribing internal pressure on the inner edge of these two elements and in selective printout of stress and strain results). In a similar manner, we named the node set EDGE2 to consist of the eleven nodes lying on the right edge of the model, and the node set EDGE1 to be the eleven nodes on the left edge. These two node sets are subsequently used in the TRANSFORMATION option.

NOTE

A set must be DEFINE'd before it can be referenced.

Notice we did not specify the thickness of the model this time. For an analysis using plane strain elements (such as MARC Elements 6, 11, and 27), the default element thickness is unity. Recall that a plane strain model typically represents a 2-D (X,Y) slice of a long structure (such as this thick-walled cylinder), where the displacements and strains associated with the Z-direction may be assumed to be zero.

Material Properties

The material properties of the wedge model are entered using the ISOTROPIC option. *See MARC Volume C.* The only properties required for this linear elastic analysis are Young's modulus and Poisson's ratio. The blank line following "ISOTROPIC" means you don't have to count how many lines are in this block. The 1, following the blank line associates these properties with material id number 1. Young's modulus is 30E6 and Poisson's ratio is 0.3. The last line in this block says these properties are applicable to all the elements (element set ALLE).

Pressure Loading and Prescribed Displacement Boundary Conditions

The uniform internal pressure acts on the inner edge of the two innermost elements (element set INNER). The DIST LOADS option is used as in Example 1. Again, a blank line is used; you don't have to give a count of how many lines follow. The detailed description of Element 11 in Volume B shows that the inner edge corresponds to the 4-1 face of these two elements, and the appropriate Load Type for this face is 10. Positive pressure means the pressure acts toward the element. The last line in this block shows that the 1.0 psi pressure acts on element set INNER.

The next block is TRANSFORMATION, an option which defines nodal coordinates for calculation of a direction cosine matrix, which is then used for transforming the global DOFs of a specified node to a new local coordinate system. *See MARC Volume C.* After the blank line, the three zeroes refer to the global (xyz) coordinates of the first point used to define the local (1,2) plane. As seen in the sketch, the local x'-axis points from its origin at (0,0,0) up the included EDGE2, and the local y'-axis is perpendicular to EDGE2. In the first field of this line, instead of naming a node number, we left the field blank and began the data with a 0. The next line ("EDGE2") then tell MARC all the node numbers in node set EDGE2 for which this transformation applies.

The FIXED DISP block follows. *See MARC Volume C.* After the blank line, the "0.," line followed by the "1," line means that a zero value is prescribed for the first DOF (or X-translation) of the nodes to be specified. The next line names the nodes by using the node set EDGE1 (i.e., nodes 2, 5, 8, 12, 14, 18, 20, 23, 27, 29, and 32). In other words, the left edge (EDGE1) of the model has been placed on rollers; only translations along the global Y-axis are permitted. Then, in the same manner, the "0.," line followed by the "2," and "EDGE2" lines indicate we are restraining the second DOF (local y'-translation, or perpendicular to the inclined EDGE2) of the eleven nodes of EDGE2.

NOTE

Prescribed nodal boundary conditions are given with respect to the local system defined by the TRANSFORMATION option.

Then comes the ORIENTATION block, which allows you to define the orientation of the element material system with respect to the global coordinate system. *See MARC Volume C.* Use of ORIENTATION combined with the PREF (preferred system) option in PRINT ELEM lets you print physical stress output in the preferred material system defined by the ORIENTATION option. After the blank line, the "EDGE 1-2, 0.," line specifies the orientation angle type to be EDGE 1-2 and the orientation angle to be zero. This means that the material system will be parallel to EDGE 1-2 for all elements. The last line ("ALLE") means that all twenty elements are designated by element set ALLE are associated with this orientation angle. This is done so that we can examine the stresses in a cylindrical system as opposed to a Cartesian system.

Output Controls

The POST block creates a post-processor tape for later post-processing by Mentat II. The next line "2,,,1" is the second line series in the POST block. The 2 means the number of element variables to be written to tape; the 1 in the fourth field indicates that we want a formatted post tape. (A formatted post tape is often preferable because a binary tape is system-dependent and may not be portable to other computers.) The "12" on the third line refers to the post code assigned for the second component of stress (global σ_{YY}). The "111" on the next line refers to the post code assigned to the first component of stress in the local material system defined by the ORIENTATION option; this stress component is like a radial stress. A complete description of the POST option is given in *MARC Volume C.*

The PRINT ELEM option allows us to reduce printout quantity by selectively printing certain quantities for certain elements. *See MARC Volume C.* The "STRESS,STRAIN,PREF" line after the blank line indicates we would like to output the total stress and total strain, as well as the stress in the preferred system (labeled STRESSP in the output). The "INNER" line names the elements (1,2) for which we want results printed, and the "1 TO 4" line means we would like the results at all four integration points in each element.

The PRINT NODE option is used for the first time. It lets us choose which nodes and what nodal quantities are to be printed. *See MARC Volume C*. We could select any of these nodal quantities:

- INCR (incremental displacement)
- TOTA (total displacement)
- VELO (velocity)
- ACCE (acceleration)
- LOAD (total applied load)
- REAC (reaction/residual force)
- TEMP (temperature)
- FLUX
- STRESS (average generalized stresses at nodes)
- VOLT (voltage, in Joule heating analysis)
- PRES (pressure, in bearing analysis)
- COOR (coordinates)
- ALL (all relevant quantities)

Here, we selected TOTAL,LOAD,REACTION as the three nodal quantities we want printed, for nodes 1 TO 33. Notice in this line, as well as in the "STRESS,STRAIN,PREF" line before, a comma may be used as a delimiter instead of a blank. The "END OPTION" line terminates the input for this example.

Output

The last two pages of the printout are shown, representing the element stress and strain output for elements 1 and 2 and the requested nodal results (total displacements, nodal loads, reactions).

The element stress and strain format appears the same as that of Example 1, except for the line labeled "STRESSP" below the line for "STRESS". This "STRESSP" line, of course, gives the stresses in the preferred system as designated in the ORIENTATION data. Remember that in the local coordinate system for this problem, the first physical component of stress in the "STRESSP" line is parallel to the inclined edge (or radial), and the second stress component is tangential (or hoop).

The nodal total displacements, loads, and reactions are in the same format as before. Note that for the nodes defined by EDGE2, the nodal results are given with respect to the local coordinate system.

```
                          i n p u t   d a t a

    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
-------------------------------------------------------------------------------------------------
TITLE,THICK CYLINDER USING PLANE STRAIN ELEMENT 11
SIZING,100000,
ELEMENTS      11
END
CONNECTIVITY
   20     0     0
    1    11     1     4     5     2
    2    11     3     6     4     1
    3    11     6     9     7     4
    4    11     4     7     8     5
    5    11     7    10    12     8
    6    11     9    11    10     7
    7    11    11    15    13    10
    8    11    10    13    14    12
    9    11    15    17    16    13
   10    11    13    16    18    14
   11    11    17    21    19    16
   12    11    16    19    20    18
   13    11    21    24    22    19
   14    11    19    22    23    20
   15    11    22    25    27    23
   16    11    24    26    25    22
   17    11    26    30    28    25
   18    11    25    28    29    27
   19    11    28    31    32    29
   20    11    30    33    31    28
COORDINATES
    2    33     0     0
    1 0.87156-1    0.99619
    2 0.0000000    1.00000
    3    0.17365   0.98481
    4 0.92398-1    1.05611
    5 0.0000000    1.06014
    6    0.18409   1.04404
    7 0.98950-1    1.13101
    8 0.0000000    1.13533
    9    0.19715   1.11808
   10    0.10714   1.22462
   11    0.21347   1.21063
-------------------------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
   12 0.0000000   1.22930
   13   0.11738   1.34165
   14 0.0000000   1.34677
   15   0.23386   1.32631
   16   0.13018   1.48793
   17   0.25936   1.47092
   18 0.0000000   1.49361
   19   0.14617   1.67078
   20 0.0000000   1.67716
   21   0.29124   1.65168
   22   0.16617   1.89934
   23 0.0000000   1.90660
   24   0.33108   1.87763
   25   0.19117   2.18504
   26   0.38088   2.16007
   27 0.0000000   2.19339
   28   0.22241   2.54217
   29 0.0000000   2.55188
   30   0.44313   2.51311
   31   0.26147   2.98858
   32 0.0000000   3.00000
   33   0.52094   2.95442
DEFINE    ELEMENT   SET       ALLE
    1 TO      20
DEFINE    ELEMENT   SET       INNER
    1 TO       2
DEFINE    NODE      SET       EDGE2
    3    6    9   11   15   17   21   24   26   30   33
DEFINE    NODE      SET       EDGE1
    2    5    8   12   14   18   20   23   27   29   32
ISOTROPIC

1,
30.E6,.3,
ALLE
DIST LOADS

10,1.,
INNER
COMMENT, TRANSFORM ANGLED EDGE OF WEDGE (NODE SET EDGE2) TO LOCAL
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
-------------------------------------------------------------------------------------------------
COMMENT,  COORDINATE SYSTEM NORMAL TO EDGE SO THAT BOUNDARY CONDITIONS
COMMENT,  MAY BE EASILY APPLIED.
TRANSFORMATION

0,0.,0.,0.,0.,0.,0.,
EDGE2
FIXED DISP

0.,
1,
EDGE1
0.,
2,
EDGE2
COMMENT, DEFINE A LOCAL COORDINATE SYSTEM AT EACH INTEGRATION POINT
COMMENT,  PARALLEL TO EDGE 1-2 OF THE ELEMENT.  THIS COORDINATE SYSTEM
COMMENT,  APPROXIMATES A TRUE CYLINDRICAL COORDINATE SYSTEM.  THE ERROR
COMMENT,  IN THE APPROXIMATION DECREASES AS THE MESH IS REFINED.
ORIENTATION

EDGE 1-2,0.,
ALLE
POST
2,,,1
12,,STRESS IN GLOBAL Y
111,,STRESS, 1-2 EDGE - RADIAL
COMMENT, TO REQUEST PRINT OUT OF THE STRESSES IN THE COORDINATE SYSTEM
COMMENT,  DEFINED BY THE ORIENTATION OPTION, USE THE -PREF- OPTION
COMMENT,  OF PRINT ELEM.
PRINT ELEM

STRESS,STRAIN,PREF
INNER
1 TO 4
PRINT NODE

TOTAL,LOAD,REAC
1 TO 33
END OPTION
-------------------------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
-------------------------------------------------------------------------------------------------
```

Transform Nodal Degrees of Freedom

RESULTS

```
MARC          output for increment    0.    thick cylinder using plane strain element 11

element with highest stress relative to yield is    2  where equivalent stress is 0.187E-19  of yield
```

If Yield Stress is not entered, default is 1 X 10^{20}

```
        tresca     mises     mean   p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
      intensity intensity  normal   minimum intermediate maximum      1          2          3          4          5          6
                           intensity

element    1  point   1       integration pt. coordinate=     0.696E-01    0.101E+01
stress   2.152E+00 1.865E+00 1.460E-01-9.077E-01 1.011E-01 1.245E+00 1.238E+00-9.009E-01 1.011E-01-1.208E-01
stressp  2.152E+00 1.865E+00 1.460E-01-9.077E-01 1.011E-01 1.245E+00-9.056E-01 1.243E+00 1.011E-01-6.675E-02
strain   9.327E-08 5.396E-08 1.947E-09-4.372E-08 0.000E+00 4.956E-08 4.926E-08-4.342E-08 0.000E+00-1.047E-08

element    1  point   2       integration pt. coordinate=     0.720E-01    0.104E+01
stress   2.081E+00 1.803E+00 6.878E-02-9.612E-01 4.762E-02 1.120E+00 1.113E+00-9.546E-01 4.762E-02-1.168E-01
stressp  2.081E+00 1.803E+00 6.878E-02-9.612E-01 4.762E-02 1.120E+00-9.592E-01 1.118E+00 4.762E-02-6.454E-02
strain   9.018E-08 5.209E-08 9.171E-10-4.371E-08 0.000E+00 4.647E-08 4.618E-08-4.343E-08 0.000E+00-1.012E-08

element    1  point   3       integration pt. coordinate=     0.187E-01    0.101E+01
stress   2.152E+00 1.865E+00 1.461E-01-9.077E-01 1.011E-01 1.245E+00 1.243E+00-9.056E-01 1.011E-01-6.675E-02
stressp  2.152E+00 1.865E+00 1.461E-01-9.077E-01 1.011E-01 1.245E+00-9.009E-01 1.238E+00 1.011E-01-1.208E-01
strain   9.327E-08 5.396E-08 1.947E-09-4.372E-08 0.000E+00 4.956E-08 4.947E-08-4.363E-08 0.000E+00-5.785E-09

element    1  point   4       integration pt. coordinate=     0.193E-01    0.105E+01
stress   2.081E+00 1.803E+00 6.878E-02-9.612E-01 4.762E-02 1.120E+00 1.118E+00-9.592E-01 4.762E-02-6.454E-02
stressp  2.081E+00 1.803E+00 6.878E-02-9.612E-01 4.762E-02 1.120E+00-9.546E-01 1.113E+00 4.762E-02-1.168E-01
strain   9.018E-08 5.209E-08 9.171E-10-4.371E-08 0.000E+00 4.647E-08 4.638E-08-4.363E-08 0.000E+00-5.593E-09

element    2  point   1       integration pt. coordinate=     0.157E+00    0.100E+01
stress   2.153E+00 1.865E+00 1.461E-01-9.077E-01 1.011E-01 1.245E+00 1.201E+00-8.637E-01 1.011E-01-3.047E-01
stressp  2.153E+00 1.865E+00 1.461E-01-9.077E-01 1.011E-01 1.245E+00-9.057E-01 1.243E+00 1.011E-01-6.663E-02
strain   9.328E-08 5.396E-08 1.948E-09-4.372E-08 0.000E+00 4.956E-08 4.765E-08-4.181E-08 0.000E+00-2.640E-08

element    2  point   2       integration pt. coordinate=     0.163E+00    0.103E+01
stress   2.081E+00 1.803E+00 6.877E-02-9.612E-01 4.761E-02 1.120E+00 1.077E+00-9.187E-01 4.761E-02-2.946E-01
stressp  2.081E+00 1.803E+00 6.877E-02-9.612E-01 4.761E-02 1.120E+00-9.592E-01 1.118E+00 4.761E-02-6.441E-02
strain   9.018E-08 5.209E-08 9.170E-10-4.372E-08 0.000E+00 4.647E-08 4.462E-08-4.187E-08 0.000E+00-2.553E-08
```

Note the symmetry between σ_x (1,3) and σ_θ (2,1); and also σ_y (1,4) and σ_r (2,2)

```
element    2  point   3       integration pt. coordinate=     0.107E+00    0.101E+01
```

```
stress  2.152E+00 1.865E+00 1.461E-01-9.077E-01 1.011E-01 1.245E+00 1.215E+00-8.777E-01 1.011E-01-2.522E-01
stressp 2.152E+00 1.865E+00 1.461E-01-9.077E-01 1.011E-01 1.245E+00-9.009E-01 1.238E+00 1.011E-01-1.207E-01
strain  9.327E-08 5.396E-08 1.948E-09-4.372E-08 0.000E+00 4.956E-08 4.826E-08-4.242E-08 0.000E+00-2.186E-08

element    2  point    4        integration pt. coordinate=     0.110E+00    0.104E+01
stress  2.081E+00 1.803E+00 6.879E-02-9.612E-01 4.762E-02 1.120E+00 1.091E+00-9.322E-01 4.762E-02-2.439E-01
stressp 2.081E+00 1.803E+00 6.879E-02-9.612E-01 4.762E-02 1.120E+00-9.546E-01 1.113E+00 4.762E-02-1.167E-01
strain  9.018E-08 5.209E-08 9.172E-10-4.372E-08 0.000E+00 4.647E-08 4.521E-08-4.246E-08 0.000E+00-2.114E-08
```

Note the symmetric results along EDGE1 and EDGE2

```
                         n o d a l   p o i n t   d a t a


                         t o t a l   d i s p l a c e m e n t s


     1  4.42122E-09  5.05356E-08          2       0.       5.07284E-08          3  5.07284E-08      0.
     4  4.19219E-09  4.79171E-08          5       0.       4.81003E-08          6  4.81000E-08      0.
     7  3.94169E-09  4.50538E-08          8       0.       4.52259E-08          9  4.52259E-08      0.
    10  3.67419E-09  4.19962E-08         11  4.21564E-08       0.              12      0.      4.21565E-08
    13  3.39583E-09  3.88143E-08         14       0.       3.89627E-08         15  3.89627E-08      0.
    16  3.11449E-09  3.55981E-08         17  3.57341E-08       0.              18      0.      3.57342E-08
    19  2.83886E-09  3.24493E-08         20       0.       3.25732E-08         21  3.25732E-08      0.
    22  2.57869E-09  2.94746E-08         23       0.       2.95871E-08         24  2.95872E-08      0.
    25  2.34289E-09  2.67789E-08         26  2.68811E-08       0.              27      0.      2.68811E-08
    28  2.13987E-09  2.44589E-08         29       0.       2.45523E-08         30  2.45524E-08      0.
    31  1.97734E-09  2.26010E-08         32       0.       2.26873E-08         33  2.26873E-08      0.



                 total equivalent nodal forces (distributed plus point loads)


     1  7.59500E-03  8.68250E-02          2  1.90500E-03  4.35780E-02          3  4.35780E-02  1.90627E-03
     4       0.           0.              5       0.           0.              6       0.           0.
     7       0.           0.              8       0.           0.              9       0.           0.
    10       0.           0.             11       0.           0.             12       0.           0.
    13       0.           0.             14       0.           0.             15       0.           0.
    16       0.           0.             17       0.           0.             18       0.           0.
    19       0.           0.             20       0.           0.             21       0.           0.
    22       0.           0.             23       0.           0.             24       0.           0.
    25       0.           0.             26       0.           0.             27       0.           0.
    28       0.           0.             29       0.           0.             30       0.           0.
    31       0.           0.             32       0.           0.             33       0.           0.



                 reaction forces at fixed boundary conditions, residual load correction elsewhere
```

```
         1 -1.37911E-16  1.66533E-16              2 -3.73441E-02 -2.70617E-16              3 -3.46945E-16 -3.73497E-02
         4  1.66533E-16 -1.45717E-16              5 -7.55711E-02  9.02056E-17              6  1.15186E-15 -7.55703E-02
         7  6.93889E-18 -1.40860E-15              8 -8.36900E-02  4.16334E-17              9  1.73472E-16 -8.36898E-02
        10 -6.93889E-18 -4.78784E-16             11 -1.24900E-16 -9.11438E-02             12 -9.11438E-02 -5.34295E-16
        13  4.85723E-17  2.77556E-17             14 -9.76663E-02  2.67147E-16             15 -1.80411E-16 -9.76645E-02
        16  1.45717E-16 -3.43475E-16             17 -1.63064E-16 -0.10312                 18 -0.10312       2.08167E-16
        19 -9.02056E-17 -4.33681E-17             20 -0.10759      -8.32667E-17             21 -1.38778E-16 -0.10759
        22  1.31839E-16 -1.52656E-16             23 -0.11140      -1.17961E-16             24 -8.32667E-17 -0.11140
        25  6.93889E-17  1.39645E-16             26  1.66533E-16 -0.11513                 27 -0.11513       9.71445E-17
        28       0.     -2.08167E-17             29 -0.11957       9.54098E-18             30 -4.59702E-17 -0.11957
        31 -1.04083E-16 -1.69136E-16             32 -5.77721E-02  6.25585E-17             33  6.59195E-17 -5.77728E-02


                 summary of externally applied loads


 0.15190E-01    0.17365E+00


                 summary of reaction/residual forces


-0.15190E-01   -0.17365E+00




             distributed load    type    current
             list number                 magnitude


                   1           10      1.000          0.           0.


                 e n d   o f   i n c r e m e n t   0


                 formatted post data at increment    0.   0    on tape 19
                  time =        1.02



                 *** end of input deck - job ends



                       marc exit number 3004
```

Results

The closed-form solutions for the tangential stress, radial stress, and radial displacement in a thickwalled cylinder loaded by internal pressure may be found in any standard text on strength of materials. For the parameters of this problem, with a 1.00 psi internal pressure and an inner radius of 1.00 in. and an outer radius of 3.00 in., the theoretical answers (maximum at the *inner* radius) are:

From R. J. Roark and W. C. Young, *Formulas for Stress and Strain* (5th ed.), McGraw-Hill, 1976, p. 504):

tangential stress:	$\sigma_T = +1.25$ psi
radial stress:	$\sigma_R = -1.00$ psi
radial displacement:	$v = +5.16667E-8$ in.

(Note that these theoretical values are calculated at r = 1.00 in. In order to make an exact correlation with the MARC-calculated values, the values should really be obtained using the radius at the particular integration point of element 1.)

The maximum MARC-calculated values for this coarse 20-element model are:

At element 1, integration point 3:

$\sigma_T = +\mathbf{1.243}$ psi	(approximately half a percent off the theoretical value at r = 1.00 in.)

At element 1, integration point 4:

$\sigma_R = -\mathbf{0.9592}$ psi	(approximately 4% off the theoretical value at r = 1.00 in.)

At the inner edge (nodes 2 and 3):

$v = +\mathbf{5.07281E-8}$ in	(approximately 2% off theoretical value)

The correlation is excellent even for this coarse model. Now, let's go on to Example 2B, which shows you how to solve the same problem using an axisymmetric solution.

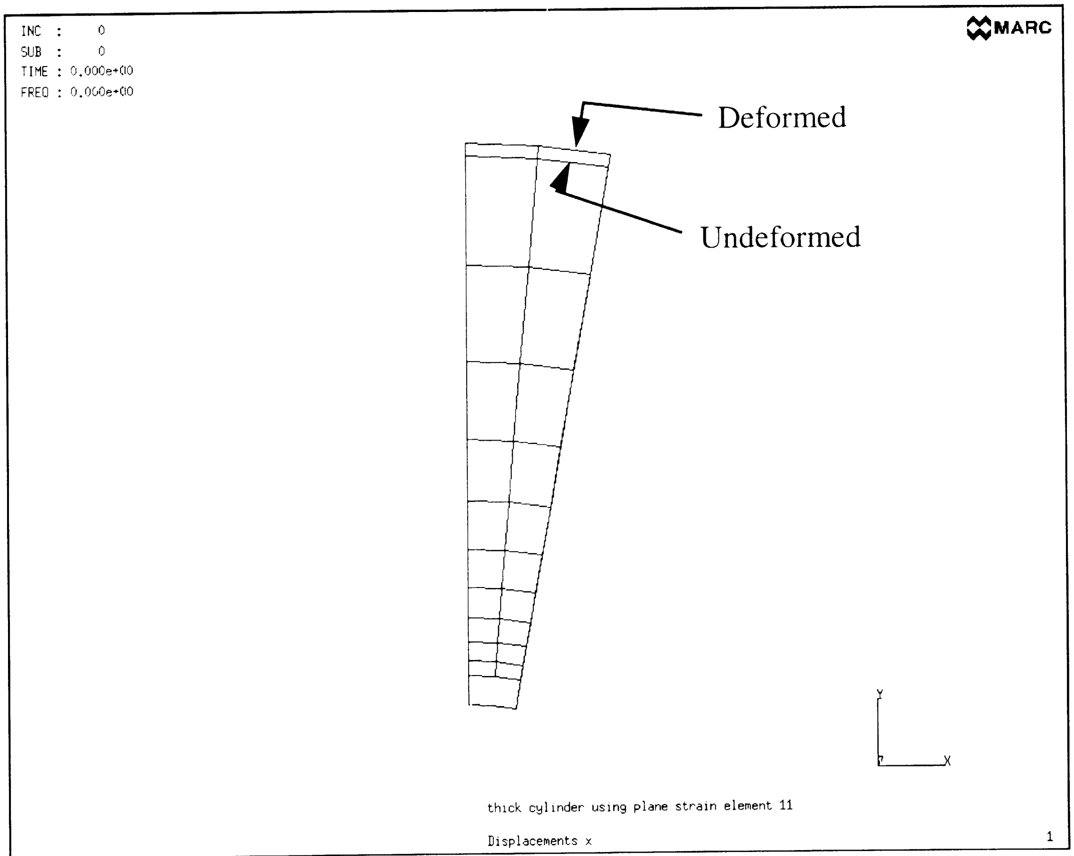


Figure 2A.2 **Radial Displacement**

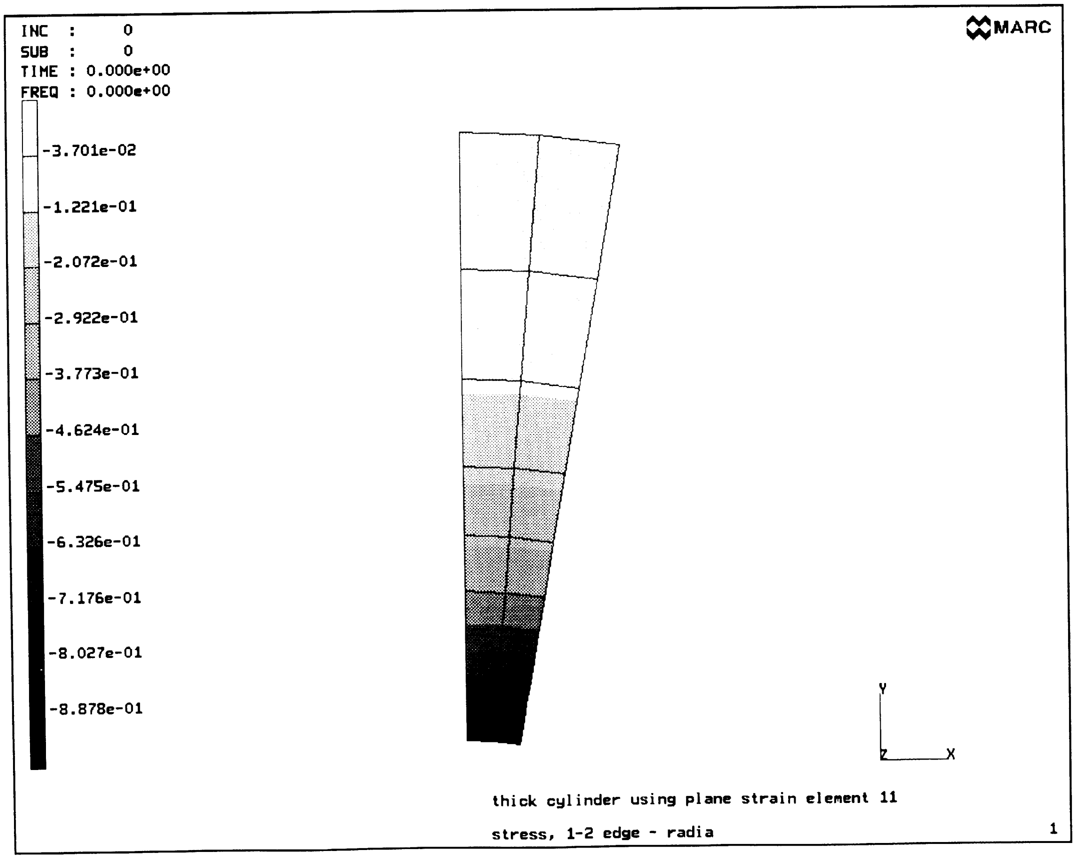


Figure 2A.3 **Radial Stress**

Example 2B

Thick Cylinder Under Internal Pressure–Axisymmetric Solution

Sketch

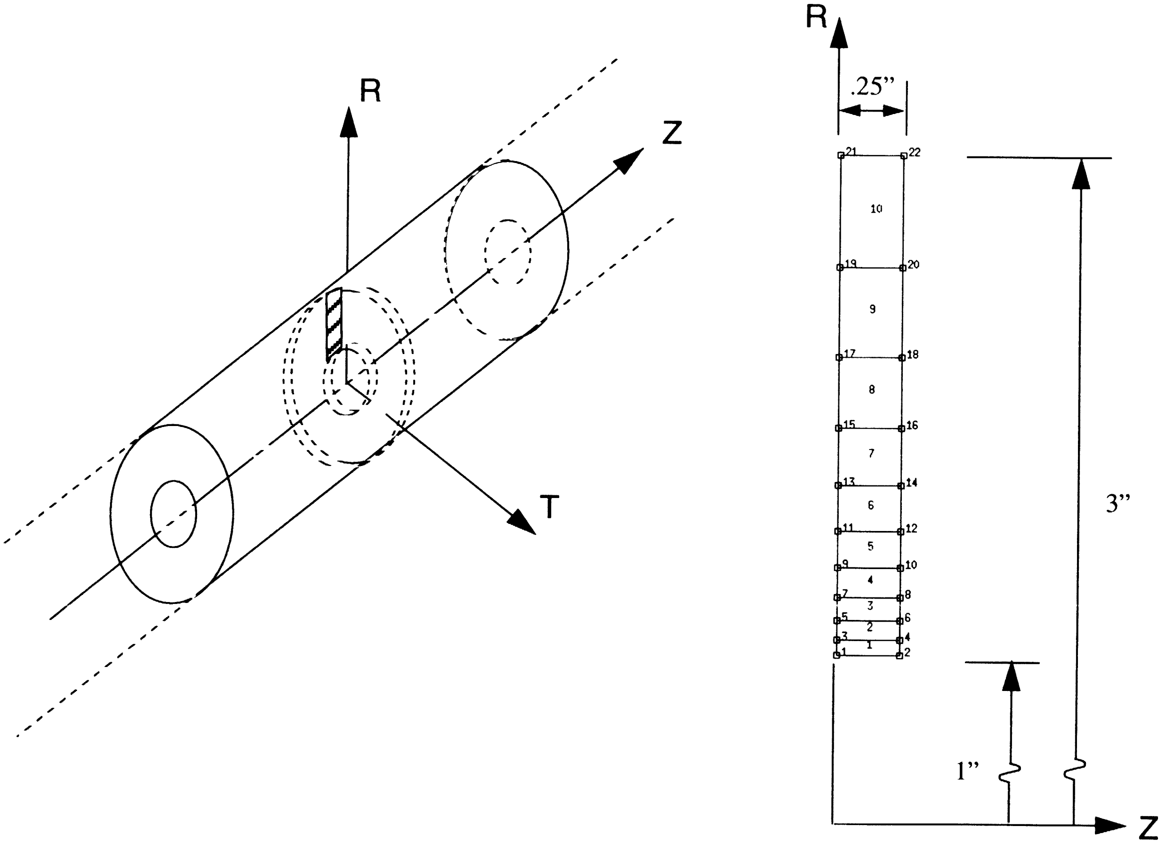


Figure 2B.1 Thick Cylinder, Axisymmetric Model

Model

The axisymmetric model is a thin (0.25 in.) slice taken in the axial direction of a long thick-walled cylinder. The model has ten elements through the thickness, like the plane-strain model in Example 2A, and 22 nodes. The geometry (inner radius = 1 in., outer radius = 3 in.) and the material properties are the same as before.

MARC element 10 is a four-noded linear axisymmetric element, with two translational DOFs at each node and four Gaussian integration points. A complete description of Element 10 is found in *MARC Volume B*. The origin of the global coordinate system (Z,R) is at the axis of the cylinder. The axis of symmetry is along the Z-axis. Radial displacement is therefore along the R-axis, and the tangential (hoop) direction is out-of-plane. As before, the radial and tangential stresses are expected to be the greatest at the inner radius; therefore, the mesh is refined there.

(Alternatively, we could have selected Element 2, a triangular axisymmetric ring element, or Element 28, an 8-noded second order quadrilateral axisymmetric element.)

Properties

Identical to Example 2A.

Loads

Identical to Example 2A.

Boundary Conditions

The boundary conditions for this model are simple; no displacements are permitted in the Z-direction for all the nodes on both the left and right edges of the model.

Input

A complete input echo from the printout is included. Most of the input is self-explanatory, or is similar to the previous examples. Here, we'll just explain those features in the input data which are being seen for the first time.

The DEFINE MODEL DEFINITION option is again used to define two element sets named "ALLE" (all ten elements) and "INNER" (element 1), and a node set named "ALLN" (all 22 nodes).

Four new output control options are seen for the first time. The NO PRINT option (*See MARC Volume C*) suppresses element and nodal output. We use this option here because all we want are the summary-type tables shown. The SUMMARY option (*MARC Volume C*) produces a summary of the results of an increment and outputs them in a report format. The next two options tell MARC what results we would like to have in a report format. The ELEM SORT option (*MARC Volume C*) sorts various element quantities. The "1" line which follows means we would like to sort one element quantity. The "2,,,10," line indicates that the code we want sorted is 2 (highest absolute value of the second component of stress, which in this example is the radial stress), and the 10 means we want ten items on the sorted list. Leaving the remaining fields blank results in a list in descending order of the absolute magnitudes. The NODE SORT option (*MARC Volume C*) sorts various nodal quantities. The "1," line says we want to sort one nodal quantity here. The "14,,,10" line means we want code number 14 (highest absolute value of the second component of total displacement, which in this example is the total radial displacement), and there are ten items in the sorted list.

NOTE

The output of the SUMMARY and SORT options can be optionally directed to a file other than the standard line printer to be used directly in reports.

The POST block creates a post-processor file for later post-processing by Mentat II (*see MARC Volume C*). The "4,,,1" line means the number of element variables to be written to file is four, and the "1" in the fourth field indicates we want a formatted post file. The "11", "12", "13", and "14" on the next four lines refer to the four post codes assigned to the first four components of stress (i.e., the normal stresses in the axial, radial, and hoop directions, and the shear stress in the Z-R plane).

Output

In this example, only the summary tables of the printout are included here. They are self-explanatory. The first summary table appears on one and a half pages and summarizes all the stress and strain quantities. The second table ranks the ten highest "second component of stress" in descending order (the default element sorting order), listing the values at the four integration points in element 1, followed by those in element 2, then at integration points 3 and 4 for element 3. The third table summarizes the maximum/minimum incremental and total displacements, and reaction forces, the last table ranks the ten highest "second component of total displacement", listing the values in descending order (the default node sorting order).

input data

```
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
--------------------------------------------------------------------------------
TITLE      THICK CYLINDER USING AXISYMMETRIC ELEMENT 10
SIZING,100000
ELEMENTS     10
END
CONNECTIVITY
   10    0    0
    1   10    1    2    4    3
    2   10    3    4    6    5
    3   10    5    6    8    7
    4   10    7    8   10    9
    5   10    9   10   12   11
    6   10   11   12   14   13
    7   10   13   14   16   15
    8   10   15   16   18   17
    9   10   17   18   20   19
   10   10   19   20   22   21
COORDINATES
    2   22    0    0
    1   0.00000   1.00000
    2   0.25000   1.00000
    3   0.00000   1.06014
    4   0.25000   1.06014
    5   0.00000   1.13533
    6   0.25000   1.13533
    7   0.00000   1.22930
    8   0.25000   1.22930
    9   0.00000   1.34677
   10   0.25000   1.34677
   11   0.00000   1.49361
   12   0.25000   1.49361
   13   0.00000   1.67716
   14   0.25000   1.67716
   15   0.00000   1.90660
   16   0.25000   1.90660
   17   0.00000   2.19339
   18   0.25000   2.19339
   19   0.00000   2.55188
   20   0.25000   2.55188
   21   0.00000   3.00000
--------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
   22   0.25000   3.00000
COMMENT, DEFINE SETS FOR LATER USE
DEFINE    ELEMENT   SET       ALLE
    1 TO      10
DEFINE    NODE      SET       ALLN
    1 TO      22
DEFINE    ELEMENT   SET       INNER
    1
ISOTROPIC

1,
30.E6,.3,
ALLE
COMMENT, DIST LOADS APPLIES PRESSURE TO FACE 1-2 OF SET INNER
DIST LOADS

0,1.,
INNER
COMMENT, TO SIMULATE PLANE STRAIN CONDITIONS IN THE PLANE OF THE
COMMENT,  CYLINDER PERPENDICULAR TO THE AXIAL DIRECTION, RESTRAIN ALL
COMMENT,  DOFS IN THE AXIAL DIRECTION.
FIXED DISP

0.,
1,
ALLN
COMMENT, SUPPRESS ALL OUTPUT OF ELEMENT INTEGRATION POINT QUANTITIES AND
COMMENT,  NODAL QUANTITIES.  ASK MARC FOR SUMMARY OF RESULTS AND SORTED
COMMENT,  TABLES OF LARGEST QUANTITIES.
NO PRINT
SUMMARY
COMMENT, SORT 10 HIGHEST ABSOLUTE VALUES OF RADIAL STRESS
ELEM SORT
1,
2,,,10,
COMMENT, SORT 10 HIGHEST ABSOLUTE VALUES OF RADIAL DISPLATEMENT
NODE SORT
1,
14,,,10
COMMENT, 1 IN 4TH FIELD OF POST DATA CARD MEANS WRITE FORMATTED POST TAPE
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
            --------------------------------------------------------------------------------
card   80   POST
            4,,,1
            11,
            12,
            13,
card   85   14,
            END OPTION
            --------------------------------------------------------------------------------
                5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
            --------------------------------------------------------------------------------
            --------------------------------------------------------------------------------
                          ●                                            ●
                          ●                                            ●
                          ●                                            ●
                          ●                                            ●
```

Results from Summary Option

```
***********************************************************************
***********************************************************************
*                                                                     *
* thick cylinder using axisymmetric element 10                        *
*                                                                     *
*     increment     0                                  marc k5        *
*                                                                     *
***********************************************************************
*                                    *              *     *     *     *
*              quantity              *    value     * elem.* int.*layer*
*                                    *              *number*point*     *
*                                    *              *     *     *     *
***********************************************************************
*                                    *              *     *     *     *
* max first  comp. of stress         *  0.10257E+00 *    3 *  1  *   1 *
* min first  comp. of stress         *  0.46968E-01 *    2 *  4  *   1 *
*                                    *              *     *     *     *
* max second comp. of stress         * -0.10178E-01 *   10 *  2  *   1 *
* min second comp. of stress         * -0.96154E+00 *    1 *  4  *   1 *
*                                    *              *     *     *     *
* max third  comp. of stress         *  0.12451E+01 *    1 *  2  *   1 *
* min third  comp. of stress         *  0.24888E+00 *   10 *  3  *   1 *
*                                    *              *     *     *     *
* max fourth comp. of stress         *  0.63375E-14 *    1 *  1  *   1 *
* min fourth comp. of stress         * -0.49631E-15 *    8 *  1  *   1 *
*                                    *              *     *     *     *
* max   equivalent    stress         *  0.18659E+01 *    1 *  2  *   1 *
* min   equivalent    stress         *  0.24601E+00 *   10 *  4  *   1 *
*                                    *              *     *     *     *
* max       mean       stress        *  0.14816E+00 *    3 *  1  *   1 *
* min       mean       stress        *  0.67843E-01 *    2 *  4  *   1 *
*                                    *              *     *     *     *
* max      tresca      stress        *  0.21531E+01 *    1 *  2  *   1 *
* min      tresca      stress        *  0.27956E+00 *   10 *  4  *   1 *
*                                    *              *     *     *     *
* max first  comp. of total strain   *  0.00000E+00 *    1 *  1  *   1 *
* min first  comp. of total strain   *  0.00000E+00 *    1 *  1  *   1 *
*                                    *              *     *     *     *
* max second comp. of total strain   * -0.41660E-08 *   10 *  2  *   1 *
* min second comp. of total strain   * -0.43730E-07 *    1 *  2  *   1 *
*                                    *              *     *     *     *
* max third  comp. of total strain   *  0.49573E-07 *    1 *  2  *   1 *
* min third  comp. of total strain   *  0.79482E-08 *   10 *  3  *   1 *
*                                    *              *     *     *     *
* max fourth comp. of total strain   *  0.54925E-21 *    1 *  1  *   1 *
* min fourth comp. of total strain   * -0.43013E-22 *    8 *  1  *   1 *
*                                    *              *     *     *     *
***********************************************************************
```

Radial Stress

Hoop Stress

```
************************************************************************
*                                                                      *
* thick cylinder using axisymmetric element 10                         *
*                                                                      *
*     increment      0                                    marc k5      *
*                                                                      *
************************************************************************
*                                    *              *      *      *      *
*              quantity              *    value     * elem.* int.*layer*
*                                    *              *number*point*      *
*                                    *              *      *      *      *
************************************************************************
*                                    *              *      *      *      *
* max   equivalent    total strain   *  0.53974E-07 *    1 *  2   *    1 *
* min   equivalent    total strain   *  0.73271E-08 *   10 *  3   *    1 *
*                                    *              *      *      *      *
* max      mean       total strain   *  0.19754E-08 *    3 *  1   *    1 *
* min      mean       total strain   *  0.90458E-09 *    2 *  4   *    1 *
*                                    *              *      *      *      *
* max     tresca      total strain   *  0.93303E-07 *    1 *  2   *    1 *
* min     tresca      total strain   *  0.12114E-07 *   10 *  4   *    1 *
*                                    *              *      *      *      *
* max temperature                    *  0.00000E+00 *    1 *  1   *    1 *
* min temperature                    *  0.00000E+00 *    1 *  1   *    1 *
************************************************************************
```

Result of Element Sort

```
************************************************************
*                                                          *
* thick cylinder using axisymmetric element 10            *
*     increment     0                        marc k5       *
*                                                          *
************************************************************
*                                                          *
*  highest  absolute value of second comp. of stress       *
*                                                          *
************************************************************
*          *                   *          *       *        *
*   rank   *       value       *  element *  int. * layer  *
*          *                   *   number * point *        *
*          *                   *          *       *        *
************************************************************
*          *                   *          *       *        *
*     1    *     -0.96154E+00  *       1  *   4   *    1   *
*     2    *     -0.96154E+00  *       1  *   3   *    1   *
*     3    *     -0.90801E+00  *       1  *   2   *    1   *
*     4    *     -0.90801E+00  *       1  *   1   *    1   *
*     5    *     -0.83581E+00  *       2  *   4   *    1   *
*     6    *     -0.83581E+00  *       2  *   3   *    1   *
*     7    *     -0.78049E+00  *       2  *   2   *    1   *
*     8    *     -0.78049E+00  *       2  *   1   *    1   *
*     9    *     -0.70707E+00  *       3  *   4   *    1   *
*    10    *     -0.70707E+00  *       3  *   3   *    1   *
*          *                   *          *       *        *
************************************************************
```

Result of Summary

```
******************************************************************
* thick cylinder using axisymmetric element 10                   *
*      increment      0                             marc k5      *
******************************************************************
*              quantity                *      value      *  node   *
*                                      *                 * number  *
******************************************************************
* max second comp. of incremental disp *   0.50759E-07   *     2   *
* min second comp. of incremental disp *   0.22697E-07   *    21   *
*                                      *                 *         *
* max second comp. of total disp.      *   0.50759E-07   *     2   *
* min second comp. of total disp.      *   0.22697E-07   *    21   *
*                                      *                 *         *
* max first  comp. of reaction force   *   0.49488E+00   *    20   *
* min first  comp. of reaction force   *  -0.49488E+00   *    19   *
******************************************************************
```

Result of Node Sort

NOTE

Nodes with applied boundary conditions are not included when sorting displacements

```
*******************************************
* thick cylinder using axisymmetric element 10*
*     increment      0              marc k5  *
*******************************************
*                                          *
*  highest  absolute value of second comp. of *
*              total disp.                 *
*                                          *
*******************************************
*         *                    *           *
*  rank   *       value        *   node    *
*         *                    *  number   *
*         *                    *           *
*******************************************
*     1   *     0.50759E-07    *     2     *
*     2   *     0.50759E-07    *     1     *
*     3   *     0.48129E-07    *     4     *
*     4   *     0.48129E-07    *     3     *
*     5   *     0.45253E-07    *     6     *
*     6   *     0.45253E-07    *     5     *
*     7   *     0.42181E-07    *     8     *
*     8   *     0.42181E-07    *     7     *
*     9   *     0.38985E-07    *    10     *
*    10   *     0.38985E-07    *     9     *
*******************************************
```

Note the Symmetry

Results

The theoretical results (at R = 1.00 in.) were given in Example 2A. The calculated results at integration point 2 of element 1 (which is close to R = 1.00 but not exactly at that radius) are:

At element 1, integration point 2:

$\sigma_T = +1.25$ psi (approximately 0.4% off theory)

At element 1, integration point 2:

$\sigma_R = 0.9080$ psi (approximately 9% off theory)

At the inner edge (nodes 1 and 2), the radial displacement is:

$v = +5.0759$ E-8 in. (approximately 2% off theory)

We see that, for this coarse 10-element axisymmetric model, the correlation with theory is good. The stresses in the radial direction oscillate, owing to the 1/r dependence of the hoop strain. This cannot be adequately represented in a linear displacement element. Centroidal values often give better results. Therefore, we have demonstrated in Example 2 that the two solutions to the thick cylinder give essentially the same results.

You have now seen how to:

- set up a plane strain and an axisymmetric analysis;
- apply the appropriate boundary conditions;
- use TRANSORMATION to facilitate skewed geometries; and
- customize your printout to display only items you want to see.

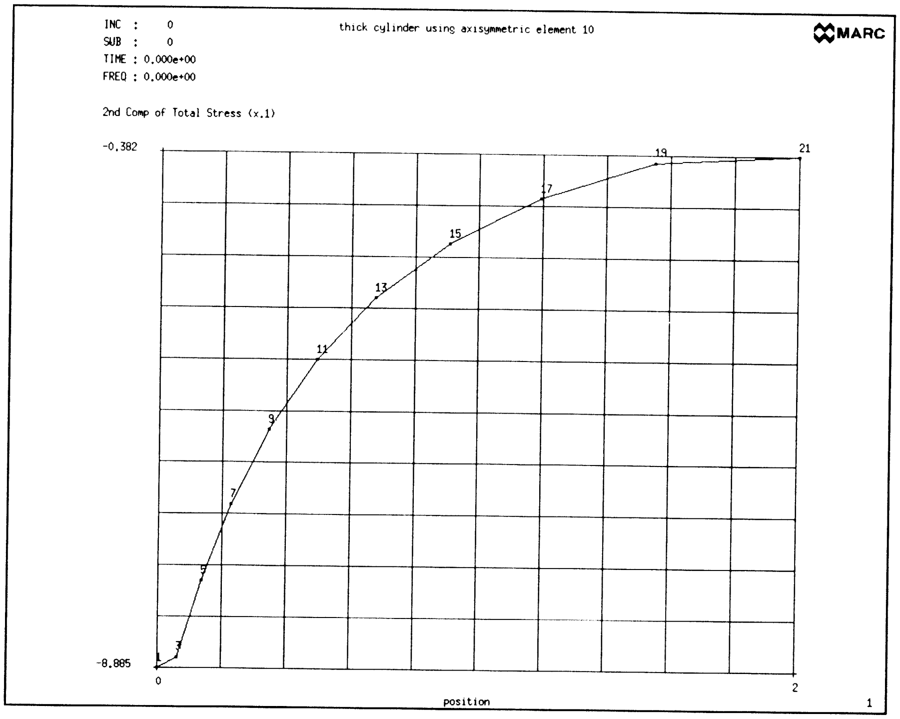


Figure 2B.2 **Radial Stress Through Radius**

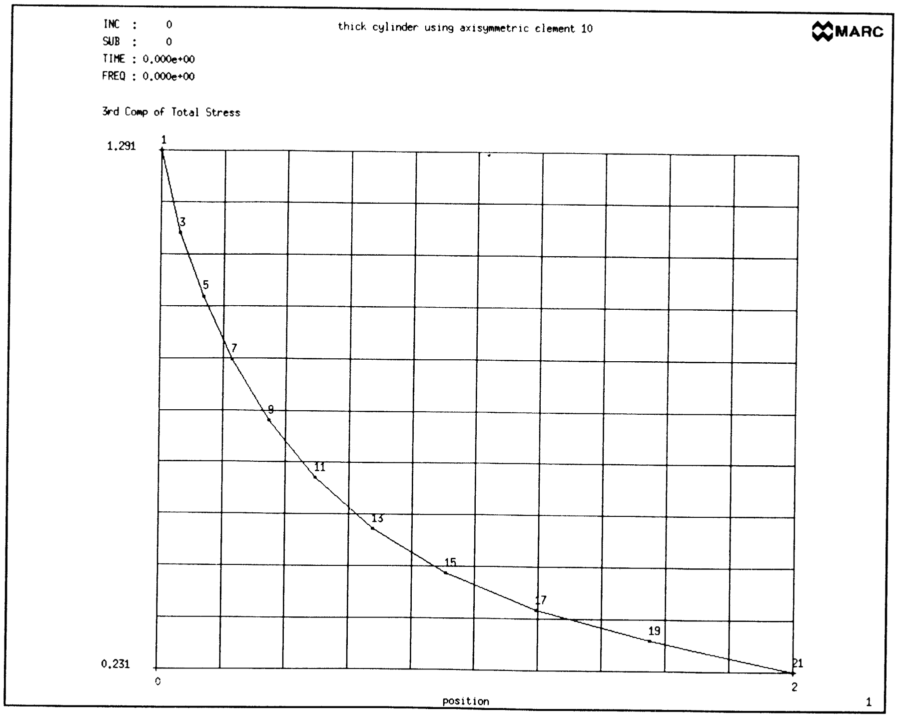


Figure 2B.3 **Hoop Stress Through Radius**

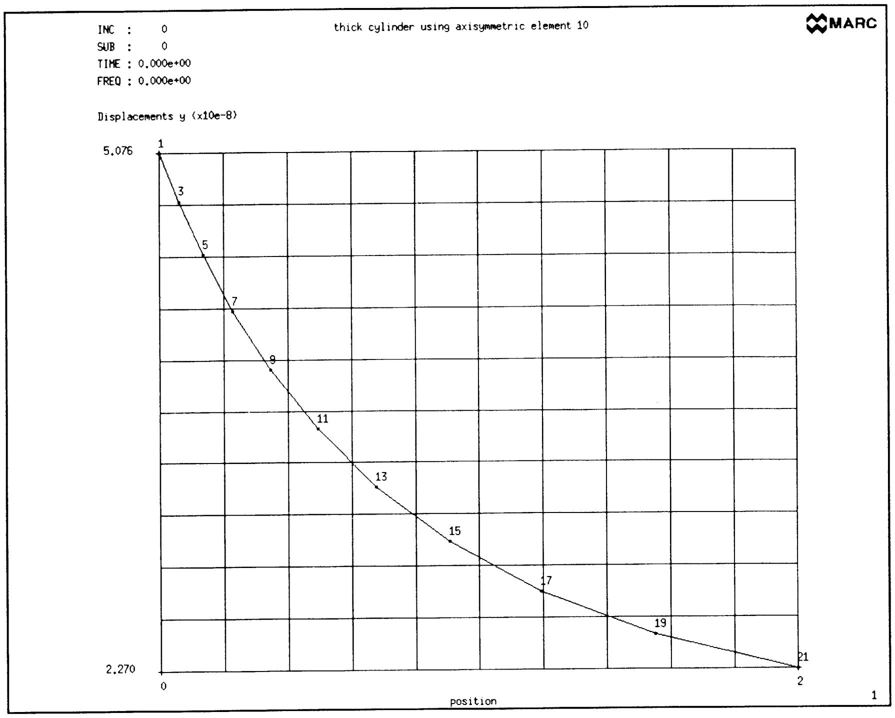


Figure 2B.4 Radial Displacement Through Radius

Example 3A

Modal Analysis of a Cantilevered Beam

The solution of the undamped free vibration of a cantilevered beam can be found in any textbook on vibration theory. This example first illustrates the eigenvalue extraction (modal analysis) of a cantilevered beam. Then, a linear dynamic analysis (undamped) using direct integration follows, with the beam subjected to a ramp-type uniform load. Finally, a damped modal superposition solution of the beam subjected to initial conditions is shown.

Example 3A:	Modal Analysis
Example 3B:	Linear Dynamic Analysis using Direct Integration
Example 3C:	Damped Modal Superposition Response Subjected to Initial Conditions

Sketch

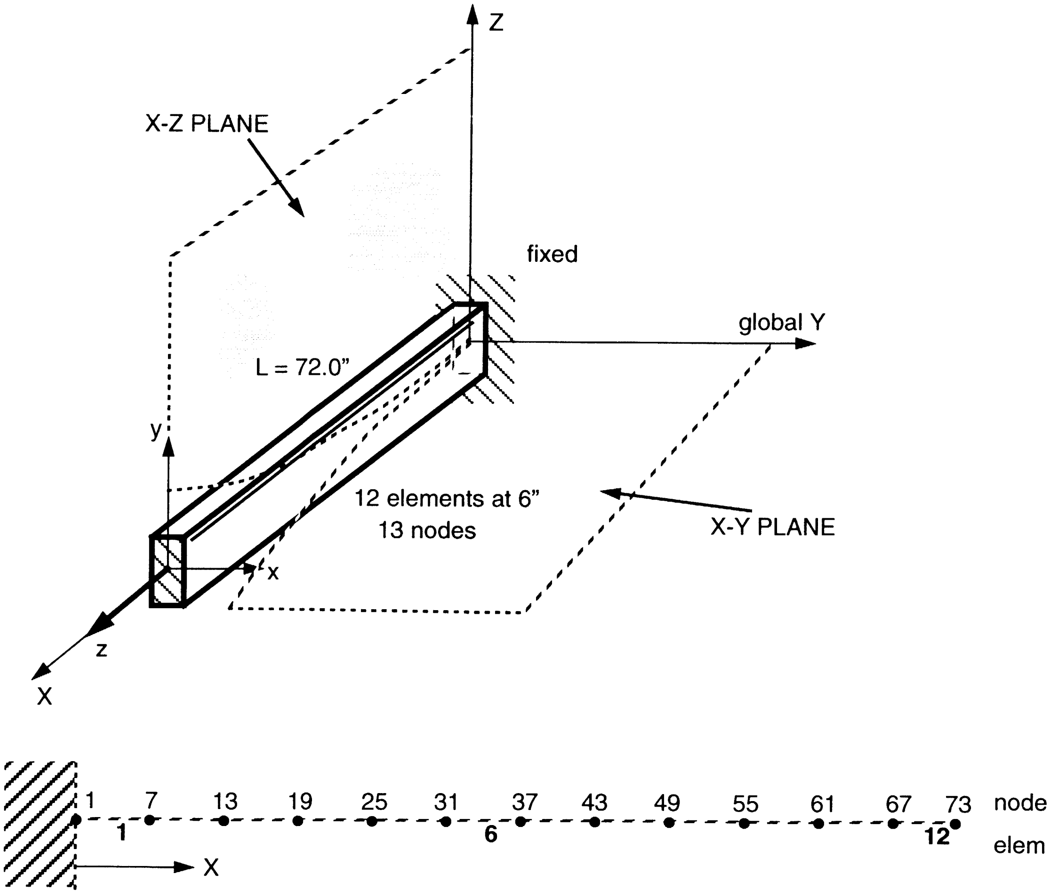


Figure 3A.1 Three-Dimensional Cantilevered Beam Model

Model

The finite element model is a simple cantilevered beam consisting of twelve elements (MARC Element 52). The material properties are isotropic and linear elastic: same Young's modulus (30 x 10^6psi or 30,000,000) and Poisson's ratio (0.3) as in Examples 1 and 2, except this time we also have to input a mass density of 7.6754 x 10^{-4} lb-sec^2 /in^4 for an eigenvalue or dynamic analysis problem.

The left end of the beam is fixed against all displacements and rotations. This is the only boundary condition necessary for this problem. Element 52 is a 3-D elastic beam element, with two nodes and three Gaussian integration points along the length. Each node has six DOFs: three translations and three rotations. A description of Element 52 is found in *MARC Volume B.*

For this 3-D beam element, the following geometry properties have to be input: cross-sectional area, moments of inertia in the major and minor axes, and the orientation of the local axis.

Properties

As before, for an isotropic linear elastic material, we only need to input Young's modulus and Poisson's ratio. In addition, since this is an eigenvalue problem, we also have to input the (volumetric) mass density in units consistent with the problem description.

Loads

None, since this is only an eigenvalue extraction problem.

Boundary Conditions

The left end (node 1) of the beam is fully clamped. Hence, all six degrees of freedom are suppressed.

Input

A complete input echo is included. Items of interest in the input follow.

PARAMETER Section

The "SIZING" line sets 100,000 words as the workspace. The "ELEMENTS" line informs MARC that Element 52 will be used. (This element is a 3-D elastic beam element, and is integrated exactly.)

The next line is seen for the first time. The "DYNAMIC" option (*see MARC Volume C*) indicates that we want to perform a dynamic analysis, where the "1" means eigenvalue extraction and the "5" means we desire five mode shapes and frequencies.

As usual, the "END" line terminates the input of the PARAMETER section.

MODEL DEFINITION Section

The MODEL DEFINITION options in this example consist of:

- the FE mesh topology (element connectivity, nodal coordinates, definition of node set and element set)
- material properties
- boundary conditions
- geometric properties
- output controls

FE Mesh Topology

The FE mesh in this example consists of: 12 beam elements and 13 nodes spaced every 6.00 in. along the global X-axis. The CONNECTIVITY and COORDINATES data are self-explanatory. The DEFINE option is used to name a node set FIXME (consisting of only node 1) and an element set ALLE (all 12 elements). Note that in this example, the node numbers are not given in consecutive order. This is for user convenience since MARC internally allocates only 13 nodes. There is no additional computational cost to having non-consecutive element numbers or node numbers.

Material Properties

The ISOTROPIC option is used to enter the material properties. The "1," line following the blank line indicates the material properties which follow to be designated as material property set number 1. Young's modulus is 30E6, Poisson's ratio is 0.3, and the mass density is 7.6754E-4 units. The last line in this block, "ALLE", says these properties apply to all twelve elements in the model.

Boundary Conditions

In the FIXED DISP block, the blank line is followed by a option with six zeroes, which refer to the six zero-valued displacements being prescribed. The next "1 TO 6" line means the list of six DOFs (3 translations, 3 rotations) for which the zero displacements are applicable. And the last "FIXME" line indicates the pertinent node set–in this case, consisting of node 1 only.

Geometric Properties

The element geometry is entered using the GEOMETRY block (*see MARC Volume C*). The blank line as usual means you don't have to count how many distinct element geometries are input. The next line gives six data items for this beam element (*see MARC Volume B*): the first field has a value of 6.0 in^2, which is the cross-sectional area of the beam; the second field is 16.0 in^4 which is I_{xx}, the moment of inertia about the local x-axis (in this case, the major or strong axis of bending); the third field is 5.33 in^4, which is I_{yy}, the moment of inertia about the local y-axis (in this case, the minor or weak axis); the fourth field is 0.0, the fifth field is 1.0, and the sixth field is 0.0–these three being components of a vector representing the local x-axis and aligning it with the positive global Y-axis (see sketch). The last line in the block, "ALLE", tells MARC that these element geometry properties apply to element set ALLE, which represents all twelve elements of the model.

Output Controls

The POST block creates a post-processor file for later post-processing by Mentat II. The ",,,1" line means that we would like to have a formatted post file. The "END OPTION" line ends the MODEL DEFINITION options.

HISTORY DEFINITION Section

Here, HISTORY DEFINITION options are specified for the first time in this Primer. Two such options are illustrated: MODAL SHAPE, and RECOVER.

The "MODAL SHAPE" line (*see MARC Volume C*) tells MARC we want to extract eigenmodes, and is required since we set the first field equal to 1 on the "DYNAMIC" parameter line. Also, the fact we have left the third field blank on the "DYNAMIC" parameter line means we have opted for the inverse power sweep method of eigenvalue extraction. The blank line which follows indicates we want to use default values for the six parameters needed for the inverse power sweep method (the use of defaults is highly recommended if you are a novice):

Field/Variable	Meaning	Default Value
1 NCYCM	max. no. iterations per mode	40
2 FLAMB	convergence tolerance	0.0001
3 SHFLAM	initial shift frequency (cycles per time)	0
4 SHFMAX	max. frequency to be extracted (cycles per time)	no. of modes on DYNAMIC Parameter line
5 INCSHF	no. modes expected per shift	5
6 SHFSCL	auto shift parameter	1.0

Of these six parameters, only the first two are usually needed by the novice. The "CONTINUE" line ends the MODAL SHAPE input block, and the modal extraction now begins.

RECOVER (*see MARC Volume C*) is an option for:

1. the storing of eigenvectors on post file
2. the recovery of modal reaction forces
3. the recovery of modal stresses and reactions for a specified number of modes during a modal or buckling analysis.

This option should be used after the modal shapes and frequencies have been extracted. The "1,5" line which follows means that modes 1 through 5 are to be written to the post file. The next "CONTINUE" line indicates that the data set for this load increment has now been fully defined.

Output

In addition to the input echo, selected portions of the output are included. (Increment 0 is a null increment in this type of analysis, and values for incremental displacements, total equivalent nodal forces, and reactions forces at fixed boundary conditions are all zero.)

After the underlined "CONTINUE" line, results of using the inverse power sweep method to extract eigenvalues of the first mode are shown. The solution converged in five iterations, yielding the first eigenvalue to be a frequency of 126.12 rad/sec, or 20.114 cycles/sec. This is then followed by a printout of the familiar first mode shape (eigenvector), and we notice that the largest displacements occur in the Y-direction (second column). We go on to see that the Mode 2 frequency is 218.51 rad /sec, with the mode shape vibrating in the Z-direction. Mode 3 has a frequency of 789.32 rad/sec, with the mode shape vibrating in the Y-direction. Mode 4 has a frequency of 1,367.58 rad/sec, with the mode shape vibrating in the Z-direction. And Mode 5 has a frequency of 2,217.88 rad/sec, with the mode shape vibrating in the Y-direction. (After the Mode 5 eigenvector printout, one additional frequency of 2,676.83 rad/sec is shown along with its eigenvector; this sixth mode is a torsional mode, and the eigenvector of interest shows rotations around the global X-axis.)

After the eigenvalue extraction, the RECOVER option instructs MARC to store all the eigenvectors on the post file. MARC proceeds to do so, and labels the five eigenvectors INCREMENT 0, (SUBINCREMENT) 1 (through 5). It then tells you that the eigenvectors have been stored on unit 19, and makes a normal job exit (exit number 3004).

input data

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
TITLE       EIGENVALUE EXTRACTION OF A CANTILEVERED BEAM
SIZING        100000   12   13
ELEMENTS      52
DYNAMIC,1,5,
END
CONNECTIVITY
   12    0    0
    1   52    1    7
    2   52    7   13
    3   52   13   19
    4   52   19   25
    5   52   25   31
    6   52   31   37
    7   52   37   43
    8   52   43   49
    9   52   49   55
   10   52   55   61
   11   52   61   67
   12   52   67   73
COORDINATES
    3   13    0    0
    1   1.00000   0.00000   0.00000
    7   7.00000   0.00000   0.00000
   13  13.00000   0.00000   0.00000
   19  19.00000   0.00000   0.00000
   25  25.00000   0.00000   0.00000
   31  31.00000   0.00000   0.00000
   37  37.00000   0.00000   0.00000
   43  43.00000   0.00000   0.00000
   49  49.00000   0.00000   0.00000
   55  55.00000   0.00000   0.00000
   61  61.00000   0.00000   0.00000
   67  67.00000   0.00000   0.00000
   73  73.00000   0.00000   0.00000
DEFINE    NODE      SET       FIXME
    1
DEFINE    ELEMENT   SET       ALLE
1 TO 12
ISOTROPIC
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

Eigenvalue Extraction of Five Modes

```
              5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
           ------------------------------------------------------------------------------

card  40
           1,
           30.E6,.3,7.6754E-4
           ALLE
           FIXED DISP
card  45
           0.,0.,0.,0.,0.,0.,
           1 TO 6
           FIXME
           GEOMETRY
card  50
           6.,16.,5.33,0.,1.,0.,
           ALLE
           POST
           ,,,1
card  55   END OPTION
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
```

HISTORY DEFINITION options

```
           MODAL SHAPE


           CONTINUE
```

Extracts Modes

```
           RECOVER
card  60   1,5
           CONTINUE
```

Places Modal Shapes in Post File

```
           ------------------------------------------------------------------------------
              5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
           ------------------------------------------------------------------------------

           ------------------------------------------------------------------------------
                           •
                           •
                           •
                           •

                   end of increment 0

                   formatted post data at increment   0.   0   on tape 19
                    time =       0.58


       modal shape
       ----------


         max       conv      initial      max     modes     auto
        cycles  tolerance     shift    frequency   per      shift
                            frequency             shift   parameter
          40    1.000E-04  0.000E+00  0.000E+00     5     1.000E+00
       continue
       ----------


                   power sweep for eigenvalue
```

During Increment Zero, only matrices are assembled. Sinc this is the only function performed, all results are zer

Default options used to control extraction iteration.

Inverse power sweep method used by default.

```
eigenvalues extracted with shift point of  0.0000E+00


iteration   single              double
 number   eigenvalue          eigenvalue
          estimate            estimate



   1       3.736E+05


   2       1.715E+04


   3       1.605E+04       1.591E+04 4.778E+04


   4       1.592E+04       1.591E+04 4.775E+04


   5       1.591E+04       1.591E+04 4.775E+04



double eigenvalue estimates have converged


frequency in radians per time=  1.26118E+02  in cycles per time=   2.00723E+01


  e i g e n v e c t o r


 1       0.           0.           0.           0.           0.           0.
 7  2.21605E-14 -1.17254E-02  1.56485E-11  9.49363E-13 -4.98482E-12 -3.83090E-03
13  4.39413E-14 -4.50409E-02  5.70641E-11  1.88246E-12 -8.59527E-12 -7.19697E-03
19  6.49696E-14 -9.71648E-02  1.16171E-10  2.78331E-12 -1.08993E-11 -1.01012E-02
25  8.48853E-14 -0.16534      1.85522E-10  3.63648E-12 -1.20404E-11 -1.25506E-02
31  1.03348E-13 -0.24689      2.58763E-10  4.42738E-12 -1.22377E-11 -1.45584E-02
37  1.20041E-13 -0.33920      3.31033E-10  5.14247E-12 -1.17651E-11 -1.61451E-02
43  1.34679E-13 -0.43984      3.99197E-10  5.76953E-12 -1.09178E-11 -1.73397E-02
49  1.47013E-13 -0.54657      4.61857E-10  6.29786E-12 -9.97280E-12 -1.81807E-02
55  1.56832E-13 -0.65740      5.19128E-10  6.71843E-12 -9.15208E-12 -1.87171E-02
61  1.63967E-13 -0.77068      5.72215E-10  7.02407E-12 -8.59247E-12 -1.90083E-02
67  1.68298E-13 -0.88515      6.22832E-10  7.20957E-12 -8.32447E-12 -1.91252E-02
73  1.69750E-13 -1.0000       6.72528E-10  7.27175E-12 -8.26004E-12 -1.91495E-02
```

Typical iterations

Lowest frequency

Using the Inverse Power Sweep Method, the vector printed is normalized such that the largest absolute value is one.

```
frequency in radians per time=  2.18511E+02  in cycles per time=   3.47771E+01
e i g e n v e c t o r


 1       0.           0.           0.           0.           0.           0.
```

Second mode frequency

```
 7 -5.14206E-12 -2.36136E-07 -1.17254E-02 -2.20895E-10  3.83090E-03 -6.88670E-08
13 -1.01960E-11 -7.09507E-07 -4.50409E-02 -4.38005E-10  7.19697E-03 -7.94276E-08
19 -1.50754E-11 -1.07989E-06 -9.71648E-02 -6.47612E-10  1.01012E-02 -3.55182E-08
25 -1.96966E-11 -1.04273E-06 -0.16534     -8.46126E-10  1.25506E-02  5.46611E-08
31 -2.39805E-11 -3.56489E-07 -0.24689     -1.03015E-09  1.45584E-02  1.78400E-07
37 -2.78540E-11  1.13308E-06 -0.33920     -1.19654E-09  1.61451E-02  3.19570E-07
43 -3.12507E-11  3.47805E-06 -0.43984     -1.34244E-09  1.73397E-02  4.60644E-07
49 -3.41127E-11  6.62751E-06 -0.54657     -1.46537E-09  1.81807E-02  5.85262E-07
55 -3.63910E-11  1.04427E-05 -0.65740     -1.56323E-09  1.87170E-02  6.80935E-07
61 -3.80467E-11  1.47277E-05 -0.77068     -1.63434E-09  1.90083E-02  7.41521E-07
67 -3.90515E-11  1.92741E-05 -0.88515     -1.67750E-09  1.91252E-02  7.69168E-07
73 -3.93884E-11  2.39143E-05  -1.0000     -1.69197E-09  1.91495E-02  7.75594E-07



                          mode    1     generalized mass= 2.363E-02


                          mode    2     generalized mass= 2.363E-02


                          end of calculations for mode    2


                          norm of remaining search vector 0.972E+00


                          eigenvalues extracted with shift point of  0.0000E+00



                          iteration   single             double
                           number   eigenvalue          eigenvalue
                                     estimate            estimate


                              1       5.803E+07

                              2       9.747E+05

                              3       6.234E+05       6.205E+05 6.850E+06

                              4       6.228E+05       6.231E+05 3.344E+06

                              5       6.230E+05       6.231E+05 2.070E+06

                              6       6.230E+05       6.230E+05 1.894E+06
```

```
                     single eigen estimate has converged



                     frequency in radians per time=   7.89318E+02  in cycles per time=   1.25624E+02



                      e i g e n v e c t o r


  1      0.           0.           0.           0.           0.           0.
  7  8.52149E-10  6.66679E-02  2.52623E-05  2.47792E-07 -7.77661E-06  2.05271E-02
 13  1.68972E-09  0.22617      8.56614E-05  4.91344E-07 -1.17325E-05  3.09966E-02
 19  2.49837E-09  0.41961      1.58830E-04  7.26489E-07 -1.20901E-05  3.19929E-02
 25  3.26428E-09  0.59357      2.24474E-04  9.49204E-07 -9.32675E-06  2.47695E-02
 31  3.97434E-09  0.70426      2.65974E-04  1.15568E-06 -4.18530E-06  1.12791E-02
 37  4.61639E-09  0.72135      2.71830E-04  1.34238E-06  2.38556E-06 -5.98374E-03
 43  5.17945E-09  0.63040      2.36574E-04  1.50611E-06  9.34592E-06 -2.42827E-02
 49  5.65390E-09  0.43312      1.60889E-04  1.64407E-06  1.57117E-05 -4.10265E-02
 55  6.03160E-09  0.14535      5.08025E-05  1.75390E-06  2.07107E-05 -5.41802E-02
 61  6.30610E-09 -0.20753     -8.40258E-05  1.83372E-06  2.39262E-05 -6.26433E-02
 67  6.47270E-09 -0.59711     -2.32794E-04  1.88216E-06  2.54101E-05 -6.65499E-02
 73  6.52855E-09 -1.0000      -3.86617E-04  1.89840E-06  2.57579E-05 -6.74658E-02



                    mode    3     generalized mass= 2.331E-02



                    end of calculations for mode    3



                    norm of remaining search vector 0.999E+00



                    eigenvalues extracted with shift point of  0.0000E+00
```

```
                    iteration   single               double
                     number  eigenvalue             eigenvalue
                              estimate              estimate


                       1      1.879E+06


                       2      1.871E+06


                       3      1.870E+06      1.870E+06 5.040E+06


                       4      1.870E+06      1.870E+06 4.978E+06


                    single eigen estimate has converged




                    frequency in radians per time=   1.36758E+03  in cycles per time=   2.17657E+02


                     e i g e n v e c t o r


 1       0.            0.            0.            0.            0.            0.
 7 -3.28917E-09 -2.62291E-04 -6.66659E-02 -1.58766E-05  2.05265E-02 -7.48531E-05
13 -6.52206E-09 -7.53579E-04 -0.22616     -3.14815E-05  3.09959E-02 -7.80040E-05
19 -9.64336E-09 -1.08781E-03 -0.41960     -4.65478E-05  3.19925E-02 -2.66028E-05
25 -1.25997E-08 -1.02488E-03 -0.59356     -6.08176E-05  2.47695E-02  4.84645E-05
31 -1.53404E-08 -5.32771E-04 -0.70425     -7.40469E-05  1.12795E-02  1.10612E-04
37 -1.78186E-08  2.15488E-04 -0.72134     -8.60091E-05 -5.98302E-03  1.30181E-04
43 -1.99920E-08  9.18407E-04 -0.63039     -9.64998E-05 -2.42820E-02  9.53692E-05
49 -2.18232E-08  1.26971E-03 -0.43313     -1.05339E-04 -4.10260E-02  1.63255E-05
55 -2.32811E-08  1.07981E-03 -0.14535     -1.12376E-04 -5.41800E-02 -7.97281E-05
61 -2.43407E-08  3.43910E-04  0.20753     -1.17491E-04 -6.26435E-02 -1.60855E-04
67 -2.49837E-08 -7.75832E-04  0.59711     -1.20595E-04 -6.65502E-02 -2.05888E-04
73 -2.51993E-08 -2.05746E-03   1.0000     -1.21635E-04 -6.74661E-02 -2.17898E-04


                     mode     4     generalized mass= 2.330E-02


                     end of calculations for mode     4
```

```
                      norm of remaining search vector 0.100E+01


                      eigenvalues extracted with shift point of  0.0000E+00



                      iteration   single             double
                       number  eigenvalue           eigenvalue
                               estimate             estimate



                          1      4.960E+06


                          2      4.938E+06


                          3      4.928E+06      4.919E+06 7.165E+06


                          4      4.923E+06      4.919E+06 7.165E+06


                      double eigenvalue estimates have converged



                      frequency in radians per time=   2.21788E+03  in cycles per time=   3.52987E+02


                       e i g e n v e c t o r



   1       0.            0.            0.            0.            0.            0.
   7  2.67277E-07 -0.17145      1.22142E-04 -1.72247E-07 -3.74585E-05 -4.92685E-02
  13  5.29980E-07 -0.50043      4.10921E-04 -3.41547E-07 -5.56801E-05 -5.35191E-02
  19  7.83615E-07 -0.74348      7.54566E-04 -5.05003E-07 -5.61098E-05 -2.31063E-02
  25  1.02384E-06 -0.74539      1.05442E-03 -6.59819E-07 -4.16989E-05  2.32769E-02
  31  1.24655E-06 -0.47713      1.23381E-03 -8.03345E-07 -1.67543E-05  6.33800E-02
  37  1.44793E-06 -3.49574E-02  1.24455E-03 -9.33126E-07  1.36532E-05  7.89581E-02


  43  1.62454E-06  0.40467      1.06920E-03 -1.04694E-06  4.44841E-05  6.23535E-02
  49  1.77335E-06  0.65879      7.18502E-04 -1.14284E-06  7.15042E-05  1.90277E-02
  55  1.89182E-06  0.61078      2.24627E-04 -1.21919E-06  9.18740E-05 -3.52535E-02
  61  1.97791E-06  0.25203     -3.68298E-04 -1.27468E-06  1.04492E-04 -8.17005E-02
  67  2.03017E-06 -0.32723     -1.01513E-03 -1.30835E-06  1.10125E-04 -0.10766
  73  2.04769E-06  -1.0000     -1.68092E-03 -1.31964E-06  1.11410E-04 -0.11461
```

```
                      frequency in radians per time=   2.67683E+03  in cycles per time=   4.26031E+02



                      e i g e n v e c t o r


 1       0.             0.             0.            0.              0.            0.
 7 -1.14293E-05 -1.98598E-05 -2.79893E-04 -0.13053      7.92746E-05 -4.66454E-06
13 -2.26631E-05 -3.54915E-05 -7.90268E-04 -0.25882      7.87599E-05  6.71960E-07
19 -3.35091E-05 -5.11546E-06 -1.10342E-03 -0.38268      1.82696E-05  9.23469E-06
25 -4.37817E-05  6.37605E-05 -9.60090E-04 -0.50000     -6.65933E-05  1.22341E-05
31 -5.33053E-05  1.20615E-04 -3.40368E-04 -0.60876     -1.33841E-04  5.12418E-06
37 -6.19167E-05  1.11739E-04  5.41289E-04 -0.70711     -1.49798E-04 -8.42376E-06
43 -6.94688E-05  2.61808E-05  1.32662E-03 -0.79335     -1.01762E-04 -1.87006E-05
49 -7.58322E-05 -8.80340E-05  1.65823E-03 -0.86603     -2.63327E-06 -1.70672E-05
55 -8.08981E-05 -1.52172E-04  1.32078E-03 -0.92388      1.14946E-04 -2.66624E-06
61 -8.45798E-05 -1.09984E-04  3.18856E-04 -0.96593      2.13194E-04  1.66125E-05
67 -8.68144E-05  3.51276E-05 -1.14659E-03 -0.99144      2.67414E-04  3.02115E-05
73 -8.75635E-05  2.32263E-04 -2.80665E-03  -1.0000      2.81824E-04  3.43929E-05



                      mode    5     generalized mass= 2.278E-02



                      end of calculations for mode    5



                      eigenvalue extraction complete - all requested modes have been found

                      e n d   o f   i n c r e m e n t   0
                       time =        0.72
```

NOTE

The RECOVER option puts eigenvectors into the post file. It may also be used to obtain modal stresses and forces

```
recover
----------


write eigenvectors on post tape for mode      1 to    5
zero reference amplitude entered, no scaling of eigenvectors will be done.


continue
----------



                 s t a r t   o f   i n c r e m e n t  0.   1







                 e n d   o f   i n c r e m e n t  0.   1

                 formatted post data at increment   0.   1   on tape 19
                  time =        0.77

modal recovery will be performed at this stage for mode      2


                 s t a r t   o f   i n c r e m e n t  0.   2
```

Continues until all five modes are placed on the post tape

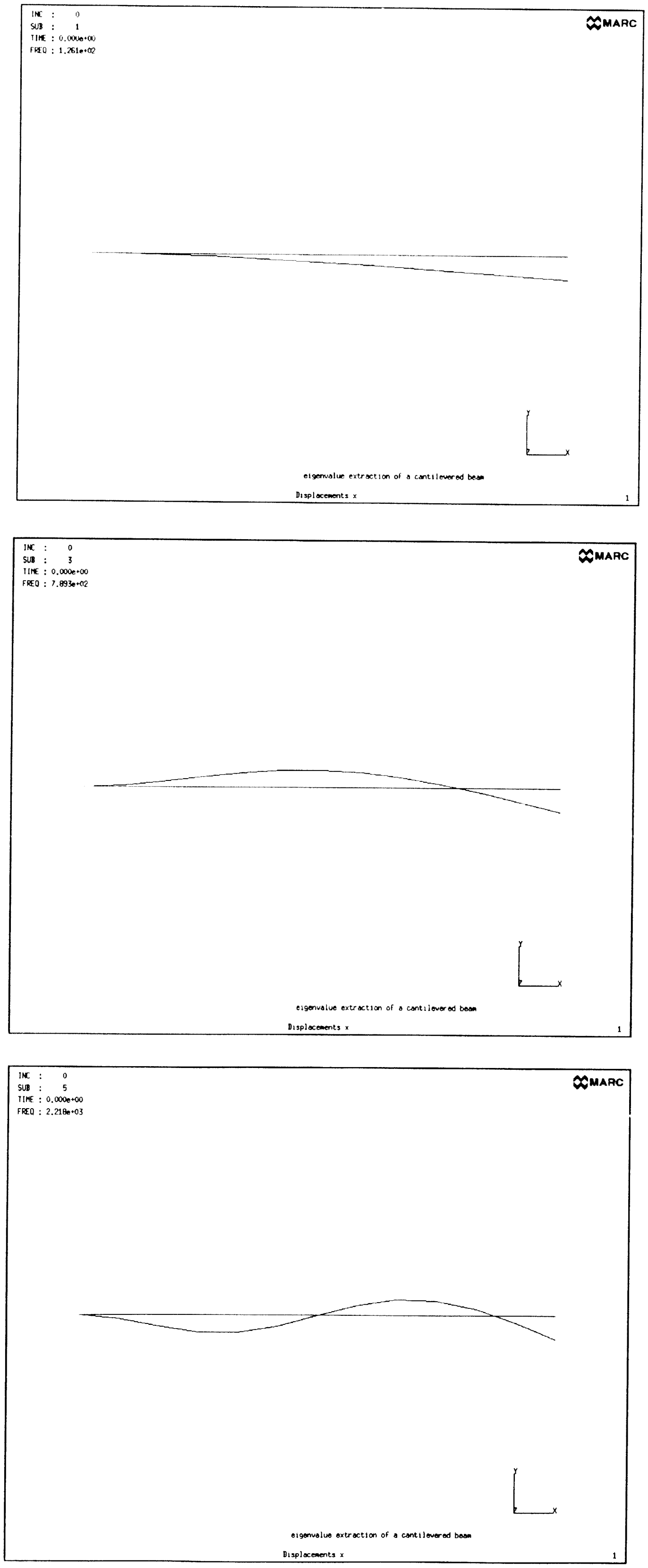
Results

Let's compare the calculated eigenvalues (frequencies) with theoretical values (e.g., R. J. Roark and W. C. Young, *Formulas for Stress and Strain,* 5th ed., McGraw-Hill, 1976, p. 576).

To be able to correlate the calculated frequencies and theory, we must first remember that MARC Element 52 is a 3-D beam, and the inverse power sweep method will extract successive modes using values of I_{xx} as well as I_{yy}. In other words, consecutive modes can vibrate in-plane or out-of-plane. In our case, I_{xx} (16.0) is three times greater than I_{yy} (5.33). This means we must use the correct moment of inertia in calculating the theoretical frequencies in order to make a meaningful comparison.

Mode	Vibrating In	Theor. Freq. (rad/sec)	Using	MARC Calc. (rad/sec)	% Difference
1	X-Y Plane	126.56	I_{yy}	126.12	-0.35
2	X-Z Plane	219.20	I_{xx}	218.51	-0.32
3	X-Y Plane	790.97	I_{yy}	789.32	+0.21
4	X-Z Plane	1370.0	I_{xx}	1367.58	+0.18
5	X-Y Plane	2218.2	I_{yy}	2217.88	-0.14

The agreement is excellent. All the calculated frequencies are within four-tenths of one percent of the theoretical values. The following plots show the first three X-Y plane mode shapes, which correspond to the MARC-calculated Modes 1, 3, and 5 using I_{yy} .



Mode	**rad/sec**
1	**126.12**
3	**789.32**
5	**2217.88**

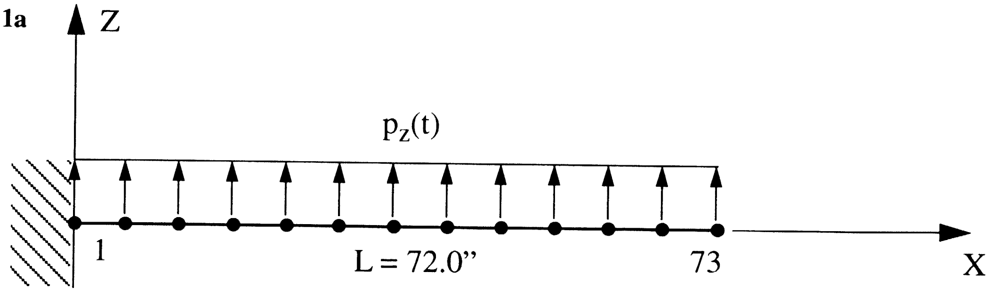
Figure 3A.2 Eigenmodes

Example 3B

Linear Dynamic Analysis Using Direct Integration

Sketch

3B.1a



3B.1b

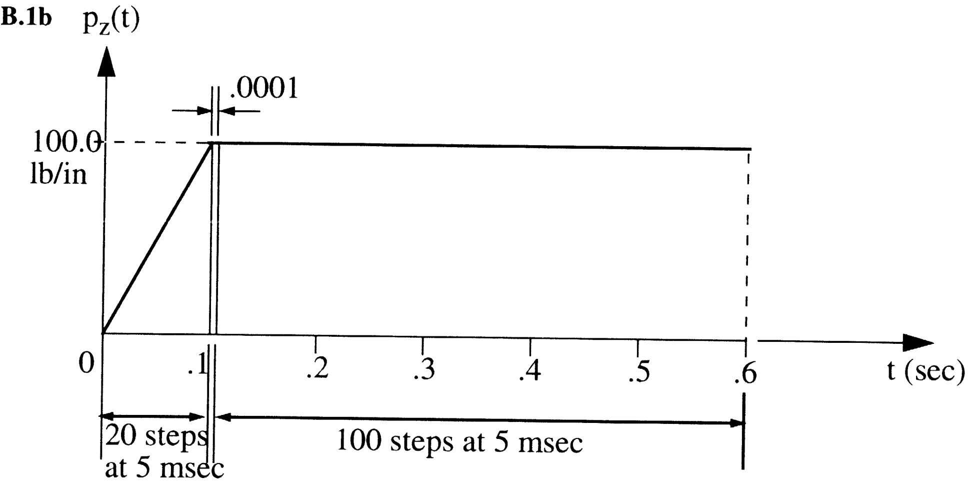


Figure 3B.1a Cantilevered Beam

Figure 3B.1b Time History of Load

Model

The Example 3A finite element beam model is used again. The material properties, boundary conditions, and geometric properties remain the same. Here, we'll just highlight the input options needed to describe the dynamic load and the transient analysis using direct integration.

Loads

A uniform distributed load is applied in the positive global Z-direction, along the complete length of the beam. The load magnitude is ramped up from 0 lb/in to 100 lb/in in 0.1 second using twenty 5-msec time steps. The load is then held constant for the remainder of the analysis. The dynamic analysis is carried out over a total elapsed time of 0.6 sec.

Input

A complete input echo is included. Notice how this input file is identical to Example 3A in the FE mesh topology definition (including the same node and element set definitions), material properties assignment, boundary conditions specification, and geometric properties description–but is very different in the remaining portions.

PARAMETER Section

All the PARAMETER options are the same as those of Example 3A except the DYNAMIC option. On the "DYNAMIC" line, instead of putting in a "1" in the second field to denote eigenvalue extraction, we now use a "2" to mean we've selected the Newmark-beta method of direct integration of the equations of motion. The Newmark-beta method is unconditionally stable in linear dynamic analysis, exhibits no numerical damping, and is probably the most popular direct integration method used in finite element analysis. It offers a variable time-stepping algorithm. (See *MARC Volume A* for details of this method.)

MODEL DEFINITION Section

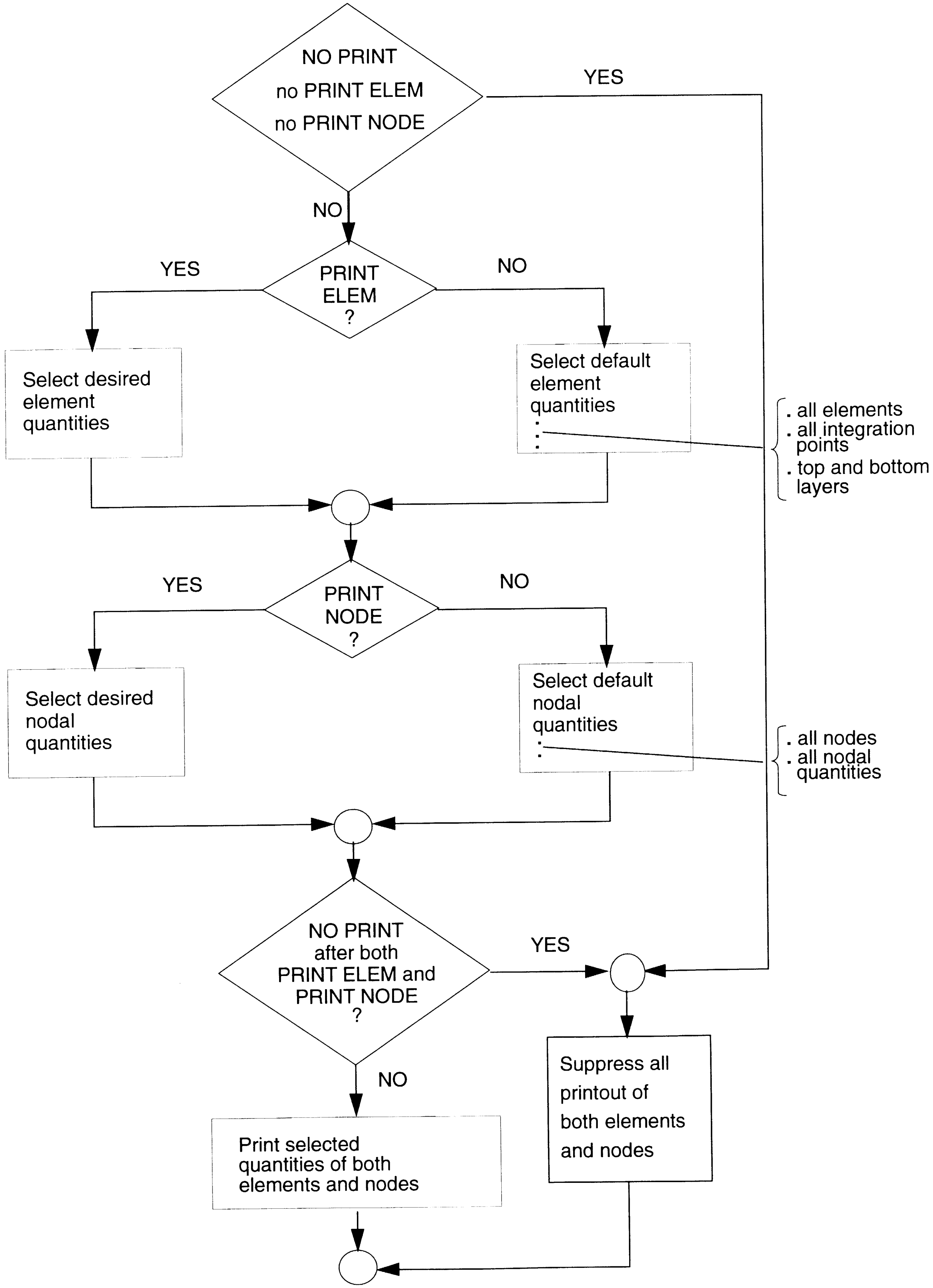
The first part of the MODEL DEFINITION options, from line 7 ("CONNECTIVITY") through line 56 is identical to that of Example 3A. We'll now explain line 57 ("DIST LOADS") through line 70 ("END OPTION"), which terminates the MODEL DEFINITION options.

The first block is the DIST LOADS block (See *MARC Volume C*). It is important to realize that these loads are incremental values. This block of data allows pressure loads (surface and volumetric) to be specified. The blank line merely means we don't have to count how many lines follow. The next line (3,0.,) indicates that the Load Type is 3, which for Element 52 means a uniform load per unit length in the global Z-direction, and the initial magnitude of this distributed load is 0. The last line ("1 TO 12") is the 4th line series and refers to the fact that the distributed load is applied over elements 1 to 12. (Alternatively, we could have used ALLE here, since we had previously defined element set ALLE to comprise all twelve elements in the model.) This block is needed to specify load type and applicable elements so that the program can allocate memory for these loads. Even though the distributed load is not applied until increment one, it must be defined in the MODEL DEFINITION section. The distributed loads are changed by the program into equivalent nodal loads.

The second block is the PRINT ELEM block. We first saw the use of this option for selective printing of element quantities in Example 1. The "STRESS" line means total stresses are to be output. The blank lines mean we do not want to print results for the elements in the model.

The third and last block in the MODEL DEFINITION options is the PRINT NODE block. The "TOTAL" line designates the total nodal displacement to be printed out for the node(s) to be named. The 73 on the last line in this block means we only want nodal quantities to be printed out for node 73.

The flowchart on the next page diagrams what happens when you choose the PRINT ELEM or PRINT NODE selective printing option in MARC. Note that the ordering of input between PRINT ELEM/NODE and NO PRINT determines which takes place. (Also, you should avoid the use of the PRINT CHOICE option!).


NO PRINT
no PRINT ELEM
no PRINT NODE
YES
NO
PRINT ELEM ?
YES
NO
Select desired element quantities
Select default element quantities
. all elements
. all integration points
. top and bottom layers
PRINT NODE ?
YES
NO
Select desired nodal quantities
Select default nodal quantities
. all nodes
. all nodal quantities
NO PRINT after both PRINT ELEM and PRINT NODE ?
YES
NO
Print selected quantities of both elements and nodes
Suppress all printout of both elements and nodes

The "END OPTION" line ends the MODEL DEFINITION section.

HISTORY DEFINITION Options

Here, we will illustrate two HISTORY DEFINITION options: DIST LOADS (same function as its counterpart in MODEL DEFINITION) and DYNAMIC CHANGE.

If you examine the input file carefully, you'll notice that the HISTORY DEFINITION part of the input (lines 71 through 87) actually consists of three distinct portions (lines 71 to 77, lines 78 to 84, lines 85 to 87), each ending with a CONTINUE line, with the first two portions containing a DIST LOADS block and all three containing a DYNAMIC CHANGE block. The rationale for this careful step-by-step input of the dynamic load will now be explained. The procedure adopted will allow us to input the ramp forcing function successfully, minimize the effects of the sharp "knee" at 0.2 sec., and avoid potential numerical difficulties in the dynamic analysis.

In the first portion (lines 71 to 77), we first see the DIST LOADS block (see *MARC Volume C*). This option, like its MODEL DEFINITION counterpart, allows surface and volumetric pressure loads to be specified. Again, remember that these loads are *incremental* values. The blank line merely means we do not have to bother counting how many lines are in this block. On the third line (line 73), the 3 again means Load Type 3, which for Element 52 refers to uniform load per unit length in the global Z-direction. The 5. on the same line is the value of the incremental distributed load per step, 5.0 lb/in. The 1 to 12 on the next line represents the twelve elements to which this load applies (again, we could have used the element set named ALLE). Then come the two lines in the DYNAMIC CHANGE block. The DYNAMIC CHANGE option (see *MARC Volume C*) specifies the parameters required for the time integration (using either the direct integration procedure or the modal superposition procedure). On the ".005,.1,20," line (line 76), the .005 is the time step size (5 msec.), the .1 is the period of time for this dynamic change condition, and 20 is the number of time steps in this set of boundary conditions. In other words, to climb the ramp portion of the dynamic load (which occurs from 0 to 0.1 sec.), we have used a total of twenty 5-msec time steps. The purpose of using such small time steps in this initial 0.1 sec. interval is to capture any significant dynamic response in the beam as the load is first applied. The "CONTINUE" line ends this first portion.

The second portion (lines 78 to 84) is a safe, practical method for turning the "knee" of the ramp load without incurring numerical difficulties, since in real life a discontinuity like this sharp knee does not exist. The DIST LOADS block is similar to that in the first portion, except that the third line in this block (3,0.,) shows a 0. in the second field. This means the incremental change in load for this interval is zero. Notice then, in the DYNAMIC CHANGE block which follows, the ".0001,.0001,1" line represents a single very small time step size of .0001 sec (100 microseconds), which is applicable to this zero pressure load change. This is good modeling practice in order to simulate such forcing functions with sharp discontinuities. The "CONTINUE" line terminates this second portion.

The third and final portion (lines 85 to 87) applies the constant pressure load for the remainder of the analysis, to 0.6 sec. The only block is DYNAMIC CHANGE. Notice the absence of a DIST LOADS block here means the previous incremental pressure load change of zero is still applicable. That is, a zero load change from before indicates that the previous pressure load of 5.0 lb/in. remains on the beam. The ".005,.5,100," line means this interval is to have a time step size of 0.005 sec., for a duration of 0.5 sec, and there is a total of 100 such time steps. Since there was a 0.0001 sec. duration in the second portion, the total elapsed time at the end of the transient analysis is actually the sum of 0.10, 0.0001, and 0.5–or a total of 0.6001 sec. The final "CONTINUE" lines ends the HISTORY DEFINITION input as well as the entire input file.

Output

Only a portion of the printed output is included: the printed echo, the printed output of increments 0 and 1, and finally, the results of the final increment (121). Increment 0 is a null step because no preload exists in this example. At the end of each increment, the "total dynamic transient time" is printed out: 0.005 sec. after increment 1, 0.6001 sec. after increment 121. Also shown is the elapsed MARC solution "TIME": 0.66 sec. after increment 1, 17.75 sec. after increment 121. After each increment, MARC informs you that the formatted post data has been stored on tape 19.

input data

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
TITLE          LINEAR DYNAMICS BY DIRECT INTEGRATION
SIZING         100000    12    13
ELEMENTS      52
COMMENT         USE NEWMARK BETA DIRECT INTEGRATOR
DYNAMIC,2,
SHELL SECT,3
END
CONNECTIVITY
   12     0     0
    1    52     1     7
    2    52     7    13
    3    52    13    19
    4    52    19    25
    5    52    25    31
    6    52    31    37
    7    52    37    43
    8    52    43    49
    9    52    49    55
   10    52    55    61
   11    52    61    67
   12    52    67    73
COORDINATES
    3    13     0     0
    1   1.00000   0.00000    0.00000
    7   7.00000   0.00000    0.00000
   13  13.00000   0.00000    0.00000
   19  19.00000   0.00000    0.00000
   25  25.00000   0.00000    0.00000
   31  31.00000   0.00000    0.00000
   37  37.00000   0.00000    0.00000
   43  43.00000   0.00000    0.00000
   49  49.00000   0.00000    0.00000
   55  55.00000   0.00000    0.00000
   61  61.00000   0.00000    0.00000
   67  67.00000   0.00000    0.00000
   73  73.00000   0.00000    0.00000
DEFINE     NODE      SET       FIXME
    1
DEFINE     ELEMENT   SET       ALLE
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

Direct integration by Newmark-beta

```
               5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
          --------------------------------------------------------------------------------
card 40   1 TO 12
          ISOTROPIC

          1,
          30.E6,.3,7.6754E-4
card 45   ALLE
          FIXED DISP

          0.,0.,0.,0.,0.,0.,
          1 TO 6
card 50   FIXME
          GEOMETRY

          6.,16.,5.33,0.,1.,0.,
          ALLE
card 55   POST
          ,,,1
          DIST LOADS

          3,0.,
card 60   1 TO 12
          PRINT ELEM

          STRESS

card 65
          PRINT NODE

          TOTAL
          73
card 70   END OPTION
          DIST LOADS

          3,5.,
          1 TO 12
card 75   DYNAMIC CHANGE
          .005,.1,20,
          CONTINUE
          DIST LOADS

card 80   3,0.,
          1 TO 12
          DYNAMIC CHANGE
          .0001,.0001,1
          CONTINUE
          --------------------------------------------------------------------------------
               5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

Suppress Element Output

Ramp Up Load

Take small time steps when load changes

```
            5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
         --------------------------------------------------------------------------------
card   85    DYNAMIC CHANGE
             .005,.5,100,
             CONTINUE
         --------------------------------------------------------------------------------
            5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
         --------------------------------------------------------------------------------
         --------------------------------------------------------------------------------
```

Finish the time period

```
                maximum connectivity is     2  at node     7


                maximum half-bandwidth is     2  between nodes     1  and     7

                 number of profile entries including fill-in is         25

                 number of profile entries excluding fill-in is         25


                total workspace needed with in-core matrix storage =      20963



                load increments associated with each degree of freedom
                summed over the whole model

                distributed loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                point loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                increment zero is a null step


                total dynamic transient time = 0.00000E+00
```

At time = zero, nothing happens

```
          distributed load    type    current
          list number                 magnitude


                 1            3         0.          0.          0.

                  e n d   o f   i n c r e m e n t   0

                  formatted post data at increment   0.   0   on tape 19
                   time =        0.44



      dist loads
      ----------

      read from unit     5
      type index distributed load
         3    0  0.5000000E+01  0.0000000E+00  0.0000000E+00
      from element     1 to element    12 by     1

      dynamic change
      ----------


         time       time     maximum   assembly  max iter
      increment    period     steps    interval  mcreep
       5.000E-03 1.000E-01     20          0      5

      continue
      ----------



      auto control specified for time of 0.100E+00


                   s t a r t   o f   i n c r e m e n t   1
```

Start of a typical increment

```
                    load increments associated with each degree of freedom
                    summed over the whole model


                    distributed loads
 0.000E+00 0.000E+00 3.600E+02 0.000E+00-3.553E-15 0.000E+00



                    point loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                    start of assembly
                     time =       0.48




                    start of matrix solution
                     time =       0.55




                    singularity ratio    1.2887E-03



                    end of matrix solution
                     time =       0.56
```

Test for convergence

```
                   maximum residual force  at node   43 degree of freedom    3 is equal to    0.544E-10
                   maximum reaction force  at node    1 degree of freedom    3 is equal to    0.185E+03
                   convergence ratio                                                          0.295E-12
```

Convergence ratio indicates error of less than .01% - which is very good!

```
 MARC         output for increment    1.       linear dynamics by direct integration




                   dynamic change has reached time of 0.500E-02  of total time period 0.100E+00




                   total dynamic transient time = 5.00000E-03
```

```
          distributed load    type    current
          list number                 magnitude


                      1           3      5.000          0.          0.

                        e n d   o f   i n c r e m e n t   1

                        formatted post data at increment   1.   0   on tape 19
                         time =        0.66
                                ●                                          ●
                                ●                                          ●
                                ●                                          ●
                                ●                                          ●

                        s t a r t   o f   i n c r e m e n t 120




                        load increments associated with each degree of freedom
                        summed over the whole model

                        distributed loads
    0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00



                        point loads
    0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                        end of matrix back substitution
                         time =       17.51


                        maximum residual force  at node   67 degree of freedom    3 is equal to    0.161E-09
                        maximum reaction force  at node    1 degree of freedom    3 is equal to    0.708E+04
                        convergence ratio                                                          0.228E-13



MARC          output for increment  120.        linear dynamics by direct integration
```

NOTE

In linear dynamic analysis, if the time step does not change, reassembling the stiffness matrix is not necessary.

```
                      dynamic change has reached time of 0.495E+00  of total time period 0.500E+00


                      total dynamic transient time = 5.95100E-01




                      n o d a l   p o i n t   d a t a




                      t o t a l   d i s p l a c e m e n t s


    73       0.          0.       0.67892          0.     -1.25573E-02       0.

                      summary of externally applied loads

 0.00000E+00    0.00000E+00    0.72000E+04    0.00000E+00   -0.22737E-12    0.00000E+00

                      summary of reaction/residual forces

 0.00000E+00    0.00000E+00   -0.70792E+04    0.00000E+00    0.25253E+06    0.00000E+00



                 distributed load    type    current
                 list number                 magnitude


                      1           3     100.0          0.          0.

                      e n d   o f   i n c r e m e n t 120

                      formatted post data at increment 120.   0   on tape 19
                       time =      17.61


                      s t a r t   o f   i n c r e m e n t 121
```

```
                         load increments associated with each degree of freedom
                         summed over the whole model


                         distributed loads
   0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00



                         point loads
   0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                         end of matrix back substitution
                          time =      17.66


                        maximum residual force  at node   43 degree of freedom    3 is equal to    0.818E-10
                        maximum reaction force  at node    1 degree of freedom    3 is equal to    0.697E+04
                        convergence ratio                                                          0.117E-13


MARC            output for increment  121.       linear dynamics by direct integration




                          dynamic change has reached time of 0.500E+00  of total time period 0.500E+00



                          total dynamic transient time = 6.00100E-01





                          n o d a l   p o i n t   d a t a





                          t o t a l   d i s p l a c e m e n t s






   73        0.             0.       0.66468          0.     -1.22902E-02       0.

                           summary of externally applied loads
```

```
0.00000E+00     0.00000E+00     0.72000E+04     0.00000E+00    -0.22737E-12     0.00000E+00

                      summary of reaction/residual forces

0.00000E+00     0.00000E+00    -0.69689E+04     0.00000E+00     0.24757E+06     0.00000E+00



                distributed load    type    current
                list number                 magnitude


                        1            3       100.0          0.          0.

                         end  of  increment 121

                         formatted post data at increment 121.   0   on tape 19
                          time =       17.75



                         *** end of input deck - job ends




                                   marc exit number 3004
```

Results

The following plots show the deflected shape of the entire beam at the end of the transient analysis, and a time history plot of the Z-displacement at the beam tip node. The smoothness of the history plot indicates that the time steps we selected were adequate to characterize the dynamic response; a very jagged plot would have meant we needed to reduce the time step sizes.

Note that in the second figure, the small oscillations about a certain constant displacement value of approximately 0.70 are real, and represent the true dynamic response of the beam about the static displacement value. The dynamic response in this example is undamped; Example 3C will demonstrate the damped behavior of the same beam subjected to initial conditions but solved using modal superposition.

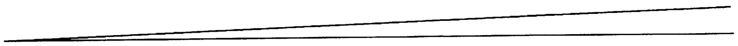


Figure 3B.2 Deflected Shape

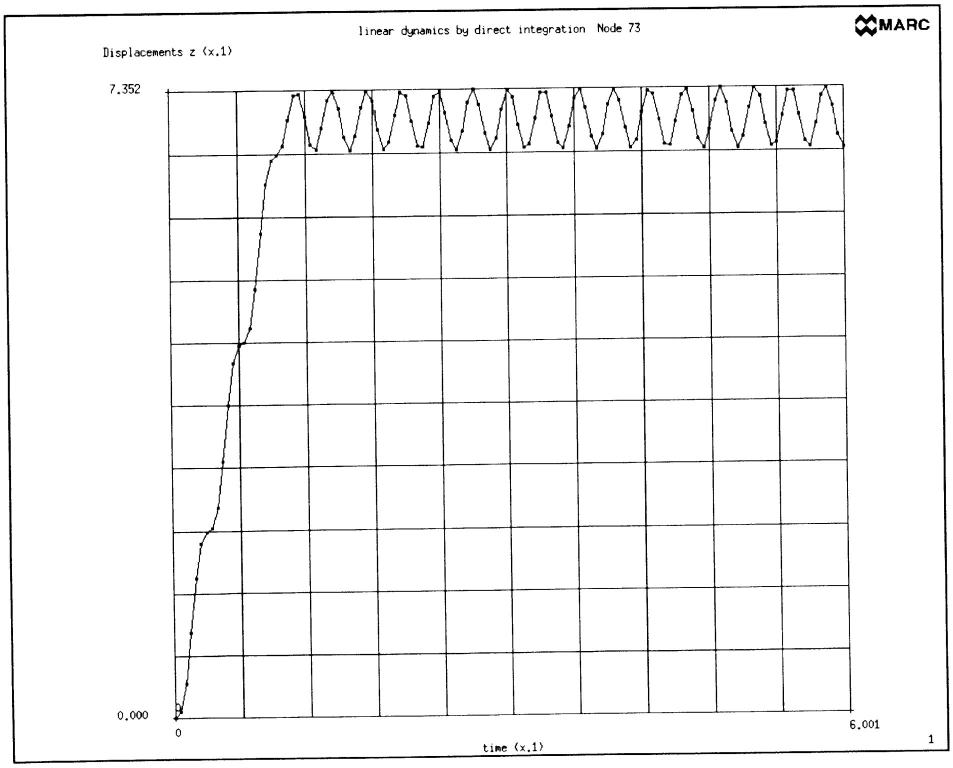


Figure 3B.3 Time History of Tip Deflection

Example 3C

Damped Modal Superposition Response Subjected to Initial Conditions

Sketch

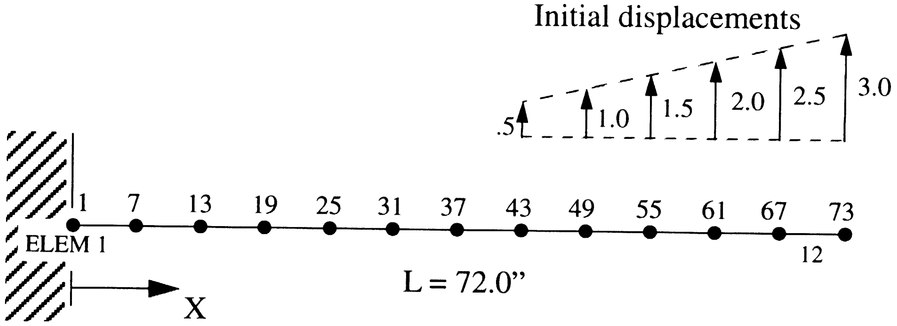


Figure 3C.1 **Cantilevered Beam with Initial Displacements**

Model

Same as for Examples 3A and 3B.

Initial Conditions

Instead of using the time-varying distributed load of Example 3B, here we will specify some initial conditions–consisting of initial nodal displacements applied to the six nodes on the right half of the cantilevered beam. These applied initial nodal displacements resemble the first bending mode of the beam.

Input

A complete input echo is included. Notice how this input file is identical to those of Examples 3A and 3B in the mesh definition, material properties assignment, boundary conditions specification, and geometric properties description (through line 54)–but is different in the remaining portions due to addition of damping to the beam, application of the initial displacements, and the modal superposition solution.

PARAMETER Section

All the PARAMETER options are the same as before except the "DYNAMIC" line. On the DYNAMIC line, the "1" in the second field indicates we want to do modal superposition dynamic response analysis, the "5" in the third field means five modes are to be used in the modal superposition analysis, and the "1" in the fourth field says that we want to use the Lanczos method for eigenvalue extraction prior to the modal superposition analysis. The first part (lines 8 through 54) of the MODEL DEFINITION options is identical to those of Examples 3A and 3B.

MODEL DEFINITION Section

The DAMPING block (see *MARC Volume C*) in this example consists of the next two lines. This block allows the input of damping factors for use with the dynamic analysis options. For the modal superposition case (as in this example), you input the fraction of critical damping associated with each mode of the solution. On line 58 following the "DAMPING" line, the "0.5" shown in the first field is the DAMP(1) parameter, or the fraction of the critical damping factor for the first mode, that is, 5%. The blank second field means we want zero damping for the second mode, that is, DAMP(2) is 0%. The "0.05" value in the third field indicates that we want the damping factor for the third mode to be 0.05 also (or 5%). Since we had specified on the DYNAMIC parameter option that we want to use five modes and we did not specify damping factors for modes 4 and 5, MARC will assume that they are zero.

The POST block (lines 59, 60) is the same as in Examples 3A and 3B; it tells MARC to create a post processor file for later post processing by Mentat II. The line (line 60) says that we would like to have a formatted post file.

The next block of data (lines 61 to 76) in the MODEL DEFINITION options is the INITIAL DISP block (*see MARC Volume C*). This block prescribes the initial displacements to be put on the model. line 64 ("6,") means NSET is six–the number of sets of prescribed displacements to follow. The next twelve lines are the six pairs of lines prescribing initial displacements on nodes 43, 49, 55, 61, 67, and 73–that is, the six nodes on the right half of the beam model. For instance, line 67 ("0.,1.,") after the line "43" means for node 43, apply an initial displacement of zero in the global X-direction (first degree of freedom) and 1.0 in the global Y-direction (second degree of freedom). This initial applied displaced shape resembles approximately the first bending mode of the cantilevered beam. (The reason for choosing this shape will be explained later under the *Results* section.)

The "NO PRINT" option (line 77) suppresses element and nodal output. The "END OPTION" line terminates the MODEL DEFINITION section.

HISTORY DEFINITION Section

In this example, the HISTORY DEFINITION section consists of three blocks: MODAL SHAPE, RECOVER, and DYNAMIC CHANGE.

The MODAL SHAPE option was first discussed in Example 3A, when we used it to specify eigenvalue extraction using the inverse power sweep method. This option is explained in *MARC Volume C*. Here, we had specified the Lanczos method for eigenvalue extraction, by using a "1" in the fourth field of the DYNAMIC PARAMETER option. Now, on line 80 after the "MODAL SHAPE" line, the "0.,0.,5" is interpreted as follows: the first field refers to the lowest modal frequency to be extracted–or the initial shift point in the Lanczos solution–and is specified to be zero here; the second field is the highest modal frequency to be extracted and is also zero; and the "5" in the third field means the number of requested modes to extract. As the first two entries are zero, MARC will calculate the lowest five modes. The "CONTINUE" line ends this MODAL SHAPE input block.

The RECOVER option (*see MARC Volume C*) was first used in Example 3A and is again used here for the same purpose. The "1,5" line means modes 1 through 5 are to be written to the post file.

The DYNAMIC CHANGE block ends the input for this example. We first encountered this block in Example 3B. This option (*see MARC Volume C*) specifies the parameters needed for the time integration in dynamic analysis. On line 88 (".001,.5,500,"), the ".001" is the time step size (1 msec.) chosen for the modal superposition analysis, the "0.5" (sec.) is the period of time for this dynamic change condition, and the "500" is the number of time steps (increments) using this set of boundary conditions.

NOTE

The time step size does not influence the accuracy of the solution when using modal superposition. The accuracy is determined by the number of modes used to represent the problem.

The "CONTINUE" line ends this block, as well as the input to this problem.

The time step size of 0.001 was chosen to give adequate resolution of the beam response. In this particular case, it is known that the response will occur primarily in modes 1 and 3. Therefore, the period of the highest significant mode (mode 3) is divided into eight time steps, giving a time step of 0.001.

Output

Only a portion of the output is included: the input echo; the pages showing the Lanczos eigenvalue extraction results; increment 1 modal superposition results; and increments 499 and 500 modal superposition results. (The analysis ends after increment 500, at which time the total elapsed time period is 0.5 sec.) After each increment, MARC informs you that the formatted post data has been stored on tape 19 and informs you of the total dynamic transient time.

input data

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
TITLE       MODAL SUPERPOSITION
SIZING        100000   12   13
ELEMENTS     52
COMMENT, THE DYNAMIC OPTION SPECIFIES MODAL DYNAMICS USING LANCZOS PLUS
COMMENT,  RECOVER TO PUT THE MODES ON THE POST TAPE
DYNAMIC,1,5,1,1
END
CONNECTIVITY
   12    0    0
    1   52    1    7
    2   52    7   13
    3   52   13   19
    4   52   19   25
    5   52   25   31
    6   52   31   37
    7   52   37   43
    8   52   43   49
    9   52   49   55
   10   52   55   61
   11   52   61   67
   12   52   67   73
COORDINATES
    3   13    0    0
    1   1.00000   0.00000   0.00000
    7   7.00000   0.00000   0.00000
   13  13.00000   0.00000   0.00000
   19  19.00000   0.00000   0.00000
   25  25.00000   0.00000   0.00000
   31  31.00000   0.00000   0.00000
   37  37.00000   0.00000   0.00000
   43  43.00000   0.00000   0.00000
   49  49.00000   0.00000   0.00000
   55  55.00000   0.00000   0.00000
   61  61.00000   0.00000   0.00000
   67  67.00000   0.00000   0.00000
   73  73.00000   0.00000   0.00000
DEFINE    NODE      SET       FIXME
    1
DEFINE    ELEMENT   SET       ALLE
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
--------------------------------------------------------------------------------
1 TO 12
ISOTROPIC

1,
30.E6,.3,7.6754E-4
ALLE
FIXED DISP

0.,0.,0.,0.,0.,0.,
1 TO 6
FIXME
GEOMETRY

6.,16.,5.33,0.,1.,0.,
ALLE
COMMENT, DAMPING FOR MODAL DYNAMICS SPECIFIES THE FRACTION OF CRITICAL
COMMENT,  DAMPING TO APPLY TO THE MODES IN THE PROBLEM.
DAMPING
0.05,,0.05
POST
,,,1
COMMENT, INITIAL DISPLACEMENT SPECIFIES THE DEFLECTED SHAPE OF THE
COMMENT,  BEAM WHEN THE TRANSIENT BEGINS.
INITIAL DIS
6,
0.,.5,
43,
0.,1.,
49,
0.,1.5,
55,
0.,2.,
61,
0.,2.5,
67,
0.,3.,
73,
NO PRINT
END OPTION
MODAL SHAPE
0.,0.,5
CONTINUE
--------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

The initial displacements are the excitation in this problem.

Extract Modes

```
                  5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                --------------------------------------------------------------------------------
                COMMENT, RECOVER INSTRUCTS MARC TO WRITE THE EIGENVECTORS ON THE POST TAPE
                RECOVER
                1,5
card    85      CONTINUE
                COMMENT,  BEGIN MODAL DYNAMICS
                DYNAMIC CHANGE
                .001,.5,500,
                CONTINUE
                --------------------------------------------------------------------------------
                  5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                --------------------------------------------------------------------------------
                --------------------------------------------------------------------------------
                          ●                                      ●
                          ●                                      ●
                          ●                                      ●
                          ●                                      ●
                     total workspace needed with in-core matrix storage =     22673


                     load increments associated with each degree of freedom
                     summed over the whole model

                     distributed loads
   0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                     point loads
   0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                     start of assembly
                      time =       0.39



                     start of matrix solution
                      time =       0.48



                     singularity ratio    5.7870E-04


                     end of matrix solution
                      time =       0.49
```

```
MARC           output for increment    0.      modal superposition


element with highest stress relative to yield is    1  where equivalent stress is 0.100E-19  of yield
```

Increment zero is a null increment.

```
                     total dynamic transient time = 0.00000E+00



                     e n d   o f   i n c r e m e n t   0

                     formatted post data at increment    0.   0   on tape 19
                      time =        0.59


         modal shape
         ----------


         lanczos eigenvalue procedure used
         ---------------------------------
         minimum frequency (cys/s)           0.00000E+00
         shift point (cys/s)                 0.00000E+00
         maximum frequency (cys/s)           0.00000E+00
         number of requested modes           5
         sturm sequence flag                 0
         restart option flag                 0
```

Requests 5 modes to be extracted using the Lanczos method.

```
         continue
         ----------
```

```
                     perform lanczos  iteration
       start lanczos run   1 with   10 vectors
                           time =       0.62

       start lanczos run   2 with   15 vectors
                           time =       0.65
```

Typical iteration procedure.

```
                     number of lanczos runs =    2
```

```
          mode     1      generalized mass= 1.000E+00
          frequency in radians per time = 1.261E+02
          frequency in cycles per time  = 2.007E+01

          mode     2      generalized mass= 1.000E+00
          frequency in radians per time = 2.185E+02
          frequency in cycles per time  = 3.478E+01

          mode     3      generalized mass= 1.000E+00
          frequency in radians per time = 7.893E+02
          frequency in cycles per time  = 1.256E+02

          mode     4      generalized mass= 1.000E+00
          frequency in radians per time = 1.368E+03
          frequency in cycles per time  = 2.177E+02

          mode     5      generalized mass= 1.000E+00
          frequency in radians per time = 2.218E+03
          frequency in cycles per time  = 3.530E+02

          e n d   o f   i n c r e m e n t   0
           time =       0.70
```

The NO PRINT option suppresses the printing of eigenvectors.

```
comment, recover instructs marc to write the eigenvectors on the post tape

recover
----------

write eigenvectors on post tape for mode     1 to    5
zero reference amplitude entered, no scaling of eigenvectors will be done.

continue
----------



          s t a r t   o f   i n c r e m e n t   0.   1




          e n d   o f   i n c r e m e n t   0.   1
```

```
                    formatted post data at increment    0.   1   on tape 19
                     time =        0.76

  modal recovery will be performed at this stage for mode     2


                    s t a r t   o f   i n c r e m e n t   0.   2






                    e n d   o f   i n c r e m e n t   0.   2

                    formatted post data at increment    0.   2   on tape 19
                     time =        0.80

  modal recovery will be performed at this stage for mode     3


                    s t a r t   o f   i n c r e m e n t   0.   3






                    e n d   o f   i n c r e m e n t   0.   3

                    formatted post data at increment    0.   3   on tape 19
                     time =        0.85

  modal recovery will be performed at this stage for mode     4


                    s t a r t   o f   i n c r e m e n t   0.   4






                    e n d   o f   i n c r e m e n t   0.   4

                    formatted post data at increment    0.   4   on tape 19
```

```
                  time =        0.89

     modal recovery will be performed at this stage for mode      5


                  s t a r t   o f   i n c r e m e n t   0.   5




                  e n d   o f   i n c r e m e n t   0.   5

                  formatted post data at increment   0.   5   on tape 19
                   time =        0.94

     comment,  begin modal dynamics

     dynamic change
     ----------
```

Define Time Step

```
       time        time      maximum   assembly  max iter
     increment    period      steps    interval  mcreep
      1.000E-03 5.000E-01     500     500000       5

     continue
     ----------


     auto control specified for time of 0.500E+00


                  s t a r t   o f   i n c r e m e n t   1


                  load increments associated with each degree of freedom
                  summed over the whole model

                  distributed loads
  0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
```

```
                    point loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00




                    i n i t i a l   m o d a l   c o m p o n e n t s


 mode     displacement    velocity     acceleration



   1       0.6660E+00      0.0000E+00    -0.1057E+05
   2      -0.1946E-11      0.0000E+00     0.9291E-07
   3      -0.2184E+00      0.0000E+00     0.1357E+06
   4      -0.4721E-14      0.0000E+00     0.8829E-08
   5       0.4100E-02      0.0000E+00    -0.2017E+05
```

We observe that modes 1 and 2 have the dominant modal participation factors based on initial displacement conditions.

```
                    dynamic response based on    5  modes

                    m o d a l   c o m p o n e n t s
          incremental
 mode        load       displacement    velocity     acceleration



   1      0.000E+00      0.6608E+00    -0.1047E+02   -0.1035E+05
   2      0.000E+00     -0.1900E-11     0.9218E-10    0.9070E-07
   3      0.000E+00     -0.1556E+00     0.1174E+03    0.8736E+05
   4      0.000E+00     -0.9528E-15     0.6323E-11    0.1782E-08
   5      0.000E+00     -0.2472E-02    -0.7255E+01    0.1216E+05
```

NOTE

Modal components will be printed every increment. This is a clear indication of which modes are relevant in the analysis.

```
MARC          output for increment    1.     modal superposition


                    dynamic change has reached time of 0.100E-02  of total time period 0.500E+00


                    total dynamic transient time = 1.00000E-03
```

```
                  e n d   o f   i n c r e m e n t   1

                  formatted post data at increment   1.   0   on tape 19
                   time =        1.10
                       ●                                             ●
                       ●                                             ●
                       ●                                             ●
                       ●                                             ●

                  s t a r t   o f   i n c r e m e n t 499




                  load increments associated with each degree of freedom
                  summed over the whole model

                  distributed loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                  point loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00







                  dynamic response based on    5  modes

                  m o d a l   c o m p o n e n t s
          incremental
mode          load    displacement    velocity      acceleration



   1     0.000E+00     0.3026E-01   -0.8028E-01    -0.4537E+03
   2     0.000E+00     0.1181E-11    0.3380E-09    -0.5638E-07
   3     0.000E+00    -0.5460E-03   -0.3029E-06    -0.2837E-03
   4     0.000E+00     0.3633E-14   -0.4122E-11    -0.6795E-08
   5     0.000E+00     0.2598E-02   -0.7034E+01    -0.1278E+05
```

```
                    dynamic change has reached time of 0.499E+00  of total time period 0.500E+00


                    total dynamic transient time = 4.99000E-01



                    e n d   o f   i n c r e m e n t 499

                    formatted post data at increment 499.   0   on tape 19
                     time =       69.81



                    s t a r t   o f   i n c r e m e n t 500 <------------- Last increment



                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                    point loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00



                    dynamic response based on    5  modes

                    m o d a l   c o m p o n e n t s
         incremental
 mode        load     displacement    velocity     acceleration



    1     0.000E+00     0.2995E-01    -0.5293E+00    -0.4432E+03
    2     0.000E+00     0.1488E-11     0.2740E-09    -0.7105E-07
    3     0.000E+00    -0.5460E-03    -0.4610E-06    -0.1939E-04
    4     0.000E+00    -0.2219E-14    -0.5699E-11     0.4150E-08
    5     0.000E+00    -0.4097E-02    -0.3568E+00     0.2015E+05
```

```
dynamic change has reached time of 0.500E+00  of total time period 0.500E+00


total dynamic transient time = 5.00000E-01



end  of  increment 500

formatted post data at increment 500.   0   on tape 19
 time =      69.95



*** end of input deck - job ends




          marc exit number 3004
```

Results

The damped dynamic response of the top of the beam, subjected to the applied initial displacements, is shown below.

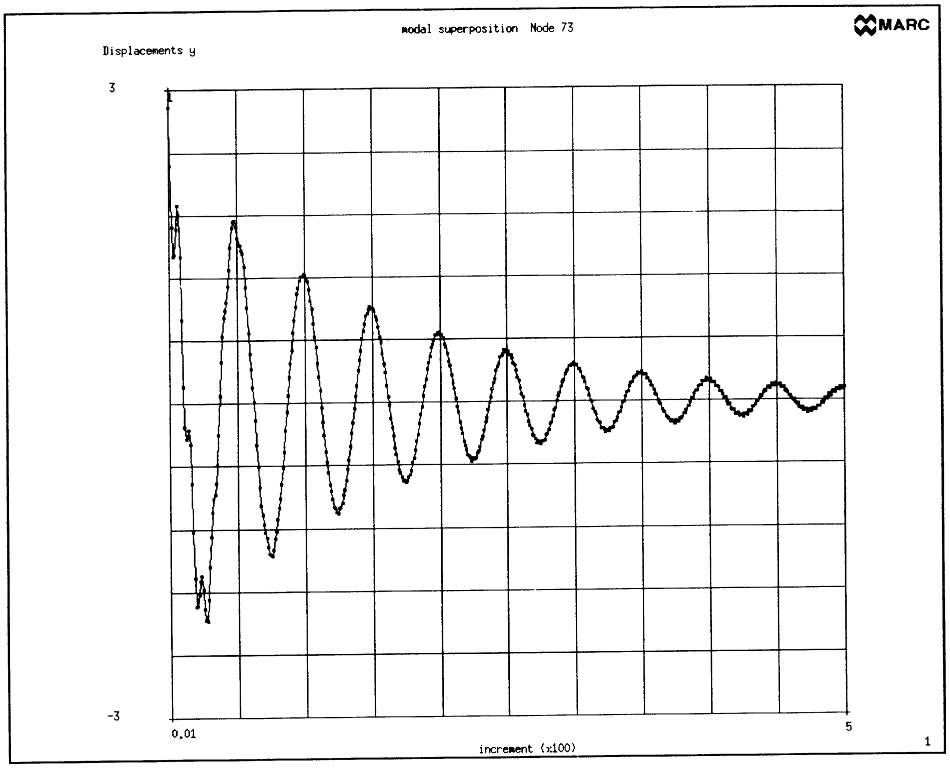


Figure 3C.2 Time History of Tip Deflection

Lets first examine how the eigenvalue extraction using the Lanczos method performed, prior to the modal superposition analysis.

Mode/ Vibrating In	Theoretical Frequency (rad/sec)	Using	Inverse Power Sweep (rad/sec)	Lanczos (rad/sec)	% Difference
1 X-Y plane	126.56	I_{yy}	126.38	126.1	-0.36
2 X-Z plane	219.20	I_{xx}	218.97	218.5	-0.11
3 X-Y plane	790.97	I_{yy}	792.03	789.3	+0.21
4 X-Z plane	1370.0	I_{xx}	1372.28	1368.0	+0.15
5 X-Y plane	2218.2	I_{yy}	2217.96	2218.0	-0.01

The agreement with theoretical frequencies is again excellent, with the Lanczos method performing even slightly better than the inverse power sweep method. Notice the worst error in frequency calculation is only about four-tenths of one percent!

The use of the RECOVER option allowed us to store eigenvectors on the post tape, and recover modal reactions forces and modal stresses and reactions for five modes in this modal superposition analysis.

Why did we use only two modes (1 and 3) to describe the transient response of the cantilevered beam? Since the imposed initial shape "looks" like the first mode of the beam, our experience tells us that the response will be predominantly in the first mode, with perhaps a little "spillover" into the third mode. Also, since the initial displacements are orthogonal to the other three modes (2, 4, and 5), these three modes will not be excited. This discussion illustrates the importance of the analyst's experience and judgment in using finite element techniques to solve linear/nonlinear static and dynamic problems. In modal superposition analysis (e.g., Bathe, K.J. and E.L. Wilson, *Numerical Methods in Finite Element Analysis,* Prentice-Hall, 1976, pp. 326-343), the analyst needs to specify the number of modes (five, in this case) to be used in the analysis–thus making the inherent assumption that the higher modes are probably not as accurate and therefore unimportant in the finite element analysis. In such cases, modal superposition is better than direct time integration. (Note that in this example, modes 2, 4, and 5 are just as accurate as modes 1 and 3–but they are still not important because of the nature of the excitation.)

Notice that the dominance of modes 1 and 3 in the dynamic response is borne out by the table entitled "Initial Modal Components" in the printout. The displacement values for these two modes are orders of magnitudes higher than those of modes 2,4, and 5.

Why did we apply damping to the first and third modes, and not for modes 2, 4, and 5? Another case of the analyst's experience! Again, we know from experience and knowledge of the cantilevered beam's dynamic behavior that the first and third modes (in-plane vibration) will probably be sufficient to capture the dynamic response of this problem. Therefore, we chose to apply damping to only these two modes. Experience also tells us that the response is orthogonal to modes 2, 4, and 5; therefore, they are unimportant in this problem, and no damping needs to be applied to these modes.

Exercises

Try doing Example 3C: (1) undamped–and compare the undamped modal superposition response with the damped; (2) using more than five modes, say seven or ten modes; or (3) using a different damping factor, say 3% or 10%, for the first and third modes–and see the difference in response.

Example 4

Stiffened Composite Roof Under Uniform Pressure

This example illustrates the modeling and linear analysis of a composite structure. The model consists of a cylindrical roof panel modeled by thin shell elements stiffened by beam elements (BEAM SECTION PARAMETER option). It is loaded by uniform pressure. The example demonstrates: the input of composite layup properties (COMPOSITE option); the use of two different types of MARC elements in a single model; and the input of a material orientation angle which is not aligned with one of the global axes (ORIENTATION option).

Sketch

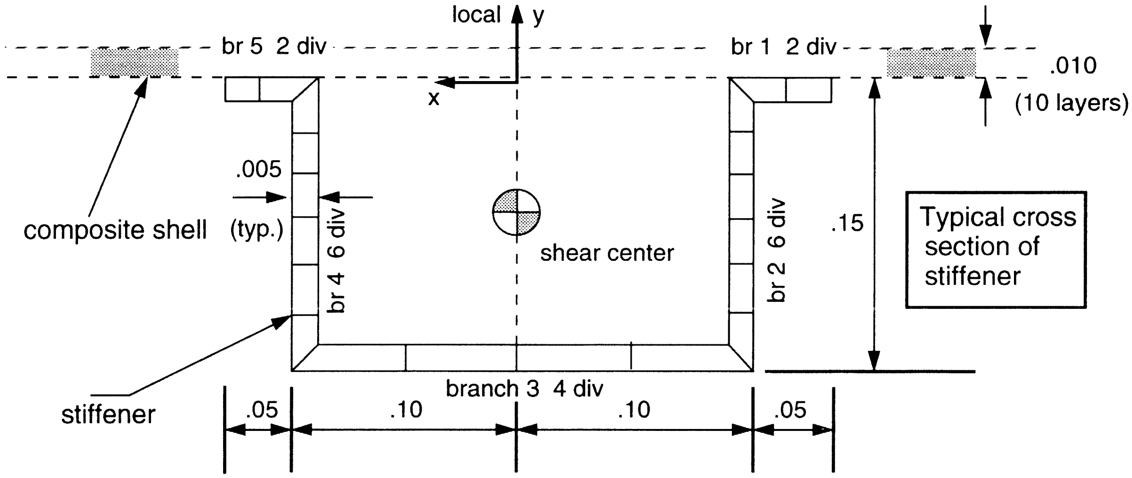


Figure 4.1a **Beam Cross Section**

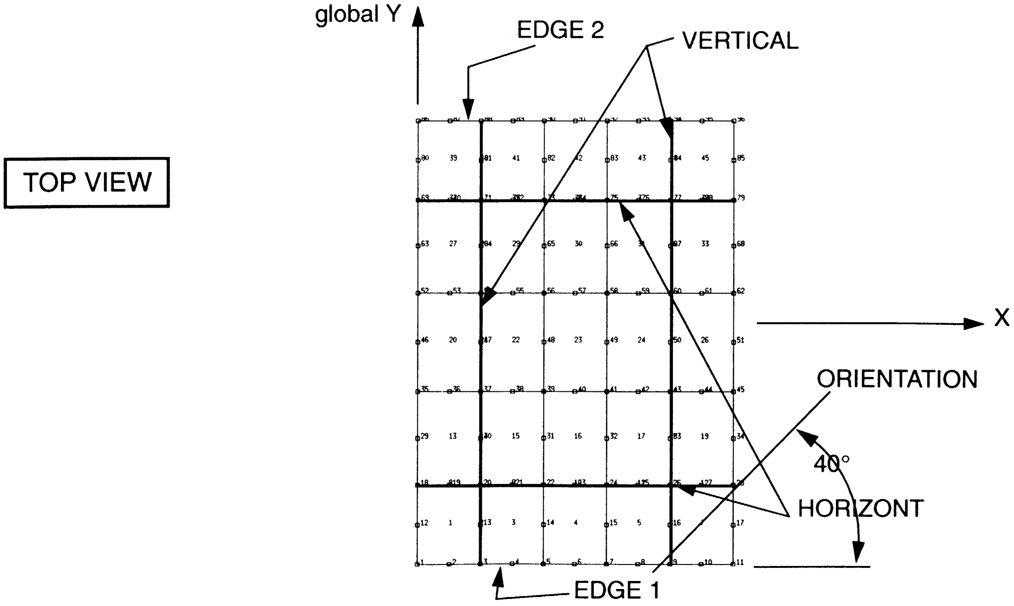


Figure 4.1b **Beam and Material Orientation**

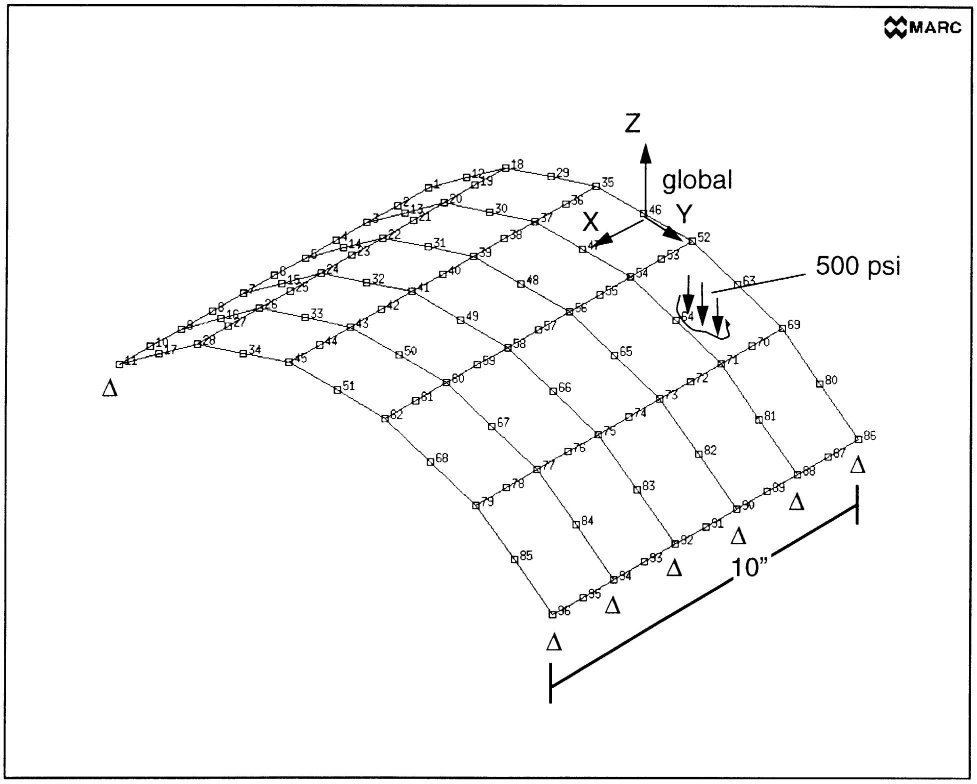


Figure 4.2a Node Numbers

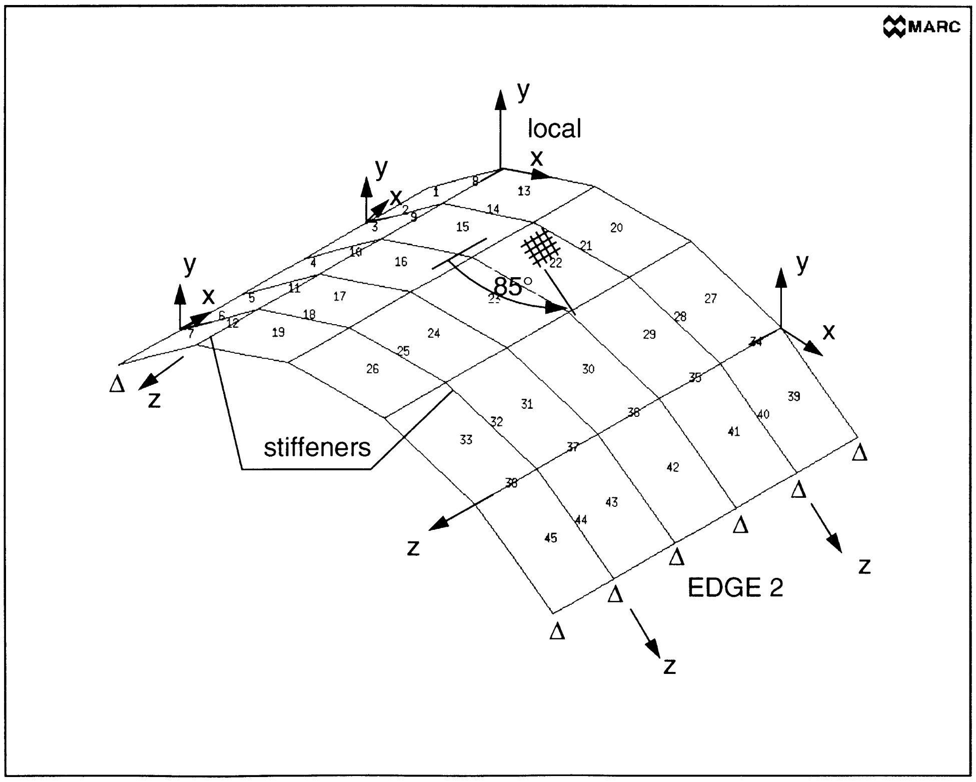


Figure 4.2b Element Numbers

Model

The cylindrical panel extends 10 inches along the X-direction. It has a thickness of 0.010 inch (10 layers of 0.001 in.). It is stiffened by four U-shaped stiffeners: two in the axial (X) direction running from nodes 18 to 28, and 69 to 79; and another two in the circumferential direction, running from nodes 3 to 88, and 9 to 94. The model consists of 96 nodes and a total of 45 elements: 25 shell elements (MARC Element 72) and 20 beam elements (Element 77). Element 72 (p. B72.1-1) is an eight-noded thin shell element with straight edges. It has three displacement DOFs at the four corner nodes, and one rotation DOF (of the edge about itself) at each of the midside nodes. It has four Gaussian integration points. Element 77 (p. B77.1-1) is a 3-D, 3-noded beam element which can include warping and twist effects of the open cross section. Output may be requested at either the centroid or the two Gaussian integration points. This element is designed to be compatible with Element 72 to simulate stiffened shell structures.

Geometric Properties

Geometric properties need to be specified for the beam stiffeners. This is actually done by flagging the EGEOM2 parameter for Element 77, which references a section number described in the BEAM SECT PARAMETER options. The orientation of the beam cross section is defined using the EGEOM3, EGEOM4 and EGEOM5 fields. The shell thickness is specified using the COMPOSITE option, in this particular case.

Material Properties

The beam stiffener elements are assigned material-id 1, with the following isotropic properties: Young's modulus of 20.0 x 10^{10} psi and Poisson's ratio of 0.3.

The orthotropic layers of the composite shell are assigned material-id 2, with the following properties: Young's moduli E_{11} of 30.0 x 10^{10} psi and E_{22} of 3.0 x 10^{10} psi; Poisson's ratio ν_{12} of 0.4; and shear modulus G_{12} of 1.0 x 10^{10} psi.

Each of the ten layers of the composite shell is 0.001 in. thick. The ten layers are stacked together in a symmetrical layup with the following ply angles: 45/-45/0/90/0/0/90/0/-45/45 degrees. However, the major material axis is oriented +40 degrees from the X-axis. This means the first ply is (40 + 45), or 85 degrees from the X-axis, and so forth.

Loads

The shell is loaded in the negative Z-direction by a uniform gravity load of 500.0 psi.

Boundary Conditions

Along node set EDGE1 (nodes 1, 3, 5, 7, 9, 11) and node set EDGE2 (nodes 86, 88, 90, 92, 94, 96), in other words, the two longitudinal edges, each node is restrained against translation in all three global directions.

Input

A complete input listing is included.

PARAMETER Section

The "TITLE" line is self-explanatory. The "SIZING" line says to set a workspace of 200,000 words. The "ELEMENTS" line informs MARC that two types of elements will be used: MARC Element 72 (shell) and Element 77 (beam).

The next PARAMETER option is BEAM SECT (see *MARC Volume A and Volume C*), which allows you to input sectional properties for a beam element. It is one of the few PARAMETER options to require so much data input. In this example, the block consists of a total of fourteen lines, beginning with the "BEAM SECT" line, followed by the section title line "U SECTION STIFFENER", the eleven data lines, and ending with the "LAST" line. The "5,2,6,4,6,2," tells MARC that there are a total of five "branches"–with two "divisions" (intervals) in the first branch, six in the second, four in the third, six in the fourth, and two in the fifth and last branch (see sketch). The number of divisions must be *even.* For each branch, there are two data lines. For instance, the first data line that follows "-.15,0.,1.,0.,-1,0.,1.,0.," gives data for the first branch: the X-coordinate of beginning of branch is -0.15 in. and the Y-coordinate is zero; DX/DS at beginning of branch is 1.0 (where S is the distance along the branch) and DY/DS is 0.0; the X-coordinate of end of branch is -0.1 with the Y-coordinate being zero; DX/DS at end of branch is 1.0 and DY/DS is 0.0. The next line ".05,.005," says that this first branch has a length of 0.05 and a thickness of the beginning branch of 0.005. (Note that since we left the third field blank on this line, the thickness of the end branch will default to the thickness of the beginning branch, or 0.005.) And, so forth for branches 2 to 5.

Several things are important to remember in the description of the local coordinate system of such beam stiffeners. First, the origin of the local coordinate system is the node location. The beam cross section can also be given an offset with respect to the node by choosing the section geometry such that the beam shear center is not at the local origin (0,0). in this example, the shear center is at (0.,-0.0875) with respect to the local coordinate system. The coordinates of section points printed in the output are with respect to the shear center. Second, the cross section orientation is defined with respect to a local coordinate system defined by either the GEOMETRY option or the COORDINATES option in the MODEL DEFINITION section. The local z-direction is always along the beam from the first node to the last node (based on the CONNECTIVITY list). The local x-axis is defined by prescribing the direction cosines with respect to the global system using the GEOMETRY Model Definition option.

The "END" line terminates the section.

MODEL DEFINITION Section

This section consists of:

- the FE mesh topology--including the CONNECTIVITY, COORDINATES, and DEFINE options
- geometric properties

- material properties
- composite layup
- orientation angle
- boundary conditions
- loads
- bandwidth minimization
- output controls

FE Mesh Topology

The CONNECTIVITY option (*see MARC Volume C*) provides components of the element data block. The first line ("45 0 0") informs MARC that the connectivity of 45 elements will be defined. The next line ("1 72 1 3 20 18 2 13 19 12") is typical of the connectivity lines for Element 72: the first 1 is the element number; 72 is the element type; 1, 3, 20, and 18 are the corner nodes of this element (counterclockwise); and 2, 13, 19, and 12 are the mid-side nodes. The following line ("2 77 3 13 20") is a typical connectivity line for Element 77: the 2 refers to element number 2; the 77 is the element type; and 3, 13, 20 define the three nodes for this beam element.

The COORDINATES option (*see MARC Volume C*) gives the coordinates of each node. After the usual blank line, the global X, Y, and Z coordinates of nodes 1 through 96 are given. Our model was generated using Mentat II.

The DEFINE option (*see MARC Volume C*) is used to define element sets ELEM72, VERTICAL, and HORIZONT, and node sets EDGE1 and EDGE2.

Geometric Properties

The GEOMETRY option (*see MARC Volume C*) lets us input element geometry data. The floating point value of the beam section number (1 in this case) is entered as explained in *MARC Volume A*. For the stiffener beam elements that appear in the vertical direction (VERTICAL) in the top view of the model, the local x-axis is given a direction cosine of (-1.,0.,0.), while the beam elements that appear in the horizontal direction (HORIZONT) are given a direction cosine of (0.,1.,0.). For both sets of beams, the second field references the BEAM SECT data while the 4, 5, and 6 fields define direction cosines. As opposed to other examples, the thickness of the shell elements will be defined using the COMPOSITE option.

Material Properties

Two material property options are used: ISOTROPIC and ORTHOTROPIC.

The ISOTROPIC option (*see MARC Volume C*) provides for the input of isotropic material properties. The blank line means we don't need to count how many lines follow. The "1," line assigns the material-id number to be 1. The "20.E10,.3," line says Young's modulus is 20.0E10 and Poisson's ratio is 0.3. And the "VERTICAL

AND HORIZONT" line gives the element sets to which these properties apply, that is, all twenty beam elements.

The ORTHOTROPIC option (*see MARC Volume C*) lets you define properties for an orthotropic material (whose properties vary in different directions). After the usual blank line, the "2," line assigns this material-id number to be 2. The "30.El0,3.El0,,.4," line means: E_{11} is 30.0E10, E_{22} is 3.0E10, and ν_{12} is 0.4. The next line "l.El0," says G_{12} is 1.0E10. Note the comment explaining the lack of an element list which is associated with this material, because in the following COMPOSITE option the shell will refer to this material in its layer descriptions.

Composite Layup

The COMPOSITE option (*see MARC Volume C*) allows you to define the layer-by-layer material identifications, layer thicknesses, and orientation angles for a laminated composite material, and to associate this material with an element number. The blank line, as usual, means we don't need to give a count of the number of lines that follow. In the "1,10," line, the 1 is the composite group number while the 10 refers to the fact there are ten layers in this group. (Since we left the third field blank on this line, the default is to input actual layer thicknesses in the following data lines.) It is often more convenient to specify percentage thickness where the total thickness is given through the GEOMETRY option or the NODAL THICKNESS option. This is especially true if the shell has varying thicknesses. The next ten data lines define for each of the ten layers: material-id number, layer thickness, and ply orientation angle in degrees. For instance, on the first of these lines ("2,.001,45.,"), the 2 refers to the orthotropic material-id of 2, the 0.001 is the layer thickness, and 45 is the ply angle. Finally, the "ELEM72" line ends this block, and assigns these composite layup descriptions to all 25 elements in element set ELEM72.

Orientation Angle

The ORIENTATION option (*see MARC Volume C*) lets you specify the reference material orientation. All of the ply orientation will be with respect to this orientation. After the blank line, the "EDGE 1-2,40." line indicates that the orientation angle is 40 degrees from edge 1-2, which in this case is aligned parallel to the X-axis (going from the first to the second node in the connectivity specification of the shell elements). The "ELEM72" line means this orientation angle applies to all 25 shell elements in element set ELEM72.

Boundary Conditions

The FIXED DISP option (*see MARC Volume C*) is for prescribing fixed displacements on the model. After the blank line, the "0.," line means the value of the first DOF is zero. (Note that since we left the second and third fields blank on this line, MARC will assume the second and third DOFs will also be zero.) The "l 2 3" line gives the three applicable DOFs. And, the "EDGE1 AND EDGE2" line names the two longitudinal node sets for which these fixed displacements apply. Note that no constraint was placed on the midside nodes which have a single rotational DOF.

Loads

The DIST LOADS option (*see MARC Volume C*) allows pressure loads to be specified. After the blank line, the "1,500.," line indicates the type of load is 1 (uniform gravity load per surface area in the -Z-direction), and the load intensity is 500. The "ELEM72" line says this pressure load applies to all 25 shell elements in element set ELEM72.

Bandwidth Minimization

The OPTIMIZE option (*see MARC Volume C*) lets you choose a bandwidth minimization scheme. In this case, the 2 means the Cuthill-McKee algorithm (default). The 5, line says we want a maximum of five numbering schemes (iterations).

Output Controls

The POST option (*see MARC Volume C*) tells MARC to create a post-processor file for later processing by Mentat II. The "3 ... 1" line means three post variables are desired. The next three lines indicate that we want to store the three stress components in the preferred coordinate system (defined in the ORIENTATION and COMPOSITE options) for layer 1.

The PRINT ELEM option (*see MARC Volume C*) lets you specify which element quantities you wish to be printed, and for which elements. After the blank line, the "STRESS PREF" line flags the total stresses and stresses in the preferred system to be printed. The "ELEM7" line tells MARC we want this done for all 25 shell elements in element set ELEM72. The "1 TO 4" line means we want all four integration points to be printed. And the "1" line indicates layer 1.

The PRINT NODE option (*see MARC Volume C*) does the same thing for nodal quantities. After the blank line, the "TOTAL" line means we want to print the total displacement. And the "1" line means node 1.

The "END OPTION" line terminates the MODEL DEFINITION section as well as the entire input file.

Output

On the first page of the output (after the input echo), you'll see the results of the BEAM SECT input in the PARAMETER section. The branch definitions for the five branches are shown, as are the coordinates of the 21 points (with respect to the shear center), thicknesses, and warping functions for each portion of the open cross section. This part of the printout begins with U SECTION and ends with LAST, and precedes the usual program sizing and options table.

Also included is the MARC interpretation of the composite layup definition. Each of the ten layers uses material 2, and its thickness is 0.001 in. The input ply angles are shown.

Then, the element outputs for elements 1, 22, and 45 are shown. For each integration point, element quantities are printed out, as are the stresses in the global and preferred directions.

input data

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
TITLE, STIFFENED COMPOSITE ROOF UNDER LOAD
SIZING        200000
ELEMENTS     72   77
COMMENT,  BEAM SECT DEFINES THE BEAM SECTION FOR ELEMENT TYPE 77
BEAM SECT
U SECTION STIFFENER
5,2,6,4,6,2,
-.15,0.,1.,0.,-.1,0.,1.,0.,
.05,.005,
-.1,0.,0.,-1.,-.1,-.15,0.,-1.,
.15,.005,                                   Define Beam Cross Section
-.1,-.15,1.,0.,.1,-.15,1.,0.,
.2,.005,
.1,-.15,0.,1.,.1,0.,0.,1.,
.15,.005,
.1,0.,1.,0.,.15,0.,1.,0.,
.05,.005
LAST
END
CONNECTIVITY
   45    0    0
    1   72    1    3   20   18    2   13   19   12
    2   77    3   13   20
    3   72    3    5   22   20    4   14   21   13
    4   72    5    7   24   22    6   15   23   14
    5   72    7    9   26   24    8   16   25   15
    6   77    9   16   26
    7   72    9   11   28   26   10   17   27   16
    8   77   18   19   20
    9   77   20   21   22
   10   77   22   23   24
   11   77   24   25   26
   12   77   26   27   28
   13   72   18   20   37   35   19   30   36   29
   14   77   20   30   37
   15   72   20   22   39   37   21   31   38   30
   16   72   22   24   41   39   23   32   40   31
   17   72   24   26   43   41   25   33   42   32
   18   77   26   33   43
   19   72   26   28   45   43   27   34   44   33
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                      5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                    --------------------------------------------------------------------------------
                      20   72   35   37   54   52   36   47   53   46
                      21   77   37   47   54
                      22   72   37   39   56   54   38   48   55   47
                      23   72   39   41   58   56   40   49   57   48
card      45          24   72   41   43   60   58   42   50   59   49
                      25   77   43   50   60
                      26   72   43   45   62   60   44   51   61   50
                      27   72   52   54   71   69   53   64   70   63
                      28   77   54   64   71
card      50          29   72   54   56   73   71   55   65   72   64
                      30   72   56   58   75   73   57   66   74   65
                      31   72   58   60   77   75   59   67   76   66
                      32   77   60   67   77
                      33   72   60   62   79   77   61   68   78   67
card      55          34   77   69   70   71
                      35   77   71   72   73
                      36   77   73   74   75
                      37   77   75   76   77
                      38   77   77   78   79
card      60          39   72   69   71   88   86   70   81   87   80
                      40   77   71   81   88
                      41   72   71   73   90   88   72   82   89   81
                      42   72   73   75   92   90   74   83   91   82
                      43   72   75   77   94   92   76   84   93   83
card      65          44   77   77   84   94
                      45   72   77   79   96   94   78   85   95   84
                    COORDINATES

                       1   0.00000  -7.07107   7.07107
card      70           2   1.00000  -7.07107   7.07107
                       3   2.00000  -7.07107   7.07107
                       4   3.00000  -7.07107   7.07107
                       5   4.00000  -7.07107   7.07107
                       6   5.00000  -7.07107   7.07107
card      75           7   6.00000  -7.07107   7.07107
                       8   7.00000  -7.07107   7.07107
                       9   8.00000  -7.07107   7.07107
                      10   9.00000  -7.07107   7.07107
                      11  10.00000  -7.07107   7.07107
card      80          12   0.00000  -5.80549   7.99056
                      13   2.00000  -5.80549   7.99056
                      14   4.00000  -5.80549   7.99056
                      15   6.00000  -5.80549   7.99056
                    --------------------------------------------------------------------------------
                      5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
--------------------------------------------------------------------------------
   16   8.00000  -5.80549   7.99056
   17  10.00000  -5.80549   7.99056
   18   0.00000  -4.53991   8.91006
   19   1.00000  -4.53991   8.91006
   20   2.00000  -4.53991   8.91006
   21   3.00000  -4.53991   8.91006
   22   4.00000  -4.53991   8.91006
   23   5.00000  -4.53991   8.91006
   24   6.00000  -4.53991   8.91006
   25   7.00000  -4.53991   8.91006
   26   8.00000  -4.53991   8.91006
   27   9.00000  -4.53991   8.91006
   28  10.00000  -4.53991   8.91006
   29   0.00000  -3.05213   9.39347
   30   2.00000  -3.05213   9.39347
   31   4.00000  -3.05213   9.39347
   32   6.00000  -3.05213   9.39347
   33   8.00000  -3.05213   9.39347
   34  10.00000  -3.05213   9.39347
   35   0.00000  -1.56435   9.87688
   36   1.00000  -1.56435   9.87688
   37   2.00000  -1.56435   9.87688
   38   3.00000  -1.56435   9.87688
   39   4.00000  -1.56435   9.87688
   40   5.00000  -1.56435   9.87688
   41   6.00000  -1.56435   9.87688
   42   7.00000  -1.56435   9.87688
   43   8.00000  -1.56435   9.87688
   44   9.00000  -1.56435   9.87688
   45  10.00000  -1.56435   9.87688
   46   0.00000   0.00000   9.87688
   47   2.00000   0.00000   9.87688
   48   4.00000   0.00000   9.87688
   49   6.00000   0.00000   9.87688
   50   8.00000   0.00000   9.87688
   51  10.00000   0.00000   9.87688
   52   0.00000   1.56434   9.87688
   53   1.00000   1.56434   9.87688
   54   2.00000   1.56434   9.87688
   55   3.00000   1.56434   9.87688
   56   4.00000   1.56434   9.87688
   57   5.00000   1.56434   9.87688
   58   6.00000   1.56434   9.87688
--------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

```
    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
-------------------------------------------------------------------------------
    59   7.00000   1.56434   9.87688
    60   8.00000   1.56434   9.87688
    61   9.00000   1.56434   9.87688
    62  10.00000   1.56434   9.87688
    63   0.00000   3.05212   9.39347
    64   2.00000   3.05212   9.39347
    65   4.00000   3.05212   9.39347
    66   6.00000   3.05212   9.39347
    67   8.00000   3.05212   9.39347
    68  10.00000   3.05212   9.39347
    69   0.00000   4.53990   8.91006
    70   1.00000   4.53990   8.91006
    71   2.00000   4.53990   8.91006
    72   3.00000   4.53990   8.91006
    73   4.00000   4.53990   8.91006
    74   5.00000   4.53990   8.91006
    75   6.00000   4.53990   8.91006
    76   7.00000   4.53990   8.91006
    77   8.00000   4.53990   8.91006
    78   9.00000   4.53990   8.91006
    79  10.00000   4.53990   8.91006
    80   0.00000   5.80548   7.99057
    81   2.00000   5.80548   7.99057
    82   4.00000   5.80548   7.99057
    83   6.00000   5.80548   7.99057
    84   8.00000   5.80548   7.99057
    85  10.00000   5.80548   7.99057
    86   0.00000   7.07107   7.07107
    87   1.00000   7.07106   7.07107
    88   2.00000   7.07106   7.07107
    89   3.00000   7.07106   7.07107
    90   4.00000   7.07106   7.07107
    91   5.00000   7.07106   7.07107
    92   6.00000   7.07106   7.07107
    93   7.00000   7.07106   7.07107
    94   8.00000   7.07106   7.07107
    95   9.00000   7.07106   7.07107
    96  10.00000   7.07107   7.07107
DEFINE    ELEMENT   SET       ELEM72
DEFINE    ELEMENT   SET       VERTICAL
    2    6   14   18   21   25   28   32   40   44
DEFINE    ELEMENT   SET       HORIZONT
    1    3    4    5    7   13   15   16   17   19   20   22   23   24   26 C
-------------------------------------------------------------------------------
    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
  8 TO 12 AND 34 TO 38
   27   29   30   31   33   39   41   42   43   45
DEFINE    NODE      SET       EDGE1
    1    3    5    7    9   11
DEFINE    NODE      SET       EDGE2
   86   88   90   92   94   96
GEOMETRY

0.,1.,0.,0.,1.,0.,
HORIZONT
0.,1.,0.,-1.,0.,0.,
VERTICAL
COMMENT,  DEFINE THE BEAM PROPERTIES
ISOTROPIC

1,
20.E10,.3,
VERTICAL AND HORIZONT
COMMENT,  NO ELEMENT LIST IN ORTHOTROPIC SINCE THESE PROPERTIES ARE
COMMENT,   USED IN THE COMPOSITE LAYUP
ORTHOTROPIC

2,
30.E10,3.E10,,.4
1.E10,


COMMENT,  DEFINE THE COMPOSITE LAYUP HERE
COMPOSITE

1,10,
2,.001,45.
2,.001,-45.
2,.001,0.,
2,.001,90.,
2,.001,0.,
2,.001,0.,
2,.001,90.,
2,.001,0.,
2,.001,-45.,
2,.001,45.,
ELEM72
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

Define Composite Layup

```
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
--------------------------------------------------------------------------------
FIXED DISP

0.,
1 2 3
EDGE1 AND EDGE2
DIST LOADS

1,500.,
ELEM72
OPTIMIZE,2,
5,
POST
3,,,1
111,1,
112,1,
113,1,
PRINT ELEM

STRESS PREF
ELEM72
1 TO 4
1
PRINT NODE

TOTAL
1
COMMENT, ORIENT THE LAYUP TO 40 DEG FROM THE ELEMENTS 1-2 EDGE
ORIENTATION

EDGE 1-2,40.
ELEM72
END OPTION
--------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
--------------------------------------------------------------------------------
--------------------------------------------------------------------------------
```

```
                          u section



 no.of branches    5     intervals per branch    2    6    4    6    2




                      branch definition


 branch    x1       y1      x1p      y1p       x2       y2      x2p      y2p       pl       t1       t2
 1    -0.150    0.000    1.000    0.000   -0.100    0.000    1.000    0.000    0.050    0.005    0.005
 2    -0.100    0.000    0.000   -1.000   -0.100   -0.150    0.000   -1.000    0.150    0.005    0.005
 3    -0.100   -0.150    1.000    0.000    0.100   -0.150    1.000    0.000    0.200    0.005    0.005
 4     0.100   -0.150    0.000    1.000    0.100    0.000    0.000    1.000    0.150    0.005    0.005
 5     0.100    0.000    1.000    0.000    0.150    0.000    1.000    0.000    0.050    0.005    0.005



                         section  1  (open)
```

with respect to shear center

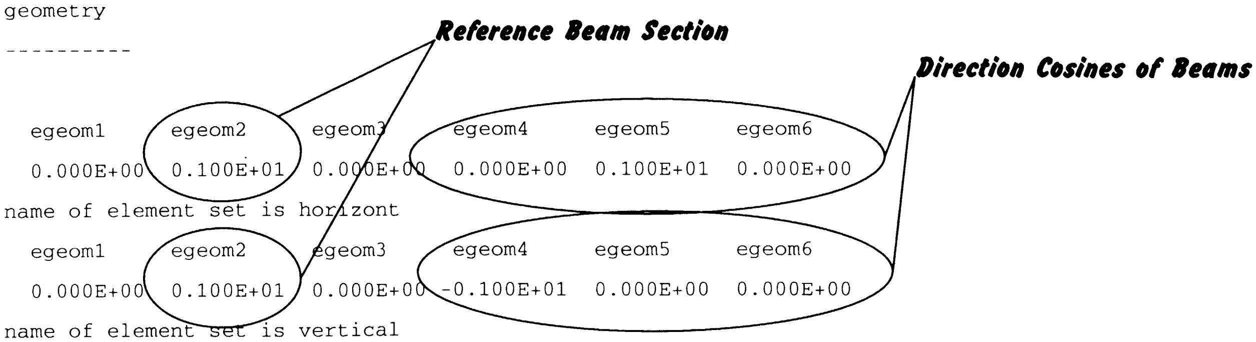
```
  point no,  coordinates in section,      thickness,   warping ftn.,      weight
     1       -0.15000         0.08750      0.00500       0.01688        0.00004
     2       -0.12500         0.08750      0.00500       0.01906        0.00017
     3       -0.10000         0.08750      0.00500       0.02125        0.00008
     4       -0.10000         0.06250      0.00500       0.01875        0.00017
     5       -0.10000         0.03750      0.00500       0.01625        0.00008
     6       -0.10000         0.01250      0.00500       0.01375        0.00017
     7       -0.10000        -0.01250      0.00500       0.01125        0.00008
     8       -0.10000        -0.03750      0.00500       0.00875        0.00017
     9       -0.10000        -0.06250      0.00500       0.00625        0.00013
    10       -0.05000        -0.06250      0.00500       0.00313        0.00033
    11        0.00000        -0.06250      0.00500       0.00000        0.00017
    12        0.05000        -0.06250      0.00500      -0.00313        0.00033
    13        0.10000        -0.06250      0.00500      -0.00625        0.00013
    14        0.10000        -0.03750      0.00500      -0.00875        0.00017
    15        0.10000        -0.01250      0.00500      -0.01125        0.00008
    16        0.10000         0.01250      0.00500      -0.01375        0.00017
    17        0.10000         0.03750      0.00500      -0.01625        0.00008
    18        0.10000         0.06250      0.00500      -0.01875        0.00017
    19        0.10000         0.08750      0.00500      -0.02125        0.00008
    20        0.12500         0.08750      0.00500      -0.01906        0.00017
    21        0.15000         0.08750      0.00500      -0.01688        0.00004



                          last
```

Beam Cross Section Data

```
          *************************************************

          *************************************************


           program sizing and options requested as follows


           element type requested*************************   72

           element type requested*************************   77

           number of elements in mesh*********************   45

           number of nodes in mesh************************   96

           max number of elements in any dist load list***   25

           maximum number of boundary conditions**********   36

           load correction flagged or set*****************

           number of lists of distributed loads***********    3

           stresses stored at all integration points******

           beam section sizes specified by user***********

           tape no.for input of coordinates + connectivity    5

           no.of different materials    2 max.no of slopes    5

           maximum elements variables per point on post tp   33

           number of points on shell section *************   10

           option for terminal debug**********************

           max number of composite groups*****************    1

           new style input format will be used************

           maximum number of set names is*****************   10

           number of processors used *********************    1

           vector length used ****************************    1


           end of parameters and sizing
          *************************************************

          *************************************************


key to stress, strain and displacement output


          element type   72


4+4 - node shell element


generalized strains in local coordinates
  1=local x membrane
  2=local y membrane
  3=local xy shear
stresses correspond to strains in each fiber


displacements in global directions at corner nodes
  1=u global x direction
  2=v global y direction
  3=w global z direction
```

```
rotation around edge at mid-side nodes
  1=rotation, 2 and 3 not used


          element type   77


3-node thin walled, open section beam (including warping)


strains -
  1=axial stretch
  2=local xx curvature
  3=local yy curvature
  4=warping of section
  5=twist of section


section forces-
  1=axial force
  2=local xx moment
  3=local yy moment
  4=bimoment
  5=axial torque


displacements at end nodes -
  1=u global x direction
  2=v global y direction
  3=w global z direction
  4=theta x rotation about global x axis
  5=theta y rotation about global y axis
  6=theta z rotation about global z axis
  7=warping


twist rotation at middle node -
  1=rotation around axis from lowest to highest end node
```

⋮ ⋮



Reference Beam Section

Direction Cosines of Beams

```
geometry
----------


  egeom1      egeom2      egeom3      egeom4      egeom5      egeom6
  0.000E+00   0.100E+01   0.000E+00   0.000E+00   0.100E+01   0.000E+00
name of element set is horizont
  egeom1      egeom2      egeom3      egeom4      egeom5      egeom6
  0.000E+00   0.100E+01   0.000E+00  -0.100E+01   0.000E+00   0.000E+00
name of element set is vertical
comment,  define the beam properties
```

⋮ ⋮

```
 isotropic
 ----------


 isotropic material  material id =     1
  von mises  yield criteria
  isotropic  hardening rule
    e          nu         rho       alpha       yield      yield2
   0.200E+12  0.300E+00  0.000E+00  0.000E+00  0.100E+21  0.100E+21
name of element set is vertical
       and
name of element set is horizont
comment,  no element list in orthotropic since these properties are
comment,   used in the composite layup

orthotropic
----------


orthotropic material  material id =     2
 von mises  yield criteria
 isotropic  hardening rule
   e11        e22        e33        xu12       xu23       xu31       rho
  0.300E+12  0.300E+11  0.000E+00  0.400E+00  0.000E+00  0.000E+00  0.000E+00
   g12        g23        g31        a11        a22        a33
  0.100E+11  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
   yield      yield2     yrdir(1-3)                       yrshr(1-3)
  0.100E+21  0.100E+21  0.100E+01  0.100E+01  0.100E+01  0.100E+01  0.100E+01  0.100E+01
comment,  define the composite layup here

composite
----------


composite group number =     1
number of layers =    10

actual layer thickness is given below
layer matid  thickness ply angle
    1     2   0.100E-02  0.450E+02
    2     2   0.100E-02 -0.450E+02
    3     2   0.100E-02  0.000E+00
    4     2   0.100E-02  0.900E+02
    5     2   0.100E-02  0.000E+00
    6     2   0.100E-02  0.000E+00
    7     2   0.100E-02  0.900E+02
    8     2   0.100E-02  0.000E+00
    9     2   0.100E-02 -0.450E+02
```

Composite shell ply angle and thickness data

```
          10     2   0.100E-02  0.450E+02
        name of element set is elem72
                     •                                            •
                     •                                            •
                     •                                            •
                     •                                            •


        comment, orient the layup to 40 deg from the elements 1-2 edge

        orientation
        ----------

        orientation angle type = edge 1-2
             orientation angle =  40.000
             user vector 1     =     0.000E+00     0.000E+00     0.000E+00
             user vector 2     =     0.000E+00     0.000E+00     0.000E+00
        name of element set is elem72

        end option
        ----------
                     •                                            •
                     •                                            •
                     •                                            •
                     •                                            •


 MARC          output for increment    0.    stiffened composite roof under load

      element with highest stress relative to yield is    2  where equivalent stress is 0.474E-13  of yield

        tresca     mises    mean   p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
      intensity intensity  normal  minimum intermediate maximum     1         2         3         4         5         6
                                 intensity

element    1  point   1      integration pt. coordinate=     0.423E+00   -0.654E+01    0.746E+01
section thickness = 0.100E-01
average membrane
stress  5.570E+05 4.897E+05-2.503E+05-5.570E+05-1.940E+05 0.000E+00-1.957E+05-5.553E+05-2.487E+04
moment  9.744E+01 9.300E+01 2.606E+01-9.631E+00 0.000E+00 8.781E+01-8.732E+00 8.691E+01 9.313E+00
stretch 5.064E-06 5.263E-06 0.000E+00-4.720E-06 0.000E+00 3.441E-07 3.341E-07-4.710E-06 4.492E-07
curvatr 8.451E-05 6.336E-05 0.000E+00-2.205E-05 0.000E+00 6.247E-05-2.189E-05 6.231E-05 7.252E-06
layer  1
stress  1.325E+06 1.302E+06-4.577E+05-1.325E+06-4.812E+04 0.000E+00-5.572E+04-1.318E+06-9.820E+04
stressp 1.325E+06 1.302E+06-4.577E+05-1.325E+06-4.812E+04 0.000E+00-1.325E+06-4.825E+04-1.285E+04

element    1  point   2      integration pt. coordinate=     0.158E+01   -0.654E+01    0.746E+01
section thickness = 0.100E-01
average membrane
```

```
stress  4.187E+05 3.699E+05-1.851E+05-4.187E+05-1.365E+05 0.000E+00-1.371E+05-4.182E+05-1.236E+04
moment  9.476E+01 8.658E+01 3.818E+01 0.000E+00 1.977E+01 9.476E+01 2.112E+01 9.341E+01 9.948E+00
stretch 3.952E-06 3.978E-06 0.000E+00-3.605E-06 0.000E+00 3.469E-07 3.341E-07-3.593E-06 4.492E-07
curvatr 6.254E-05 7.577E-05 0.000E+00 0.000E+00 5.765E-06 6.254E-05 5.997E-06 6.231E-05 7.252E-06
layer  1
stress  9.851E+05 9.702E+05-3.385E+05-9.851E+05-3.053E+04 0.000E+00-3.598E+04-9.797E+05-7.190E+04
stressp 9.851E+05 9.702E+05-3.385E+05-9.851E+05-3.053E+04 0.000E+00-9.850E+05-3.066E+04-1.112E+04

element    1  point    3       integration pt. coordinate=     0.423E+00   -0.507E+01    0.852E+01
section thickness = 0.100E-01
average membrane
stress  5.077E+05 4.747E+05-1.941E+05-5.077E+05-7.474E+04 0.000E+00-7.499E+04-5.074E+05-1.042E+04
moment  3.365E+02 3.237E+02 1.213E+02 0.000E+00 2.731E+01 3.365E+02 3.058E+01 3.332E+02 3.164E+01
stretch 5.974E-06 4.887E-06 0.000E+00-4.718E-06 0.000E+00 1.255E-06 1.247E-06-4.710E-06 4.492E-07
curvatr 2.529E-04 2.549E-04 0.000E+00-2.194E-05 0.000E+00 2.309E-04-2.189E-05 2.309E-04 7.252E-06
layer  1
stress  1.082E+06 1.077E+06-3.645E+05-1.082E+06-1.103E+04 0.000E+00-1.705E+04-1.076E+06-8.008E+04
stressp 1.082E+06 1.077E+06-3.645E+05-1.082E+06-1.103E+04 0.000E+00-1.082E+06-1.119E+04-1.311E+04

element    1  point    4       integration pt. coordinate=     0.158E+01   -0.507E+01    0.852E+01
section thickness = 0.100E-01
average membrane
stress  3.703E+05 3.624E+05-1.289E+05-3.703E+05-1.639E+04 0.000E+00-1.640E+04-3.703E+05 2.087E+03
moment  3.434E+02 3.188E+02 1.334E+02 0.000E+00 5.675E+01 3.434E+02 6.043E+01 3.397E+02 3.227E+01
stretch 4.860E-06 3.657E-06 0.000E+00-3.603E-06 0.000E+00 1.257E-06 1.247E-06-3.593E-06 4.492E-07
curvatr 2.309E-04 2.702E-04 0.000E+00 0.000E+00 5.939E-06 2.309E-04 5.997E-06 2.309E-04 7.252E-06
layer  1
stress  7.490E+05 7.457E+05-2.453E+05-7.424E+05 0.000E+00 6.575E+03 2.693E+03-7.385E+05-5.378E+04
stressp 7.490E+05 7.457E+05-2.453E+05-7.424E+05 0.000E+00 6.575E+03-7.422E+05 6.401E+03-1.139E+04

                                •                                   •
                                •                                   •
                                •                                   •
                                •                                   •

element   22  point    1       integration pt. coordinate=     0.242E+01   -0.903E+00    0.988E+01
section thickness = 0.100E-01
average membrane
stress  5.578E+05 5.423E+05-1.967E+05-5.578E+05-3.238E+04 0.000E+00-3.240E+04-5.577E+05-2.959E+03
moment  2.849E+02 2.665E+02-1.088E+02-2.849E+02-4.168E+01 0.000E+00-4.431E+01-2.822E+02-2.515E+01
stretch 7.217E-06 5.478E-06 0.000E+00-5.387E-06 0.000E+00 1.830E-06 1.819E-06-5.375E-06 5.657E-07
curvatr 1.942E-04 2.230E-04 0.000E+00-1.935E-04 0.000E+00 7.113E-07 7.056E-07-1.934E-04 2.106E-06
layer  1
stress  1.850E+06 1.839E+06-6.240E+05-1.850E+06-2.244E+04 0.000E+00-3.311E+04-1.839E+06-1.392E+05
stressp 1.850E+06 1.839E+06-6.240E+05-1.850E+06-2.244E+04 0.000E+00-1.849E+06-2.265E+04-1.967E+04

element   22  point    2       integration pt. coordinate=     0.358E+01   -0.903E+00    0.988E+01
section thickness = 0.100E-01
average membrane
stress  4.812E+05 4.809E+05-1.600E+05-4.806E+05 0.000E+00 6.109E+02 5.763E+02-4.806E+05 4.081E+03
moment  2.858E+02 2.659E+02-1.104E+02-2.858E+02-4.554E+01 0.000E+00-4.822E+01-2.831E+02-2.523E+01
```

```
stretch 6.590E-06 4.801E-06 0.000E+00-4.759E-06 0.000E+00 1.831E-06 1.819E-06-4.746E-06 5.657E-07
curvatr 1.935E-04 2.251E-04 0.000E+00-1.935E-04-2.940E-06 0.000E+00-2.945E-06-1.934E-04 2.106E-06
layer  1
stress  1.660E+06 1.652E+06-5.582E+05-1.660E+06-1.518E+04 0.000E+00-2.466E+04-1.650E+06-1.245E+05
stressp 1.660E+06 1.652E+06-5.582E+05-1.660E+06-1.518E+04 0.000E+00-1.659E+06-1.539E+04-1.855E+04

element   22  point   3      integration pt. coordinate=     0.242E+01    0.903E+00    0.988E+01
section thickness = 0.100E-01
average membrane
stress  5.627E+05 5.417E+05-2.025E+05-5.627E+05-4.482E+04 0.000E+00-4.486E+04-5.627E+05-4.450E+03
moment  2.854E+02 2.669E+02-1.090E+02-2.854E+02-4.176E+01 0.000E+00-4.439E+01-2.827E+02-2.520E+01
stretch 7.123E-06 5.499E-06 0.000E+00-5.387E-06 0.000E+00 1.736E-06 1.725E-06-5.375E-06 5.657E-07
curvatr 1.945E-04 2.234E-04 0.000E+00-1.938E-04 0.000E+00 7.113E-07 7.056E-07-1.938E-04 2.106E-06
layer  1
stress  1.851E+06 1.839E+06-6.256E+05-1.851E+06-2.532E+04 0.000E+00-3.601E+04-1.841E+06-1.393E+05
stressp 1.851E+06 1.839E+06-6.256E+05-1.851E+06-2.532E+04 0.000E+00-1.851E+06-2.553E+04-1.951E+04

element   22  point   4      integration pt. coordinate=     0.358E+01    0.903E+00    0.988E+01
section thickness = 0.100E-01
average membrane
stress  4.855E+05 4.797E+05-1.658E+05-4.855E+05-1.187E+04 0.000E+00-1.188E+04-4.855E+05 2.590E+03
moment  2.863E+02 2.664E+02-1.106E+02-2.863E+02-4.561E+01 0.000E+00-4.830E+01-2.836E+02-2.528E+01
stretch 6.496E-06 4.816E-06 0.000E+00-4.759E-06 0.000E+00 1.737E-06 1.725E-06-4.746E-06 5.657E-07
curvatr 1.938E-04 2.255E-04 0.000E+00-1.938E-04-2.940E-06 0.000E+00-2.945E-06-1.938E-04 2.106E-06
layer  1
stress  1.661E+06 1.652E+06-5.598E+05-1.661E+06-1.806E+04 0.000E+00-2.755E+04-1.652E+06-1.245E+05
stressp 1.661E+06 1.652E+06-5.598E+05-1.661E+06-1.806E+04 0.000E+00-1.661E+06-1.827E+04-1.839E+04
```

maximum stress in the preferred direction

⋮

```
element   45  point   1      integration pt. coordinate=     0.842E+01    0.507E+01    0.852E+01
section thickness = 0.100E-01
average membrane
stress  3.703E+05 3.624E+05-1.289E+05-3.703E+05-1.639E+04 0.000E+00-1.640E+04-3.703E+05 2.080E+03
moment  3.434E+02 3.188E+02 1.334E+02 0.000E+00 5.675E+01 3.434E+02 6.043E+01 3.397E+02 3.227E+01
stretch 4.860E-06 3.657E-06 0.000E+00-3.603E-06 0.000E+00 1.257E-06 1.247E-06-3.593E-06 4.490E-07
curvatr 2.309E-04 2.701E-04 0.000E+00 0.000E+00 5.939E-06 2.309E-04 5.997E-06 2.309E-04 7.253E-06
layer  1
stress  7.490E+05 7.457E+05-2.453E+05-7.424E+05 0.000E+00 6.574E+03 2.692E+03-7.386E+05-5.379E+04
stressp 7.490E+05 7.457E+05-2.453E+05-7.424E+05 0.000E+00 6.574E+03-7.423E+05 6.401E+03-1.139E+04

element   45  point   2      integration pt. coordinate=     0.958E+01    0.507E+01    0.852E+01
section thickness = 0.100E-01
average membrane
stress  5.077E+05 4.747E+05-1.941E+05-5.077E+05-7.474E+04 0.000E+00-7.499E+04-5.074E+05-1.043E+04
moment  3.365E+02 3.237E+02 1.213E+02 0.000E+00 2.731E+01 3.365E+02 3.058E+01 3.332E+02 3.164E+01
stretch 5.974E-06 4.888E-06 0.000E+00-4.718E-06 0.000E+00 1.255E-06 1.247E-06-4.710E-06 4.490E-07
curvatr 2.529E-04 2.549E-04 0.000E+00-2.194E-05 0.000E+00 2.309E-04-2.189E-05 2.309E-04 7.253E-06
```

```
layer  1
stress  1.082E+06 1.077E+06-3.645E+05-1.082E+06-1.103E+04 0.000E+00-1.705E+04-1.076E+06-8.008E+04
stressp 1.082E+06 1.077E+06-3.645E+05-1.082E+06-1.103E+04 0.000E+00-1.082E+06-1.119E+04-1.311E+04

element   45  point   3      integration pt. coordinate=     0.842E+01    0.654E+01    0.746E+01
section thickness = 0.100E-01
average membrane
stress  4.187E+05 3.699E+05-1.851E+05-4.187E+05-1.365E+05 0.000E+00-1.371E+05-4.182E+05-1.237E+04
moment  9.475E+01 8.658E+01 3.818E+01 0.000E+00 1.977E+01 9.475E+01 2.112E+01 9.341E+01 9.948E+00
stretch 3.952E-06 3.978E-06 0.000E+00-3.605E-06 0.000E+00 3.469E-07 3.341E-07-3.593E-06 4.490E-07
curvatr 6.254E-05 7.576E-05 0.000E+00 0.000E+00 5.764E-06 6.254E-05 5.997E-06 6.231E-05 7.253E-06
layer  1
stress  9.851E+05 9.702E+05-3.386E+05-9.851E+05-3.053E+04 0.000E+00-3.598E+04-9.797E+05-7.191E+04
stressp 9.851E+05 9.702E+05-3.386E+05-9.851E+05-3.053E+04 0.000E+00-9.850E+05-3.066E+04-1.112E+04

element   45  point   4      integration pt. coordinate=     0.958E+01    0.654E+01    0.746E+01
section thickness = 0.100E-01
average membrane
stress  5.570E+05 4.897E+05-2.503E+05-5.570E+05-1.940E+05 0.000E+00-1.957E+05-5.553E+05-2.488E+04
moment  9.743E+01 9.299E+01 2.606E+01-9.630E+00 0.000E+00 8.780E+01-8.732E+00 8.691E+01 9.313E+00
stretch 5.064E-06 5.263E-06 0.000E+00-4.720E-06 0.000E+00 3.441E-07 3.341E-07-4.710E-06 4.490E-07
curvatr 8.451E-05 6.336E-05 0.000E+00-2.205E-05 0.000E+00 6.247E-05-2.189E-05 6.231E-05 7.253E-06
layer  1
stress  1.325E+06 1.302E+06-4.578E+05-1.325E+06-4.812E+04 0.000E+00-5.572E+04-1.318E+06-9.821E+04
stressp 1.325E+06 1.302E+06-4.578E+05-1.325E+06-4.812E+04 0.000E+00-1.325E+06-4.825E+04-1.284E+04




                          n o d a l   p o i n t   d a t a



                          t o t a l   d i s p l a c e m e n t s



    1      0.         0.          0.         0.         0.         0.         0.

                          summary of externally applied loads

  0.00000E+00   0.00000E+00   -0.78217E+05   0.00000E+00   0.00000E+00   0.00000E+00   0.00000E+00

                          summary of reaction/residual forces

 -0.31704E-11   0.54570E-11   0.78217E+05   0.47510E-11   -0.16080E-11   -0.61590E-11   -0.28941E-12
```

```
distributed load    type    current
list number                 magnitude



        1           1      500.0          0.          0.

         e n d   o f   i n c r e m e n t   0

             formatted post data at increment   0.   0   on tape 19
              time =      11.55

             *** end of input deck - job ends


                       marc exit number 3004
```

Results

For the composite roof panel, the maximum first component of the preferred stress in layer 1 occurs in element 22 (integration point 3) and is -1.851×10^6 psi.

Figure 4.3 is the deformed geometry, and Figure 4.4 shows a Z-displacement contour plot. Note that the deformation is magnified by the automatic scaling of Mentat II. Figure 4.5 shows a contour plot of the first component of preferred stress in layer 1 of the composite shell roof.

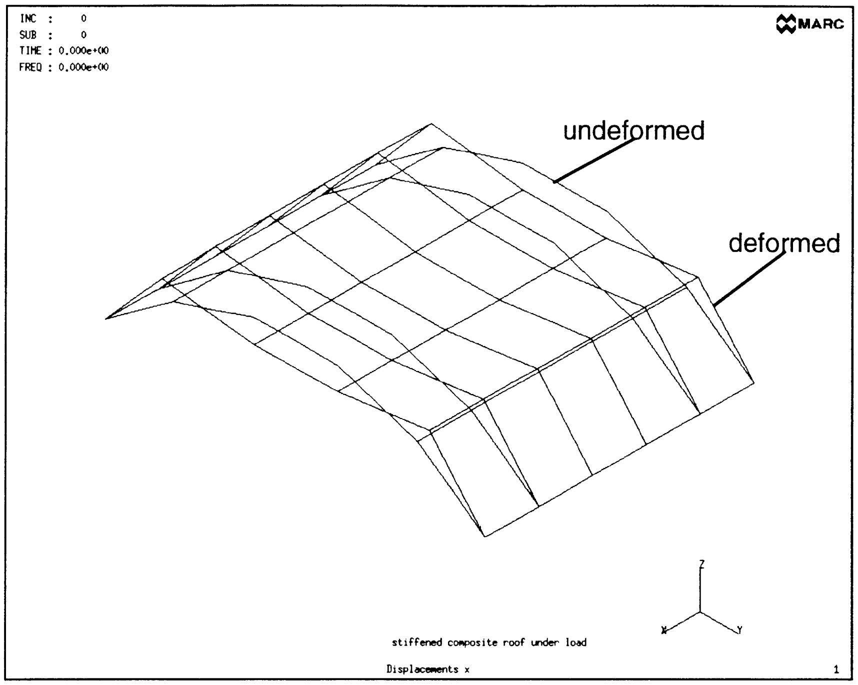


Figure 4.3 **Deformed geometry**

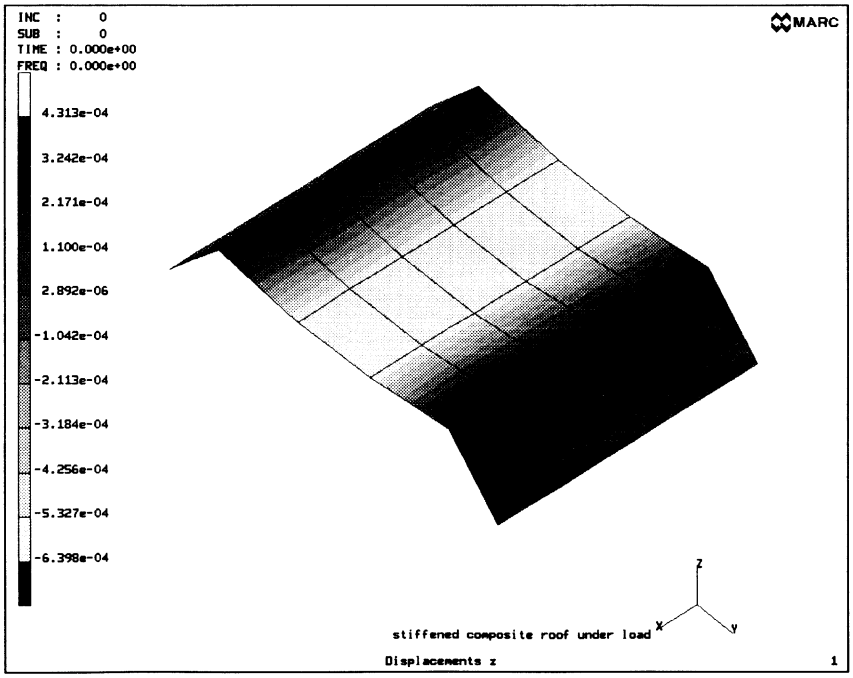


Figure 4.4 **Z-displacement contours**

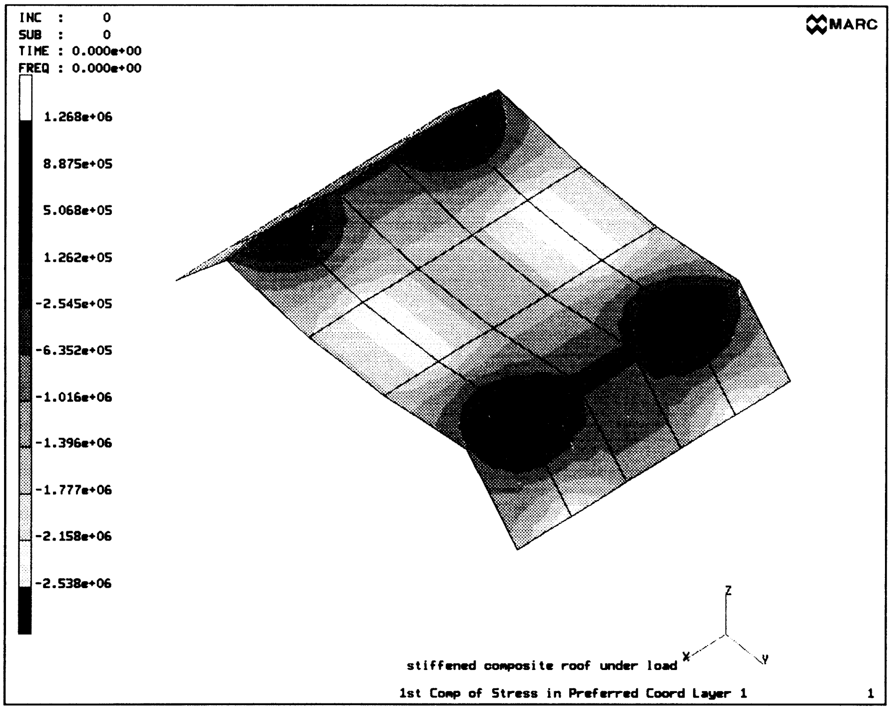


Figure 4.5 **First component of stress in the preferred direction - Layer 1**

Exercises

What do you think will happen if the material orientation angle in layer 1 is parallel to the global X-axis?

The contribution of the beam stiffeners to the bending stiffness of the roof is obviously quite significant. Try varying the beam section properties and see how the deformed geometry changes.

Composite materials are often used for thick shell structures. For this type of analysis, either element 22 or element 75 should be used. In addition, the inclusion of the TSHEAR parameter chard results in a more realistic transverse shear calculation and the calculation of the interlaminar shear stress.

CHAPTER 3: Nonlinear Structural Problems – Plasticity, Large Deformation, and Post-Buckling Analysis

In this chapter, we begin using MARC to analyze *nonlinear* structural problems. You should recognize at the outset that, by its very nature, a nonlinear structural problem is *more complex* – and therefore more difficult (and expensive) to analyze – than a linear problem. The principle of superposition no longer applies! A problem is nonlinear if the force-displacement relationship depends on the *current state* (i.e., current displacement, force, and stress-strain relations). Commonly, nonlinearities arise from three sources:

• **geometric** nonlinearity	large deformations (displacements), finite strains, buckling/collapse, snap-through, etc.
• **material** nonlinearity	plasticity (isotropic/kinematic/combined hardening), rigid plastic flow, creep, viscoelasticity, viscoplasticity, etc.
• **boundary** nonlinearity	opening/closing of gaps, contact/friction, etc.

You should *not* attempt to perform nonlinear analysis until you have a firm grasp of linear FE analysis fundamentals. In general, the solution of nonlinear problems usually requires *incremental* solution schemes, and often requires *iterations* (called "cycles" in MARC output) within each load or time increment to ensure that equilibrium is properly satisfied. A good idea is to first do a "dry run": make a small FE model; try making a MARC run using the nonlinear analysis option(s); and examine how MARC converges in the solution process. Once you have an understanding of how MARC solves your type of nonlinear problem, you'll be better prepared to tackle your actual problem.

This chapter presents three examples which illustrate plasticity, large deformation, and post-buckling behavior. Again, the FE model in each case (a beam in Example 5, a square plate in Example 6, a spherical cap in Example 7) is intentionally kept simple. Typical input concepts for these types of nonlinear structural analyses are introduced and explained in detail. Selective output and results are then discussed. [Later, Chapter 4 will discuss a heat transfer analysis, followed by a thermal stress problem. Finally, Chapter 5 will illustrate the analyses of contact and rubber (elastomer) problems.] The three examples covered in this chapter are:

Example 5	Cantilevered Beam Loaded by Tip Load
Example 6	Large Displacement and Plastic Analysis of a Simply-Supported Square Plate
Example 7	Post-Buckling Analysis of a Spherical Cap Under Apex Load

Example 5

Cantilevered Beam Loaded by Tip Load

The purpose of this example is to illustrate the elastic-plastic analysis of a cantilevered beam. Large deformation is not considered. As the tip load is applied gradually, the beam cross section yields further and further until a plastic hinge forms at the restrained end and the beam loses its capacity to resist any more load. New MARC options seen for the first time include: SCALE, POINT LOAD, CONTROL, PROPORTIONAL INCREMENT, and AUTO LOAD.

Sketch

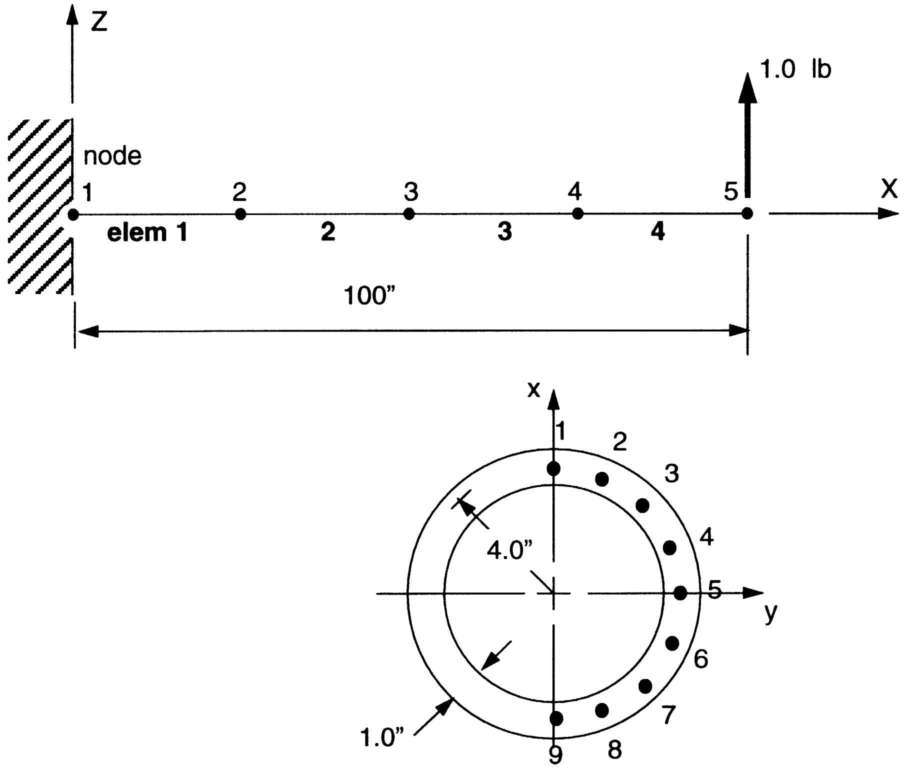


Figure 5.1 **Cantilevered Beam**

Model

This cantilevered beam is 100 inches long. The finite element model has five nodes and four beam elements (MARC Element 25), each 25 inches long. The left end of the beam is fixed against all displacements and rotations.

Young's modulus is 30E6 (30 x 10^6) psi, Poisson's ratio is 0.3, and the tensile yield stress is 20,000 psi.

MARC Element 25 is a 3-D, thin-walled, closed section beam (the default section being a hollow circular cylinder – as used in this example). It is a straight element which does not allow warping of the sections but does allow twist; it can handle large displacements (but not finite strains). It has two nodes and three Gaussian integration points along the length. Each node has seven DOFs: three translations, three rotations, and a seventh DOF which measures the rates of change of displacement along the beam axis. For element input, you need to specify the wall thickness, radius of the cross section, and the orientation of the cross section. You may request output of element quantities either at the centroid or at the three Gaussian integration points along the length.

In the sketch of the beam cross section on the previous page, 9 of the 16 integration points are shown. In later discussions, these points also correspond to the "layers."

Properties

This example is the first one where you'll encounter nonlinear material properties. In this case, the plasticity behavior of the metal is defined using the von Mises yield criterion and isotropic work hardening model. You'll see later when we describe the input file that both of these theories are specified within the ISOTROPIC model definition block.

The von Mises yield criterion, sometimes called the Huber-von Mises yield criterion, is the most widely used yield criterion for metals implemented in FE codes. "A yield criterion is a hypothesis concerning the limit of elasticity under any possible combination of stresses." (Reference: W. Johnson and P.B. Mellor, *Engineering Plasticity*, Van Nostrand Reinhold Co., London, 1973, Chapter 4.) The von Mises criterion states that yielding occurs when the effective stress equals the yield stress as measured in a uniaxial test. The success of the von Mises criterion is due to: the continuous nature of the function that defines this criterion, and its good agreement with data from extensive experiments on ductile metals. (In MARC, *the von Mises criterion is the default yield criterion.*)

The isotropic work-hardening rule assumes that during plastic flow, the yield surface expands uniformly about the origin in stress space, maintaining the same shape, center, and orientation. You should be aware, however, that this rule does not account for the Bauschinger effect exhibited by most structural materials, that is, unloading effects in cyclic loading problems. (In MARC, *the isotropic hardening rule – with a hardening slope of zero – is the default hardening rule*. This means that the *default material* is elastic-perfectly plastic, with a constant yield stress as specified by the data lines.

Loads

The applied load at the beam tip (node 5) is 1.0 lb. in the positive global Z-direction.

Boundary Conditions

The left end (node 1) of the beam is fully clamped. All six degrees of freedom are suppressed.

Input

A complete input listing (with comments) is included.

PARAMETER Section

The “SIZING” line sets 100,000 words as the workspace. In addition, the third field also informs MARC that the maximum number of elements is 4, and the fourth field says that the maximum number of nodes is 5.

The “ELEMENTS” line indicates that MARC Element 25 will be used.

The “SCALE” option is used for the first time. In elastic-plastic analysis, this convenient option scales the linear elastic solution to first yield in the highest stressed element in the model. This option applies only to small displacement, elastic-plastic, quasi-static analysis where element properties are not temperature dependent, because only for these types of problems can the solution be scaled linearly.

The “END” line terminates the PARAMETER options.

MODEL DEFINITION Section

The MODEL DEFINITION options in this example consist of:

FE mesh topology

Geometric properties

Material properties

Boundary conditions

Loads

Nonlinear analysis controls

Output controls

FE Mesh Topology

The FE model is simple: five nodes numbered 1 to 5 from the clamped end to the tip, equidistant at 25.0 inches, and connected by four beam elements (MARC Element 25). The CONNECTIVITY and COORDINATES data are self-explanatory. The beam model lies in the global X-Z plane.

Geometric Properties

The GEOMETRY block is used to enter element geometric properties. The blank line means we do not need to count the number of geometric properties input. The next line (“1.,4., , 0.,0.,1.,”) shows six data items:

The first field says the thickness of the hollow circular cylinder is 1.0 in.;

The second field prescribes a radius of 4.0 in.;

The third field is not used for MARC Element 25 and is blank;

The fourth, fifth, and sixth fields are the three components of a vector representing the local x-axis, which in this case is aligned with the positive global Z-axis.

The “1 TO 4” line is the last line in this block, and assigns these geometric properties to elements 1 through 4.

Material Properties

In this example, all the material properties are input using the ISOTROPIC block. This option lets you define material properties, a yield criterion, and a strain (work) hardening law for an isotropic material.

The next line ("1,VON MISES,") sets the material identification number to be 1 and says the von Mises yield criterion shall be used.

NOTE

The material in this example happens to be elastic-perfectly plastic, which means that the yield stress is constant and the hardening slope is zero

This is followed by the "30.E6,.3,,,20000," line. The first two fields refer to Young's modulus and Poisson's ratio. The third field is for mass density while the fourth field is for coefficient of thermal expansion; these two material properties are not needed for this problem and are left blank. The fifth field sets the equivalent (von Mises) tensile yield stress to be 20,000 psi.

The last line in this block ("1 TO 4") says that the above material properties apply to elements 1 to 4.

Boundary Conditions

In the FIXED DISP block, the blank line is followed by a line with six zeroes, which refer to the six zero-valued displacements being prescribed. The next "1 TO 6" line is the list of the six DOFs (three translations, three rotations) for which the zero displacements are applicable. And the last line ("1") indicates these boundary conditions apply to node 1 only. You should not constrain the seventh DOF of nodes along the beam as this represents an axial strain.

Loads

The tip load in this example is applied using the POINT LOAD option. The "0.,0.,1.," line specifies the magnitude of the point load in the first three DOFs (global X, Y, Z directions). The last line ("5") in this block means the point load is applied at node 5, which is the tip of the beam. In other words, a tip load of 1.0 lb. is applied in the positive Z-direction. Of course, this initial load will be scaled, so that at least one integration point is at the yield stress.

Nonlinear Analysis Controls

The CONTROL block allows you to input parameters for controlling the convergence and accuracy of the nonlinear analysis. The "40,6," line means that 40 is the maximum number of load steps we are allowing in this analysis, and the maximum number of iterations required to achieve convergence during an increment due to plasticity is 6. The ".1," line refers to the maximum allowed relative error in residual forces; this value also happens to be the default value in nonlinear stress analysis. The program will iterate until the maximum error in equilibrium is less than 10 percent of the maximum reaction force.

Output Controls

Output controls for this example consist of two separate blocks: PRINT ELEM and POST.

The PRINT ELEM block allows us to specify which elements and what element quantities for those elements are to be printed out. The blank line means we do not need to enter the number of sets which follow. In this example, we are going to request printed output for two element quantities: STRESS (total stress) and PLASTIC (plastic strain), for two groups of elements. After the "STRESS PLAST" line, the first group of three lines follows. The "2 TO 4" line refers to the list of elements to be printed. The "2," line is the integration point to be printed for these three elements. The third line of this group ("1") means we only want layer 1 (i.e., integration point 2 in the beam cross section) to be printed for these three elements. Next, we have a second "STRESS PLAST" line, followed by another group of three lines. The next "1" line means element 1. The next "1," line says we want integration point number 1 to be printed for element 1. and finally, the "1 TO 9" line means we would like layer 1 printed for element 1.

The POST block creates a post-processor file for later post-processing by Mentat II. The ",,,1" line means we would like to have a formatted post file. The next four lines tell MARC four post variables are to be stored in a file: the first component of stress for the beam itself, and also for layers 1, 5, and 9.

The "END OPTION" line terminates the MODEL DEFINITION options.

LOAD INCREMENTATION section

In this example, the LOAD INCREMENTATION options consist of two options: PROPORTIONAL INCREMENT and AUTO LOAD.

The PROPORTIONAL INCREMENT option scales the previous load increment up or down for use in the current load increment. The "0,.1" line means that the minimum number of iterations will default to the value specified on the CONTROL option, and the ratio of the next load increment to the present increment is to be 0.1 (this ratio being known as the "load factor" in the MARC output). This 0.1 ratio means that the incremental load is equal to 10 percent of the load which causes yield. MARC knows this load from the results of scaling in increment zero.

The AUTO LOAD option is a convenient feature for nonlinear analysis with proportional loads. It generates a specified number of increments, each with the same load increment. Here, the "10," line tells MARC to apply the tip load over ten equal load increments. Thus, the total desired applied load in this analysis is twice the load required to produce the initial yield.

The "CONTINUE" line ends the LOAD INCREMENTATION options and the input file.

Output

Since this is the very first nonlinear analysis example, the entire output listing is included for your information. Notice in the MARC output, the description of MARC Element 25 shows a twisting strain, a torque capability about the beam axis, and a seventh DOF which is labeled "stretch." Under the MARC interpretation of CONTROL, we see the message: "Full Newton-Raphson technique chosen." We are by default using the "full Newton-Raphson" technique as the nonlinear solution procedure, because we have not flagged any of the procedures in the sixth field on the first line following CONTROL ("40,6,"). You should always use the defaults provided by MARC unless you have a special reason not to! For more details regarding the solution of nonlinear problems, please see Volume A.

At the beginning of the output for increment 0, we see the consequence of using the SCALE option. MARC informs us that the "increment 0" results have been scaled by 0.103E5, in order to reach yield stress in element 1 (which is the element closest to the clamped end). If we examine the stresses for each layer in element 1 (integration point 1), the point nearest the fixed end where the stresses are the highest, we see that layers 1 and 9 have indeed yielded, but yielding has not occurred anywhere else in the beam cross section. The use of SCALE instructs MARC to search for the highest von Mises stress at all integration points in the model and make the material yield at that location, by scaling the applied load appropriately. Notice that it does not matter at all what value you input as the magnitude of the tip load. Instead of 1.0 lb., we could have input a tip load value of 34.567, or 1009, or whatever. All that would change would be the scale factor you see printed out before increment 0 results in the output; the results for increment 0 would be identical.

As the nonlinear solution proceeds, MARC informs us that the solution converged in increment 3, but not in increment 4. At the beginning of increment 4 output, MARC informs us that "Failure to converge to tolerance–Increment will be recycled." This message occurs a total of six times, each time showing the "convergence ratio." (Recall that we had originally specified the maximum number of recycles to be six on the "CONTROL" line.) Finally, MARC tells us: "Failure to converge to tolerance***ERROR–Too many recycles–Job ends at this increment." MARC then proceeds to print out the results for increment 4 (which *do not* satisfy equilibrium to the tolerance requested...), and ends the job with the messages "Analysis failed to converge during this increment" and "MARC exit number 3002" (which means convergence has not occurred within the allowable number of recycles). Oops! Is this a bad run? What is really happening?

Let's go back and look carefully at the output of the last "good" increment: increment 3. We see that layers 1, 2, 3, 7, 8, and 9 of the beam cross section had indeed yielded, and layers 4 and 6 had von Mises stresses of 18,330 psi – very close to the yield stress of 20,000 psi. (Layer 5, of course, should be unstressed because it is at the neutral axis of the beam.) An examination of the (unconverged) increment 4 output reveals that the entire cross section (all layers except 5) has yielded. The conclusion is obvious: a plastic hinge has formed at the left end, and the beam has lost its ability to resist the load any further! MARC cannot converge to equilibrium because no unique equilibrium state exists! Another indication that a plastic hinge has formed is that in increment 4, the Z-displacements are huge.

Let's examine how the singularity ratio decreases from increment to increment:

Increment	Singularity Ratio
0	1.5625E-02
1	1.4726E-02
2	1.1206E-02
3	7.5594E-03
4 cycle 1	5.7177E-03
cycle 2	5.2343E-03
cycle 3	3.2148E-03
cycle 4	1.4614E-03
cycle 5	6.5630E-04
cycle 6	2.9310E-04
cycle 7	1.3057E-04

Notice that as prescribed in the CONTROL block, MARC went through six recycles in increment 4 before informing you that it failed to converge due to too many recycles. Recall in Chapter 1, when discussing singularity ratios in interpreting analysis messages, we had said that a value between 10^{-4} and 10^{-8} means to watch out for possible numerical problems. A singularity ratio is a measure of the conditioning number (or accuracy) in the solution of the linear equations. Therefore, in nonlinear analysis, it is always a good idea to watch this parameter from increment to increment and see whether it decreases to unacceptably low values.

input data

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
TITLE    BENDING OF A BEAM UNTIL A PLASTIC HINGE FORMS
SIZING        100000    4    5
ELEMENTS     25
COMMENT, SCALE CARD TELLS MARC TO SCALE INCREMENT 0 RESULTS UP TO
COMMENT,  POINT OF FIRST YIELD IN THE MODEL.
SCALE
END
CONNECTIVITY
    4    0    0
    1   25    1    2
    2   25    2    3
    3   25    3    4
    4   25    4    5
COORDINATES
    3    5    0    0
    1   0.00000   0.00000   0.00000
    2  25.00000   0.00000   0.00000
    3  50.00000   0.00000   0.00000
    4  75.00000   0.00000   0.00000
    5 100.00000   0.00000   0.00000
COMMENT, SPECIFY THICKNESS, RADIUS, AND ORIENTATION OF CROSS SECTION
GEOMETRY

1.,4.,,0.,0.,1.,
1 TO 4
COMMENT, SPECIFY PLASTICITY MODEL IN ISOTROPIC OPTION
COMMENT, VON MISES YIELD CRITERIA AND ISOTROPIC HARDENING ARE USED
COMMENT,  FOR NORMAL ENGINEERING PLASTICITY
ISOTROPIC

1,VON MISES,ISOTR HARD,
30.E6,.3,,,20000.,
1 TO 4
FIXED DISP

0.,0.,0.,0.,0.,0.,
1 TO 6
1
POINT LOAD

0.,0.,1.,
5
COMMENT, CONTROL OPTION IS USED TO SPECIFY CONTROL PARAMETERS
COMMENT,  GOVERNING THE MARC'S SOLUTION OF THE NONLINEAR EQUILIBRIUM
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
COMMENT,  EQUATIONS.
CONTROL
40,6,
.1,
PRINT ELEM

STRESS PLAST
2 TO 4
2,
1
STRESS PLAST
1
1,
1 TO 9
POST
,,,1
11,
11,1,
11,5,
11,9,
END OPTION
COMMENT, SPECIFY 10 ADDITIONAL LOAD STEPS WITH EACH STEP
COMMENT,  ADDING 10 PERCENT OF THE LOAD TO CAUSE INITIAL YIELD
PROPORTIONAL INC
0,.1,
AUTO LOAD
10,
CONTINUE
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
--------------------------------------------------------------------------------
```

```
                    *************************************************
                    *************************************************


                     program sizing and options requested as follows



                     element type requested*************************   25
                     number of elements in mesh*********************    4
                     number of nodes in mesh************************    5
                     max number of elements in any dist load list***    0
                     maximum number of boundary conditions**********    6
                     scaling to first yield was flagged*************
                     load correction flagged or set*****************
                     number of lists of distributed loads***********    3
                     stresses stored at all integration points******
```

```
           tape no.for input of coordinates + connectivity    5
           no.of different materials    1 max.no of slopes    5
           maximum elements variables per point on post tp   33
           number of points on shell section *************   11
           new style input format will be used************
           maximum number of set names is*****************   10
           number of processors used *********************    1
           vector length used ****************************    1


           end of parameters and sizing
          *************************************************
          *************************************************



 key to stress, strain and displacement output



           element type   25
```

NOTE

Default cross section for element type 25 is a circular pipe with the radius and thickness given through the GEOMETRY option.

```
 2-node thin walled,closed section (no warping) beam


 strains -
   1=axial stretch
   2=local xx curvature
   3=local yy curvature
   4=twist


 section forces -
   1=axial force
   2=local xx moment
   3=local yy moment
   4=torque about beam axis


 displacements-
   1=u global x direction
   2=v global y direction
   3=w global z direction
   4=theta x rotation about global x axis
   5=theta y rotation about global y axis
   6=theta z rotation about global z axis
   7=stretch
```

This degree of freedom is like an axial strain - DO NOT put a boundary condition on it.

```
 workspace needed for input and stiffness assembly     29255
```

```
                 internal core allocation parameters
                 degrees of freedom per node (ndeg)  7
                 coords per node (ncrd)  6
                 strains per integration point (ngens)  4
                 max. nodes per element (nnodmx)  2
                 max.stress components per int. point (nstrmx)  32
                 max. invariants per int. points (neqst) 16


                 flag for element storage (ielsto)  0
                 elements in core, words per element (nelsto)      5668
                                   total space required           22672
                 vectors in core, total space required       510


                 words per track on disk set to 4096


                 internal element variables



                 internal element number  1  library code type 25
                 number of nodes=  2
                 stresses stored per integration point = 33
                 direct continuum components stored =  1
                 shear continuum components stored =  1
                 shell/beam flag = 1
                 curvilinear  coord. flag = 0
                 int.points for elem. stiffness  3
                 number of local inertia directions  4
                 int.point for print if all points not flagged  2
                 int. points for dist. surface loads (pressure)  3
                 library code type = 25
                 no local rotation flag = 1
                 generalized displ. flag = 0
                 large disp. row counts     6    0    0    0



                 residual load correction is invoked



connectivity
----------

meshr1,iprnt
    5      0
elem no., type,        nodes
    1      25         1    2
```

```
    2      25         2    3
    3      25         3    4
    4      25         4    5

coordinates
----------

ncrd1 ,meshr1,iprnt
    3      5      0
 node      coordinates
    1      0.          0.          0.
    2  25.000          0.          0.
    3  50.000          0.          0.
    4  75.000          0.          0.
    5  100.00          0.          0.
comment, specify thickness, radius, and orientation of cross section

geometry
----------

  egeom1     egeom2     egeom3     egeom4     egeom5     egeom6
  0.100E+01  0.400E+01  0.000E+00  0.000E+00  0.000E+00  0.100E+01
from element     1 to element     4 by     1
comment, specify plasticity model in isotropic option
comment, von mises yield criteria and isotropic hardening are used
comment,  for normal engineering plasticity

isotropic
----------
```

Yield stress of 20,000 psi.

```
isotropic material  material id =     1
 von mises  yield criteria
 isotropic  hardening rule
   e          nu         rho        alpha       yield      yield2
  0.300E+08  0.300E+00  0.000E+00  0.000E+00  0.200E+05  0.200E+05
from element     1 to element     4 by     1

fixed disp
----------

fixed displacement =  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
from degrees of freedom    1 to degrees of freedom    6 by     1
a list of nodes given below
     1



fixed boundary condition summary.
total fixed degrees of freedom read so far =     6
```

```
 b.c.    node   degree of  magnitude             b.c.    node   degree of  magnitude
number           freedom                        number           freedom


    1       1         1    0.000E+00               2       1         2    0.000E+00
    3       1         3    0.000E+00               4       1         4    0.000E+00
    5       1         5    0.000E+00               6       1         6    0.000E+00

point load
----------

read from unit     5
 0.000E+00 0.000E+00 0.100E+01 0.000E+00 0.000E+00 0.000E+00 0.000E+00
a list of nodes given below
     5
comment, control option is used to specify control parameters
comment,  governing the marc's solution of the nonlinear equilibrium
comment,  equations.

control
----------

max.     max.     min.
incs   recycles recycles
   40       6       0

maximum allowed relative error in residual forces      0.10000E+00


full newton-raphson technique chosen

print elem
----------



values will be printed at integration points

element quantities printed every    1 increments

stress plast
from element     2 to element     4 by     1
a list of integration points given below
     2
a list of layers given below
     1

stress plast
a list of elements given below
     1
a list of integration points given below
     1
```

```
  from layer     1 to layer     9 by     1

  post
  ----------

  *** note - format of post code cards has changed.
            in k4, enter code in first field and  layer number in second field

  elem vars,post tape,prev tape,  type  , conn fl ,post tape, prev tape,  repost ,frequency, k2post
         0        16        17        1        1        19        20        0        1        0
  element variables appear on post-processor tape 16 in following order
  post variable   1 is post code   11 =
  post variable   2 is post code   11 at layer   1 =
  post variable   3 is post code   11 at layer   5 =
  post variable   4 is post code   11 at layer   9 =

 ***maximum record length on formatted post file=   80     approximate no. of records per increment on file=      21

  end option
  ----------

                  maximum connectivity is    2  at node    2


                  maximum half-bandwidth is    2  between nodes    1  and    2

                   number of profile entries including fill-in is          9

                   number of profile entries excluding fill-in is          9


                  total workspace needed with in-core matrix storage =     30277


                  load increments associated with each degree of freedom
                  summed over the whole model

                  distributed loads
0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                  point loads
0.000E+00 0.000E+00 1.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                  start of assembly
                   time =        0.97


                  start of matrix solution
                   time =        1.09
```

```
                              singularity ratio    1.5625E-02


                              end of matrix solution
                               time =        1.10

MARC       output for increment    0.    ending of a beam until a plastic hinge forms

                         solution given below has been scaled by 0.103E+05  to reach yield stress in element     1

         tresca     mises    mean   p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
       intensity intensity  normal   minimum intermediate maximum     1        2        3        4        5        6
                         intensity

element     1  point   1
 axial force    0.00000E+00     local xx moment    9.18781E-12     local yy moment    1.00531E+06  axial torque    0.00000E+00
axial strain    2.16625E-15  local xx curvature   -1.41759E-16  local yy curvature    1.66667E-04         twist    0.00000E+00
layer  1
stress  2.000E+04 2.000E+04-6.667E+03-2.000E+04 0.000E+00 0.000E+00-2.000E+04 0.000E+00
layer  2
stress  1.848E+04 1.848E+04-6.159E+03-1.848E+04 0.000E+00 0.000E+00-1.848E+04 0.000E+00
layer  3
stress  1.414E+04 1.414E+04-4.714E+03-1.414E+04 0.000E+00 0.000E+00-1.414E+04 0.000E+00
layer  4
stress  7.654E+03 7.654E+03-2.551E+03-7.654E+03 0.000E+00 0.000E+00-7.654E+03 0.000E+00
layer  5
stress  1.500E-07 1.500E-07 5.001E-08 0.000E+00 0.000E+00 1.500E-07 1.500E-07 0.000E+00
layer  6
stress  7.654E+03 7.654E+03 2.551E+03 0.000E+00 0.000E+00 7.654E+03 7.654E+03 0.000E+00
layer  7
stress  1.414E+04 1.414E+04 4.714E+03 0.000E+00 0.000E+00 1.414E+04 1.414E+04 0.000E+00
layer  8
stress  1.848E+04 1.848E+04 6.159E+03 0.000E+00 0.000E+00 1.848E+04 1.848E+04 0.000E+00
layer  9
stress  2.000E+04 2.000E+04 6.667E+03 0.000E+00 0.000E+00 2.000E+04 2.000E+04 0.000E+00

element     2  point   2
 axial force    4.59391E-12     local xx moment    2.29695E-11     local yy moment    6.46535E+05  axial torque    0.00000E+00
axial strain    1.39316E-15  local xx curvature   -9.11679E-17  local yy curvature    1.07187E-04         twist    0.00000E+00
layer  1
stress  1.286E+04 1.286E+04-4.287E+03-1.286E+04 0.000E+00 0.000E+00-1.286E+04 0.000E+00


element     3  point   2
 axial force    2.29695E-12     local xx moment    2.75634E-11     local yy moment    3.87921E+05  axial torque    0.00000E+00
axial strain    8.35894E-16  local xx curvature   -5.47007E-17  local yy curvature    6.43120E-05         twist    0.00000E+00
layer  1
stress  7.717E+03 7.717E+03-2.572E+03-7.717E+03 0.000E+00 0.000E+00-7.717E+03 0.000E+00


element     4  point   2
 axial force   -2.87119E-12     local xx moment    0.00000E+00     local yy moment    1.29307E+05  axial torque    0.00000E+00
axial strain    2.78631E-16  local xx curvature   -1.82336E-17  local yy curvature    2.14373E-05         twist    0.00000E+00
layer  1
stress  2.572E+03 2.572E+03-8.575E+02-2.572E+03 0.000E+00 0.000E+00-2.572E+03 0.000E+00
```

The solution has been scaled such that Layer 1 and Layer 9 are at yiel

Yield Stress

```
                       n o d a l   p o i n t   d a t a



                       i n c r e m e n t a l   d i s p l a c e m e n t s


  1       0.           0.           0.          0.         0.          0.       2.22905E-15
  2  4.87605E-14 -4.17853E-14  4.91272E-02      0.    -3.75153E-03 -3.19088E-15  1.67179E-15
  3  8.35894E-14 -1.51946E-13  0.17864          0.    -6.43120E-03 -5.47007E-15  1.11453E-15
  4  1.04487E-13 -3.07692E-13  0.36176          0.    -8.03900E-03 -6.83759E-15  5.57263E-16
  5  1.11453E-13 -4.86229E-13  0.57166          0.    -8.57494E-03 -7.29343E-15       0.



                       t o t a l   d i s p l a c e m e n t s


  1       0.           0.           0.          0.         0.          0.       2.22905E-15
  2  4.87605E-14 -4.17853E-14  4.91272E-02      0.    -3.75153E-03 -3.19088E-15  1.67179E-15
  3  8.35894E-14 -1.51946E-13  0.17864          0.    -6.43120E-03 -5.47007E-15  1.11453E-15
  4  1.04487E-13 -3.07692E-13  0.36176          0.    -8.03900E-03 -6.83759E-15  5.57263E-16
  5  1.11453E-13 -4.86229E-13  0.57166          0.    -8.57494E-03 -7.29343E-15       0.



                 total equivalent nodal forces (distributed plus point loads)


  1       0.           0.           0.          0.         0.          0.          0.
  2       0.           0.           0.          0.         0.          0.          0.
  3       0.           0.           0.          0.         0.          0.          0.
  4       0.           0.           0.          0.         0.          0.          0.
  5       0.           0.         10345.        0.         0.          0.          0.

                 reaction forces at fixed boundary conditions, residual load correction elsewhere


  1 -9.95346E-12  2.84674E-12  -10345.          0.     1.03446E+06  2.91647E-12 -3.31178E-11
  2  8.42216E-12 -1.66060E-12  8.72842E-11      0.    -5.88020E-10  5.93436E-11 -2.06907E-10
  3 -2.29695E-12 -1.18614E-12 -1.51599E-10      0.     1.83756E-09  1.81446E-11  1.10796E-10
  4  5.67060E-12  7.11685E-13  8.38388E-11      0.     2.38883E-09  3.23760E-11 -2.92464E-11
  5 -1.84235E-12 -7.11685E-13 -6.54632E-11      0.     2.14535E-09  5.83346E-12  9.73204E-12
```

Moment at built-in end is force times length

```
                      summary of externally applied loads

0.00000E+00    0.00000E+00    0.10345E+05    0.00000E+00    0.00000E+00    0.00000E+00

0.00000E+00
```

```
                       summary of reaction/residual forces

-0.12117E-26    0.20195E-27   -0.10345E+05    0.00000E+00    0.10345E+07    0.11861E-09

-0.14874E-09

                       e n d   o f   i n c r e m e n t   0

                       formatted post data at increment   0.   0   on tape 19
                        time =       1.26

      comment, specify 10 additional load steps with each step
      comment,  adding 10 percent of the load to cause initial yield

      proportional inc
      ----------



      min. recycles    load factor

          0            0.1000000E+00

      auto load
      ----------


      iotnum,incasm
         10    0

      continue
      ----------


      equal load incs specified for  10  increments


                       s t a r t   o f   i n c r e m e n t   1




                       load increments associated with each degree of freedom
                       summed over the whole model

                       distributed loads
   0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
```

```
                         point loads
            0.000E+00 0.000E+00 1.034E+03 0.000E+00 0.000E+00 0.000E+00 0.000E+00
```

NOTE

MARC first tried to do back substitution only. It then predicted further plasticity so MARC reassembled the stiffness matrix.

```
                         load increments associated with each degree of freedom
                         summed over the whole model

                         distributed loads
            0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00



                         point loads
            0.000E+00 0.000E+00 1.034E+03 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                         start of assembly
                          time =       1.30

                         start of matrix solution
                          time =       1.39

                         singularity ratio    1.4726E-02

                         end of matrix solution
                          time =       1.40
```

Singularity ratio is smaller than before due to plasticity.

```
                        maximum residual force  at node    2 degree of freedom    3 is equal to    0.638E+03
                        maximum reaction force  at node    1 degree of freedom    3 is equal to    0.107E+05
                        convergence ratio                                                          0.593E-01
 MARC      output for increment    1.    ending of a beam until a plastic hinge forms

          tresca     mises    mean   p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
        intensity intensity  normal  minimum intermediate maximum      1         2         3         4         5         6
                            intensity



element     1  point  1
 axial force  -4.64206E-08     local xx moment   6.71862E-08     local yy moment   1.08597E+06  axial torque   0.00000E+00
axial strain   2.92928E-15  local xx curvature  -2.41119E-17  local yy curvature   1.87699E-04         twist   0.00000E+00
layer  1
stress  2.000E+04 2.000E+04-6.667E+03-2.000E+04 0.000E+00 0.000E+00-2.000E+04 0.000E+00
plas.st 8.413E-05 6.869E-05 0.000E+00-8.413E-05 0.000E+00 0.000E+00-8.413E-05 0.000E+00
layer  2
stress  2.000E+04 2.000E+04-6.667E+03-2.000E+04 0.000E+00 0.000E+00-2.000E+04 0.000E+00
plas.st 2.698E-05 2.203E-05 0.000E+00-2.698E-05 0.000E+00 0.000E+00-2.698E-05 0.000E+00
```

Development of plastic strains.

```
layer  3
stress  1.593E+04 1.593E+04-5.309E+03-1.593E+04 0.000E+00 0.000E+00-1.593E+04 0.000E+00
layer  4
stress  8.620E+03 8.620E+03-2.873E+03-8.620E+03 0.000E+00 0.000E+00-8.620E+03 0.000E+00
layer  5
stress  1.999E-07 1.999E-07 6.664E-08 0.000E+00 0.000E+00 1.999E-07 1.999E-07 0.000E+00
layer  6
stress  8.620E+03 8.620E+03 2.873E+03 0.000E+00 0.000E+00 8.620E+03 8.620E+03 0.000E+00
layer  7
stress  1.593E+04 1.593E+04 5.309E+03 0.000E+00 0.000E+00 1.593E+04 1.593E+04 0.000E+00
layer  8
stress  2.000E+04 2.000E+04 6.667E+03 0.000E+00 0.000E+00 2.000E+04 2.000E+04 0.000E+00
plas.st 2.698E-05 2.203E-05 0.000E+00 0.000E+00 0.000E+00 2.698E-05 2.698E-05 0.000E+00
layer  9
stress  2.000E+04 2.000E+04 6.667E+03 0.000E+00 0.000E+00 2.000E+04 2.000E+04 0.000E+00
plas.st 8.413E-05 6.869E-05 0.000E+00 0.000E+00 0.000E+00 8.413E-05 8.413E-05 0.000E+00

element    2  point  2
 axial force  -3.51429E-09     local xx moment   0.00000E+00     local yy moment   7.11188E+05  axial torque   0.00000E+00
axial strain   1.52780E-15  local xx curvature  -1.00287E-16  local yy curvature   1.17905E-04         twist   0.00000E+00
layer  1
stress  1.415E+04 1.415E+04-4.716E+03-1.415E+04 0.000E+00 0.000E+00-1.415E+04 0.000E+00

element    3  point  2
 axial force  -6.07542E-10     local xx moment   2.54659E-11     local yy moment   4.26713E+05  axial torque   0.00000E+00
axial strain   9.18675E-16  local xx curvature  -6.01740E-17  local yy curvature   7.07432E-05         twist   0.00000E+00
layer  1
stress  8.489E+03 8.489E+03-2.830E+03-8.489E+03 0.000E+00 0.000E+00-8.489E+03 0.000E+00

element    4  point  2
 axial force  -1.20963E-10     local xx moment   5.45697E-12     local yy moment   1.42238E+05  axial torque   0.00000E+00
axial strain   3.06337E-16  local xx curvature  -2.00569E-17  local yy curvature   2.35811E-05         twist   0.00000E+00
layer  1
stress  2.830E+03 2.830E+03-9.432E+02-2.830E+03 0.000E+00 0.000E+00-2.830E+03 0.000E+00


                              n o d a l   p o i n t   d a t a


                              i n c r e m e n t a l   d i s p l a c e m e n t s


     1       0.           0.           0.           0.           0.           0.      1.00692E-15
     2  8.70680E-15  2.19296E-14  5.77732E-03      0.     -4.14130E-04  8.57852E-16  1.99067E-16
     3  1.22674E-14  4.03362E-14  1.97035E-02      0.     -6.82097E-04  6.29863E-16  1.16903E-16
     4  1.43704E-14  5.41823E-14  3.89889E-02      0.     -8.42877E-04  4.93031E-16  5.66932E-17
     5  1.50697E-14  6.57484E-14  6.09541E-02      0.     -8.96470E-04  4.47449E-16  3.19823E-19
```

```
                          t o t a l   d i s p l a c e m e n t s


  1        0.            0.            0.            0.            0.            0.       3.23597E-15
  2   5.74673E-14 -1.98557E-14  5.49046E-02         0.      -4.16566E-03 -2.33302E-15  1.87086E-15
  3   9.58568E-14 -1.11610E-13  0.19835             0.      -7.11330E-03 -4.84021E-15  1.23143E-15
  4   1.18857E-13 -2.53509E-13  0.40074             0.      -8.88188E-03 -6.34456E-15  6.13956E-16
  5   1.26522E-13 -4.20480E-13  0.63262             0.      -9.47141E-03 -6.84598E-15  3.19823E-19



                  total equivalent nodal forces (distributed plus point loads)


  1        0.            0.            0.            0.            0.            0.            0.
  2        0.            0.            0.            0.            0.            0.            0.
  3        0.            0.            0.            0.            0.            0.            0.
  4        0.            0.            0.            0.            0.            0.            0.
  5        0.            0.         11379.           0.            0.            0.            0.



                  reaction forces at fixed boundary conditions, residual load correction elsewhere


  1   7.43239E-09 -1.55300E-08  -10742.             0.       1.12650E+06 -2.77676E-07 -1.81467E-07
  2  -7.43542E-09  1.55326E-08  -637.51             0.       -4539.6      -1.10532E-07  5.80667E-08
  3   6.06330E-12 -2.81797E-12  2.00089E-11         0.       1.16415E-10  1.13275E-11 -9.32267E-11
  4  -4.13062E-12 -5.04886E-13 -1.27329E-11         0.      -2.32831E-10 -6.06862E-12  2.92894E-11
  5   1.09897E-12  6.92750E-13  4.27463E-11         0.       6.18456E-11 -3.69505E-12 -1.20367E-11
```

Although relatively small, there is an error in satisfying equilibrium.

```
                          summary of externally applied loads

 0.00000E+00    0.00000E+00    0.11379E+05    0.00000E+00    0.00000E+00    0.00000E+00

 0.00000E+00

                          summary of reaction/residual forces

-0.31019E-24   -0.12940E-23   -0.11379E+05    0.00000E+00    0.11220E+07   -0.38821E-06

-0.12348E-06

                          e n d   o f   i n c r e m e n t   1

                          formatted post data at increment   1.   0   on tape 19
                           time =       1.54
```

```
                      s t a r t   o f   i n c r e m e n t   2

                      ●                                           ●
                      ●                                           ●
                      ●                                           ●
                      ●                                           ●

                      singularity ratio    1.1206E-02

                      ●                                           ●
                      ●                                           ●
                      ●                                           ●
                      ●                                           ●

                      e n d   o f   i n c r e m e n t   2

                      formatted post data at increment   2.   0   on tape 19
                       time =       1.79



                      s t a r t   o f   i n c r e m e n t   3



                      load increments associated with each degree of freedom
                      summed over the whole model

                      distributed loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                      point loads
 0.000E+00 0.000E+00 1.034E+03 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                      start of assembly
                       time =       1.80



                      start of matrix solution
                       time =       1.89



                      singularity ratio    7.5594E-03


                      end of matrix solution
                       time =       1.90


                     maximum residual force  at node    2 degree of freedom    1 is equal to    0.684E-07
                     maximum reaction force  at node    1 degree of freedom    3 is equal to    0.134E+05
                     convergence ratio                                                          0.509E-11
```

```
 MARC       output for increment    3.    ending of a beam until a plastic hinge forms

          tresca     mises    mean  p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
        intensity intensity  normal   minimum intermediate maximum     1         2         3         4         5         6
                             intensity


element    1  point   1
 axial force   2.74522E-08     local xx moment   8.63874E-08     local yy moment   1.25877E+06  axial torque   0.00000E+00
axial strain   7.30662E-15  local xx curvature  -1.35233E-15  local yy curvature   3.99178E-04         twist   0.00000E+00
layer  1
stress  2.000E+04 2.000E+04-6.667E+03-2.000E+04 0.000E+00 0.000E+00-2.000E+04 0.000E+00
plas.st 9.300E-04 7.594E-04 0.000E+00-9.300E-04 0.000E+00 0.000E+00-9.300E-04 0.000E+00
layer  2
stress  2.000E+04 2.000E+04-6.667E+03-2.000E+04 0.000E+00 0.000E+00-2.000E+04 0.000E+00
plas.st 8.085E-04 6.601E-04 0.000E+00-8.085E-04 0.000E+00 0.000E+00-8.085E-04 0.000E+00
layer  3
stress  2.000E+04 2.000E+04-6.667E+03-2.000E+04 0.000E+00 0.000E+00-2.000E+04 0.000E+00
plas.st 4.624E-04 3.775E-04 0.000E+00-4.624E-04 0.000E+00 0.000E+00-4.624E-04 0.000E+00
layer  4
stress  1.833E+04 1.833E+04-6.110E+03-1.833E+04 0.000E+00 0.000E+00-1.833E+04 0.000E+00
layer  5
stress  3.014E-07 3.014E-07 1.005E-07 0.000E+00 0.000E+00 3.014E-07 3.014E-07 0.000E+00
layer  6
stress  1.833E+04 1.833E+04 6.110E+03 0.000E+00 0.000E+00 1.833E+04 1.833E+04 0.000E+00
layer  7
stress  2.000E+04 2.000E+04 6.667E+03 0.000E+00 0.000E+00 2.000E+04 2.000E+04 0.000E+00
plas.st 4.624E-04 3.775E-04 0.000E+00 0.000E+00 0.000E+00 4.624E-04 4.624E-04 0.000E+00
layer  8
stress  2.000E+04 2.000E+04 6.667E+03 0.000E+00 0.000E+00 2.000E+04 2.000E+04 0.000E+00
plas.st 8.085E-04 6.601E-04 0.000E+00 0.000E+00 0.000E+00 8.085E-04 8.085E-04 0.000E+00
layer  9
stress  2.000E+04 2.000E+04 6.667E+03 0.000E+00 0.000E+00 2.000E+04 2.000E+04 0.000E+00
plas.st 9.300E-04 7.594E-04 0.000E+00 0.000E+00 0.000E+00 9.300E-04 9.300E-04 0.000E+00

element    2  point   2
 axial force   5.04770E-08     local xx moment   1.45519E-11     local yy moment   8.40495E+05  axial torque   0.00000E+00
axial strain   1.87804E-15  local xx curvature  -1.18519E-16  local yy curvature   1.39343E-04         twist   0.00000E+00
layer  1
stress  1.672E+04 1.672E+04-5.574E+03-1.672E+04 0.000E+00 0.000E+00-1.672E+04 0.000E+00

element    3  point   2
 axial force   8.70205E-09     local xx moment   1.09139E-11     local yy moment   5.04297E+05  axial torque   0.00000E+00
axial strain   1.09819E-15  local xx curvature  -7.11131E-17  local yy curvature   8.36056E-05         twist   0.00000E+00
layer  1
stress  1.003E+04 1.003E+04-3.344E+03-1.003E+04 0.000E+00 0.000E+00-1.003E+04 0.000E+00

element    4  point   2
 axial force   1.73986E-09     local xx moment   1.81899E-12     local yy moment   1.68099E+05  axial torque   0.00000E+00
axial strain   3.64533E-16  local xx curvature  -2.37038E-17  local yy curvature   2.78685E-05         twist   0.00000E+00
layer  1
stress  3.344E+03 3.344E+03-1.115E+03-3.344E+03 0.000E+00 0.000E+00-3.344E+03 0.000E+00
```

```
                         n o d a l   p o i n t   d a t a


                         i n c r e m e n t a l   d i s p l a c e m e n t s


 1       0.            0.            0.            0.            0.            0.        4.05498E-15
 2  3.96099E-14 -2.64309E-13  3.69997E-02         0.      -2.15919E-03 -1.59209E-14  5.66237E-17
 3  4.28228E-14 -6.65372E-13  9.45523E-02         0.      -2.42715E-03 -1.61489E-14  9.24469E-17
 4  4.48657E-14 -1.07099E-12  0.15746            0.      -2.58793E-03 -1.62855E-14  5.23696E-17
 5  4.55529E-14 -1.47889E-12  0.22306            0.      -2.64153E-03 -1.63311E-14 -1.12226E-18




                         t o t a l   d i s p l a c e m e n t s


 1       0.            0.            0.            0.            0.            0.        8.00169E-15
 2  1.22029E-13 -3.13489E-13  0.10394            0.      -7.05864E-03 -1.84850E-14  1.71628E-15
 3  1.66191E-13 -8.15119E-13  0.32686            0.      -1.05422E-02 -2.14479E-14  1.37037E-15
 4  1.93164E-13 -1.37601E-12  0.61944            0.      -1.26324E-02 -2.32258E-14  7.10584E-16
 5  2.02181E-13 -1.96653E-12  0.94686            0.      -1.33291E-02 -2.38184E-14 -4.62084E-18




                  total equivalent nodal forces (distributed plus point loads)


 1       0.            0.            0.            0.            0.            0.            0.
 2       0.            0.            0.            0.            0.            0.            0.
 3       0.            0.            0.            0.            0.            0.            0.
 4       0.            0.            0.            0.            0.            0.            0.
 5       0.            0.         13448.           0.            0.            0.            0.




                  reaction forces at fixed boundary conditions, residual load correction elsewhere


 1  6.84134E-08  3.75729E-13  -13448.            0.        1.32395E+06  7.44563E-07  2.85028E-07
 2 -6.84170E-08 -3.75729E-13  7.27596E-12        0.          20841.     -7.44559E-07  2.84979E-07
 3  9.39811E-12  9.39322E-13  1.12777E-10        0.       -1.16415E-10  3.09420E-11 -1.15101E-10
 4 -5.60855E-12 -3.99212E-13  1.45519E-11        0.       -1.68802E-09  4.47153E-12 -1.39845E-11
 5 -1.51582E-13 -5.40110E-13  7.91260E-11        0.       -8.00355E-11  7.23139E-12 -1.78596E-12

                         summary of externally applied loads
```

```
0.00000E+00    0.00000E+00    0.13448E+05    0.00000E+00    0.00000E+00    0.00000E+00

0.00000E+00

                          summary of reaction/residual forces

0.20680E-23    0.00000E+00   -0.13448E+05    0.00000E+00    0.13448E+07    0.46379E-10

0.56988E-06

                          e n d   o f   i n c r e m e n t   3

                          formatted post data at increment   3.   0   on tape 19
                           time =       2.05




                          s t a r t   o f   i n c r e m e n t   4





                          load increments associated with each degree of freedom
                          summed over the whole model

                          distributed loads
    0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                          point loads
    0.000E+00 0.000E+00 1.034E+03 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                          start of assembly
                           time =       2.06



                          start of matrix solution
                           time =       2.15



                          singularity ratio    5.7177E-03



                          end of matrix solution
                           time =       2.16
```

```
                  maximum residual force  at node    2 degree of freedom    3 is equal to    0.275E+04
                  maximum reaction force  at node    1 degree of freedom    3 is equal to    0.117E+05
                  convergence ratio                                                          0.235E+00

                   failure to converge to tolerance

                   increment will be recycled
```

Convergence not satisfied - try again!

```
                   load increments associated with each degree of freedom
                   summed over the whole model

                   distributed loads
  0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                   point loads
  0.000E+00 0.000E+00 1.034E+03 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                   start of assembly
                    time =       2.26



                   start of matrix solution
                    time =       2.36



                   singularity ratio    5.2343E-03


                   end of matrix solution
                    time =       2.36


                  maximum residual force  at node    2 degree of freedom    3 is equal to    0.144E+04
                  maximum reaction force  at node    1 degree of freedom    3 is equal to    0.130E+05
                  convergence ratio                                                          0.110E+00

                   failure to converge to tolerance

                   increment will be recycled
```

```
                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                    point loads
 0.000E+00 0.000E+00 1.034E+03 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                    start of assembly
                     time =       2.47



                    start of matrix solution
                     time =       2.56



                    singularity ratio    3.2148E-03


                    end of matrix solution
                     time =       2.57


                   maximum residual force  at node    2 degree of freedom    3 is equal to    0.139E+04
                   maximum reaction force  at node    1 degree of freedom    3 is equal to    0.131E+05
                   convergence ratio                                                          0.106E+00

                    failure to converge to tolerance

                    increment will be recycled





                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                    point loads
 0.000E+00 0.000E+00 1.034E+03 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                    start of assembly
                     time =       2.67
```

```
                    start of matrix solution
                     time =       2.77



                    singularity ratio    1.4614E-03


                    end of matrix solution
                     time =       2.77


                   maximum residual force  at node    2 degree of freedom    3 is equal to    0.138E+0
                   maximum reaction force  at node    1 degree of freedom    3 is equal to    0.131E+0
                    convergence ratio                                                         0.106E+0

                    failure to converge to tolerance

                    increment will be recycled





                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                    point loads
 0.000E+00 0.000E+00 1.034E+03 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                    start of assembly
                     time =       2.88



                    start of matrix solution
                     time =       2.98



                    singularity ratio    6.5630E-04


                    end of matrix solution
```

```
           time =       2.98


      maximum residual force  at node    2 degree of freedom    3 is equal to    0.138E+04
      maximum reaction force  at node    1 degree of freedom    3 is equal to    0.131E+05
       convergence ratio                                                         0.105E+00

       failure to converge to tolerance

       increment will be recycled




                load increments associated with each degree of freedom
                summed over the whole model

                distributed loads
0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                point loads
0.000E+00 0.000E+00 1.034E+03 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                start of assembly
                 time =       3.09



                start of matrix solution
                 time =       3.18



                singularity ratio    2.9310E-04


                end of matrix solution
                 time =       3.19


      maximum residual force  at node    2 degree of freedom    3 is equal to    0.138E+04
      maximum reaction force  at node    1 degree of freedom    3 is equal to    0.131E+05
       convergence ratio                                                         0.105E+00

       failure to converge to tolerance

       increment will be recycled
```

```
                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                    point loads
 0.000E+00 0.000E+00 1.034E+03 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                    start of assembly
                     time =        3.30



                    start of matrix solution
                     time =        3.39
```

Singularity ratio keeps getting smaller.

```
                    singularity ratio    1.3057E-04


                    end of matrix solution
                     time =        3.40


                  maximum residual force  at node    2 degree of freedom    3 is equal to    0.138E+04
                  maximum reaction force  at node    1 degree of freedom    3 is equal to    0.131E+05
                  convergence ratio                                                           0.105E+00

                    failure to converge to tolerance
```

Convergence ratio is NOT improving.

```
                    *** error - too many recycles - job ends at this increment
```

MARC senses that convergence will not be obtained under CONTROL constraints.

******WARNING******
Results are for a solution which has not converged.

```
 MARC       output for increment    4.    ending of a beam until a plastic hinge forms



          tresca     mises    mean   p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
        intensity intensity  normal   minimum intermediate maximum     1         2         3         4         5         6
                             intensity


element    1  point  1
 axial force    3.55716E-06     local xx moment    1.76385E-05     local yy moment    1.28017E+06  axial torque    0.00000E+00
axial strain    1.57531E-12  local xx curvature   -7.30755E-13  local yy curvature    7.45033E-02          twist    0.00000E+00
layer  1
stress  2.000E+04 2.000E+04-6.667E+03-2.000E+04 0.000E+00 0.000E+00-2.000E+04 0.000E+00
plas.st 2.973E-01 2.428E-01 0.000E+00-2.973E-01 0.000E+00 0.000E+00-2.973E-01 0.000E+00
layer  2
stress  2.000E+04 2.000E+04-6.667E+03-2.000E+04 0.000E+00 0.000E+00-2.000E+04 0.000E+00
plas.st 2.747E-01 2.243E-01 0.000E+00-2.747E-01 0.000E+00 0.000E+00-2.747E-01 0.000E+00
layer  3
stress  2.000E+04 2.000E+04-6.667E+03-2.000E+04 0.000E+00 0.000E+00-2.000E+04 0.000E+00
plas.st 2.101E-01 1.715E-01 0.000E+00-2.101E-01 0.000E+00 0.000E+00-2.101E-01 0.000E+00
layer  4
stress  2.000E+04 2.000E+04-6.667E+03-2.000E+04 0.000E+00 0.000E+00-2.000E+04 0.000E+00
plas.st 1.134E-01 9.257E-02 0.000E+00-1.134E-01 0.000E+00 0.000E+00-1.134E-01 0.000E+00
layer  5
stress  5.195E-06 5.195E-06 1.732E-06 0.000E+00 0.000E+00 5.195E-06 5.195E-06 0.000E+00 ◄──────
layer  6
stress  2.000E+04 2.000E+04 6.667E+03 0.000E+00 0.000E+00 2.000E+04 2.000E+04 0.000E+00
plas.st 1.134E-01 9.257E-02 0.000E+00 0.000E+00 0.000E+00 1.134E-01 1.134E-01 0.000E+00
layer  7
stress  2.000E+04 2.000E+04 6.667E+03 0.000E+00 0.000E+00 2.000E+04 2.000E+04 0.000E+00
plas.st 2.101E-01 1.715E-01 0.000E+00 0.000E+00 0.000E+00 2.101E-01 2.101E-01 0.000E+00
layer  8
stress  2.000E+04 2.000E+04 6.667E+03 0.000E+00 0.000E+00 2.000E+04 2.000E+04 0.000E+00
plas.st 2.747E-01 2.243E-01 0.000E+00 0.000E+00 0.000E+00 2.747E-01 2.747E-01 0.000E+00
layer  9
stress  2.000E+04 2.000E+04 6.667E+03 0.000E+00 0.000E+00 2.000E+04 2.000E+04 0.000E+00
plas.st 2.973E-01 2.428E-01 0.000E+00 0.000E+00 0.000E+00 2.973E-01 2.973E-01 0.000E+00


element    2  point  2
 axial force    6.63751E-06     local xx moment   -1.08994E-08     local yy moment    9.07280E+05  axial torque    0.00000E+00
axial strain    1.07583E-14  local xx curvature   -1.29740E-16  local yy curvature    1.50415E-04          twist    0.00000E+00
layer  1
stress  1.805E+04 1.805E+04-6.017E+03-1.805E+04 0.000E+00 0.000E+00-1.805E+04 0.000E+00


element    3  point  2
 axial force    1.17295E-06     local xx moment    0.00000E+00     local yy moment    5.43089E+05  axial torque    0.00000E+00
```

Plasticity throughout beam cross section except at neutral axis.

```
axial strain   2.72592E-15  local xx curvature  -7.65840E-17  local yy curvature   9.00368E-05         twist   0.00000E+00
layer  1
stress  1.080E+04 1.080E+04-3.601E+03-1.080E+04 0.000E+00 0.000E+00-1.080E+04 0.000E+00

element    4  point  2
 axial force   2.34585E-07     local xx moment  -5.45697E-12     local yy moment   1.81030E+05  axial torque   0.00000E+00
axial strain   7.01215E-16  local xx curvature  -2.55285E-17  local yy curvature   3.00123E-05         twist   0.00000E+00
layer  1
stress  3.601E+03 3.601E+03-1.200E+03-3.601E+03 0.000E+00 0.000E+00-3.601E+03 0.000E+00


                              n o d a l   p o i n t   d a t a



                              i n c r e m e n t a l   d i s p l a c e m e n t s


   1       0.           0.            0.           0.          0.           0.      1.90848E-12
   2  1.81393E-11 -1.62934E-10   16.562          0.      -0.92637     -9.10630E-12 -6.30561E-14
   3  1.79813E-11 -3.90596E-10   39.725          0.      -0.92665     -9.10659E-12 -1.03885E-14
   4  1.79576E-11 -6.18262E-10   62.893          0.      -0.92681     -9.10672E-12 -1.79722E-15
   5  1.79532E-11 -8.45931E-10   86.064          0.      -0.92686     -9.10677E-12 -6.17642E-16
```

Huge displacements (...obviously incorrect)

```
                              t o t a l   d i s p l a c e m e n t s


   1       0.           0.            0.           0.          0.           0.      1.91648E-12
   2  1.82613E-11 -1.63247E-10   16.666          0.      -0.93343     -9.12479E-12 -6.13398E-14
   3  1.81475E-11 -3.91411E-10   40.052          0.      -0.93719     -9.12803E-12 -9.01816E-15
   4  1.81508E-11 -6.19638E-10   63.513          0.      -0.93944     -9.12995E-12 -1.08663E-15
   5  1.81554E-11 -8.47897E-10   87.011          0.      -0.94019     -9.13059E-12 -6.22263E-16



                     total equivalent nodal forces (distributed plus point loads)


   1       0.          0.          0.          0.          0.          0.          0.
   2       0.          0.          0.          0.          0.          0.          0.
   3       0.          0.          0.          0.          0.          0.          0.
   4       0.          0.          0.          0.          0.          0.          0.
   5       0.          0.        14482.        0.          0.          0.          0.
```

```
               reaction forces at fixed boundary conditions, residual load correction elsewhere


  1  1.57391E-07 -6.24096E-07  -13101.          0.      1.37347E+06 -1.60108E-05  5.09266E-07
  2 -2.55904E-07  6.24097E-07  -1380.9          0.       40250.      4.08415E-07  1.64714E-06
  3  9.85083E-08 -1.87864E-12 -1.09535E-06      0.     -7.74418E-06  7.83421E-12 -7.68962E-07
  4  6.78331E-12  1.87864E-12 -3.45790E-08      0.      1.42655E-06 -1.86707E-12 -5.30683E-11
  5 -2.23584E-12 -7.51458E-13  6.67596E-08      0.      4.11652E-07  2.92570E-12  1.43245E-11

                             summary of externally applied loads

0.00000E+00    0.00000E+00    0.14482E+05    0.00000E+00    0.00000E+00    0.00000E+00

0.00000E+00

                             summary of reaction/residual forces

0.00000E+00    0.10960E-22   -0.14482E+05    0.00000E+00    0.14137E+07   -0.15602E-04

0.13874E-05

                             e n d   o f   i n c r e m e n t   4

                             formatted post data at increment    4.   0    on tape 19
                              time =       3.55



                             analysis failed to converge during this increment
                             specific message given before element printout




                                      marc exit number 3002
```

Results

Let's track the von Mises stress intensity and plastic strain from layer 1 (top) to layer 5 (neutral axis), at element 1 integration point 1 (closest to the clamped left end) from increment 0 through increment 4:

Table 1-1: von Mises Stress (psi)

	Increment				
Layer	**0**	**1**	**2**	**3**	**4**
1	20,000	20,000	20,000	20,000	20,000
2	18,480	20,000	20,000	20,000	20,000
3	14,140	15,930	20,000	20,000	20,000
4	7,654	8,620	10,950	18,330	20,000
5	0.000	0.000	0.000	0.000	5.2E-6

Table 1-2: Plastic Strain (in/in)

	Increment				
Layer	**0**	**1**	**2**	**3**	**4**
1	–	8.413E-5	2.868E-4	9.300E-4	2.973E-1
2	–	2.698E-5	2.143E-4	8.085E-4	2.747E-1
3	–	–	7.562E-6	4.624E-4	2.101E-1
4	–	–	–	–	1.134E-1
5	–	–	–	–	–

We can thus visualize the onset of plasticity across the beam cross section, from increment to increment. By increment 3, most of the beam cross section has yielded, and by increment 4, all eight layers have yielded (except for the neutral axis "layer 5"). We also now understand that MARC's nonconvergence message after increment 4 is due to the fact that after six recycles, the convergence ratio is still 0.105 – which is slightly higher than the 0.100 tolerance allowed. In increment 4, also notice that layer 5 now shows a von Mises stress of 5.2E-6 psi, rather than the zero value it should be at the neutral axis. Therefore, recognize the fact that increment 4 results did not converge (although coming close), and static equilibrium is not satisfied.

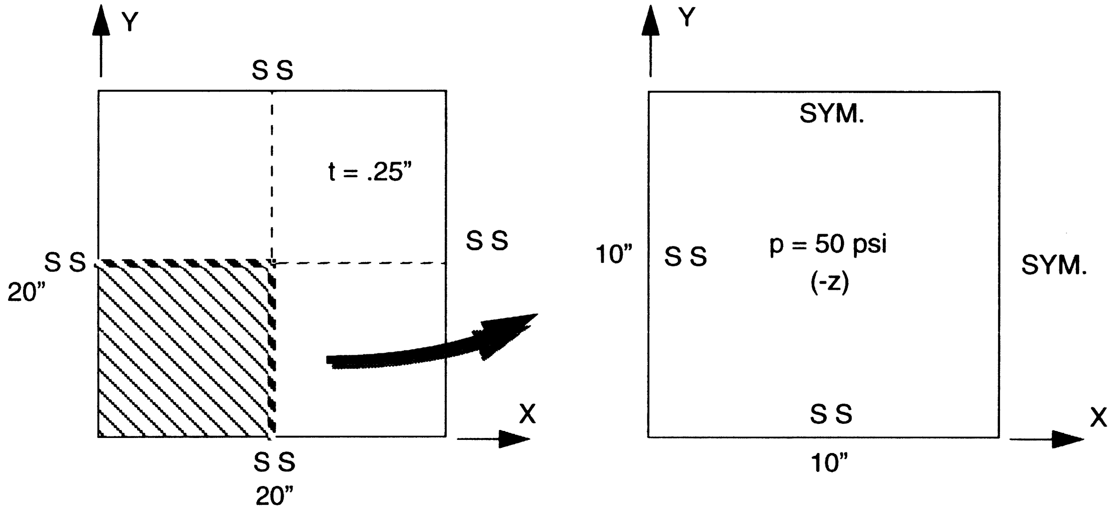
In summary, the beam has yielded at the left end, thereby losing its capacity to resist any bending altogether. The MARC nonlinear analysis output, although appearing unsuccessful at first glance, is actually correct in predicting the plastic behavior of the beam until its eventual collapse. The lesson here, as in all nonlinear analysis, is to: think; proceed carefully; and examine the results very closely.

Example 6

Large Displacement and Plasticity Analysis of a Simply-Supported Square Plate

The purpose of this example is to illustrate a typical large displacement elastic-plastic analysis. The model is a square plate under uniform pressure. The LARGE DISP option is introduced for the first time.

Sketch



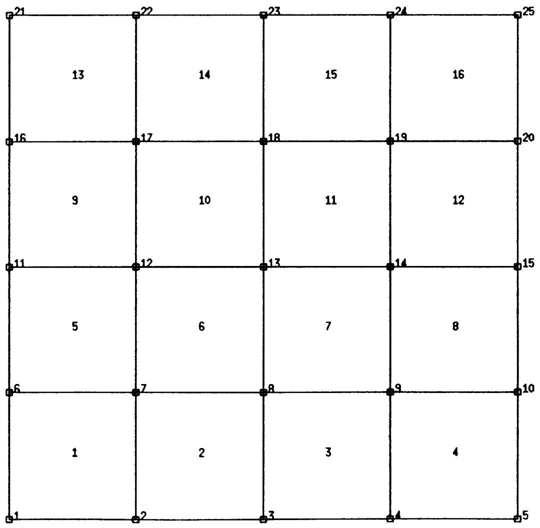


Figure 6.1 **Square Plate**

Model

As in Example 1, only one quarter of the plate needs to be modeled. This quarter model has 25 nodes and 16 thick shell elements (MARC Element 75), and is 10.0 by 10.0 inches square. Each of the 16 elements is 2.5 by 2.5 inches square. The plate thickness is 0.25 inch. The top and right edges have symmetry boundary conditions. The left and bottom edges are simply-supported.

Young's modulus is 10.0E6 psi and Poisson's ratio is 0.3. The tensile yield stress is 20,000 psi.

MARC Element 75 is a 4-noded bilinear thick shell element. The element description is summarized in Chapter 1, and is given in detail in Volume B. The geometry in this problem results in a length-to-thickness ratio of 20:(.25) = 80:1, which indicates that only thin shell theory is required. The use of Element 75 is acceptable because this element can also represent thin shell theory.

Properties

Like the cantilevered beam in Example 5, this plate is assumed to obey the von Mises yield criterion, and exhibit elastic-perfectly plastic behavior.

Loads

The loading on the plate is a uniform pressure of 50.0 psi, acting in the -Z direction. For reasons to be discussed later, this pressure load will be applied over ten load steps at 5.0 psi each.

Boundary Conditions

The square plate is simply-supported at all four edges. Since there are two planes of symmetry, only a quarter model is needed. For the quarter model shown in the sketch, the LEFT and BOTTOM edges are the two simply-supported edges: there are no displacements permitted in the X, Y, and Z-directions; the BOTTOM edge can rotate about the X-axis; and the LEFT edge can rotate about the Y-axis. The TOP and RIGHT edges represent the "symmetry planes." Along the TOP edge, there can be no displacements across the plane of symmetry – that is, Y-displacements must be zero. Also, there should be no rotations about the X-axis. Along the RIGHT edge, X-displacements and Y-rotations are both zero. The actual implementation of these boundary conditions will be discussed later in the input description.

LARGE DISP Option

The LARGE DISP parameter option flags the MARC program control for large displacement (and buckling) problems. The total Lagrangian approach is then used, which is based on the *initial* element geometry. It is useful for problems in plasticity and creep, with moderately large rotations and small strains (such as this plate bending problem). See Volume A.

Input

A complete input listing (with comments) is included.

PARAMETER Section

The "TITLE" line is self-explanatory. The "SIZING" line sets 100,000 words as the workspace. The "ELEMENTS" line tells MARC that Element 75 will be used.

The "LARGE DISP" line flags the program control for large displacement (or buckling) analysis. This instructs MARC to calculate the geometric stiffness matrix and the initial stress stiffness matrix. It also automatically switches off the scaling option (if it exists).

The "END" line terminates the PARAMETER section.

MODEL DEFINITION Section

The MODEL DEFINITION options in this example consists of:

a. FE mesh topology – including CONNECTIVITY, COORDINATES, and DEFINE blocks
b. Geometric properties
c. Loads
d. Material properties
e. Boundary conditions
f. Output controls.

FE Mesh Topology

The mesh is square and measures 10 by 10 inches, with 25 nodes and 16 square elements (MARC Element 75). Each element is identical in size and shape, 2.5 by 2.5 inches square. The FE mesh is laid out in the global X-Y plane, with the TOP and RIGHT edges being the "symmetry planes." Therefore, the center of the full plate is node 25, where we would expect the maximum deflection when the plate is loaded by uniform pressure.

The CONNECTIVITY and COORDINATES blocks are self-explanatory. On the first line after the CONNECTIVITY line, the "16" in the first field refers to the number of elements in the mesh. On the first line after "COORDINATES" line, the "3" in the first field means the maximum number of coordinate directions to be read in per node, and the "25" in the second field represents the number of nodes in the mesh.

Next, five DEFINE blocks follow: all 16 elements are placed in the element set named ALLE; and the five nodes corresponding to each edge are placed in the node sets named BOTTOM, LEFT, TOP, and RIGHT.

Geometric Properties

The GEOMETRY block allows you to enter the element geometric properties. For this plate example, the only geometric property which needs to be entered is the constant plate thickness of 0.25 inch. The blank line following the "GEOMETRY" line means we do not have to count the number of sets of geometric properties to be input. The next line (".25,") gives a thickness of 0.25 inch in the first field. (This element actually allows you to input a bilinear variation of the thickness; the fact that we left the second, third, and fourth fields blank on this line implies that the element thickness is assumed to be uniform.) The last line in this block ("ALLE") assigns the 0.25 thickness to all the elements in element set ALLE – in other words, all 16 elements in the model.

Loads

The DIST LOADS block allows pressure loads to be specified. The blank line means we do not have to count how many sets of distributed loads are entered. The "2,0.0," line indicates that the traction type is 2, which for MARC Element 75 is a uniform pressure load which is positive in the Z direction (acting toward plate), with a value of zero. Why zero? This is merely a method for us to delay application of the pressure load until increment 1, so that the full Newton-Raphson iterative procedure occurs from the start of loading. Increment zero should be constrained to purely linear (material and geometric) behavior. The "ALLE" line says this pressure load is applied to the element set ALLE (all 16 elements).

Material Properties

As in Example 5, all the material properties in this example are prescribed using the ISOTROPIC block. This option lets you define material properties, a yield criterion, and a strain (work) hardening law for an isotropic material. Again, the blank line following the "ISOTROPIC" line means you do not need to specify the number of sets of isotropic material data to follow.

The next line ("10.E6,.3,,,20000.,") gives the Young's modulus, Poisson's ratio, and the equivalent (von Mises) tensile yield stress. (The third field is for mass density while the fourth field is for coefficient of thermal expansion; these properties are not needed for this problem and are therefore left blank.)

The last line ("ALLE") in this block assigns the above material properties to all the elements in the model.

Boundary Conditions

The FIXED DISP block defines the fixed displacement that each named DOF must take during the first and subsequent increments. The blank line means we do not need to count how many sets of boundary conditions are coming next. What follows are four sets of three lines each, all ending with a node set name.

Line 81 ("0.,0.,0.,0.,0.,") gives zero-valued prescribed displacements for the five DOFs to come on the next line. Line 82 ("1 2 3 5 6") names the five DOFs: the X-, Y-, and Z-displacements, and the Y- and Z-rotations. Then, line 83 ("BOTTOM") assigns these prescribed displacements to the node set BOTTOM (i.e., nodes 1 to 5). In other words, the edge named BOTTOM is a simply-supported edge, which permits X-rotation only, with all the remaining five DOFs prescribed to be zero during the analysis.

In a similar fashion, simply-supported boundary conditions are also applied to node set LEFT (permitting Y-rotation only). Also, symmetry boundary conditions are prescribed for node sets TOP and RIGHT – with no displacements permitted across the plane of symmetry and zero slope.

Notice that the sixth DOF (out-of-plane rotation about the Z-axis) is usually prescribed to be zero in a plate/shell problem of this type. This special boundary condition of assuming the Z-rotation to be zero is standard practice in finite element analysis – the rationale being that the plate/shell is considered infinitely stiff in the Z-rotation direction. In practice, this assumption is valid in nearly all circumstances; it certainly falls within the accuracy range of most finite element assumptions regarding material properties, boundary conditions, and loads.

Output Controls

The output controls in this example consists of three blocks: PRINT ELEM, PRINT NODE, and POST. They end the MODEL DEFINITION portion of the input.

The PRINT ELEM option allows you to print selective element quantities for certain elements. The blank line, again, means you do not need to enter the number of sets to follow. The "STRESS STRAIN PLASTIC" line tells MARC you would like to print out total stress and total strain as well as the plastic strain for the elements to be named on the next line. Other element quantities you could have selected include: CREEP (creep, swelling, and viscoelastic strain); THERMAL (thermal strain); ENERGY (strain energy); CRACK (cracking strain); CAUCHY (Cauchy or true stress); STATE (state variables); PREFER (stresses in preferred system); and ALL (all of the above). The "1 TO 16" on the following line indicates elements 1 to 16 are to be printed. The "1 TO 4" line means we want results for integration points 1 to 4 to be printed. And the "1 11" line says we want layers 1 and 11 printed.

The PRINT NODE option lets you print selective nodal quantities for certain nodes. The blank line means you do not need to enter the number of sets to follow. The "ALL" line tells MARC to print all relevant nodal quantities. (See Volume C or Example 2A input description for a list of the optional nodal quantities which can be printed.) And the "25" line indicates that the quantities for node 25 are to be printed. Note that the use of the ALL option turned on the nodal stress option. These values are obtained by extrapolating integration point values, and averaging between elements.

The POST option creates a post-processor file for later post-processing by Mentat II. The "10,,,1" line means ten element variables are to be written in the file, and we want a formatted post file. The next ten lines indicate the five post codes which are to be stored in the file for both layers 1 and 11: 7, 11, 12, 13, and 17. These post codes mean equivalent plastic strain, first/second/third components of stress, and equivalent Mises stress, respectively.

The "END OPTION" line terminates the MODEL DEFINITION section.

LOAD INCREMENTATION Section

The LOAD INCREMENTATION options consist of two options in this example: DIST LOADS and AUTO LOAD.

Recall that earlier we had already encountered a DIST LOADS block (which occurred after the GEOMETRY block). The purpose of that block was to delay application of the pressure load until increment 1, so that there would be iteration from the start of loading. The reason that DIST LOADS in increment 0 is 0.0 is that MARC assumes that increment 0 is linear-elastic, and therefore does not perform either convergence checking or iteration. In order to have MARC perform an equilibrium check from the very start of the analysis, the loading should begin in increment 1 instead of increment 0.

The DIST LOADS block allows pressure loads to be specified. The blank line says that you do not need to count the number of sets of distributed loads to be entered. The "2,5.0," line means a uniform pressure load of 5.0 psi is to be applied in the -Z-direction, that is, downward towards the plate (modeled in the global X-Y plane). The "ALLE" line denotes that this pressure is applied to element set ALLE, or all 16 elements.

The AUTO LOAD block generates a specified number of increments, each with the same load increment. In this example, the "10," line means applying the 5 psi pressure load over each of ten load steps. In other words, 5 psi pressure will be applied from increments 1 to 10, ending with a total pressure of 50.0 psi on the plate at the end of the analysis.

The "CONTINUE" line ends the LOAD INCREMENTATION section as well as the input file.

Output

The selective printout included for this example consists of:

- the input echo
- program sizing and options summary table
- increment 0 results (a null step – where the distributed pressure load was zero)
- results for increments 1 (pressure = 5.0 psi) and 10 (pressure = 50.0 psi)

In an analysis where the LARGE DISP option is used (without UPDATE or FINITE), the strains reported are the Green-Lagrange strains, not "Engineering strains." In addition, the stresses are the second Piola-Kirchhoff stresses, not the "Engineering stress." Care must be taken to correctly interpret the output. If any work hardening data was included, they must be given using these measures. See Volume A for more details.

```
                              i n p u t   d a t a


    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
TITLE,  NONLINEAR ANALYSIS OF A SIMPLY SUPPORTED SQUARE PLATE
SIZING,100000
ELEMENTS     75
COMMENT, LARGE DISP PARAMETER CARD INCLUDES EFFECTS OF GEOMETRICALLY
COMMENT,  LARGE DISPLACEMENTS.
LARGE DISP
END
CONNECTIVITY
   16    0    0
    1   75    1    2    7    6
    2   75    2    3    8    7
    3   75    3    4    9    8
    4   75    4    5   10    9
    5   75    6    7   12   11
    6   75    7    8   13   12
    7   75    8    9   14   13
    8   75    9   10   15   14
    9   75   11   12   17   16
   10   75   12   13   18   17
   11   75   13   14   19   18
   12   75   14   15   20   19
   13   75   16   17   22   21
   14   75   17   18   23   22
   15   75   18   19   24   23
   16   75   19   20   25   24
    COORDINATES
    3   25    0    0
    1   0.00000   0.00000   0.00000
    2   2.50000   0.00000   0.00000
    3   5.00000   0.00000   0.00000
    4   7.50000   0.00000   0.00000
    5  10.00000   0.00000   0.00000
    6   0.00000   2.50000   0.00000
    7   2.50000   2.50000   0.00000
    8   5.00000   2.50000   0.00000
    9   7.50000   2.50000   0.00000
   10  10.00000   2.50000   0.00000
   11   0.00000   5.00000   0.00000
   12   2.50000   5.00000   0.00000
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

Flags large displacement.

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
   13   5.00000   5.00000   0.00000
   14   7.50000   5.00000   0.00000
   15  10.00000   5.00000   0.00000
   16   0.00000   7.50000   0.00000
   17   2.50000   7.50000   0.00000
   18   5.00000   7.50000   0.00000
   19   7.50000   7.50000   0.00000
   20  10.00000   7.50000   0.00000
   21   0.00000  10.00000   0.00000
   22   2.50000  10.00000   0.00000
   23   5.00000  10.00000   0.00000
   24   7.50000  10.00000   0.00000
   25  10.00000  10.00000   0.00000
DEFINE    ELEMENT   SET       ALLE
    1 TO    16
DEFINE    NODE      SET       BOTTOM
    1 TO     5
DEFINE    NODE      SET       LEFT
    1    6   11   16   21
DEFINE    NODE      SET       TOP
   21 TO    25
DEFINE    NODE      SET       RIGHT
    5   10   15   20   25
GEOMETRY

.25,
ALLE
COMMENT,  DELAY APPLICATION OF LOAD UNTIL INCREMENT ONE SO THAT
COMMENT,  LOAD CORRECTION OCCURS FROM START OF LOADING
DIST LOADS

2,0.0,
ALLE
COMMENT,  MATERIAL NONLINEARITY IS SPECIFIED BY YIELD STRESS.
ISOTROPIC

1,VON MISES,ISOTR HARD
10.E6,.3,,,20000.,
ALLE
FIXED DISP

0.,0.,0.,0.,0.,
1 2 3 5 6
BOTTOM
0.,0.,0.,0.,0.,
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                --------------------------------------------------------------------------------
card    85      1 2 3 4 6
                LEFT
                0.,0.,0.,
                2 4 6
                TOP
card    90      0.,0.,0.,
                1 5 6
                RIGHT
                PRINT ELEM

card    95      STRESS STRAIN PLASTIC
                1 TO 16
                1 TO 4
                1 11
                PRINT NODE
card   100
                ALL
                25
                POST
                10,,,1
card   105      11,1
                12,1
                13,1
                17,1
                7,1,
card   110      11,11,
                12,11,
                13,11,
                17,11,
                7,11,
card   115      END OPTION
                COMMENT,  APPLY 10 LOAD STEPS OF 5 PSI EACH
                DIST LOADS

                2,5.0,
card   120      ALLE
                AUTO LOAD
                10,
                CONTINUE
                --------------------------------------------------------------------------------
                    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                --------------------------------------------------------------------------------
                --------------------------------------------------------------------------------
```

```
               ***********************************************
               ***********************************************


                program sizing and options requested as follows




                element type requested*************************   75
                number of elements in mesh*********************   16
                number of nodes in mesh************************   25
                max number of elements in any dist load list***   16
                maximum number of boundary conditions**********   80
                large displacement analysis flagged************
                load correction flagged or set*****************
                number of lists of distributed loads***********    3
                stresses stored at all integration points******
                tape no.for input of coordinates + connectivity    5
                no.of different materials    1 max.no of slopes    5
                maximum elements variables per point on post tp   33
                number of points on shell section *************   11
                option for terminal debug**********************
                new style input format will be used************
                maximum number of set names is*****************   10
                number of processors used *********************    1
                vector length used ****************************    1


                end of parameters and sizing
               ***********************************************
               ***********************************************
```

By default, thi
element has 1
layers.

```
key to stress, strain and displacement output


          element type   75


4-node shell element

generalized strains in local coordinates
  1=local x membrane
  2=local y membrane
  3=local xy shear
  4=local yz transverse shear
  5=local zx transverse shear
stresses correspond to strains in each fiber
```

This is a thick shell element which includes transverse shear behavior.

```
        transverse shear strain distribution is constant
         through the thickness

        displacements in global directions
          1=u global x direction
          2=v global y direction
          3=w global z direction
          4=theta x rotation about global x axis
          5=theta y rotation about global y axis
          6=theta z rotation about global z axis


        workspace needed for input and stiffness assembly     40099


        internal core allocation parameters
        degrees of freedom per node (ndeg)  6
        coords per node (ncrd)  3
        strains per integration point (ngens) 11
        max. nodes per element (nnodmx)  4
        max.stress components per int. point (nstrmx)  55
        max. invariants per int. points (neqst) 11
```

ELSTO turned on

```
        flag for element storage (ielsto)  1
        elems out of core, words per elem (nelsto) 9496
        elems per buffer (mxels)    1

out-of-core space needed for element storage =   196608  based on record size of  4096
    vectors in core, total space required      1968


        words per track on disk set to 4096


        internal element variables


        internal element number  1  library code type 75
        number of nodes=  4
        stresses stored per integration point = 55
        direct continuum components stored =  2
        shear continuum components stored =  3
        shell/beam flag = 1
        curvilinear  coord. flag = 0
        int.points for elem. stiffness  4
        number of local inertia directions  6
        int.point for print if all points not flagged  5
```

```
                int. points for dist. surface loads (pressure)  4
                library code type = 75
                no local rotation flag = 1
                generalized displ. flag = 0
                large disp. row counts     6    6   11    0    0    0    0    0    0    0    0



                residual load correction is invoked




connectivity
----------

meshr1,iprnt
    5      0
elem no., type,        nodes
    1      75          1    2    7    6
    2      75          2    3    8    7
    3      75          3    4    9    8
    4      75          4    5   10    9
    5      75          6    7   12   11
    6      75          7    8   13   12
    7      75          8    9   14   13
    8      75          9   10   15   14
    9      75         11   12   17   16
   10      75         12   13   18   17
   11      75         13   14   19   18
   12      75         14   15   20   19
   13      75         16   17   22   21
   14      75         17   18   23   22
   15      75         18   19   24   23
   16      75         19   20   25   24

coordinates
----------

ncrd1 ,meshr1,iprnt
    3      5      0
 node      coordinates
    1      0.           0.          0.
    2  2.5000           0.          0.
    3  5.0000           0.          0.
    4  7.5000           0.          0.
    5  10.000           0.          0.
    6      0.       2.5000          0.
    7  2.5000       2.5000          0.
```

```
    8  5.0000      2.5000          0.
    9  7.5000      2.5000          0.
   10  10.000      2.5000          0.
   11      0.      5.0000          0.
   12  2.5000      5.0000          0.
   13  5.0000      5.0000          0.
   14  7.5000      5.0000          0.
   15  10.000      5.0000          0.
   16      0.      7.5000          0.
   17  2.5000      7.5000          0.
   18  5.0000      7.5000          0.
   19  7.5000      7.5000          0.
   20  10.000      7.5000          0.
   21      0.      10.000          0.
   22  2.5000      10.000          0.
   23  5.0000      10.000          0.
   24  7.5000      10.000          0.
   25  10.000      10.000          0.

define    element   set       alle
----------

from element     1 to element    16 by     1

define    node      set       bottom
----------

from node     1 to node     5 by     1

define    node      set       left
----------

a list of nodes given below
     1      6     11      16     21

define    node      set       top
----------

from node    21 to node    25 by     1

define    node      set       right
----------

a list of nodes given below
     5     10     15      20     25
```

```
geometry
----------

  egeom1     egeom2     egeom3     egeom4     egeom5     egeom6
  0.250E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
name of element set is alle
comment,  delay application of load until increment one so that
comment,  load correction occurs from start of loading

dist loads
----------

read from unit     5
type index distributed load
    2    0  0.0000000E+00  0.0000000E+00  0.0000000E+00
name of element set is alle
comment,  material nonlinearity is specified by yield stress.

isotropic
----------



isotropic material  material id =     1
 von mises  yield criteria
 isotropic  hardening rule
   e          nu         rho       alpha       yield      yield2
  0.100E+08  0.300E+00  0.000E+00  0.000E+00  0.200E+05  0.200E+05
name of element set is alle

fixed disp
----------

fixed displacement =  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
a list of degrees of freedom given below
     1      2      3      5      6
name of node set is bottom
fixed displacement =  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
a list of degrees of freedom given below
     1      2      3      4      6
name of node set is left
fixed displacement =  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
a list of degrees of freedom given below
     2      4      6
name of node set is top
fixed displacement =  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
a list of degrees of freedom given below
     1      5      6
name of node set is right
```

```
fixed boundary condition summary.
total fixed degrees of freedom read so far =   80

 b.c.    node   degree of  magnitude               b.c.    node   degree of  magnitude
number           freedom                          number           freedom



    1       1          1   0.000E+00                  2       1          2   0.000E+00
    3       1          3   0.000E+00                  4       1          5   0.000E+00
    5       1          6   0.000E+00                  6       2          1   0.000E+00
    7       2          2   0.000E+00                  8       2          3   0.000E+00
    9       2          5   0.000E+00                 10       2          6   0.000E+00
   11       3          1   0.000E+00                 12       3          2   0.000E+00
   13       3          3   0.000E+00                 14       3          5   0.000E+00
   15       3          6   0.000E+00                 16       4          1   0.000E+00
   17       4          2   0.000E+00                 18       4          3   0.000E+00
   19       4          5   0.000E+00                 20       4          6   0.000E+00
   21       5          1   0.000E+00                 22       5          2   0.000E+00
   23       5          3   0.000E+00                 24       5          5   0.000E+00
   25       5          6   0.000E+00                 26       1          1   0.000E+00
   27       1          2   0.000E+00                 28       1          3   0.000E+00
   29       1          4   0.000E+00                 30       1          6   0.000E+00
   31       6          1   0.000E+00                 32       6          2   0.000E+00
   33       6          3   0.000E+00                 34       6          4   0.000E+00
   35       6          6   0.000E+00                 36      11          1   0.000E+00
   37      11          2   0.000E+00                 38      11          3   0.000E+00
   39      11          4   0.000E+00                 40      11          6   0.000E+00
   41      16          1   0.000E+00                 42      16          2   0.000E+00
   43      16          3   0.000E+00                 44      16          4   0.000E+00
   45      16          6   0.000E+00                 46      21          1   0.000E+00
   47      21          2   0.000E+00                 48      21          3   0.000E+00
   49      21          4   0.000E+00                 50      21          6   0.000E+00
   51      21          2   0.000E+00                 52      21          4   0.000E+00
   53      21          6   0.000E+00                 54      22          2   0.000E+00
   55      22          4   0.000E+00                 56      22          6   0.000E+00
   57      23          2   0.000E+00                 58      23          4   0.000E+00
   59      23          6   0.000E+00                 60      24          2   0.000E+00
   61      24          4   0.000E+00                 62      24          6   0.000E+00
   63      25          2   0.000E+00                 64      25          4   0.000E+00
   65      25          6   0.000E+00                 66       5          1   0.000E+00
   67       5          5   0.000E+00                 68       5          6   0.000E+00
   69      10          1   0.000E+00                 70      10          5   0.000E+00
   71      10          6   0.000E+00                 72      15          1   0.000E+00
   73      15          5   0.000E+00                 74      15          6   0.000E+00
   75      20          1   0.000E+00                 76      20          5   0.000E+00
   77      20          6   0.000E+00                 78      25          1   0.000E+00
   79      25          5   0.000E+00                 80      25          6   0.000E+00
```

```
print elem
----------



values will be printed at integration points

element quantities printed every    1 increments

stress strain plastic
from element     1 to element    16 by     1
from integration point     1 to integration point     4 by     1
a list of layers given below
     1     11

print node
----------



number of sets used for selective print of nodal quantities is99999

nodal quantities printed every    1 increments

all
a list of nodes given below
    25

post
----------



*** note - format of post code cards has changed.
           in k4, enter code in first field and  layer number in second field

elem vars,post tape,prev tape,  type  , conn fl ,post tape, prev tape,  repost ,frequency, k2post
       10       16       17        1        1       19       20        0        1        0
element variables appear on post-processor tape 16 in following order
post variable   1 is post code   11 at layer   1 =
post variable   2 is post code   12 at layer   1 =
post variable   3 is post code   13 at layer   1 =
post variable   4 is post code   17 at layer   1 =
post variable   5 is post code    7 at layer   1 =
post variable   6 is post code   11 at layer  11 =
post variable   7 is post code   12 at layer  11 =
post variable   8 is post code   13 at layer  11 =
post variable   9 is post code   17 at layer  11 =
post variable  10 is post code    7 at layer  11 =

***maximum record length on formatted post file=   80     approximate no. of records per increment on file=     157
```

```
   end option
   ----------


                  maximum connectivity is     5  at node    7


                  maximum half-bandwidth is     7  between nodes     1  and     7

                   number of profile entries including fill-in is        145

                   number of profile entries excluding fill-in is         97


                  total workspace needed with in-core matrix storage =     51139



                  load increments associated with each degree of freedom
                  summed over the whole model

                  distributed loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00


                  point loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
```

No load was applied in increment zero because increment zero cannot include any nonlinearities.

```
                  increment zero is a null step



          distributed load    type    current
          list number                 magnitude


                 1             2         0.           0.           0.

                  end of increment 0

                  formatted post data at increment   0.   0   on tape 19
                   time =        2.15

   comment,  apply 10 load steps of 5 psi each
```

```
dist loads
----------

read from unit     5
type index distributed load
    2    0  0.5000000E+01  0.0000000E+00  0.0000000E+00
name of element set is alle

auto load
----------



iotnum,incasm
   10    0

continue
----------



equal load incs specified for  10  increments




                 s t a r t   o f   i n c r e m e n t   1



                 load increments associated with each degree of freedom
                 summed over the whole model

                 distributed loads
0.000E+00 0.000E+00-5.000E+02 0.000E+00 0.000E+00 0.000E+00



                 point loads
0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                 start of assembly
                  time =       2.36



                 start of matrix solution
                  time =       3.11



                 singularity ratio    1.2068E-03
```

Stiffness matrix does not include any geometrical stiffening.

```
                 end of matrix solution
                  time =       3.16
```

```
        maximum residual force  at node  14 degree of freedom   2 is equal to   0.172E+04
        maximum reaction force  at node   4 degree of freedom   2 is equal to   0.358E+04
         convergence ratio                                                      0.482E+00

         failure to converge to tolerance

          increment will be recycled




               load increments associated with each degree of freedom
               summed over the whole model

               distributed loads
 0.000E+00 0.000E+00-5.000E+02 0.000E+00 0.000E+00 0.000E+00


               point loads
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

               start of assembly
                time =        3.93



               start of matrix solution
                time =        4.67



               singularity ratio     4.1724E-03
```

Singularity ratio improves due to stress stiffening.

```
               end of matrix solution
                time =        4.72


        maximum residual force  at node  14 degree of freedom   2 is equal to   0.795E+02
        maximum reaction force  at node   4 degree of freedom   2 is equal to   0.129E+04
         convergence ratio                                                      0.618E-01
```

```
 MARC          output for increment    1.     nonlinear analysis of a simply supported square plate



tresca     mises    mean   p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
        intensity intensity  normal   minimum intermediate maximum     1          2          3          4          5          6
                  intensity

element     1  point    1       integration pt. coordinate=     0.528E+00    0.528E+00    0.000E+00
section thickness = 0.250E+00
average membrane
stress  1.840E+02 1.603E+02 9.952E+00-7.605E+01-1.999E+00 1.079E+02 1.493E+01 1.493E+01 9.098E+01-1.039E+01-1.039E+01
moment  3.775E+02 3.270E+02-5.880E+00-1.976E+02 0.000E+00 1.799E+02-8.821E+00-8.821E+00 1.887E+02 1.388E-17 1.388E-17
stretch 2.393E-05 1.399E-05 0.000E+00-1.078E-05-2.773E-07 1.315E-05 1.045E-06 1.045E-06 2.366E-05-2.700E-06-2.700E-06
curvatr 9.422E-03 5.445E-03 0.000E+00-4.830E-03 0.000E+00 4.592E-03-1.185E-04-1.185E-04 9.422E-03 0.000E+00 0.000E+00
layer  1
stress  9.242E+03 8.006E+03-1.312E+02-4.818E+03-4.877E-02 4.424E+03-1.968E+02-1.968E+02 4.621E+03-1.039E+01-1.039E+01
layer 11
stress  8.878E+03 7.692E+03 1.511E+02-4.212E+03 5.122E-02 4.665E+03 2.266E+02 2.266E+02-4.439E+03-1.039E+01-1.039E+01

element     1  point    2       integration pt. coordinate=     0.197E+01    0.528E+00    0.000E+00
section thickness = 0.250E+00
average membrane
stress  4.083E+02 3.698E+02 1.641E+02-4.020E+00 9.198E+01 4.043E+02 1.216E+02 3.706E+02 9.098E+01-3.876E+01-1.039E+01
moment  3.647E+02 3.166E+02-1.391E+01-2.032E+02 0.000E+00 1.615E+02-1.438E+01-2.736E+01 1.823E+02-2.776E-17 1.388E-17
stretch 4.084E-05 4.194E-05 0.000E+00-2.852E-06-6.783E-07 3.799E-05 1.045E-06 3.341E-05 2.366E-05-1.008E-05-2.700E-06
curvatr 9.104E-03 5.286E-03 0.000E+00-4.832E-03 0.000E+00 4.271E-03-1.185E-04-4.424E-04 9.098E-03 0.000E+00 0.000E+00
layer  1
stress  8.931E+03 7.738E+03-1.698E+02-4.720E+03-1.994E-01 4.211E+03-2.235E+02-2.860E+02 4.465E+03-3.876E+01-1.039E+01
layer 11
stress  8.585E+03 7.472E+03 4.980E+02-3.546E+03 2.385E-01 5.039E+03 4.668E+02 1.027E+03-4.283E+03-3.876E+01-1.039E+01

element     1  point    3       integration pt. coordinate=     0.528E+00    0.197E+01    0.000E+00
section thickness = 0.250E+00
average membrane
stress  4.083E+02 3.698E+02 1.641E+02-4.020E+00 9.198E+01 4.043E+02 3.706E+02 1.216E+02 9.098E+01-1.039E+01-3.876E+01
moment  3.647E+02 3.166E+02-1.391E+01-2.032E+02 0.000E+00 1.615E+02-2.736E+01-1.438E+01 1.823E+02 1.388E-17-2.776E-17
stretch 4.084E-05 4.194E-05 0.000E+00-2.852E-06-6.783E-07 3.799E-05 3.341E-05 1.045E-06 2.366E-05-2.700E-06-1.008E-05
curvatr 9.104E-03 5.286E-03 0.000E+00-4.832E-03 0.000E+00 4.271E-03-4.424E-04-1.185E-04 9.098E-03 0.000E+00 0.000E+00
layer  1
stress  8.931E+03 7.738E+03-1.698E+02-4.720E+03-1.994E-01 4.211E+03-2.860E+02-2.235E+02 4.465E+03-1.039E+01-3.876E+01
layer 11
stress  8.585E+03 7.472E+03 4.980E+02-3.546E+03 2.385E-01 5.039E+03 1.027E+03 4.668E+02-4.283E+03-1.039E+01-3.876E+01

element     1  point    4       integration pt. coordinate=     0.197E+01    0.197E+01    0.000E+00
section thickness = 0.250E+00
average membrane
stress  5.788E+02 5.115E+02 3.182E+02-5.240E+00 3.863E+02 5.735E+02 4.773E+02 4.773E+02 9.098E+01-3.876E+01-3.876E+01
moment  3.515E+02 3.062E+02-2.195E+01-2.087E+02 0.000E+00 1.428E+02-3.292E+01-3.292E+01 1.758E+02 1.110E-16 1.110E-16
stretch 4.743E-05 6.870E-05 0.000E+00-1.096E-06 2.158E-05 4.634E-05 3.341E-05 3.341E-05 2.366E-05-1.008E-05-1.008E-05
curvatr 8.774E-03 5.143E-03 0.000E+00-4.830E-03 0.000E+00 3.945E-03-4.424E-04-4.424E-04 8.774E-03 0.000E+00 0.000E+00
```

```
layer  1
stress  8.619E+03 7.471E+03-2.085E+02-4.622E+03-7.518E-01 3.997E+03-3.127E+02-3.127E+02 4.309E+03-3.876E+01-3.876E+01
layer 11
stress  8.256E+03 7.261E+03 8.449E+02-2.861E+03 1.050E+00 5.395E+03 1.267E+03 1.267E+03-4.127E+03-3.876E+01-3.876E+01

                                 •                                                 •
                                 •                                                 •
                                 •                                                 •
                                 •                                                 •

element   16  point   1       integration pt. coordinate=     0.803E+01     0.803E+01     0.000E+00
section thickness = 0.250E+00
average membrane
stress  1.846E+03 1.844E+03 1.230E+03-5.520E-03 1.843E+03 1.846E+03 1.844E+03 1.844E+03-1.331E+00-2.255E+00-2.255E+00
moment  2.613E+02 2.552E+02-1.699E+02-2.613E+02-2.485E+02 0.000E+00-2.549E+02-2.549E+02 6.396E+00-8.674E-18 0.000E+00
stretch 1.293E-04 2.582E-04 0.000E+00-1.334E-09 1.289E-04 1.293E-04 1.291E-04 1.291E-04-3.461E-07-5.864E-07-5.864E-07
curvatr 3.586E-03 6.855E-03 0.000E+00-3.586E-03-3.266E-03 0.000E+00-3.426E-03-3.426E-03 3.193E-04 0.000E+00 0.000E+00
layer  1
stress  4.426E+03 4.282E+03-2.849E+03-4.426E+03-4.122E+03 2.469E-03-4.274E+03-4.274E+03 1.522E+02-2.255E+00-2.255E+00
layer 11
stress  8.117E+03 7.967E+03 5.308E+03-1.303E-03 7.808E+03 8.117E+03 7.962E+03 7.962E+03-1.548E+02-2.255E+00-2.255E+00

element   16  point   2       integration pt. coordinate=     0.947E+01     0.803E+01     0.000E+00
section thickness = 0.250E+00
average membrane
stress  1.974E+03 1.930E+03 1.286E+03-3.121E-03 1.883E+03 1.974E+03 1.883E+03 1.974E+03-1.331E+00-9.091E-01-2.255E+00
moment  2.639E+02 2.594E+02-1.728E+02-2.639E+02-2.546E+02 0.000E+00-2.569E+02-2.616E+02 4.055E+00-2.168E-18 0.000E+00
stretch 1.409E-04 2.701E-04 0.000E+00-7.657E-10 1.291E-04 1.409E-04 1.291E-04 1.409E-04-3.461E-07-2.364E-07-5.864E-07
curvatr 3.601E-03 6.970E-03 0.000E+00-3.601E-03-3.368E-03 0.000E+00-3.426E-03-3.543E-03 2.024E-04 0.000E+00 0.000E+00
layer  1
stress  4.390E+03 4.297E+03-2.863E+03-4.390E+03-4.197E+03 1.402E-03-4.283E+03-4.305E+03 9.598E+01-9.091E-01-2.255E+00
layer 11
stress  8.293E+03 8.155E+03 5.434E+03-7.383E-04 8.009E+03 8.293E+03 8.049E+03 8.252E+03-9.864E+01-9.091E-01-2.255E+00

element   16  point   3       integration pt. coordinate=     0.803E+01     0.947E+01     0.000E+00
section thickness = 0.250E+00
average membrane
stress  1.974E+03 1.930E+03 1.286E+03-2.701E-03 1.883E+03 1.974E+03 1.974E+03 1.883E+03-1.331E+00-2.255E+00-9.091E-01
moment  2.639E+02 2.594E+02-1.728E+02-2.639E+02-2.546E+02 0.000E+00-2.616E+02-2.569E+02 4.055E+00-8.674E-18-4.770E-18
stretch 1.409E-04 2.701E-04 0.000E+00-6.659E-10 1.291E-04 1.409E-04 1.409E-04 1.291E-04-3.461E-07-5.864E-07-2.364E-07
curvatr 3.601E-03 6.970E-03 0.000E+00-3.601E-03-3.368E-03 0.000E+00-3.543E-03-3.426E-03 2.024E-04 0.000E+00 0.000E+00
layer  1
stress  4.390E+03 4.297E+03-2.863E+03-4.390E+03-4.197E+03 1.402E-03-4.305E+03-4.283E+03 9.598E+01-2.255E+00-9.091E-01
layer 11
stress  8.293E+03 8.155E+03 5.434E+03-7.383E-04 8.009E+03 8.293E+03 8.252E+03 8.049E+03-9.864E+01-2.255E+00-9.091E-01

element   16  point   4       integration pt. coordinate=     0.947E+01     0.947E+01     0.000E+00
section thickness = 0.250E+00
average membrane
stress  2.014E+03 2.013E+03 1.342E+03-8.217E-04 2.011E+03 2.014E+03 2.013E+03 2.013E+03-1.331E+00-9.091E-01-9.091E-01
moment  2.653E+02 2.636E+02-1.757E+02-2.653E+02-2.619E+02 0.000E+00-2.636E+02-2.636E+02 1.714E+00-4.337E-19-3.036E-18
stretch 1.411E-04 2.818E-04 0.000E+00-1.985E-10 1.407E-04 1.411E-04 1.409E-04 1.409E-04-3.461E-07-2.364E-07-2.364E-07
curvatr 3.586E-03 7.086E-03 0.000E+00-3.586E-03-3.500E-03 0.000E+00-3.543E-03-3.543E-03 8.555E-05 0.000E+00 0.000E+00
layer  1
stress  4.354E+03 4.314E+03-2.876E+03-4.354E+03-4.274E+03 3.867E-04-4.314E+03-4.314E+03 3.980E+01-9.091E-01-9.091E-01
layer 11
stress  8.382E+03 8.340E+03 5.560E+03-1.992E-04 8.297E+03 8.382E+03 8.340E+03 8.340E+03-4.246E+01-9.091E-01-9.091E-01
```

```
                         g e n e r a l i z e d   s t r e s s e s
```

Nodal stress quantity.

```
   25   518.57       518.57      -0.33279      -0.10442      -0.10442       -66.697       -66.697      1.20022E-03
      7.22621E-19 -9.56808E-19  2.85584E-15


      tresca        mises         mean        p r i n c i p a l   v a l u e s
    intensity    intensity       normal       minimum   intermediate   maximum
                    intensity


   25   518.90       518.57       345.71     -4.20769E-05   518.24       518.90
1

                          n o d a l   p o i n t   d a t a
```

Only center node printed.

```
                  i n c r e m e n t a l   d i s p l a c e m e n t s


   25        0.            0.     -0.17113           0.           0.           0.


                       t o t a l   d i s p l a c e m e n t s


   25        0.            0.     -0.17113           0.           0.           0.


       total equivalent nodal forces (distributed plus point loads)


   25        0.            0.      -7.8125           0.           0.           0.


       reaction forces at fixed boundary conditions, residual load correction elsewhere


   25   614.13       614.13       1.6640       81.034      -81.034      5.21706E-15

                  summary of externally applied loads

  0.00000E+00    0.00000E+00   -0.50000E+03    0.00000E+00    0.00000E+00    0.00000E+00

                  summary of reaction/residual forces

  0.45475E-12   -0.90949E-12    0.50000E+03    0.75285E+03   -0.75285E+03   -0.10936E-12
```

```
         distributed load    type    current
         list number                 magnitude


                  1            2      5.000          0.          0.

                   e n d   o f   i n c r e m e n t   1

                   formatted post data at increment   1.   0   on tape 19
                    time =       6.43
                         •                                      •
                         •                                      •
                         •                                      •
                         •                                      •

                   s t a r t   o f   i n c r e m e n t  10



                   load increments associated with each degree of freedom
                   summed over the whole model

                   distributed loads
0.000E+00 0.000E+00-5.000E+02 0.000E+00 0.000E+00 0.000E+00


                   point loads
0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

                   start of assembly
                    time =      28.62



                   start of matrix solution
                    time =      29.41



                   singularity ratio    6.7538E-03



                   end of matrix solution
                    time =      29.46


                 maximum residual force  at node    9 degree of freedom    2 is equal to    0.276E+02
                 maximum reaction force  at node    4 degree of freedom    2 is equal to    0.828E+04
                  convergence ratio                                                         0.333E-02
```

```
 MARC           output for increment   10.     nonlinear analysis of a simply supported square plate

tresca     mises    mean  p r i n c i p a l   v a l u e s        p h y s i c a l  c o m p o n e n t s
        intensity intensity  normal   minimum intermediate maximum     1          2          3          4          5          6
                   intensity

element    1  point   1       integration pt. coordinate=     0.528E+00    0.528E+00    0.000E+00
section thickness = 0.250E+00
average membrane
stress  7.501E+02 6.665E+02-1.273E+02-5.520E+02-2.805E+01 1.982E+02-1.909E+02-1.909E+02 3.610E+02-5.272E+01-5.272E+01
moment  1.129E+03 9.792E+02-3.190E+01-6.125E+02-1.891E-06 5.168E+02-4.785E+01-4.785E+01 5.646E+02 2.210E-02 2.210E-02
stretch 1.101E-04 6.892E-05 0.000E+00-6.757E-05-2.271E-06 4.256E-05-1.364E-05-1.364E-05 1.079E-04-1.390E-05-1.390E-05
curvatr 2.940E-02 1.702E-02 0.000E+00-1.536E-02 0.000E+00 1.404E-02-6.603E-04-6.603E-04 2.940E-02 0.000E+00 0.000E+00
layer  1
stress  2.305E+04 2.000E+04-8.023E+02-1.273E+04-3.518E-01 1.032E+04-1.203E+03-1.203E+03 1.153E+04-4.261E+01-4.261E+01
plas.st 7.859E-04 4.540E-04 0.000E+00-4.049E-04-1.047E-08 3.810E-04-1.194E-05-1.194E-05 7.859E-04-2.824E-06-2.824E-06
layer 11
stress  2.307E+04 2.000E+04 5.897E+02-1.065E+04 3.819E-01 1.242E+04 8.846E+02 8.846E+02-1.154E+04-4.510E+01-4.510E+01
plas.st 5.677E-04 3.279E-04 0.000E+00-2.769E-04 8.571E-09 2.908E-04 6.978E-06 6.978E-06-5.677E-04-2.179E-06-2.179E-06

element    1  point   2       integration pt. coordinate=     0.197E+01    0.528E+00    0.000E+00
section thickness = 0.250E+00
average membrane
stress  2.488E+03 2.275E+03 9.829E+02-1.714E+01 4.950E+02 2.471E+03 5.791E+02 2.370E+03 3.976E+02-1.977E+02-5.298E+01
moment  1.076E+03 9.390E+02-7.622E+01-6.524E+02-7.865E-07 4.238E+02-7.908E+01-1.496E+02 5.370E+02 2.400E-02 6.431E-03
stretch 2.614E-04 2.573E-04 0.000E+00-2.551E-05-2.984E-06 2.359E-04-1.364E-05 2.210E-04 1.079E-04-5.189E-05-1.390E-05
curvatr 2.765E-02 1.627E-02 0.000E+00-1.539E-02 0.000E+00 1.226E-02-6.603E-04-2.464E-03 2.759E-02 0.000E+00 0.000E+00
layer  1
stress  2.305E+04 2.000E+04-7.888E+02-1.271E+04-1.618E+00 1.034E+04-1.215E+03-1.151E+03 1.153E+04-1.687E+02-4.521E+01
plas.st 5.606E-04 3.238E-04 0.000E+00-2.891E-04-3.282E-08 2.716E-04-9.245E-06-8.315E-06 5.606E-04-8.025E-06-2.150E-06
layer 11
stress  2.264E+04 2.000E+04 2.624E+03-7.387E+03 2.373E+00 1.526E+04 2.383E+03 5.488E+03-1.121E+04-1.745E+02-4.675E+01
plas.st 4.299E-04 2.497E-04 0.000E+00-1.915E-04 2.640E-08 2.384E-04-4.752E-06 5.176E-05-4.261E-04-6.527E-06-1.749E-06

element    1  point   3       integration pt. coordinate=     0.528E+00    0.197E+01    0.000E+00
section thickness = 0.250E+00
average membrane
stress  2.488E+03 2.275E+03 9.829E+02-1.714E+01 4.950E+02 2.471E+03 2.370E+03 5.791E+02 3.976E+02-5.298E+01-1.977E+02
moment  1.076E+03 9.390E+02-7.622E+01-6.524E+02-7.865E-07 4.238E+02-1.496E+02-7.908E+01 5.370E+02 6.431E-03 2.400E-02
stretch 2.614E-04 2.573E-04 0.000E+00-2.551E-05-2.984E-06 2.359E-04 2.210E-04-1.364E-05 1.079E-04-1.390E-05-5.189E-05
curvatr 2.765E-02 1.627E-02 0.000E+00-1.539E-02 0.000E+00 1.226E-02-2.464E-03-6.603E-04 2.759E-02 0.000E+00 0.000E+00
layer  1
stress  2.305E+04 2.000E+04-7.888E+02-1.271E+04-1.618E+00 1.034E+04-1.151E+03-1.215E+03 1.153E+04-4.521E+01-1.687E+02
plas.st 5.606E-04 3.238E-04 0.000E+00-2.891E-04-3.282E-08 2.716E-04-8.315E-06-9.245E-06 5.606E-04-2.150E-06-8.025E-06
layer 11
stress  2.264E+04 2.000E+04 2.624E+03-7.387E+03 2.373E+00 1.526E+04 5.488E+03 2.383E+03-1.121E+04-4.675E+01-1.745E+02
plas.st 4.299E-04 2.497E-04 0.000E+00-1.915E-04 2.640E-08 2.384E-04 5.176E-05-4.752E-06-4.261E-04-1.749E-06-6.527E-06

element    1  point   4       integration pt. coordinate=     0.197E+01    0.197E+01    0.000E+00
section thickness = 0.250E+00
average membrane
stress  3.600E+03 3.261E+03 2.096E+03-2.197E+01 2.733E+03 3.578E+03 3.145E+03 3.145E+03 4.116E+02-1.983E+02-1.983E+02
moment  1.013E+03 8.954E+02-1.207E+02-6.874E+02-4.304E-09 3.252E+02-1.811E+02-1.811E+02 5.063E+02 8.366E-04 8.366E-04
stretch 2.846E-04 4.484E-04 0.000E+00-4.812E-06 1.671E-04 2.798E-04 2.210E-04 2.210E-04 1.079E-04-5.189E-05-5.189E-05
```

Notice
Plasti
Strain

```
 MARC            output for increment   10.     nonlinear analysis of a simply supported square plate

 tresca     mises     mean   p r i n c i p a l   v a l u e s         p h y s i c a l   c o m p o n e n t s
intensity intensity  normal   minimum intermediate maximum      1          2          3          4          5          6
                    intensity


 curvatr 2.579E-02 1.568E-02 0.000E+00-1.536E-02 0.000E+00 1.043E-02-2.464E-03-2.464E-03 2.579E-02 0.000E+00 0.000E+00
 layer  1
 stress  2.306E+04 2.000E+04-7.773E+02-1.269E+04-6.236E+00 1.036E+04-1.166E+03-1.166E+03 1.152E+04-1.798E+02-1.798E+02
 plas.st 3.354E-04 1.937E-04 0.000E+00-1.730E-04-8.176E-08 1.623E-04-5.403E-06-5.403E-06 3.353E-04-5.152E-06-5.152E-06
 layer 11
 stress  2.160E+04 2.000E+04 4.731E+03-3.714E+03 1.744E+01 1.789E+04 7.096E+03 7.096E+03-1.079E+04-1.800E+02-1.800E+02
 plas.st 3.099E-04 1.828E-04 0.000E+00-1.227E-04 1.060E-07 1.873E-04 3.235E-05 3.235E-05-3.098E-04-5.100E-06-5.100E-06

 element   16  point   1       integration pt. coordinate=     0.803E+01    0.803E+01    0.000E+00
 section thickness = 0.250E+00
 average membrane
 stress  1.288E+04 1.278E+04 8.522E+03-1.043E-02 1.269E+04 1.288E+04 1.278E+04 1.278E+04-9.598E+01 8.132E+00 8.132E+00
 moment  4.842E+02 4.671E+02-3.107E+02-4.842E+02-4.480E+02 8.225E-06-4.661E+02-4.661E+02 1.809E+01 4.292E-02 4.292E-02
 stretch 9.839E-04 1.934E-03 0.000E+00-2.608E-09 9.499E-04 9.839E-04 9.669E-04 9.669E-04-3.400E-05 2.226E-06 2.226E-06
 curvatr 8.227E-03 1.538E-02 0.000E+00-8.227E-03-7.144E-03 0.000E+00-7.685E-03-7.685E-03 1.082E-03 0.000E+00 0.000E+00
 layer  1
 stress  7.796E+02 6.809E+02 5.886E+01-3.013E+02-3.065E-01 4.782E+02 8.829E+01 8.829E+01 3.896E+02 8.561E+00 8.561E+00
 layer 11
 stress  2.038E+04 2.000E+04 1.333E+04-3.870E-03 1.960E+04 2.038E+04 1.999E+04 1.999E+04-3.870E+02 6.158E+00 6.158E+00
 plas.st 5.627E-04 6.113E-04 0.000E+00-3.949E-10 4.940E-04 5.627E-04 5.283E-04 5.283E-04-6.868E-05 6.246E-07 6.246E-07

 element   16  point   2       integration pt. coordinate=     0.947E+01    0.803E+01    0.000E+00
 section thickness = 0.250E+00
 average membrane
 stress  1.345E+04 1.317E+04 8.770E+03-8.155E-03 1.286E+04 1.345E+04 1.288E+04 1.343E+04-1.025E+02 6.371E+00 8.082E+00
 moment  4.731E+02 4.617E+02-3.075E+02-4.731E+02-4.495E+02 7.923E-06-4.567E+02-4.659E+02 1.086E+01 3.636E-02 4.733E-02
 stretch 1.051E-03 2.015E-03 0.000E+00-2.047E-09 9.634E-04 1.051E-03 9.669E-04 1.047E-03-3.400E-05 1.752E-06 2.226E-06
 curvatr 8.280E-03 1.577E-02 0.000E+00-8.280E-03-7.487E-03 0.000E+00-7.685E-03-8.082E-03 6.862E-04 0.000E+00 0.000E+00
 layer  1
 stress  5.427E+02 5.080E+02 2.066E+02-4.053E-01 7.796E+01 5.423E+02 1.906E+02 4.292E+02 1.992E+02 6.740E+00 8.561E+00
 layer 11
 stress  2.038E+04 2.000E+04 1.333E+04-3.084E-03 1.960E+04 2.038E+04 1.970E+04 2.027E+04-2.629E+02 4.885E+00 6.065E+00
 plas.st 6.312E-04 6.866E-04 0.000E+00-2.917E-10 5.555E-04 6.312E-04 5.656E-04 6.212E-04-5.142E-05 4.822E-07 6.489E-07

 element   16  point   3       integration pt. coordinate=     0.803E+01    0.947E+01    0.000E+00
 section thickness = 0.250E+00
 average membrane
 stress  1.345E+04 1.317E+04 8.770E+03-8.155E-03 1.286E+04 1.345E+04 1.343E+04 1.288E+04-1.025E+02 8.082E+00 6.371E+00
 moment  4.731E+02 4.617E+02-3.075E+02-4.731E+02-4.495E+02 7.923E-06-4.659E+02-4.567E+02 1.086E+01 4.733E-02 3.636E-02
 stretch 1.051E-03 2.015E-03 0.000E+00-2.047E-09 9.634E-04 1.051E-03 1.047E-03 9.669E-04-3.400E-05 2.226E-06 1.752E-06
 curvatr 8.280E-03 1.577E-02 0.000E+00-8.280E-03-7.487E-03 0.000E+00-8.082E-03-7.685E-03 6.862E-04 0.000E+00 0.000E+00
 layer  1
 stress  5.427E+02 5.080E+02 2.066E+02-4.053E-01 7.796E+01 5.423E+02 4.292E+02 1.906E+02 1.992E+02 8.561E+00 6.740E+00
 layer 11
 stress  2.038E+04 2.000E+04 1.333E+04-3.084E-03 1.960E+04 2.038E+04 2.027E+04 1.970E+04-2.629E+02 6.065E+00 4.885E+00
 plas.st 6.312E-04 6.866E-04 0.000E+00-2.917E-10 5.555E-04 6.312E-04 6.212E-04 5.656E-04-5.142E-05 6.489E-07 4.822E-07
```

```
 MARC           output for increment   10.     nonlinear analysis of a simply supported square plate

 tresca     mises    mean   p r i n c i p a l   v a l u e s         p h y s i c a l   c o m p o n e n t s
intensity intensity  normal   minimum intermediate maximum     1          2          3          4          5          6
                   intensity
 element   16  point   4       integration pt. coordinate=     0.947E+01    0.947E+01    0.000E+00
 section thickness = 0.250E+00
 average membrane
 stress  1.355E+04 1.344E+04 8.961E+03-5.948E-03 1.333E+04 1.355E+04 1.344E+04 1.344E+04-1.108E+02 6.296E+00 6.296E+00
 moment  4.562E+02 4.524E+02-3.015E+02-4.562E+02-4.485E+02 7.543E-06-4.523E+02-4.523E+02 3.867E+00 4.113E-02 4.113E-02
 stretch 1.064E-03 2.095E-03 0.000E+00-1.490E-09 1.030E-03 1.064E-03 1.047E-03 1.047E-03-3.400E-05 1.752E-06 1.752E-06
 curvatr 8.227E-03 1.616E-02 0.000E+00-8.227E-03-7.937E-03 0.000E+00-8.082E-03-8.082E-03 2.900E-04 0.000E+00 0.000E+00
 layer  1
 stress  5.405E+02 5.320E+02 3.544E+02-1.681E-01 5.229E+02 5.404E+02 5.315E+02 5.315E+02 8.678E+00 6.740E+00 6.740E+00
 layer 11
 stress  2.015E+04 2.000E+04 1.333E+04-2.368E-03 1.985E+04 2.015E+04 2.000E+04 2.000E+04-1.520E+02 4.847E+00 4.847E+00
 plas.st 6.731E-04 7.597E-04 0.000E+00-1.884E-10 6.424E-04 6.731E-04 6.577E-04 6.577E-04-3.074E-05 4.920E-07 4.920E-07


                  g e n e r a l i z e d   s t r e s s e s

    25  3416.6      3416.6     -29.133      1.4054      1.4054     -111.64     -111.64    -0.32068
         1.01344E-02  1.01344E-02  1.90502E-14


         tresca        mises         mean       p r i n c i p a l   v a l u e s
       intensity    intensity       normal      minimum  intermediate   maximum
                       intensity

     25   3445.8       3417.0       2277.7     -1.16619E-03   3387.5       3445.8

                  n o d a l   p o i n t   d a t a

                  i n c r e m e n t a l   d i s p l a c e m e n t s

     25       0.           0.     -1.94700E-02       0.          0.          0.


                  t o t a l   d i s p l a c e m e n t s

     25       0.           0.     -0.43398           0.          0.          0.
```

Total displacement of center node.

```
      total equivalent nodal forces (distributed plus point loads)


     25       0.           0.      -78.125           0.          0.          0.
```

```
      reaction forces at fixed boundary conditions, residual load correction elsewhere


  25   4131.4       4131.4     -0.22336       138.23       -138.23       2.98035E-14

                 summary of externally applied loads

 0.00000E+00    0.00000E+00   -0.50000E+04    0.00000E+00    0.00000E+00    0.00000E+00

                 summary of reaction/residual forces

-0.90949E-12   -0.27285E-11    0.50000E+04    0.16161E+04   -0.16161E+04    0.66929E-12



      distributed load    type    current
      list number                 magnitude


                 1            2      50.00          0.          0.

                 e n d   o f   i n c r e m e n t  10

                 formatted post data at increment  10.   0   on tape 19
                  time =       31.33



                 *** end of input deck - job ends



                           marc exit number 3004
```

Results

The following six figures illustrate:

1. Node 25 Z-displacement history – from increment 1 to 10.
2. Z-displacement contour plot for increment 10.
3. Layer 1 – contour plot of equivalent von Mises tensile stress for increment 10.
4. Layer 1 – contour plot of equivalent plastic strain for increment 10.
5. Layer 11 – contour plot of equivalent von Mises tensile stress for interment 10
6. Layer 11 – contour plot of equivalent plastic strain for increment 10.

Note the perfect symmetry about a 45° line for all the contour plots of displacements, stresses, and strains – as you would expect from such a problem with a symmetrical mesh and symmetrical loads and boundary conditions. (If asymmetry exists in any of these plots, that would be an indication of modeling or analysis error somewhere). Notice the nonsymmetry between the top and bottom layers (as opposed to an elastic pure bending solution). This is due to the large deformation/ initial stress stiffening effects.

From the first two figures, you can see that the Z-displacement at the center of the plate (node 25) reached a maximum value of -0.434 inch, after the full pressure load of 50.0 psi was applied in increment 10. This displacement is almost twice the value of the plate thickness of 0.25 inch! Therefore, the successful solution of a plate problem with such large deformations does require a nonlinear, large displacement formulation. Use of linear theory would have given you a wrong answer. (A rough rule of thumb says that linear theory is valid provided that the plate deflection is on the order of one-tenth the plate thickness, or less.) Also, notice in the first figure that about 40% of the displacement occurred after increment 1 (with only 5.0 psi pressure applied), and subsequently, the plate seems to exhibit stiffening behavior (the incremental displacement is getting smaller and smaller from increment to increment). This stiffening behavior (sometimes called "stress stiffening") is not unusual in such nonlinear problems.

The onset of plasticity began in increment 6 in element 16 layer 11. The plasticity zone gradually spread, until approximately two-thirds of the model had yielded by increment 10 as shown in the last two figures. Since the pressure loading is uniform, the incremental spreading of this plasticity zone is to a large extent dependent upon the exact nature of the boundary conditions along the left and bottom edges.

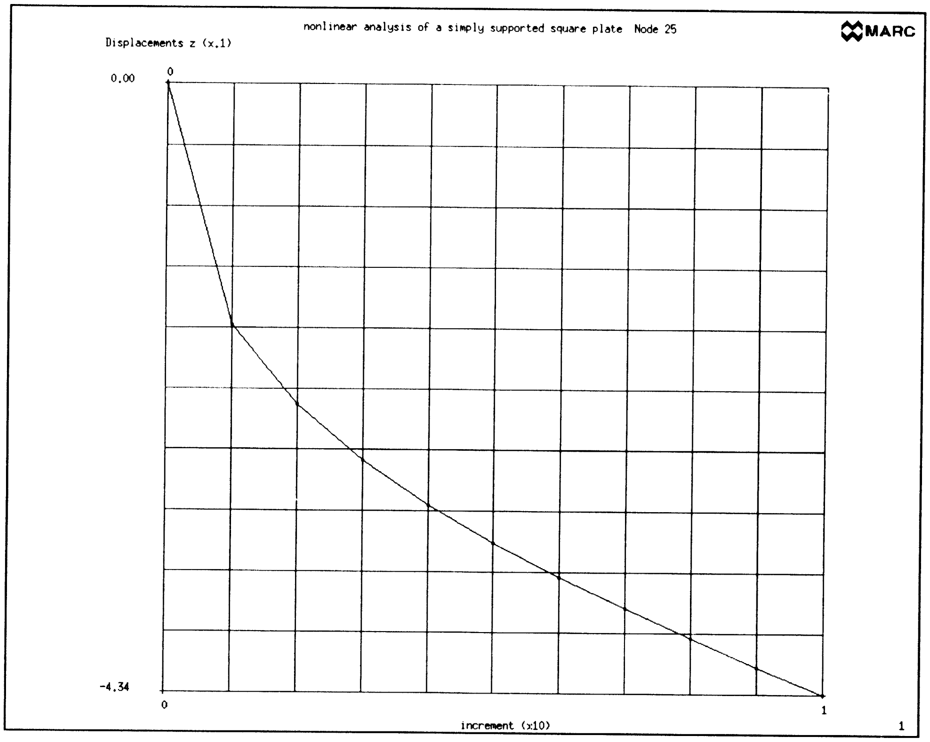


Figure 6.2 Time History of Deformation

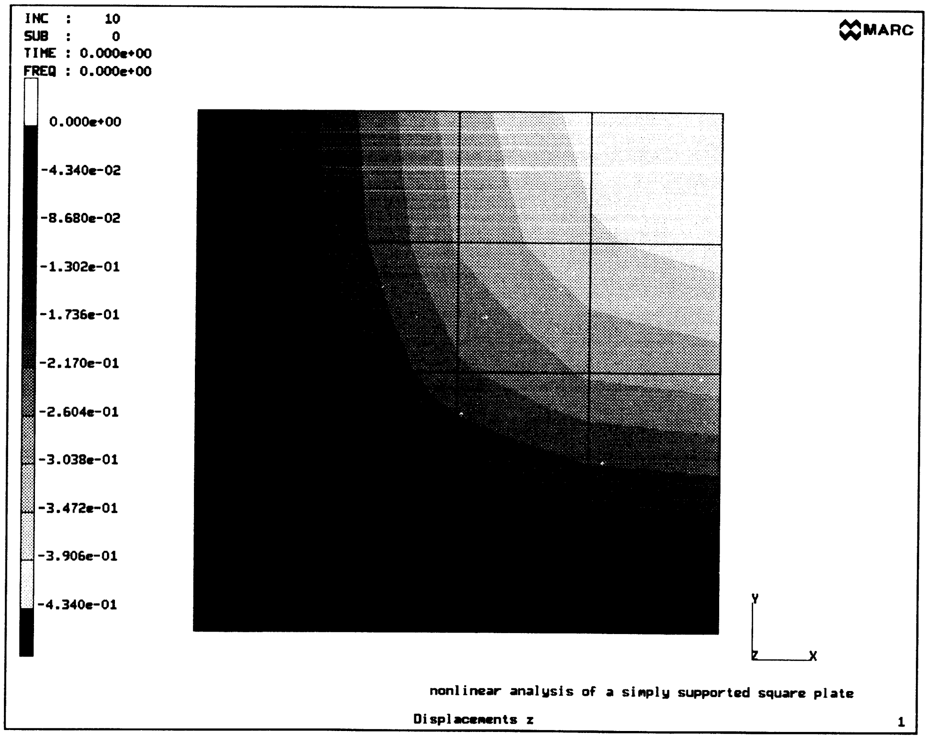


Figure 6.3 Contour Plot of Z-Displacement

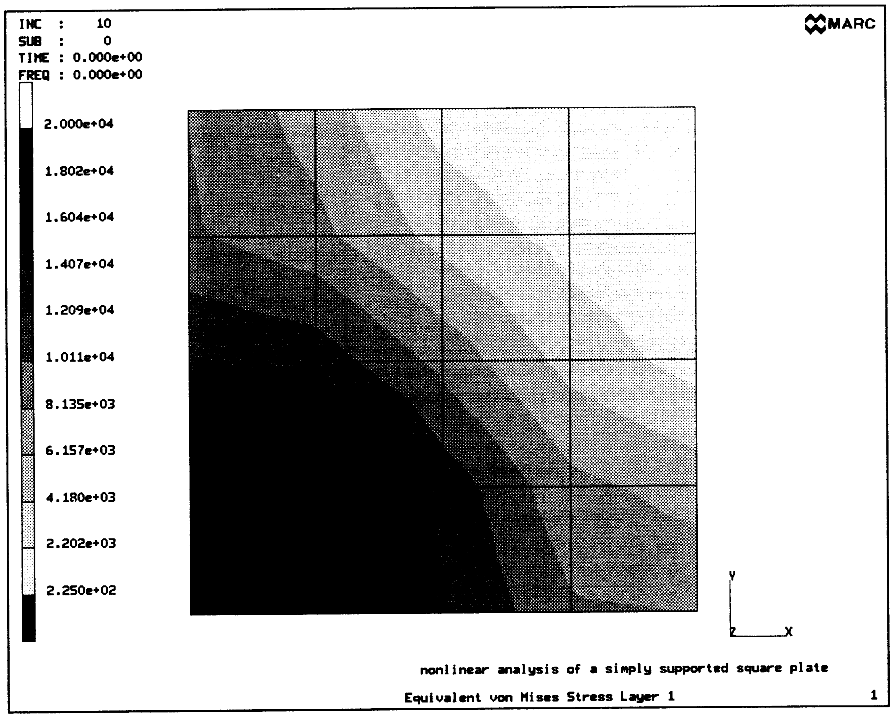


Figure 6.4 Equivalent von Mises Stress - Layer I

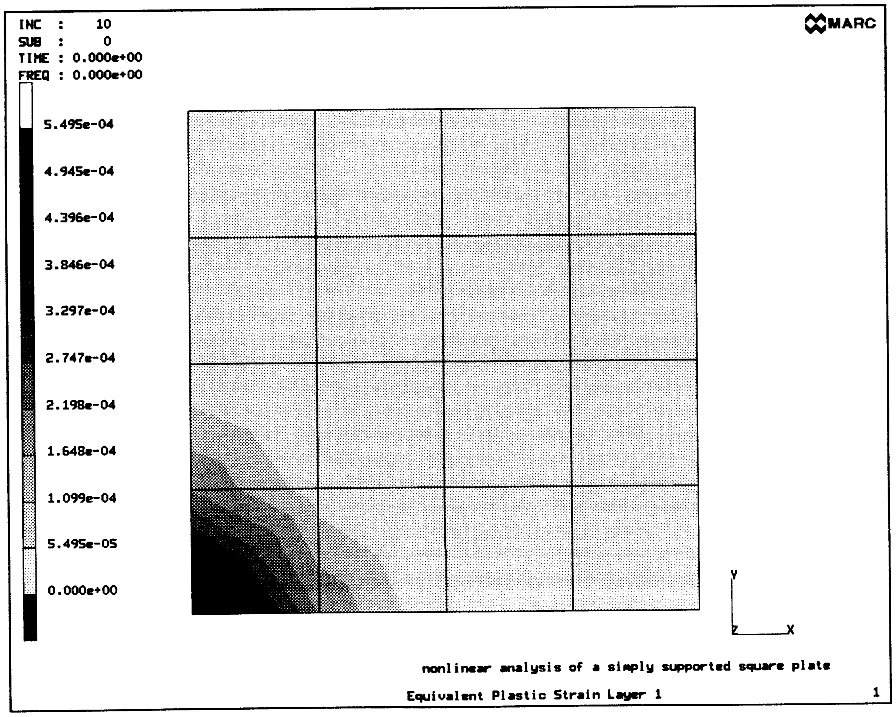


Figure 6.5 Equivalent Plastic Strain - Layer I

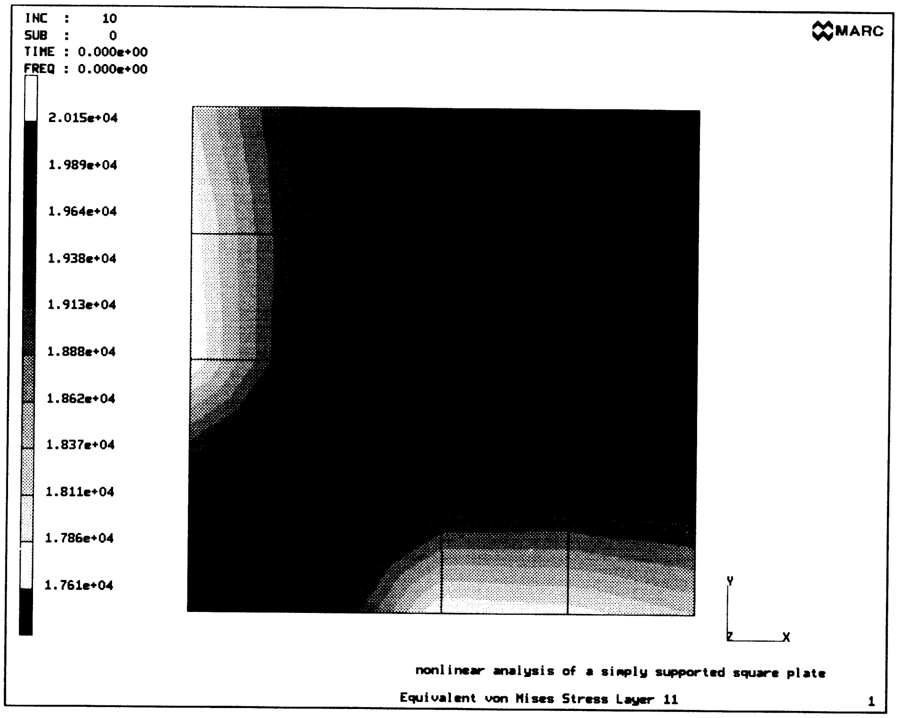


Figure 6.6 Equivalent von Mises Stress - Layer II

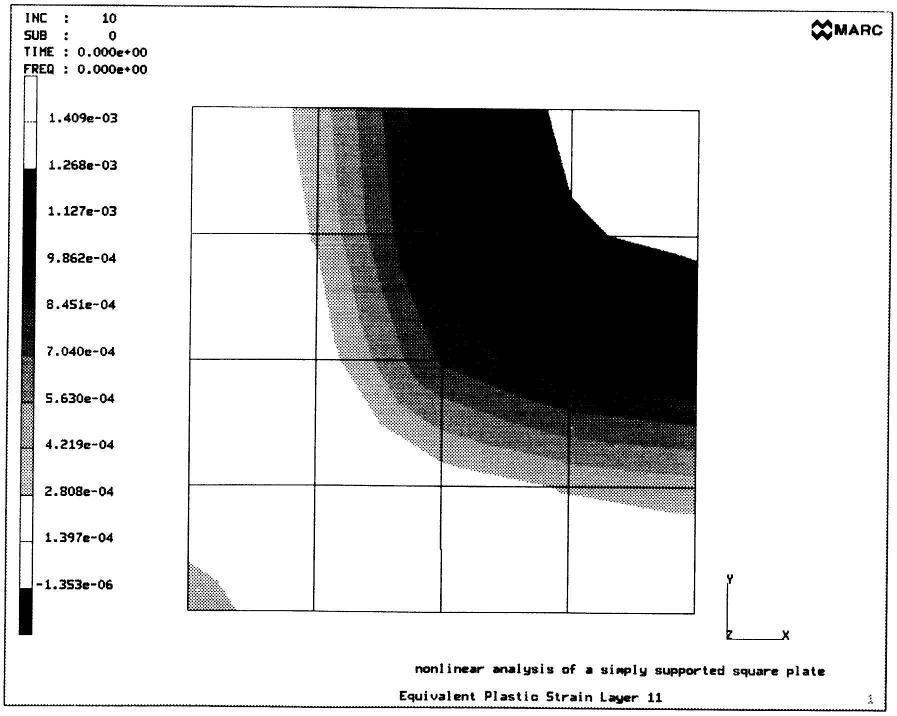


Figure 6.7 Equivalent Plastic Strain - Layer II

Exercises

1. Rerun the same problem, assuming a tensile yield stress of 25,000 psi. What, if any, is the difference in the answers?
2. Rerun the same problem, except this time assume the left and bottom edges are clamped, instead of simply-supported. Would you expect the Z-displacement at node 25 to be greater or less? How about the maximum stresses in the plate?
3. Can this example be solved using a one-eighth model? Try it.
4. Rerun the problem but require that the loads be normal to the plate surface by changing the load type and adding the FOLLOW FOR parameter card.

Example 7

Postbuckling Analysis of a Spherical Cap Under Apex Load

The purpose of this example is to demonstrate nonlinear geometric and post-buckling analysis using MARC. The FE model used is a half model of a spherical cap under a ring-type load at the apex. The objective is to track its nonlinear load-deflection behavior. This example illustrates: nonlinear geometric and post-buckling ("snap-through") behavior of a shell; the LARGE DISP option (first introduced in Example 6); use of an axisymmetric curved shell element; and automatic load incrementation.

NOTE

This example is 2-D, and does *not* consider plasticity effects or perform eigenvalue buckling analysis.

Sketch

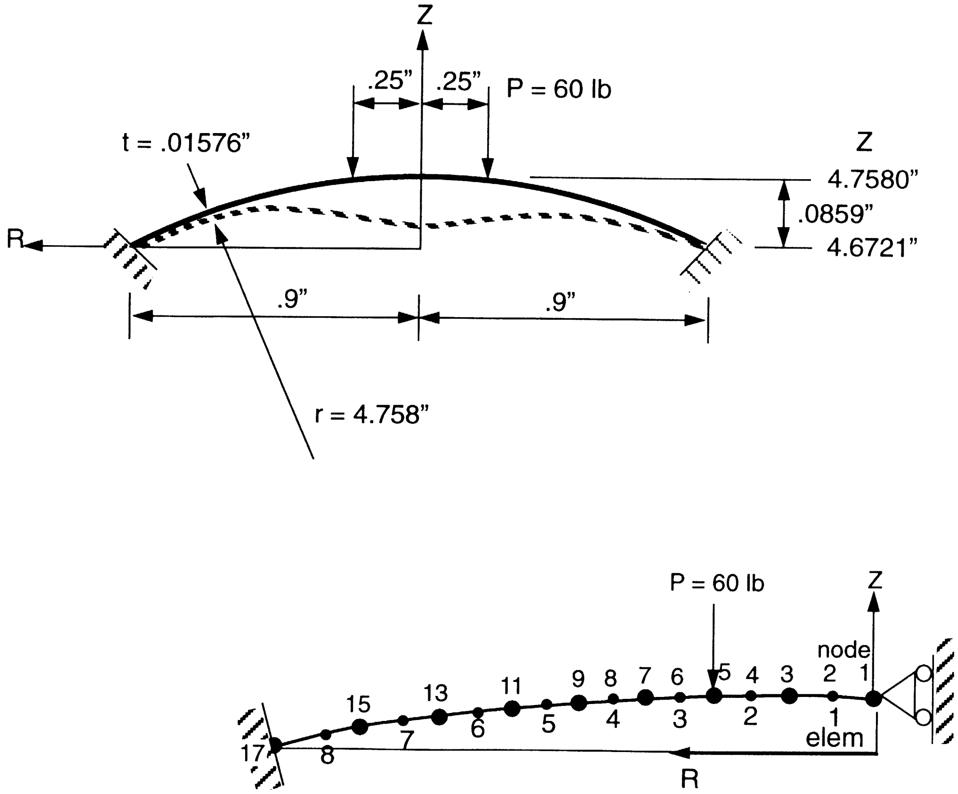


Figure 7.1 **Spherical Cap**

Model

The entire spherical cap is modeled using axisymmetric elements. The Z-axis is the longitudinal axis. The cap has a radius of 4.7580 in., a depth of 0.0859 in., and a uniform thickness of 0.01576 in. A ring load of 60 lb. is applied at a radius of 0.25 in. around the apex. Young's modulus is 10E6 (or 10,000,000) psi and Poisson's ratio is 0.3.

The model consists of 17 nodes and 8 axisymmetric curved shell elements (MARC Element 89). This element is 3-noded (with better performance than the 2- noded Element 1). It includes transverse shear deformation effects, and is therefore recommended for axisymmetric thick shell analysis. Because reduced integration is used, this element may also be used for thin shell structures as in this example. The element is suitable for large displacement analysis with small strains. Two-point Gaussian integration is used along the element for the stiffness calculation, and three-point integration is used for the mass and pressure determination. Each node has three DOFs: two displacements – in the axial (Z) and radial (R) directions, and one rotation. The element thickness can vary linearly along the length (although uniform thickness is used here). See Volume B for a complete description of Element 89.

Properties

The material is isotropic and linear elastic. Therefore, we only need to specify two material properties: Young's modulus and Poisson's ratio.

Loads

The ring load near the apex has a magnitude of 60 lbs and is applied in the Z-direction at a radius of 0.25 in. The load will be applied incrementally beginning with increment 1, where we must specify an initial step size on the AUTO INCREMENT option (to be discussed later under the LOAD INCREMENTATION input description). In MARC axisymmetric analysis, a point load is the total integrated load around the circumference.

Boundary Conditions

The end of the cap is clamped. Therefore, all three DOFs at node 17 are suppressed – that is, no translations or rotation are permitted. At the axis of symmetry (node 1), symmetry conditions are prescribed: no radial displacement and no rotation.

Input

A complete listing is included.

PARAMETER Section

The "SIZING" line sets 100,000 words as the workspace. The "ELEMENTS" line tells MARC that Element 89 will be used.

The "LARGE DISP" line flags the program control for large displacement analysis. This flag instructs MARC to calculate the geometric stiffness matrix and the initial stress stiffness matrix. It also automatically switches off the scaling option (if it exists).

The "SHELL SECT,3" line means we want three layers across the shell cross section for Simpson's rule integration of stresses. Three points are sufficient for use with linear materials, such as in this example. (Seven layers are recommended for simple plasticity or creep analysis. Eleven layers – the default value – are recommended for complex plasticity or creep problems, such as thermal plasticity.)

The "END" line terminates the PARAMETER section.

MODEL DEFINITION Section

The MODEL DEFINITION options in this example consist of:

a. FE mesh topology – including the CONNECTIVITY, COORDINATES, and DEFINE blocks
b. Material properties
c. Geometric properties
d. Boundary conditions
e. Loads
f. Nonlinear analysis controls
g. Output controls

FE Mesh Topology

The spherical cap is modeled using a model of 17 nodes and 8 curved 3-noded elements. As the nodes increase in number from 1 to 17, their Z-coordinates decrease (from 4.7580 to 4.6721 in.) and R-coordinates increase (from 0.0 to 0.9000 in.). The maximum deflection of the cap is expected to be at the apex (node 1).

The CONNECTIVITY block defines the element connectivity for each of the eight elements. On the "8 0 0" line, the "8" refers to the number of elements to be read in. Then, a typical line in this block is the next line ("1 89 1 2 3"), which is interpreted as element number 1 is of Element Type 89 and is specified by connecting nodes 1, 2, and 3.

The COORDINATES block gives the coordinates of each node. The "2 17 0 0" line means the maximum number of coordinate directions to be read in per node is two (i.e., Z and R in this example), and there are a total of 17 nodes.

Then follow four DEFINE blocks. We are placing all eight elements into an element set named ALLE, node 17 into a node set named FIXME, node 1 into node set SYMM, and node 5 into node set LOADME.

Material Properties

Again, we are using the ISOTROPIC block to input the material properties for this problem. The "1," line says only one set of material properties will follow. The "10.E6,.3," line gives the Young's modulus and Poisson's ratio, respectively. (The remaining properties on this line are not needed for this problem.) The "ALLE" line ends this block, and tells MARC to assign the specified properties to element set ALLE (in other words, all eight elements).

Geometric Properties

The GEOMETRY block defines the geometric properties needed for the elements. The blank line says we do not need to count the number of sets of geometric properties (we could have, of course, placed a "1" in the first field). The ".01576," line gives that value as the thickness of the shell elements in the model. (The fact we left the third field blank implies that constant thickness is assumed.) The "ALLE" line ends this block, and assigns this thickness to all eight elements.

Boundary Conditions

Again, the FIXED DISP option specifies the fixed displacement that each named DOF must take during the first and subsequent increments.

Two sets of fixed displacements are prescribed. In the first set, the "0.,0.,0.," line gives zero-valued displacements for the three DOFs named on the following line. The "1 TO 3" line tells MARC that DOFs 1 through 3 (translations in the Z- and R-directions, and rotation) are the DOFs we want to have zero values. Notice here we have used the list input convention "1 TO 3" rather than "1 2 3." Then, we tell MARC these fixed displacements only apply to node set FIXME (node 17). The second set begins with the "0.,0.," line. In the same manner, we can see that zero displacements are assigned to DOF 2 (R-displacement) and DOF 3 (rotation) for node set SYMM (node 1). This second set means that, on the axis of symmetry, there can be no R-displacement or rotation.

Loads

As in Example 6, we will first input a zero load for increment 0. Later, in the "LOAD INCREMENTATION" lines, we will actually use the POINT LOAD option again for increment 1 and subsequent increments, combined with the AUTO INCREMENT option. This method allows us to delay application of the concentrated ring load at the apex until increment 1.

The POINT LOAD option lets you input nodal point loads. The blank line means we do not need to count how many sets of point loads are being entered. The "0.," line assigns a zero load to the first DOF (Z-direction). The "LOADME" line tells MARC this load applies to node set LOADME (node 5).

Nonlinear Analysis Controls

The CONTROL block lets you input control parameters for regulating the convergence and accuracy of the nonlinear stress analysis. The "30,10,,,,,1" line means: the maximum number of load steps is to be 30; the maximum number of recycles will be 10; the full Newton-Raphson iterative technique will be chosen (since we left the sixth field on this blank); and solution of "non-positive definite" systems will be forced. This is needed due to the unstable nature of the physical problem. As the deflection increases and the spherical cap reached its "snap through" load, the tangent stiffness may no longer be positive definite. The ".01," line refers to the maximum allowed relative error in residual forces (which happens to be only one-tenth of that allowed in Example 5).

In most FE applications, as long as a sufficient number of boundary conditions are prescribed, the structural matrices of interest (e.g., the stiffness and mass matrices) are positive definite, real, and symmetric. In linear-elastic analysis, the system is always positive definite given sufficient boundary conditions. For more details about positive definiteness and the algebraic eigenproblem in FE structural analysis, see: David S. Burnett, *Finite Element Analysis: From Concepts to Applications*, Addison-Wesley (Reading, Massachusetts), 1987, pp. 410-414; or J. H. Wilkinson, *The Algebraic Eigenvalue Problem*, Oxford University Press (N.Y.), 1965. In this example, the stiffness matrix may not be positive definite due to the geometrically nonlinear nature of the problem. Large compressive stresses often cause a nonpositive definite system because of their negative contribution to the initial stress stiffness matrix.

The POST block follows the CONTROL option. The ",,,1" line means we would like to have a formatted post file for later post-processing by Mentat II.

The "END OPTION" line terminates the MODEL DEFINITION options.

LOAD INCREMENTATION Section

Before discussing the POINT LOAD and AUTO INCREMENT blocks, we point out that this example demonstrates the ability to insert optional printout control in the LOAD INCREMENTATION section. Note that the first block in the LOAD INCREMENTATION section is the familiar PRINT ELEM option, which you have seen many times in previous examples – but always in the MODEL DEFINITION section. It works the same way here.

The PRINT ELEM option lets you specify which element quantities at what elements are to be printed. The blank line, as usual, means you don't need to enter the number of sets which follow. The "STRESS" line says we want to see the total stress printed. The first "1" line refers to element 1. The second "1" line means integration point 1. And, the "2" line tells MARC to print layer two. Therefore, these element quantities will be printed out for every increment, starting with increment 1.

The POINT LOAD option is being used in this example a second time. Recall that it allows you to prescribe nodal point loads. The blank line says we do not need to count the number of sets of point loads. The "-60.," line says to apply a 60 lb. concentrated load in the global -Z-direction. The "LOADME" line indicates that the load is applied to node set LOADME; i.e., node 5.

NOTE

In the presence of an AUTO INCREMENT option (as in this example), the POINT LOAD, DIST LOAD, and FIXED DISP options specify *total* values.

The AUTO INCREMENT option allows for automatic load stepping (incrementation) in a quasi-static analysis. It is very useful for both geometrically nonlinear problems (such as solved here using the LARGE DISP option) and material nonlinear (elastic-plastic) problems. It can handle "snap-though" phenomena and can track post-buckling behavior accurately. The four fields on the data line are:

Field	Data Interpretation
1	***fraction*** of total load increment to be applied in the first cycle of the ***first increment*** of this AUTO INCREMENT session.
2	***maximum*** number of increments in this AUTO INCREMENT session.
3	***desired*** number of recycles per increment. Default is 5. Used to increase/decrease load steps during AUTO INCREMENT session. (More recycles may be specified using the CONTROL Model Definition option.)
4	***maximum*** fraction of total load which may be applied in any increment of this AUTO INCREMENT session. Default is 1.

Therefore, the ".2,40,3," line means: we want 20% of the total load of 60 lb. (or 12 lb.) to be applied in the first cycle of increment 1; the maximum number of steps will be 40; and the desired number of recycles per step (increment) is 3. (Leaving the fourth field blank means we will get the default value of 1 – for the maximum step size.)

The "CONTINUE" line ends the LOAD INCREMENTATION section and the input file.

Output

The selective output included here consists of: the input echo; the program sizing and options summary; and results for increment 0 (null increment), through increment 24 (final increment). As will be obvious in the results discussion, the intermediate increments are of interest because they represent relative maxima/minima in the nonlinear load-deflection behavior of the cap (Figure 7.5).

Let's examine the output for increment 1 to see how MARC's AUTO INCREMENT capability works. As prescribed, MARC first applies a load of 20% of 60 lbs. or 12 lbs. (-Z-direction), at node 5. It failed to converge in the first cycle (iteration) – to our specified "maximum allowed relative error in residual forces" value of 0.01. Thereby, MARC gives the message "Failure to converge to tolerance – Increment will be recycled." The automatic load stepping algorithm now comes in, and removes 2.66% of the load, ending up with a total applied load of (20.00 – 2.66), or 17.34%. With this load, the MARC solution converges (showing a convergence ratio of 0.000217), and prints out the requested element output, incremental nodal displacements, total displacements (showing a Z-displacement of -.00275 in. at node 1), total equivalent nodal forces (showing a nodal load of -10.402 lb. at node 5 only), and the reaction forces. Then, MARC goes on to increment 2.

The point to remember here is the importance of the convergence tolerance value you specify on the CONTROL option's third line. The default value is 0.1 – for relative residual checking purposes in stress analysis. This value is the maximum allowable value of the maximum residual force in the analysis divided by the maximum reaction force. Therefore, the tighter you specify the convergence tolerance, the more cycles or iterations MARC will have to use to attempt to converge – if it ever does. On the other hand, prescribing too large a tolerance may mean your "converged solution" may not be accurate enough! Every nonlinear problem is unique, which implies that the proper convergence tolerance is problem-dependent! Since we assume that you are a new MARC user, it is generally a good idea for you to begin your analysis with the default values in MARC options. Notice, However, in Example 7, we used a convergence tolerance of 0.01; a value of 0.1 may have been too large for this nonlinear post-buckling problem. Typically, snap-through problems like this example require a tighter tolerance, especially near the snap-through point.

```
                              i n p u t   d a t a

    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
TITLE,  POST-BUCKLING OF A SPHERICAL CAP
SIZING        100000
ELEMENTS     89
COMMENT,  LARGE DISP TURNS ON EFFECTS OF GEOMETRICALLY LARGE DISPLACEMENTS
LARGE DISP
SHELL SECT,3
END
CONNECTIVITY
    8    0    0
    1   89    1    2    3
    2   89    3    4    5
    3   89    5    6    7
    4   89    7    8    9
    5   89    9   10   11
    6   89   11   12   13
    7   89   13   14   15
    8   89   15   16   17
COORDINATES
    2   17    0    0
    1   4.75800   0.00000
    2   4.75766 0.56590-1
    3   4.75665   0.11317
    4   4.75453   0.18160
    5   4.75143   0.25000
    6   4.74887   0.29464
    7   4.74589   0.33926
    8   4.74152   0.39568
    9   4.73648   0.45204
   10   4.73076   0.50835
   11   4.72438   0.56458
   12   4.71733   0.62073
   13   4.70962   0.67679
   14   4.70124   0.73275
   15   4.69219   0.78862
   16   4.68248   0.84437
   17   4.67210   0.90000
DEFINE     ELEMENT   SET        ALLE
    1 TO       8
DEFINE     NODE      SET        FIXME
   17
DEFINE     NODE      SET        SYMM
    1
DEFINE     NODE      SET        LOADME
    5
ISOTROPIC
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
--------------------------------------------------------------------------------

1,
10.E6,.3,
ALLE
GEOMETRY

.01576,
ALLE
FIXED DISP

0.,0.,0.,
1 TO 3
FIXME
0.,0.,
2 3
SYMM
POINT LOAD

0.,
LOADME
CONTROL
30,10,,,,,1
.01,
POST
,,,1
END OPTION
PRINT ELEM

STRESS

1
2
PRINT NODE
0,
TOTAL
1 TO 6
LOADS TOTAL
5,
REAC
17,
POINT LOAD

-60.,
LOADME
AUTO INCREMENT
.2,40,3,
CONTINUE
--------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

Force solution of possible non-positive definite systems (30,10,,,,,1)

Convergence to within 1% is required (.01,)

Total load desired (-60.,)

Controls load steps adaptively (.2,40,3,)

```
 --------------------------------------------------------------------------------
 --------------------------------------------------------------------------------


                   ************************************************
                   ************************************************


                    program sizing and options requested as follows




                    element type requested*************************    89
                    number of elements in mesh*********************     8
                    number of nodes in mesh************************    17
                    max number of elements in any dist load list***     0
                    maximum number of boundary conditions**********     5
                    large displacement analysis flagged************
                    load correction flagged or set*****************
                    number of lists of distributed loads***********     3
                    stresses stored at all integration points******
                    tape no.for input of coordinates + connectivity     5
                    no.of different materials    1 max.no of slopes     5
                    maximum elements variables per point on post tp    33
                    number of points on shell section *************     3
                    new style input format will be used************
                    maximum number of set names is*****************    10
                    number of processors used *********************     1
                    vector length used ****************************     1


                   end of parameters and sizing
                  ************************************************
                  ************************************************




        key to stress, strain and displacement output



                  element type   89



        3-node curved thick axisymmetric shell element


        generalized strains
          1=meridional membrane
          2=circumferential membrane
          3=transverse shear
          4=meridional curvature
          5=circumferential curvature
        stresses
```

```
  1=meridional
  2=circumferential
  3=transverse shear

  1=u axial direction
  2=v radial direction
  3=rotational


workspace needed for input and stiffness assembly     14779



internal core allocation parameters
degrees of freedom per node (ndeg)  3
coords per node (ncrd)  2
strains per integration point (ngens)  6
max. nodes per element (nnodmx)  3
max.stress components per int. point (nstrmx)   9
max. invariants per int. points (neqst)  3



flag for element storage (ielsto)  0
elements in core, words per element (nelsto)       1120
                 total space required              8960
vectors in core, total space required        709



words per track on disk set to 4096



internal element variables



internal element number  1  library code type 89
number of nodes=  3
stresses stored per integration point = 33
direct continuum components stored =  2
shear continuum components stored =  1
shell/beam flag = 1
curvilinear  coord. flag = 0
int.points for elem. stiffness  2
number of local inertia directions  2
int.point for print if all points not flagged  2
int. points for dist. surface loads (pressure)  3
library code type = 89
no local rotation flag = 1
generalized displ. flag = 0
large disp. row counts     4    2    0    0    0    0



residual load correction is invoked
```

```
connectivity
----------

meshr1,iprnt
    5      0
elem no., type,        nodes
    1       89         1     2     3
    2       89         3     4     5
    3       89         5     6     7
    4       89         7     8     9
    5       89         9    10    11
    6       89        11    12    13
    7       89        13    14    15
    8       89        15    16    17

coordinates
----------

ncrd1 ,meshr1,iprnt
    2       5      0
 node       coordinates
    1  4.7580          0.
    2  4.7577      0.56590E-01
    3  4.7567      0.11317
    4  4.7545      0.18160
    5  4.7514      0.25000
    6  4.7489      0.29464
    7  4.7459      0.33926
    8  4.7415      0.39568
    9  4.7365      0.45204
   10  4.7308      0.50835
   11  4.7244      0.56458
   12  4.7173      0.62073
   13  4.7096      0.67679
   14  4.7012      0.73275
   15  4.6922      0.78862
   16  4.6825      0.84437
   17  4.6721      0.90000

define    element   set       alle
----------

from element      1 to element      8 by      1

define    node      set       fixme
----------

a list of nodes given below
    17
```

```
define    node      set       symm
----------

a list of nodes given below
     1

define    node      set       loadme
----------

a list of nodes given below
     5

isotropic
----------



isotropic material  material id =     1
 von mises  yield criteria
 isotropic  hardening rule
   e          nu         rho        alpha       yield      yield2
  0.100E+08  0.300E+00  0.000E+00  0.000E+00  0.100E+21  0.100E+21
name of element set is alle

geometry
----------

  egeom1     egeom2     egeom3     egeom4     egeom5     egeom6
  0.158E-01  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
name of element set is alle

fixed disp
----------

fixed displacement =  0.000E+00  0.000E+00  0.000E+00
from degrees of freedom    1 to degrees of freedom    3 by     1
name of node set is fixme
fixed displacement =  0.000E+00  0.000E+00  0.000E+00
a list of degrees of freedom given below
     2      3
name of node set is symm

fixed boundary condition summary.
total fixed degrees of freedom read so far =    5

 b.c.    node   degree of  magnitude             b.c.    node   degree of  magnitude
number            freedom                       number            freedom


    1      17         1    0.000E+00               2      17         2    0.000E+00
    3      17         3    0.000E+00               4       1         2    0.000E+00
    5       1         3    0.000E+00
```

```
point load
----------

read from unit     5
 0.000E+00 0.000E+00 0.000E+00
name of node set is loadme

control
----------

max.     max.     min.
incs   recycles recycles
   30      10       0

maximum allowed relative error in residual forces      0.10000E-01


full newton-raphson technique chosen


solution of non-positive definite equation systems will be forced

post
----------


*** note - format of post code cards has changed.
           in k4, enter code in first field and  layer number in second field

elem vars,post tape,prev tape,  type  , conn fl ,post tape, prev tape,  repost ,frequency, k2post
        0       16        17         1         1        19        20         0         1         0
element variables appear on post-processor tape 16 in following order

***maximum record length on formatted post file=   80
approximate no. of records per increment on file=      21

end option
----------



                 maximum connectivity is    3  at node    3



                 maximum half-bandwidth is    3  between nodes    1  and    3

                  number of profile entries including fill-in is          41

                  number of profile entries excluding fill-in is          41
```

```
                    total workspace needed with in-core matrix storage =     15721




                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
 0.000E+00 0.000E+00 0.000E+00



                    point loads
 0.000E+00 0.000E+00 0.000E+00



                    increment zero is a null step




                    e n d   o f   i n c r e m e n t   0

                    formatted post data at increment   0.   0   on tape 19
                     time =       1.03
  print elem
  ----------



  values will be printed at integration points

  element quantities printed every    1 increments

  stress
  a list of integration points given below
       1
  a list of layers given below
       2

  print node
  ----------



  number of sets used for selective print of nodal quantities is99999

  nodal quantities printed every    1 increments

  total
```

Change printout requirements in History Definition

```
from node      1 to node      6 by      1

loads total
a list of nodes given below
     5

reac
a list of nodes given below
    17

point load
----------

read from unit     5
-0.600E+02 0.000E+00 0.000E+00
name of node set is loadme

auto increment
----------




auto incrementation specified
initial step size                         0.2000
maximum step size                         1.0000
maximum number of steps                  40
desired number of recycles per step       3
maximum arclength multiplier              5.0000

continue
----------




automatic loadstepping specified




                start  of  increment  1
                load increments associated with each degree of freedom
                summed over the whole model

                distributed loads
 0.000E+00 0.000E+00 0.000E+00



                point loads
-1.200E+01 0.000E+00 0.000E+00

                start of assembly
                 time =       1.07
```

When used in conjunction with the AUTO INCREMENT option, loads applied are total loads, NOT incremental loads

```
                    start of matrix solution
                     time =       1.13

                    singularity ratio    3.0774E-02
                    end of matrix solution
                     time =       1.14

                     20.00 percent of total load added in this cycle

                   maximum residual force  at node    6 degree of freedom    2 is equal to    0.659E+01
                   maximum reaction force  at node   17 degree of freedom    2 is equal to    0.944E+02
                   convergence ratio                                                          0.698E-01

                    failure to converge to tolerance

                    increment will be recycled
                    distributed loads
  0.000E+00 0.000E+00 0.000E+00



                    point loads
 -1.200E+01 0.000E+00 0.000E+00

                    start of assembly
                     time =       1.19

                    start of matrix solution
                     time =       1.23

                    singularity ratio    2.8953E-02

                    end of matrix solution
                     time =       1.24

                     -2.66 percent of total load added in this cycle

                   maximum residual force  at node    5 degree of freedom    2 is equal to    0.181E-01
                   maximum reaction force  at node   17 degree of freedom    2 is equal to    0.833E+02
                   convergence ratio                                                          0.217E-03

MARC               output for increment   1.    post-buckling of a spherical cap


                    automatic stepping has reached    17.3352 percent  of total load


                    n o d a l   p o i n t   d a t a


        t o t a l   d i s p l a c e m e n t
```

Remove part of the load step

```
     1 -2.75436E-03        0.           0.                  2 -2.74351E-03 -1.30378E-05 -4.68805E-04
     3 -2.69328E-03 -2.52956E-05 -1.40565E-03               4 -2.53576E-03 -3.63577E-05 -3.52292E-03
     5 -2.17419E-03 -3.68511E-05 -7.38300E-03               6 -1.79091E-03 -2.72533E-05 -9.34646E-03



          total equivalent nodal forces (distributed plus point loads)


     5  -10.401           0.          0.
```

Clearly in equilibrium

```
          reaction forces at fixed boundary conditions, residual load correction elsewhere


   17   10.401      -83.341     -0.50813

                         summary of externally applied loads

-0.10401E+02    0.00000E+00    0.00000E+00

                         summary of reaction/residual forces

 0.10401E+02   -0.83410E+02   -0.50555E+00

                         e n d   o f   i n c r e m e n t   1
                         formatted post data at increment   1.   0   on tape 19
                          time =        1.33


                         s t a r t   o f   i n c r e m e n t   2
                         load increments associated with each degree of freedom
                         summed over the whole model

                         distributed loads
     0.000E+00 0.000E+00 0.000E+00



                         point loads
    -1.040E+01 0.000E+00 0.000E+00

                         start of assembly
                          time =        1.33

                         start of matrix solution
                          time =        1.38

                         singularity ratio    2.8935E-02
```

This is an attempt to increase the load because the last increment used only two iterations, whereas three are considered acceptable.

```
                    end of matrix solution
                     time =        1.39

                     18.12 percent of total load added in this cycle

                   maximum residual force  at node   10 degree of freedom    2 is equal to    0.889E+01
                   maximum reaction force  at node   17 degree of freedom    2 is equal to    0.172E+03
                   convergence ratio                                                          0.516E-01

                    failure to converge to tolerance

                    increment will be recycled
                    distributed loads
  0.000E+00 0.000E+00 0.000E+00


                    point loads
 -1.087E+01 0.000E+00 0.000E+00

                    start of assembly
                     time =        1.43

                    start of matrix solution
                     time =        1.47

                    singularity ratio    2.6768E-02

                    end of matrix solution
                     time =        1.48

                     -3.37 percent of total load added in this cycle

                   maximum residual force  at node    5 degree of freedom    2 is equal to    0.452E-01
                   maximum reaction force  at node   17 degree of freedom    2 is equal to    0.158E+0
                   convergence ratio                                                          0.287E-0

MARC                output for increment   2.     post-buckling of a spherical cap


                     automatic stepping has reached    32.0795 percent  of total load


                     n o d a l   p o i n t   d a t a

                     t o t a l   d i s p l a c e m e n t s


         1 -6.22414E-03       0.          0.              2 -6.17816E-03 -2.66115E-05 -1.77760E-03
         3 -6.00849E-03 -5.07760E-05 -4.39474E-03         4 -5.55769E-03 -7.06306E-05 -9.31121E-03
         5 -4.67305E-03 -6.87543E-05 -1.70886E-02         6 -3.80656E-03 -4.92028E-05 -2.08223E-02
```

```
            total equivalent nodal forces (distributed plus point loads)


        5  -19.248          0.          0.




            reaction forces at fixed boundary conditions, residual load correction elsewhere


       17   19.248      -157.55      -1.0644

                        summary of externally applied loads

-0.19248E+02    0.00000E+00    0.00000E+00

                        summary of reaction/residual forces

 0.19248E+02   -0.15776E+03   -0.10598E+01

                        e n d   o f   i n c r e m e n t   2
                        formatted post data at increment   2.   0   on tape 19
                         time =       1.57


                        s t a r t   o f   i n c r e m e n t   3
                        load increments associated with each degree of freedom
                        summed over the whole model

                        distributed loads
     0.000E+00 0.000E+00 0.000E+00



                        point loads
    -8.847E+00 0.000E+00 0.000E+00

                        start of assembly
                         time =       1.58

                        start of matrix solution
                         time =       1.62

                        singularity ratio    2.6735E-02

                        end of matrix solution
                         time =       1.63
```

```
                           14.21 percent of total load added in this cycle

                         maximum residual force  at node    8 degree of freedom    2 is equal to    0.130E+0
                         maximum reaction force  at node   17 degree of freedom    2 is equal to    0.231E+0
                          convergence ratio                                                         0.560E-0

                          failure to converge to tolerance

                          increment will be recycled
                          distributed loads
  0.000E+00 0.000E+00 0.000E+00


                          point loads
 -8.528E+00 0.000E+00 0.000E+00

                          start of assembly
                           time =       1.67

                          start of matrix solution
                           time =       1.72

                          singularity ratio    2.4358E-02

                          end of matrix solution
                           time =       1.73

                           -3.88 percent of total load added in this cycle

                         maximum residual force  at node    5 degree of freedom    2 is equal to    0.877E-0
                         maximum reaction force  at node   17 degree of freedom    2 is equal to    0.214E+0
                          convergence ratio                                                         0.409E-0
MARC                 output for increment   3.    post-buckling of a spherical cap


                          automatic stepping has reached    42.4137 percent  of total load



                          n o d a l   p o i n t   d a t a

                          t o t a l   d i s p l a c e m e n t s


       1 -1.06216E-02       0.          0.               2 -1.04957E-02 -3.94471E-05 -4.61145E-03
       3 -1.00842E-02 -7.37698E-05 -1.01116E-02          4 -9.11887E-03 -9.88766E-05 -1.85990E-02
       5 -7.47781E-03 -9.23155E-05 -2.98601E-02          6 -6.00494E-03 -6.40310E-05 -3.47454E-02
```

```
          total equivalent nodal forces (distributed plus point loads)


      5  -25.448           0.          0.




          reaction forces at fixed boundary conditions, residual load correction elsewhere


     17   25.449       -214.20      -1.6226

                        summary of externally applied loads

-0.25448E+02    0.00000E+00    0.00000E+00

                        summary of reaction/residual forces

 0.25448E+02   -0.21460E+03   -0.16178E+01

                        end of increment 3
                        formatted post data at increment   3.   0   on tape 19
                         time =       1.81



                        start of increment 4
                        load increments associated with each degree of freedom
                        summed over the whole model

                        distributed loads
     0.000E+00 0.000E+00 0.000E+00



                        point loads
    -6.201E+00 0.000E+00 0.000E+00

                        start of assembly
                         time =       1.82

                        start of matrix solution
                         time =       1.86

                        singularity ratio    2.4331E-02

                        end of matrix solution
                         time =       1.87

                          8.47 percent of total load added in this cycle
```

```
                    maximum residual force  at node    8 degree of freedom    2 is equal to    0.196E+02
                    maximum reaction force  at node   17 degree of freedom    2 is equal to    0.265E+03
                    convergence ratio                                                          0.742E-01

                     failure to converge to tolerance

                     increment will be recycled
                     distributed loads
 0.000E+00 0.000E+00 0.000E+00


                     point loads
-5.081E+00 0.000E+00 0.000E+00

                     start of assembly
                      time =       1.91

                     start of matrix solution
                      time =       1.96

                     singularity ratio    2.1777E-02

                     end of matrix solution
                      time =       1.97

                      -3.65 percent of total load added in this cycle

                    maximum residual force  at node    4 degree of freedom    2 is equal to    0.946E-01
                    maximum reaction force  at node   17 degree of freedom    2 is equal to    0.248E+03
                    convergence ratio                                                          0.382E-03

MARC                output for increment   4.    post-buckling of a spherical cap


                     automatic stepping has reached    47.2321 percent  of total load



                     n o d a l   p o i n t   d a t a

                     t o t a l   d i s p l a c e m e n t s


    1 -1.61816E-02        0.           0.              2 -1.59085E-02 -4.90560E-05 -9.70303E-03
    3 -1.50768E-02 -9.04994E-05 -1.97324E-02           4 -1.32924E-02 -1.18688E-04 -3.25362E-02
    5 -1.05977E-02 -1.09458E-04 -4.63015E-02           6 -8.37825E-03 -7.67073E-05 -5.12906E-02
```

```
          total equivalent nodal forces (distributed plus point loads)


     5  -28.339          0.          0.


          reaction forces at fixed boundary conditions, residual load correction elsewhere


    17   28.340      -248.01      -2.1276

                       summary of externally applied loads

-0.28339E+02    0.00000E+00    0.00000E+00

                       summary of reaction/residual forces

 0.28339E+02   -0.24860E+03   -0.21259E+01

                       e n d   o f   i n c r e m e n t   4
                       formatted post data at increment   4.   0   on tape 19
                        time =       2.05


                       s t a r t   o f   i n c r e m e n t   5
                       load increments associated with each degree of freedom
                       summed over the whole model

                       distributed loads
      0.000E+00 0.000E+00 0.000E+00


                       point loads
     -2.891E+00 0.000E+00 0.000E+00

                       start of assembly
                        time =       2.06

                       start of matrix solution
                        time =       2.10

                       singularity ratio    2.0896E-02

                       end of matrix solution
                        time =       2.11

                         2.37 percent of total load added in this cycle

                     maximum residual force  at node    4 degree of freedom    2 is equal to    0.281E+02
                     maximum reaction force  at node   17 degree of freedom    2 is equal to    0.273E+03
                     convergence ratio                                                           0.103E+00
```

These load steps are considerably smaller than the original ones. The applied load will barely increase in this increment.

```
                    failure to converge to tolerance


                    increment will be recycled
                    distributed loads
  0.000E+00 0.000E+00 0.000E+00



                    point loads
 -1.422E+00 0.000E+00 0.000E+00

                    start of assembly
                     time =       2.15

                    start of matrix solution
                     time =       2.20
                    non-positive definite system at user node   16 internal node   16



                    singularity ratio    1.7230E-02

                    end of matrix solution
                     time =       2.21


                     -2.25 percent of total load added in this cycle

                   maximum residual force  at node   16 degree of freedom    2 is equal to    0.222E-01
                   maximum reaction force  at node   17 degree of freedom    2 is equal to    0.264E+03
                   convergence ratio                                                          0.841E-04

MARC                output for increment   5.    post-buckling of a spherical cap


                    automatic stepping has reached    47.3571 percent  of total load




                    n o d a l   p o i n t   d a t a

                    t o t a l   d i s p l a c e m e n t s


      1 -2.31555E-02       0.           0.                2 -2.26708E-02 -5.19670E-05 -1.69634E-02
      3 -2.12476E-02 -9.74242E-05 -3.31128E-02            4 -1.83500E-02 -1.32679E-04 -5.09881E-02
      5 -1.43075E-02 -1.32384E-04 -6.64825E-02            6 -1.11934E-02 -1.04319E-04 -7.07284E-02
```

The stiffness matrix is no longer positive definite

```
               total equivalent nodal forces (distributed plus point loads)


           5  -28.414          0.          0.




               reaction forces at fixed boundary conditions, residual load correction elsewhere


          17   28.415      -263.79      -2.5783

                          summary of externally applied loads

 -0.28414E+02    0.00000E+00    0.00000E+00

                          summary of reaction/residual forces

  0.28414E+02   -0.26434E+03   -0.25827E+01

                          e n d   o f   i n c r e m e n t   5
                          formatted post data at increment   5.   0   on tape 19
                           time =        2.29


                          s t a r t   o f   i n c r e m e n t   6
                          load increments associated with each degree of freedom
                          summed over the whole model

                          distributed loads
        0.000E+00 0.000E+00 0.000E+00


                          point loads
       -7.501E-02 0.000E+00 0.000E+00

                          start of assembly
                           time =        2.30

                          start of matrix solution
                           time =        2.34
                          non-positive definite system at user node   12 internal node   12


                          singularity ratio    7.2095E-03

                          end of matrix solution
                           time =        2.35

                           -1.25 percent of total load added in this cycle
```

Applied load is removed to insure equilibrium

```
                    maximum residual force  at node    4 degree of freedom    2 is equal to    0.269E+02
                    maximum reaction force  at node   17 degree of freedom    2 is equal to    0.274E+03
                    convergence ratio                                                          0.981E-01

                     failure to converge to tolerance

                     increment will be recycled
                     distributed loads
 0.000E+00 0.000E+00 0.000E+00


                     point loads
 7.523E-01 0.000E+00 0.000E+00

                     start of assembly
                      time =       2.39

                     start of matrix solution
                      time =       2.44
                     non-positive definite system at user node   16 internal node   16


                     singularity ratio    1.3119E-02

                     end of matrix solution
                      time =       2.45

                      -0.25 percent of total load added in this cycle

                    maximum residual force  at node    8 degree of freedom    2 is equal to    0.168E+00
                    maximum reaction force  at node   17 degree of freedom    2 is equal to    0.275E+03
                    convergence ratio                                                          0.609E-03

MARC                   output for increment    6.     post-buckling of a spherical cap


                     automatic stepping has reached    45.8526 percent  of total load



                     n o d a l   p o i n t   d a t a

                     t o t a l   d i s p l a c e m e n t s


      1 -3.03582E-02       0.           0.                   2 -2.96827E-02 -4.74579E-05 -2.35451E-02
      3 -2.77198E-02 -9.40308E-05 -4.53994E-02               4 -2.37871E-02 -1.42618E-04 -6.83239E-02
      5 -1.84637E-02 -1.66601E-04 -8.58829E-02               6 -1.44798E-02 -1.54929E-04 -8.98552E-02
```

```
              total equivalent nodal forces (distributed plus point loads)


          5  -27.512           0.           0.




              reaction forces at fixed boundary conditions, residual load correction elsewhere


         17   27.512      -275.27      -2.9883

                      summary of externally applied loads

 -0.27512E+02    0.00000E+00    0.00000E+00

                      summary of reaction/residual forces

  0.27512E+02   -0.27530E+03   -0.29975E+01

                      end  of  increment  6
                      formatted post data at increment   6.   0   on tape 19
                       time =       2.53


                      start  of  increment  7
                      load increments associated with each degree of freedom
                      summed over the whole model

                      distributed loads
      0.000E+00 0.000E+00 0.000E+00



      .               point loads
      9.027E-01 0.000E+00 0.000E+00

                      start of assembly
                       time =       2.54

                      start of matrix solution
                       time =       2.59
                      non-positive definite system at user node   11 internal node   11


                      singularity ratio    6.3999E-03

                      end of matrix solution
                       time =       2.59
```

```
                     -1.43 percent of total load added in this cycle

                   maximum residual force  at node    4 degree of freedom    2 is equal to    0.245E+02
                   maximum reaction force  at node   17 degree of freedom    2 is equal to    0.289E+03
                    convergence ratio                                                         0.847E-01

                    failure to converge to tolerance

                    increment will be recycled

                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
  0.000E+00 0.000E+00 0.000E+00


                    point loads
  8.571E-01 0.000E+00 0.000E+00

                    start of assembly
                     time =        2.64

                    start of matrix solution
                     time =        2.68

                    singularity ratio    1.7137E-02

                    end of matrix solution
                     time =        2.69

                      0.46 percent of total load added in this cycle

                   maximum residual force  at node    8 degree of freedom    2 is equal to    0.322E+00
                   maximum reaction force  at node   17 degree of freedom    2 is equal to    0.294E+03
                    convergence ratio                                                         0.109E-02

MARC                    output for increment    7.    post-buckling of a spherical cap


                    automatic stepping has reached    44.8888 percent  of total load



                    nodal  point  data

                    total  displacements


     1 -3.76952E-02       0.          0.              2 -3.68830E-02 -3.87819E-05 -2.83308E-02
```

```
            3 -3.45191E-02 -8.37368E-05 -5.47300E-02                    4 -2.97677E-02 -1.47359E-04 -8.25758E-02
            5 -2.33417E-02 -2.05487E-04 -0.10342                        6 -1.85420E-02 -2.20993E-04 -0.10842



                 total equivalent nodal forces (distributed plus point loads)


           5  -26.933           0.           0.




                 reaction forces at fixed boundary conditions, residual load correction elsewhere


          17   26.933      -294.11      -3.4337

                           summary of externally applied loads

-0.26933E+02    0.00000E+00    0.00000E+00

                           summary of reaction/residual forces

 0.26933E+02   -0.29347E+03   -0.34444E+01

                           e n d   o f   i n c r e m e n t   7
                           formatted post data at increment   7.   0   on tape 19
                            time =       2.77


                           s t a r t   o f   i n c r e m e n t   8
                           load increments associated with each degree of freedom
                           summed over the whole model

                           distributed loads
       0.000E+00 0.000E+00 0.000E+00



                           point loads
       5.783E-01 0.000E+00 0.000E+00

                           start of assembly
                            time =       2.78

                           start of matrix solution
                            time =       2.82
                           non-positive definite system at user node   14 internal node   14
```

```
                    singularity ratio    4.8457E-03


                    end of matrix solution
                     time =        2.83


                     -0.45 percent of total load added in this cycle


                   maximum residual force  at node   10 degree of freedom   2 is equal to   0.292E+02
                   maximum reaction force  at node   17 degree of freedom   2 is equal to   0.318E+03
                   convergence ratio                                                        0.918E-01


                    failure to converge to tolerance


                    increment will be recycled


                    load increments associated with each degree of freedom
                    summed over the whole model


                    distributed loads
  0.000E+00 0.000E+00 0.000E+00



                    point loads
  2.705E-01 0.000E+00 0.000E+00

                    start of assembly
                     time =        2.87


                    start of matrix solution
                     time =        2.92


                    singularity ratio    2.0335E-02


                    end of matrix solution
                     time =        2.93


                      0.56 percent of total load added in this cycle


                   maximum residual force  at node   12 degree of freedom   2 is equal to   0.254E+00
                   maximum reaction force  at node   17 degree of freedom   2 is equal to   0.323E+03
                   convergence ratio                                                        0.785E-03

MARC               output for increment    8.    post-buckling of a spherical cap


                    automatic stepping has reached    44.9955 percent  of total load




                    n o d a l   p o i n t   d a t a
```

```
                    t o t a l   d i s p l a c e m e n t s


   1 -4.50233E-02        0.           0.                  2 -4.41294E-02 -2.86508E-05 -3.12725E-02
   3 -4.15098E-02 -6.93017E-05 -6.08896E-02               4 -3.61846E-02 -1.44348E-04 -9.31481E-02
   5 -2.88851E-02 -2.38015E-04 -0.11825                   6 -2.33631E-02 -2.87659E-04 -0.12556



          total equivalent nodal forces (distributed plus point loads)


   5  -26.997          0.           0.



          reaction forces at fixed boundary conditions, residual load correction elsewhere


  17   26.997      -323.10      -3.9186

                    summary of externally applied loads

-0.26997E+02    0.00000E+00    0.00000E+00

                    summary of reaction/residual forces

 0.26997E+02   -0.32195E+03   -0.39280E+01

                    e n d   o f   i n c r e m e n t   8
                    formatted post data at increment   8.   0   on tape 19
                     time =       3.01
```

NOTE

Shell has effectively “snapped through”

```
                    s t a r t   o f   i n c r e m e n t   9
                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
      0.000E+00 0.000E+00 0.000E+00



                    point loads
     -6.406E-02 0.000E+00 0.000E+00

                    start of assembly
```

```
                         time =        3.02


                        start of matrix solution
                         time =        3.07


                        singularity ratio    1.5589E-02


                        end of matrix solution
                         time =        3.07


                           0.62 percent of total load added in this cycle


                       maximum residual force  at node   10 degree of freedom    2 is equal to    0.334E+02
                       maximum reaction force  at node   17 degree of freedom    2 is equal to    0.357E+03
                        convergence ratio                                                         0.935E-01


                         failure to converge to tolerance


                         increment will be recycled


                         load increments associated with each degree of freedom
                         summed over the whole model


                         distributed loads
   0.000E+00 0.000E+00 0.000E+00



                         point loads
  -3.695E-01 0.000E+00 0.000E+00

                          start of assembly
                           time =        3.11


                          start of matrix solution
                           time =        3.16


                          singularity ratio    2.1313E-02


                          end of matrix solution
                           time =        3.17


                             0.46 percent of total load added in this cycle


                         maximum residual force  at node   10 degree of freedom    2 is equal to    0.205E+00
                         maximum reaction force  at node   17 degree of freedom    2 is equal to    0.362E+03
                          convergence ratio                                                         0.568E-03

MARC                   output for increment   9.    post-buckling of a spherical cap


                           automatic stepping has reached    46.0700 percent  of total load
```

```
                    n o d a l   p o i n t   d a t a

                    t o t a l   d i s p l a c e m e n t s


         1 -5.22639E-02       0.           0.                    2 -5.13290E-02 -1.86010E-05 -3.28234E-02
         3 -4.85662E-02 -5.29322E-05 -6.45235E-02                4 -4.28748E-02 -1.33545E-04 -0.10041
         5 -3.49251E-02 -2.57718E-04 -0.13012                    6 -2.87965E-02 -3.43370E-04 -0.14057



               total equivalent nodal forces (distributed plus point loads)


         5  -27.642           0.           0.




               reaction forces at fixed boundary conditions, residual load correction elsewhere


        17   27.642      -361.54      -4.4153

                    summary of externally applied loads

 -0.27642E+02    0.00000E+00    0.00000E+00

                    summary of reaction/residual forces

  0.27642E+02   -0.36008E+03   -0.44219E+01

                    e n d   o f   i n c r e m e n t   9
                    formatted post data at increment   9.   0   on tape 19
                     time =        3.25



                    s t a r t   o f   i n c r e m e n t  10
                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
      0.000E+00 0.000E+00 0.000E+00



                    point loads
     -6.447E-01 0.000E+00 0.000E+00
```

```
                         start of assembly
                          time =        3.26

                         start of matrix solution
                          time =        3.30

                         singularity ratio    1.6770E-02

                         end of matrix solution
                          time =        3.31

                           1.47 percent of total load added in this cycle

                        maximum residual force  at node   10 degree of freedom    2 is equal to    0.314E+02
                        maximum reaction force  at node   17 degree of freedom    2 is equal to    0.404E+03
                        convergence ratio                                                          0.778E-01

                         failure to converge to tolerance

                         increment will be recycled

                         load increments associated with each degree of freedom
                         summed over the whole model

                         distributed loads
   0.000E+00 0.000E+00 0.000E+00


                         point loads
  -8.809E-01 0.000E+00 0.000E+00

                         start of assembly
                          time =        3.35

                         start of matrix solution
                          time =        3.40

                         singularity ratio    2.0288E-02

                         end of matrix solution
                          time =        3.41

                           0.33 percent of total load added in this cycle

                        maximum residual force  at node   10 degree of freedom    2 is equal to    0.235E+00
                        maximum reaction force  at node   17 degree of freedom    2 is equal to    0.408E+03
                        convergence ratio                                                          0.576E-03

MARC                   output for increment  10.     post-buckling of a spherical cap
```

```
                    automatic stepping has reached    47.8705 percent  of total load



                    n o d a l   p o i n t   d a t a

                    t o t a l   d i s p l a c e m e n t s


        1 -5.93955E-02       0.          0.                2 -5.84463E-02 -9.28961E-06 -3.34481E-02
        3 -5.56175E-02 -3.61538E-05 -6.63725E-02           4 -4.97159E-02 -1.16698E-04 -0.10503
        5 -4.13076E-02 -2.63298E-04 -0.13919               6 -3.46916E-02 -3.82512E-04 -0.15315




               total equivalent nodal forces (distributed plus point loads)


        5  -28.722           0.          0.




               reaction forces at fixed boundary conditions, residual load correction elsewhere


       17   28.722       -407.97      -4.8882

                    summary of externally applied loads

-0.28722E+02    0.00000E+00    0.00000E+00

                    summary of reaction/residual forces

 0.28722E+02   -0.40635E+03   -0.48914E+01

                    e n d   o f   i n c r e m e n t  10
                    formatted post data at increment  10.   0   on tape 19
                     time =       3.49




                    s t a r t   o f   i n c r e m e n t  20
                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
      0.000E+00 0.000E+00 0.000E+00
```

```
                       point loads
  -1.013E+00 0.000E+00 0.000E+00

                       start of assembly
                        time =       5.66

                       start of matrix solution
                        time =       5.70

                       singularity ratio    9.7826E-03

                       end of matrix solution
                        time =       5.71

                         1.89 percent of total load added in this cycle

                      maximum residual force  at node   14 degree of freedom    2 is equal to    0.682E+02
                      maximum reaction force  at node   17 degree of freedom    2 is equal to    0.960E+03
                       convergence ratio                                                         0.711E-01

                        failure to converge to tolerance

                        increment will be recycled

                        load increments associated with each degree of freedom
                        summed over the whole model

                        distributed loads
   0.000E+00 0.000E+00 0.000E+00


                        point loads
  -1.134E+00 0.000E+00 0.000E+00

                        start of assembly
                         time =       5.75

                        start of matrix solution
                         time =       5.80

                        singularity ratio    1.6505E-02

                        end of matrix solution
                         time =       5.81

                          0.46 percent of total load added in this cycle

                       maximum residual force  at node   16 degree of freedom    2 is equal to    0.319E+00
                       maximum reaction force  at node   17 degree of freedom    2 is equal to    0.951E+03
                        convergence ratio                                                         0.336E-03
```

```
MARC                  output for increment  20.    post-buckling of a spherical cap


                           automatic stepping has reached    69.7758 percent  of total load




                 reaction forces at fixed boundary conditions, residual load correction elsewhere


          17   41.863      -951.31       1.1233

                           summary of externally applied loads

 -0.41865E+02    0.00000E+00    0.00000E+00

                           summary of reaction/residual forces

  0.41865E+02   -0.95053E+03    0.11355E+01

                           end of increment 20
                           formatted post data at increment  20.   0   on tape 19
                            time =       5.89


                           start of increment 21
                           load increments associated with each degree of freedom
                           summed over the whole model

                           distributed loads
       0.000E+00 0.000E+00 0.000E+00


                           point loads
      -1.412E+00 0.000E+00 0.000E+00

                           start of assembly
                            time =       5.90

                           start of matrix solution
                            time =       5.95

                           singularity ratio    1.0013E-02

                           end of matrix solution
                            time =       5.95

                             2.94 percent of total load added in this cycle
```

```
                    maximum residual force  at node   12 degree of freedom    2 is equal to    0.638E+02
                    maximum reaction force  at node   17 degree of freedom    2 is equal to    0.969E+03
                    convergence ratio                                                          0.658E-01

                     failure to converge to tolerance

                     increment will be recycled

                     load increments associated with each degree of freedom
                     summed over the whole model

                     distributed loads
  0.000E+00 0.000E+00 0.000E+00


                     point loads
 -1.764E+00 0.000E+00 0.000E+00

                     start of assembly
                      time =        6.00

                     start of matrix solution
                      time =        6.04

                     singularity ratio    1.6381E-02

                     end of matrix solution
                      time =        6.05

                       1.00 percent of total load added in this cycle

                     maximum residual force  at node   16 degree of freedom    2 is equal to    0.351E+00
                     maximum reaction force  at node   17 degree of freedom    2 is equal to    0.959E+03
                     convergence ratio                                                          0.366E-03

MARC                output for increment  21.    post-buckling of a spherical cap


                      automatic stepping has reached    73.7146 percent  of total load



                      n o d a l   p o i n t   d a t a

                      t o t a l   d i s p l a c e m e n t s


        1 -0.13052          0.          0.              2 -0.12975       4.20185E-05 -2.75159E-02
        3 -0.12737       7.73180E-05 -5.71975E-02       4 -0.12213       8.96238E-05 -9.77638E-02
        5 -0.11376       9.26640E-06 -0.14839           6 -0.10634      -1.46101E-04 -0.18130
```

```
           total equivalent nodal forces (distributed plus point loads)


        5  -44.229            0.           0.



           reaction forces at fixed boundary conditions, residual load correction elsewhere


       17   44.226       -958.77       4.0092

                          summary of externally applied loads

 -0.44229E+02    0.00000E+00    0.00000E+00

                          summary of reaction/residual forces

  0.44229E+02   -0.95811E+03    0.40218E+01

                          e n d   o f   i n c r e m e n t  21
                          formatted post data at increment  21.   0   on tape 19
                           time =       6.14


                          s t a r t   o f   i n c r e m e n t  22
                          load increments associated with each degree of freedom
                          summed over the whole model

                          distributed loads
     0.000E+00 0.000E+00 0.000E+00


                          point loads
    -2.363E+00 0.000E+00 0.000E+00

                          start of assembly
                           time =       6.14

                          start of matrix solution
                           time =       6.19

                          singularity ratio    1.1612E-02

                          end of matrix solution
                           time =       6.19
```

Automatic load begins to increase again at a faster rate

```
                        5.14 percent of total load added in this cycle

                     maximum residual force  at node   16 degree of freedom    2 is equal to    0.814E+02
                     maximum reaction force  at node   17 degree of freedom    2 is equal to    0.953E+03
                     convergence ratio                                                          0.854E-01

                      failure to converge to tolerance

                      increment will be recycled

                      load increments associated with each degree of freedom
                      summed over the whole model

                      distributed loads
   0.000E+00 0.000E+00 0.000E+00



                      point loads
  -3.082E+00 0.000E+00 0.000E+00

                      start of assembly
                       time =       6.24

                      start of matrix solution
                       time =       6.28

                      singularity ratio    1.7390E-02

                      end of matrix solution
                       time =       6.29

                        1.82 percent of total load added in this cycle

                      maximum residual force  at node   16 degree of freedom    2 is equal to    0.379E+00
                      maximum reaction force  at node   17 degree of freedom    2 is equal to    0.940E+03
                      convergence ratio                                                          0.403E-03

MARC                 output for increment   22.    post-buckling of a spherical cap


                       automatic stepping has reached    80.6749 percent  of total load


                       n o d a l   p o i n t   d a t a

                       t o t a l   d i s p l a c e m e n t s


      1 -0.13620           0.          0.                    2 -0.13545       4.58197E-05 -2.68290E-02
      3 -0.13313       8.60881E-05 -5.59144E-02              4 -0.12800       1.07836E-04 -9.59330E-02
      5 -0.11975       4.13212E-05 -0.14687                  6 -0.11239      -1.04308E-04 -0.18008
```

```
          total equivalent nodal forces (distributed plus point loads)


     5  -48.405           0.           0.




          reaction forces at fixed boundary conditions, residual load correction elsewhere


    17   48.401       -939.70       7.6207

                        summary of externally applied loads

-0.48405E+02    0.00000E+00    0.00000E+00

                        summary of reaction/residual forces

 0.48405E+02   -0.93915E+03    0.76339E+01

                        e n d   o f   i n c r e m e n t  22
                        formatted post data at increment  22.   0   on tape 19
                         time =       6.38


                        s t a r t   o f   i n c r e m e n t  23
                        load increments associated with each degree of freedom
                        summed over the whole model

                        distributed loads
      0.000E+00 0.000E+00 0.000E+00


                        point loads
     -4.176E+00 0.000E+00 0.000E+00

                        start of assembly
                         time =       6.38

                        start of matrix solution
                         time =       6.43

                        singularity ratio    1.3891E-02

                        end of matrix solution
                         time =       6.44

                          9.09 percent of total load added in this cycle
```

```
                    maximum residual force  at node   16 degree of freedom    2 is equal to    0.106E+03
                    maximum reaction force  at node   17 degree of freedom    2 is equal to    0.902E+03
                    convergence ratio                                                          0.118E+00

                     failure to converge to tolerance

                     increment will be recycled

                     load increments associated with each degree of freedom
                     summed over the whole model

                     distributed loads
  0.000E+00 0.000E+00 0.000E+00


                     point loads
 -5.457E+00 0.000E+00 0.000E+00

                     start of assembly
                      time =        6.48

                     start of matrix solution
                      time =        6.52

                     singularity ratio    1.9134E-02

                     end of matrix solution
                      time =        6.53

                        3.06 percent of total load added in this cycle

                    maximum residual force  at node   16 degree of freedom    2 is equal to    0.399E+00
                    maximum reaction force  at node   17 degree of freedom    2 is equal to    0.885E+03
                    convergence ratio                                                          0.451E-03

MARC                output for increment  23.    post-buckling of a spherical cap


                     automatic stepping has reached    92.8293 percent  of total load



                     n o d a l   p o i n t   d a t a

                     t o t a l   d i s p l a c e m e n t s


      1 -0.14182        0.           0.              2 -0.14110      5.14372E-05 -2.60614E-02
      3 -0.13884     9.85501E-05 -5.45281E-02        4 -0.13383      1.31996E-04 -9.40865E-02
      5 -0.12569     8.03353E-05 -0.14585            6 -0.11836     -5.72172E-05 -0.17955
```

```
          total equivalent nodal forces (distributed plus point loads)


     5  -55.698          0.          0.




          reaction forces at fixed boundary conditions, residual load correction elsewhere


    17   55.693     -885.07      12.080

                    summary of externally applied loads

-0.55698E+02    0.00000E+00    0.00000E+00

                    summary of reaction/residual forces

 0.55698E+02   -0.88464E+03    0.12094E+02

                    e n d   o f   i n c r e m e n t  23
                    formatted post data at increment  23.   0   on tape 19
                     time =       6.62


                    s t a r t   o f   i n c r e m e n t  24
                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
    0.000E+00 0.000E+00 0.000E+00



                    point loads
   -7.293E+00 0.000E+00 0.000E+00

                    start of assembly
                     time =       6.62

                    start of matrix solution
                     time =       6.67

                    singularity ratio    1.6522E-02

                    end of matrix solution
                     time =       6.68

                       7.17 percent of total load added in this cycle
```

```
                     maximum residual force  at node   16 degree of freedom    2 is equal to    0.283E+02
                     maximum reaction force  at node   17 degree of freedom    2 is equal to    0.849E+03
                     convergence ratio                                                          0.333E-01

                      failure to converge to tolerance

                      increment will be recycled

                      load increments associated with each degree of freedom
                      summed over the whole model

                      distributed loads
  0.000E+00 0.000E+00 0.000E+00


                      point loads
 -4.302E+00 0.000E+00 0.000E+00

                      start of assembly
                       time =        6.72

                      start of matrix solution
                       time =        6.76

                      singularity ratio    1.8329E-02

                      end of matrix solution
                       time =        6.77

                        0.00 percent of total load added in this cycle

                     maximum residual force  at node   12 degree of freedom    2 is equal to    0.174E+00
                     maximum reaction force  at node   17 degree of freedom    2 is equal to    0.850E+03
                     convergence ratio                                                          0.205E-03

MARC                output for increment  24.    post-buckling of a spherical cap


                      automatic stepping has reached  100.0000 percent  of total load
```

Load reaches total requested

```
                      n o d a l   p o i n t   d a t a

                      t o t a l   d i s p l a c e m e n t s


       1 -0.14415           0.           0.                 2 -0.14344       5.45369E-05 -2.57138E-02
       3 -0.14120       1.05279E-04 -5.39175E-02            4 -0.13624       1.44525E-04 -9.33219E-02
       5 -0.12814       9.94511E-05 -0.14564                6 -0.12081      -3.55167E-05 -0.17966
```

```
          total equivalent nodal forces (distributed plus point loads)


     5  -60.000          0.          0.



          reaction forces at fixed boundary conditions, residual load correction elsewhere


    17   59.998      -849.86      14.202

                    summary of externally applied loads

-0.60000E+02   0.00000E+00   0.00000E+00

                    summary of reaction/residual forces

 0.60000E+02  -0.84942E+03   0.14216E+02

                    e n d   o f   i n c r e m e n t  24
                    formatted post data at increment  24.   0   on tape 19
                     time =       6.86

                    *** end of input deck - job ends



  elapsed time information
 user time   =     0.585470E+01 seconds
 system time =     0.111510E+01 seconds
 total time  =     0.696980E+01 seconds



                         marc exit number 3004
```

Results

The highly nonlinear load-deflection behavior of the spherical cap is summarized in the following table and the four figures.

Load Displacement

Increment	% of Total Load	Node 5 Load (lb.)	Node 1 Z-deflection (in.)
1	17.34	10.401	.00275
2	32.08	19.248	.00622
3	42.41	25.448	.01062
4	47.23	28.339	.01618
5	47.35	28.414	.02315
6	45.85	27.512	.03035
7	44.88	26.933	.93769
8	44.99	26.997	.04502
9	46.07	27.642	.05226
10	47.87	28.722	.05939
11	50.16	31.637	.06642
12	52.73	31.637	.07334
13	55.38	33.232	.08167
14	57.96	34.779	.08689
15	60.33	36.198	.09351
16	62.39	37.437	.10003
17	64.16	38.494	.10642
18	65.73	39.439	.11267
19	67.42	40.451	.11877
20	69.77	41.863	.12472
21	73.71	44.229	.13052
22	80.67	48.405	.13620
23	92.83	55.398	.14182
24	100.00	60.00	.14415

Figure 7.2 in a progressive deformed geometry plot of the model for increments 6, 10, 14, 20, and 24. Figure 7.3 shows how the applied load, due to the automatic load stepping, changes from increment to increment – and actually decreases in magnitude in increments 6 and 7 before increasing again. Figure 7.4 plots Z-displacements for nodes 1, 3, 5, 7, 9, 11, and 13 from increment to increment.

Finally, Figure 7.5 is the nonlinear load-deflection curve we are after – showing the node 5 load variation versus the apex Z-deflection at node 1. Also shown is Zienkiewicz's finite element solution (reference Zienkiewicz, O.C., *The Finite Element Method*, 3rd Ed., McGraw-Hill, 1977, pp. 520-521). The correlation is excellent for such a coarse mesh. Notice that the spherical cap possesses little post-buckling strength after the first peak in the load-deflection curve. A very small change in the load produces a large deflection! This type of sensitive structural behavior is typical in such geometrically nonlinear problems. This example illustrates the ability of MARC's automatic load incrementation algorithm to "look ahead," and downsize the load step when the circumstances warrant it. (This variation in the load step size from increment to increment is also shown graphically in Figure 7.3.)

If we check to see how the singularity ratio changes from increment to increment (as we did in Example 5), we see that the numerical value of this ratio remains in an acceptable range (10^{-2} or 10^{-3}) in each increment and we never encounter any numerical problems.

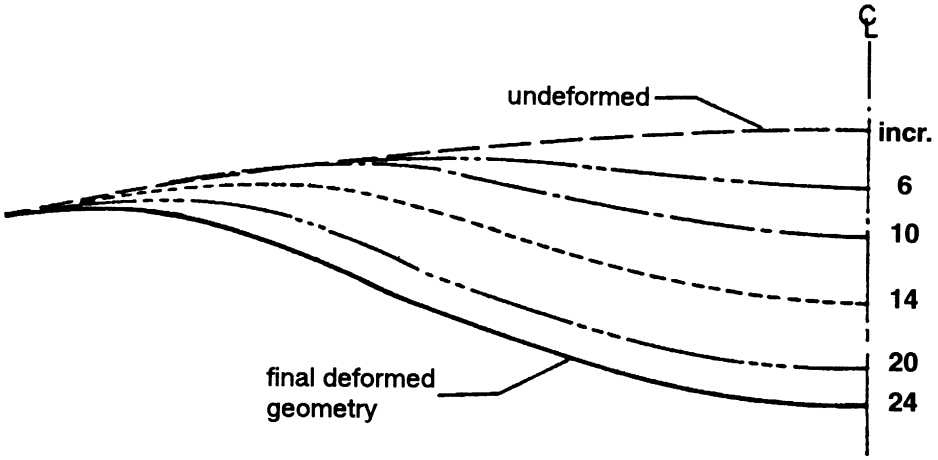


Figure 7.2 **Cap Deformed Geometry From Increment 6 to 24**

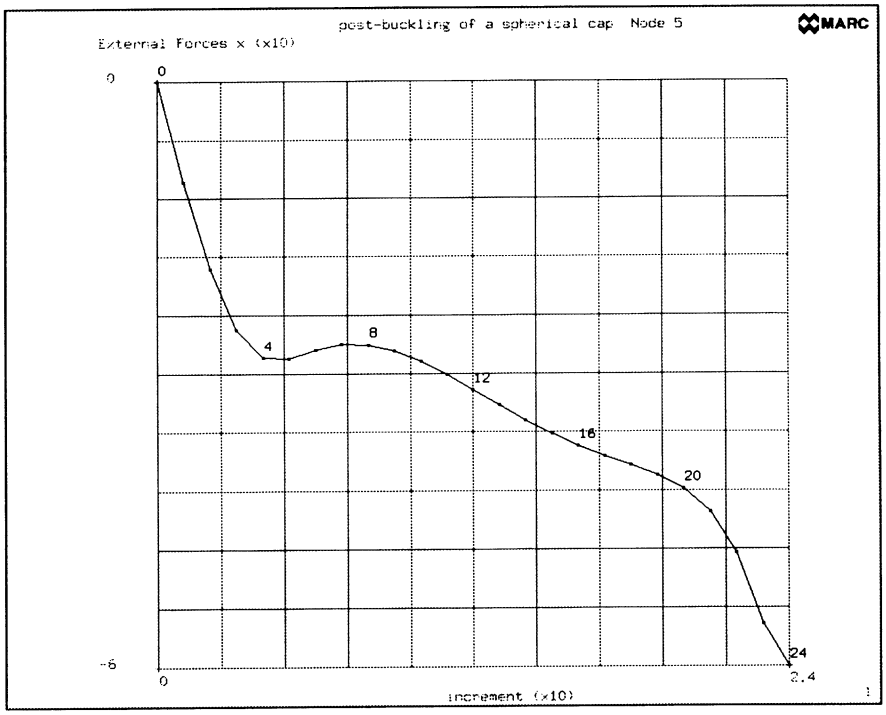


Figure 7.3 Node 5 Z-load History

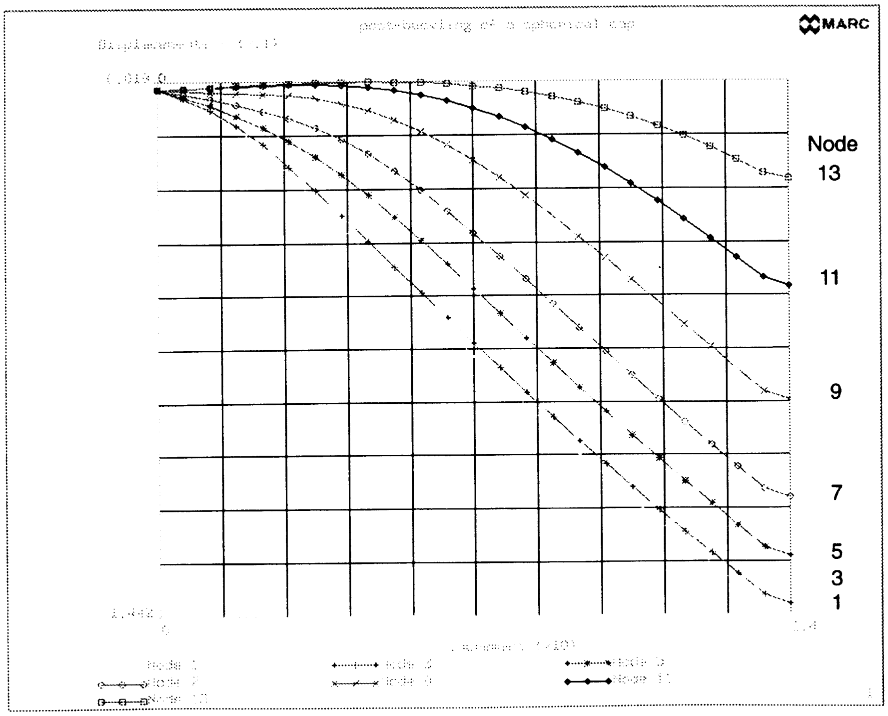


Figure 7.4 Z-displacement History – Nodes 1, 3, 5, 7, 9, 11, 13

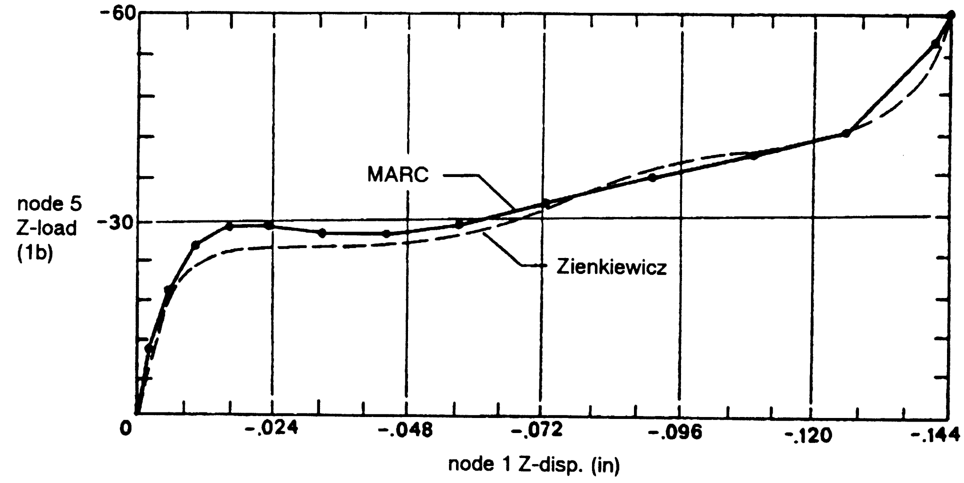


Figure 7.5 **Load-deflection Curve for Spherical Cap**

Exercises

Try Example 7 with the same load of 60 lb. located first at the apex – node 1 (since this is the axis of symmetry, you must halve the load), and then at node 9, and compare the load-deflection behavior of these two cases with the present case. Also, rerun Example 7 using a larger convergence tolerance, say 0.05, and contrast the accuracy of the analysis.

CHAPTER 4: Heat Transfer and Thermal Stress Analyses

This chapter departs from the conventional linear/nonlinear mechanical stress analysis problems you've seen so far in this Primer, and introduces you to a transient heat conduction analysis (Example 8), followed by a thermal stress analysis (Example 9) of the same model. The 2-D FE model used is the cross section of a square pipe with a circular channel in the center (which has the same geometry as the square plate with hole in Example 1).

Such a two-step heat transfer-thermal stress analysis is one of the most common types of engineering analysis. In MARC, the same solver is used to solve the field equations for both the heat transfer problem and the structural analysis problem. The major distinctions between these two classes of problems are:

- The only nodal DOF in a FE heat transfer analysis is the temperature, which is a scalar, as distinguished from the usual displacement vector comprised of three translations and three rotations found in a typical structural analysis problem.
- The common thermal properties needed in a heat transfer analysis consists of: thermal conductivity, specific heat, and mass density. In a thermal stress analysis, in addition to the usual Young's modulus and Poisson's ratio, the coefficient of thermal expansion is needed. These properties may or may not be temperature-dependent. (Needless to say, all these properties must have consistent units, either in English or SI units.)
- Thermal boundary conditions are different from structural ones, and are often trickier to handle or prescribe. These include, for instance: insulated boundary conditions; convective boundary conditions (requiring you to input a film coefficient); radiative boundary conditions; boundary layer effect; uniform or non-uniform fluxes (per unit area or unit volume); constant or time-varying boundary temperatures; steady-state or transient conditions; initial conditions; and temperature dependence of boundary conditions.

For further details on heat transfer capabilities in MARC, see Volume A, Section 5.5. For general heat transfer discussions, see any one of the many fine texts on the subject, for example: J. P. Holman. *Heat Transfer*, 6th ed., McGraw-Hill, 1986. For a description of FE implementation of heat transfer analysis, see: K. H. Huebner and E. A. Thornton, *The Finite Element Method for Engineers*, 2nd ed., John Wiley & Sons, 1982, Chapter 10.

Example 8

Transient Heat Conduction Analysis of a Square Pipe with Circular Channel

The purpose of this example is to demonstrate a typical heat transfer analysis using MARC – in this case, a transient heat conduction problem. This example illustrates: use of MARC Element 39; input of thermal properties (one of which is temperature dependent); prescribing of convective boundary conditions; and transient thermal analysis (with the use of automatic time stepping). The objective of the analysis is to obtain equilibrium nodal temperatures in the square pipe, for subsequent use in the Example 9 thermal stress analysis.

Sketch

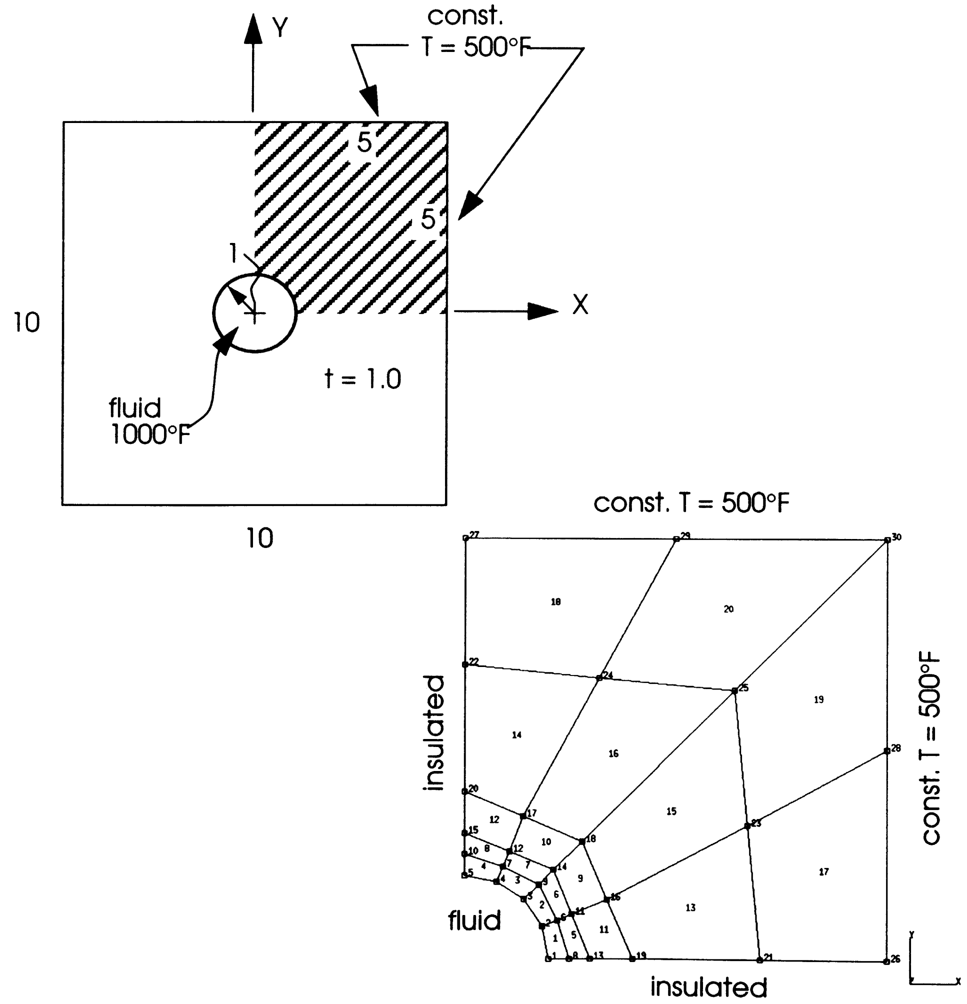


Figure 8.1 Square Plate with Circular Channel

Model

The mesh of the one-quarter FE model is identical to that used in Example 1. The difference is that here we are using a planar heat transfer element (with unit thickness), along with thermal properties and boundary conditions.

MARC Element 39 (see MARC Volume B) is a 4-node, isoparametric, quadrilateral heat transfer element. It is the heat transfer counterpart to stress Elements 3, 11, and 19. Each node is defined by global coordinates X and Y, and has one DOF – the temperature.

Properties

Three thermal properties are given; of the three, only the specific heat varies with temperature. The thermal conductivity is 0.42117×10^{-5} Btu/sec-in.-°F. The specific heat is 0.3523×10^{-2} Btu/lb-°F at 500°F, and decreases to 0.3523×10^{-3} at 1000°F and remains constant at higher temperatures. The mass density is 0.7254×10^{-3} lb/cu.in.

Boundary Conditions

No flux is transmitted across the planes of symmetry ($X = 0, Y = 0$). This condition is identical to an insulated boundary. The prescription of this boundary condition is by stipulating the flux to be zero at the nodes; as this is the default, no input is required. The initial condition for the pipe is a uniform temperature of 500°F throughout. At the two outer edges ($X = 5, Y = 5$), a constant temperature of 500°F is prescribed.

The fluid fills the circular channel, and convective boundary conditions are assumed at the channel edge. The fluid temperature is 1000°F, and the film coefficient is 0.46875×10^{-5} Btu/sec-sq.in.-°F. Therefore, this boundary condition can be expressed as:

$$q = 0.46875 \times 10^{-5} [T_S - 1000]$$

Special Features

As in Example 1, the OPTIMIZE option is used to optimize the node numbering in the FE mesh, in order to minimize the bandwidth and solution time.

The total *transient time* in the analysis is assumed to be 5.0 sec. An initial time step of 0.1 sec. is chosen for the problem. Automatic time stepping will be used to insure the required accuracy at minimal costs.

Input

A complete input listing is included.

PARAMETER Section

The "SIZING" line sets 100,000 words as the workspace. The "20" on the line means 20 elements are in the model; the "30" means 30 nodes.

The "ELEMENTS" line tells MARC that Element 39 will be used for the heat transfer analysis.

The "HEAT" line flags heat transfer analysis in MARC.

The "LUMP" line requests that the specific heat matrix be made into a diagonal (lumped) matrix. This "lumping" is beneficial when lower-order elements are used, and eliminates the "overshoot" or "undershoot" phenomena often observed in heat transfer analysis.

The "END" line terminates the PARAMETER section.

MODEL DEFINITION Section

The MODEL DEFINITION options in this example consist of:

a. FE mesh topology – including the CONNECTIVITY, COORDINATES, and DEFINE blocks
b. Geometric properties
c. Thermal properties
d. Initial and fixed temperatures
e. Bandwidth minimization
f. Convergence controls for the heat transfer analysis
g. Output controls

FE Mesh Topology

The FE mesh is exactly the same as that of Example 1. The only differences in the CONNECTIVITY block are: in this example, we used a header line which informs MARC there are 20 elements; and MARC Element 39 is named in the second field of the element definition lines (instead of Element 3 in Example 1). The COORDINATES block is the same in both examples, except for the header line this time– which tells MARC there are two coordinate directions per node and there are 30 nodes to be defined.

Next follow three DEFINE blocks to define two node sets and one element set. Along the outer edges, nodes 26 through 30 are placed in a node set named OUTEDGE. All 30 nodes in the model are placed in a node set named ALLN. Then, next to the channel, elements 1 to 4 are placed in an element set named FLUID, with the name indicating that these elements are the ones that come into direct contact with the fluid in the channel.

Geometric Properties

The GEOMETRY block defines the geometric properties needed for the elements. The blank line means we do not need to count the number of sets. The "1.0," line tells MARC the element thickness is 1.0 in. Then, the "1 TO 20" line ends this block, and means that thickness applies to elements 1 to 20 (in other words, all elements in the model).

Thermal Properties

The ISOTROPIC option for heat transfer analysis is, of course, different from that for stress analysis. The "1" line means material identifier number 1. This would be used to cross reference temperature-dependent material data. The ".42117E-5, .3523E-2, .7254E-3" line defines the thermal conductivity, specific heat, and mass density, respectively. The "1 TO 20" line indicates these properties apply to all 20 elements in the model.

The TEMPERATURE EFFECTS option for heat transfer analysis allows you to define variations in thermal conductivity, specific heat, or electrical resistance with temperature. Inserting the word DATA on the first line after TEMPERATURE EFFECTS means we are choosing the option to input the data points (rather than the slopes) directly. The "0,3," line says that we are not defining a variation in conductivity versus temperature, but instead, a variation in specific heat as defined by three data points. The next three lines show the values of specific heat at 500°F, 1000°F, and 1100°F. Note that the first value must agree with that given through the ISOTROPIC option.

In addition, we need to specify the film coefficient for convective heat transfer calculations. The FILMS option permits you to specify film coefficients and associated sink temperatures. After the usual blank line, the "0,.46875E-5,1000.," line means: the "0" refers to a uniform flux per unit thickness through the 1-2 face of the element (for MARC Element 39) – that is, the edge closest to the channel; the second number is the value of the film coefficient; and the "1000," is the reference sink temperature. The "FLUID" line ends this block, and means this film coefficient applies to the element set named FLUID; i.e., elements 1 to 4 adjacent to the channel.

Initial and Fixed Temperatures

Prescribed temperatures for this analysis are input using two thermal analysis options: INITIAL TEMP and FIXED TEMP. The INITIAL TEMP option provided initial temperatures for a heat transfer analysis. (It must precede the FIXED TEMP option, if one exists.) After the usual blank line, the "500.," line says the initial temperature is 500.0°F. (The temperature units are in degrees Fahrenheit since the thermal properties are given per degree Fahrenheit.) The "ALLN" line tells MARC that this initial temperature applies to node set ALLN, or all 30 nodes. In a transient heat transfer problem, the initial temperatures should be defined. This is not necessary for a steady-state analysis, but is recommended, especially if temperature-dependent properties exist in the model.

Then, the FIXED TEMP option described the fixed temperature each node must take during the *first and subsequent* increments (unless it is modified using the TEMP CHANGE option). After the blank line, the "500.," line denotes the prescribed temperature (in this case, in degrees Fahrenheit). The subsequent "OUTEDGE" line means these fixed temperatures are applied only to node set OUTEDGE; i.e., nodes 26 to 30.

Bandwidth Minimization

The OPTIMIZE block switches on the bandwidth minimization procedures in MARC. Since we did not select a scheme on the "OPTIMIZE" line, the default Cuthill-McKee scheme will be used. The "5," line says we want MARC to try a maximum of five different node numbering schemes, then choose the one which results in the lowest bandwidth.

Convergence Controls

The CONTROL option for heat transfer allows you to input parameters governing the convergence and accuracy in a heat transfer analysis. The "20,10" in the second line means we want the maximum number of load steps to be 20, and the maximum number of recycles within an increment due to temperature-dependent material properties to be 10. The "50.,50.," line means we want 50.0°F as the maximum nodal temperature change allowed per time step, and 50.0°F as the maximum nodal temperature change allowed before properties are reevaluated and matrices are reassembled.

NOTE

The parameter in the first field of this line is used to control the time step size. The parameter in the second field is important if temperature dependent properties are required for the analysis. If the material properties are strongly temperature dependent, or if there are latent heat effects, this value should be a small enough number to insure sufficient accuracy. If there are no temperature dependent materials, you can set this number to a large value to reduce the computational costs without any loss in accuracy.

Output Controls

The POST option informs MARC that a post-processor file is to be written for later post-processing by Mentat II. This file will also be read by MARC in the subsequent thermal stress analysis to define the temperature distribution. The "1," line means one element variable is to be written in the file. The "9," line denotes the post-code number, and refers to total temperature.

The "END OPTION" line terminates the MODEL DEFINITION section.

HISTORY DEFINITION Section

The only HISTORY DEFINITION option needed in this example is the TRANSIENT block. Automatic time-stepping will be used by default. This option controls the transient heat transfer analysis. The ".1,5.0," data line means we want the initial time step size to be 0.1 sec., and the time period for the analysis is 5.0 seconds. (Defaults will be assumed for the remaining fields on this data line.)

The “CONTINUE” line ends the HISTORY DEFINITION section as well as the input file.

NOTE

In a heat transfer analysis, if prescribed fluxes of temperatures are changed, enter TOTAL values as opposed to INCREMENTAL values used in stress analysis.

Output

The selective output included with this example consists of: the input echo; the program sizing and options summary table; and printouts for increments 1 and 16. At the end of each increment, MARC informs you that the (binary) post data have been written to Unit 16 for later post-processing. The temperatures will be later used in Example 9 for the thermal stress analysis.

Since this is a thermal analysis instead of a stress analysis, MARC’s information messages tell you how accurately the nodal temperatures are being calculated each time increment, as compared to the input parameters in the CONTROL option. Notice in increment 1, MARC went through four recycles, reducing the time step each time, before obtaining a maximum nodal temperature change (+47.1°F) at node 4 which falls within the allowable temperature change (50.0) in the CONTROL option. At the end of increment 1, the time step used in the fourth recycle was 0.00965 seconds.

```
                                    i n p u t   d a t a

    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
TITLE, HEAT TRANSFER FOR CIRCULAR CHANNEL IN SQUARE PIPE
SIZING        100000   20   30
ELEMENTS     39
HEAT_____________________________Turn on Heat Transfer Option
LUMP
END
CONNECTIVITY
   20    0    0
    1   39    2    1    8    6
    2   39    3    2    6    9
    3   39    4    3    9    7
    4   39    5    4    7   10
    5   39    6    8   13   11
    6   39    9    6   11   14
    7   39    7    9   14   12
    8   39   10    7   12   15
    9   39   14   11   16   18
   10   39   12   14   18   17
   11   39   11   13   19   16
   12   39   15   12   17   20
   13   39   16   19   21   23
   14   39   20   17   24   22
   15   39   18   16   23   25
   16   39   17   18   25   24
   17   39   23   21   26   28
   18   39   22   24   29   27
   19   39   25   23   28   30
   20   39   24   25   30   29
COORDINATES
    2   30    0    0
    1   1.00000   0.00000
    2   0.92381   0.38247
    3   0.70700   0.70700
    4   0.38247   0.92381
    5-0.25219-6   1.00000
    6   1.10190   0.45623
    7   0.45623   1.10190
    8   1.25000   0.00000
    9   0.88350   0.88350
   10-0.32459-6   1.25000
   11   1.28000   0.53000
   12   0.53000   1.28000
   13   1.50000   0.00000
   14   1.06000   1.06000
   15-0.31869-6   1.50000
--------------------------------------------------------------------------------
    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
   16   1.70000   0.70000
   17   0.70000   1.70000
   18   1.40000   1.40000
   19   2.00000   0.00000
   20-0.50439-6   2.00000
   21   3.50000   0.00000
   22-0.10206-5   3.50000
   23   3.35000   1.60000
   24   1.60000   3.35000
   25   3.20000   3.20000
   26   5.00000   0.00000
   27-0.15777-5   5.00000
   28   5.00000   2.50000
   29   2.50000   5.00000
   30   5.00000   5.00000
DEFINE    NODE      SET       OUTEDGE
   26 TO     30
DEFINE    NODE      SET       ALLN
    1 TO     30
DEFINE    ELEMENT   SET       FLUID
    1 TO      4
GEOMETRY

1.0,
    1 TO     20
ISOTROPIC
                                    Define Conductivity, Specific Heat,
    1                               and Mass Density
.42117E-5,.3523E-2,.7254E-3
    1 TO     20
TEMPERATURE EFFECTS DATA
0,3,
.3523E-2,500.,
.3523E-3,1000.,                     Define Temperature Dependent Specific Heat
.3523E-3,1100.,
FILMS

0,.46875E-5,1000.,                  Define Convection Coefficient and Sink
FLUID                               Temperature of the Fluid
INITIAL TEMP

500.,                               Initialize Temperatures
ALLN
FIXED TEMP
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
        5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
   --------------------------------------------------------------------------------
card    90
             500.,
             OUTEDGE
             OPTIMIZE
             5,
card    95   CONTROL
```

Specify the tolerances that govern time step selection

```
             20,10
             50.,50.,
             POST
             1,
card   100   9,
             END OPTION
             TRANSIENT
             .1,5.0,
```

Specify initial time step guess and total time period to be covered.

```
             CONTINUE
   --------------------------------------------------------------------------------
        5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
   --------------------------------------------------------------------------------
   --------------------------------------------------------------------------------
```

```
 **************************************************
 **************************************************

  program sizing and options requested as follows

  element type requested*************************   39
  number of elements in mesh*********************   20
  number of nodes in mesh************************   30
  max number of elements in any dist load list***    0
  maximum number of boundary conditions**********    5
  load correction flagged or set*****************
  stresses stored at all integration points******
  tape no.for input of coordinates + connectivity    5
  no.of different materials    2 max.no of slopes    5
  heat transfer analysis,  extrapolation flag, **    1
  number of elements with film coefficient b.c.**    4
  maximum number of distributed flux lists*******    3
  lumped mass or specific heat matrix will be used
  maximum elements variables per point on post tp   33
  number of points on shell section *************   11
  option for terminal debug**********************
  new style input format will be used************
  maximum number of set names is*****************   10
  number of processors used *********************    1
  vector length used ****************************    1


  end of parameters and sizing
 **************************************************
 **************************************************
```

```
          element type   39


4-node heat transfer planar element

1 degree of freedom per node - temperature


workspace needed for input and stiffness assembly     13719


internal core allocation parameters
max. degrees of freedom per node  1
max. number of coordinates per node  2
max. gradients per int. point  2
max. nodes per element   4
max. invariants per int. point  1



flag for element storage (ielsto)  0
elements in core, words per element (nelsto)       412
                  total space required            8240
vectors in core, total space required       533


words per track on disk set to 4096


internal element variables



internal element number  1  library code type 39
number of nodes  4
number of gradient components at each int. point  2
integration points for conductivity  4
integration point for print-out  5
integration points for surface b.c.s  2
no local rotation flag    1
generalized variable flag    0



residual load correction is switched off
      •                                         •
      •                                         •
      •                                         •
      •                                         •
```

```
connectivity
----------

meshr1,iprnt
    5      0
elem no., type,        nodes
    1      39          2    1    8    6
    2      39          3    2    6    9
    3      39          4    3    9    7
    4      39          5    4    7   10
    5      39          6    8   13   11
    6      39          9    6   11   14
    7      39          7    9   14   12
    8      39         10    7   12   15
    9      39         14   11   16   18
   10      39         12   14   18   17
   11      39         11   13   19   16
   12      39         15   12   17   20
   13      39         16   19   21   23
   14      39         20   17   24   22
   15      39         18   16   23   25
   16      39         17   18   25   24
   17      39         23   21   26   28
   18      39         22   24   29   27
   19      39         25   23   28   30
   20      39         24   25   30   29

coordinates
----------

ncrd1 ,meshr1,iprnt
    2      5      0
 node      coordinates
    1  1.0000          0.
    2 0.92381     0.38247
    3 0.70700     0.70700
    4 0.38247     0.92381
    5-0.25219E-06  1.0000
    6  1.1019     0.45623
    7 0.45623      1.1019
    8  1.2500          0.
    9 0.88350     0.88350
   10-0.32459E-06  1.2500
   11  1.2800     0.53000
   12 0.53000      1.2800
   13  1.5000          0.
   14  1.0600      1.0600
   15-0.31869E-06  1.5000
   16  1.7000     0.70000
   17 0.70000      1.7000
   18  1.4000      1.4000
```

```
   19  2.0000          0.
   20-0.50439E-06  2.0000
   21  3.5000          0.
   22-0.10206E-05  3.5000
   23  3.3500      1.6000
   24  1.6000      3.3500
   25  3.2000      3.2000
   26  5.0000          0.
   27-0.15777E-05  5.0000
   28  5.0000      2.5000
   29  2.5000      5.0000
   30  5.0000      5.0000

define    node      set       outedge
----------

from node    26 to node    30 by     1

define    node      set       alln
----------

from node     1 to node    30 by     1

define    element   set       fluid
----------

from element     1 to element     4 by     1

geometry
----------

  egeom1     egeom2     egeom3     egeom4     egeom5     egeom6
  0.100E+01  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
from element     1 to element    20 by     1

isotropic
----------

isotropic material  material id =     1

conduct       spht       rhoht
  0.421E-05  0.352E-02  0.725E-03
from element     1 to element    20 by     1
temperature effects data
----------

*** warning - material id unspecified. matid =     1 assumed.
material id =     1
number of data sets  for spec heat  =     3
spec heat  curve
value          temperature
```

```
0.35230E-02  0.50000E+03
0.35230E-03  0.10000E+04
0.35230E-03  0.11000E+04


films
----------


read from unit     5
face number =    0     film coefficient =       4.688E-06     sink temp. =       1.000E+03
film coefficient index =    0     sink temp. index =    0
name of element set is fluid


initial temp
----------


number of series used for initial temperatures is     0
read from unit     5
initial value  0.5000000E+03
name of node set is alln


fixed temp
----------


fixed temperature=  0.500E+03
name of node set is outedge




fixed boundary condition summary.
total fixed degrees of freedom read so far =    5

 b.c.    node   degree of  magnitude              b.c.    node   degree of  magnitude
number            freedom                        number            freedom



    1      26         1    5.000E+02                2      27         1    5.000E+02
    3      28         1    5.000E+02                4      29         1    5.000E+02
    5      30         1    5.000E+02

optimize
----------



cuthill-mckee algorithm

control
----------

max.     max.     min.
incs   recycles recycles
   21      10       0
```

```
 maximum nodal temperature change per time step = 0.50000E+02


 maximum nodal temperature change before reassembly = 0.50000E+02


 post
 ----------



 *** note - format of post code cards has changed.
            in k4, enter code in first field and  layer number in second field


 elem vars,post tape,prev tape,  type  , conn fl ,post tape, prev tape,  repost ,frequency, k2post
         1      16        17       0         1        0         0         0        1         0
 element variables appear on post-processor tape 16 in following order
 post variable   1 is post code    9 =

***maximum record length on binary post file=   30     approximate no. of words per increment on file=      96

 end option
 ----------




 transient
 ----------



    time      time     maximum   assembly  max iter
 increment   period     steps    interval  mcreep
  1.000E-01 5.000E+00     50         0       5                   Define Time Step
 continue
 ----------




 auto control specified for time of 0.500E+01



                   s t a r t   o f   i n c r e m e n t   1




 fluxes summed over the whole model

 from distributed fluxes
0.000E+00
```

```
  concentrated fluxes
0.000E+00

                   start of assembly
                    time =       0.52



                   start of matrix solution
                    time =       0.59



                   singularity ratio    7.6872E-01


                   end of matrix solution
                    time =       0.60


                   maximum nodal temperature change is  0.132E+03 at node     4

                   this is 0.265E+03  percent of change allowed on control option
```

Delta Time is too large - reduce time step

```
                   step will be recycled with time increment of 0.302E-01

  fluxes summed over the whole model

  from distributed fluxes
0.000E+00


  concentrated fluxes
0.000E+00

                   start of assembly
                    time =       0.64
```

NOTE

The operator matrix must be reassembled for each time step change.

```
                   start of matrix solution
                    time =       0.70



                   singularity ratio    8.7218E-01


                   end of matrix solution
                    time =       0.71
                   maximum nodal temperature change is  0.853E+02 at node     4
```

```
                      this is 0.171E+03  percent of change allowed on control option        Try again


                      step will be recycled with time increment of 0.142E-01





  fluxes summed over the whole model

  from distributed fluxes
0.000E+00



  concentrated fluxes
0.000E+00

                      start of assembly
                       time =       0.74



                      start of matrix solution
                       time =       0.81



                      singularity ratio    9.3263E-01


                      end of matrix solution
                       time =       0.82



                      maximum nodal temperature change is  0.588E+02 at node    4

                      this is 0.118E+03  percent of change allowed on control option        Try again


                      step will be recycled with time increment of 0.965E-02




  fluxes summed over the whole model

  from distributed fluxes
```

```
      0.000E+00


        concentrated fluxes
      0.000E+00

                            start of assembly
                             time =       0.85




                            start of matrix solution
                             time =       0.92




                            singularity ratio    9.5582E-01



                            end of matrix solution
                             time =       0.92




                            maximum nodal temperature change is  0.471E+02 at node    4


                            this is 0.941E+02  percent of change allowed on control option



                            maximum nodal temperature change since last property evaluation is    0.1174E+02


                            this is 0.23E+02  percent of change allowed on control option
  MARC          output for increment    1.    heat transfer for circular channel in square pipe





                            transient time has reached  0.965E-02  of time period  0.500E+01



                            total transient time = 9.64531E-03



    elem., point, temp.      elem., point, temp.      elem., point, temp.      elem., point, temp.      elem.,
 point, temp.



     1    1  538.5          1    2  536.7          1    3  517.2          1    4  515.7          2    1  536.7
     2    2  538.6          2    3  515.7          2    4  517.2          3    1  538.6          3    2  536.7
     3    3  517.2          3    4  515.7          4    1  536.7          4    2  538.5          4    3  515.7
     4    4  517.2          5    1  507.7          5    2  506.6          5    3  503.1          5    4  502.6
```

```
  6   1  506.6          6   2  507.7          6   3  502.6          6   4  503.1          7   1  507.7
  7   2  506.6          7   3  503.1          7   4  502.6          8   1  506.6          8   2  507.7
  8   3  502.6          8   4  503.1          9   1  501.0          9   2  501.1          9   3  500.3
  9   4  500.3         10   1  501.1         10   2  501.0         10   3  500.3         10   4  500.3
 11   1  501.1         11   2  500.9         11   3  500.3         11   4  500.3         12   1  500.9
 12   2  501.1         12   3  500.3         12   4  500.3         13   1  500.0         13   2  500.0
 13   3  500.0         13   4  500.0         14   1  500.0         14   2  500.0         14   3  500.0
 14   4  500.0         15   1  500.0         15   2  500.0         15   3  500.0         15   4  500.0
 16   1  500.0         16   2  500.0         16   3  500.0         16   4  500.0         17   1  500.0
 17   2  500.0         17   3  500.0         17   4  500.0         18   1  500.0         18   2  500.0
 18   3  500.0         18   4  500.0         19   1  500.0         19   2  500.0         19   3  500.0
 19   4  500.0         20   1  500.0         20   2  500.0         20   3  500.0         20   4  500.0


                         n o d a l   p o i n t   d a t a


                         t o t a l   n o d a l   t e m p e r a t u r e s


     1   543.68          2   547.07          3   543.71          4   547.07          5   543.68
     6   509.92          7   509.92          8   507.53          9   507.56         10   507.53
    11   501.52         12   501.52         13   501.10         14   501.12         15   501.10
    16   500.03         17   500.03         18   500.02         19   500.03         20   500.03
    21   500.00         22   500.00         23   500.00         24   500.00         25   500.00
    26   500.00         27   500.00         28   500.00         29   500.00         30   500.00

                         e n d   o f   i n c r e m e n t   1

                         binary post data at increment   1.   0   on tape 16
                          time =       1.02

                          ●                            ●
                          ●                            ●
                          ●                            ●
                          ●                            ●


                         s t a r t   o f   i n c r e m e n t  15




           fluxes summed over the whole model


           from distributed fluxes
         0.000E+00


           concentrated fluxes
         0.000E+00
```

```
                         start of assembly
                          time =        3.24



                         start of matrix solution
                          time =        3.31



                         singularity ratio    6.1463E-01


                         end of matrix solution
                          time =        3.32


                         maximum nodal temperature change is  0.105E+02 at node   18

                         this is 0.210E+02  percent of change allowed on control option


MARC          output for increment   15.    heat transfer for circular channel in square pipe


                         transient time has reached  0.500E+01  of time period  0.500E+01


                         total transient time = 5.00013E+00 ——— Total time achieved


  elem., point, temp.     elem., point, temp.     elem., point, temp.     elem., point, temp.     elem., point, temp.


      1    1  807.0           1    2  805.2           1    3  785.3           1    4  780.4           2    1  804.9
      2    2  806.9           2    3  780.0           2    4  785.2           3    1  806.9           3    2  804.9
      3    3  785.2           3    4  780.0           4    1  805.2           4    2  807.0           4    3  780.4
      4    4  785.3           5    1  770.7           5    2  763.9           5    3  752.3           5    4  743.8
      6    1  763.4           6    2  770.5           6    3  742.9           6    4  752.1           7    1  770.5
      7    2  763.4           7    3  752.1           7    4  742.9           8    1  763.9           8    2  770.7
      8    3  743.8           8    4  752.3           9    1  723.5           9    2  733.2           9    3  691.0
      9    4  700.3          10    1  733.2          10    2  723.5          10    3  700.3          10    4  691.0
     11    1  733.6          11    2  724.8          11    3  700.9          11    4  693.2          12    1  724.8
     12    2  733.6          12    3  693.2          12    4  700.9          13    1  663.2          13    2  658.7
     13    3  593.0          13    4  596.1          14    1  658.7          14    2  663.2          14    3  596.1
     14    4  593.0          15    1  649.6          15    2  660.7          15    3  569.0          15    4  585.7
     16    1  660.7          16    2  649.6          16    3  585.7          16    4  569.0          17    1  553.1
     17    2  557.7          17    3  514.2          17    4  515.5          18    1  557.7          18    2  553.1
     18    3  515.5          18    4  514.2          19    1  531.2          19    2  546.0          19    3  508.4
     19    4  512.3          20    1  546.0          20    2  531.2          20    3  512.3          20    4  508.4
```

Temperature near the fluid still has not reached the temperature of the fluid.

```
                    n o d a l   p o i n t   d a t a



                    t o t a l   n o d a l   t e m p e r a t u r e s


    1  814.02          2  815.20          3  813.69          4  815.20          5  814.02
    6  779.62          7  779.62          8  769.10          9  768.51         10  769.10
   11  748.94         12  748.94         13  733.09         14  731.81         15  733.09
   16  691.54         17  691.54         18  675.74         19  678.93         20  678.93
   21  575.37         22  575.37         23  565.14         24  565.14         25  532.71
   26  500.00         27  500.00         28  500.00         29  500.00         30  500.00

                    e n d   o f   i n c r e m e n t  15

                    binary post data at increment  15.   0   on tape 16
                     time =       3.41



                    *** end of input deck - job ends



                              marc exit number 3004
```

Results

The final equilibrium temperature distribution in the model after 5.0 seconds (increment 15) is shown in an isothermal contour plot of total nodal temperatures in Figure 8.2. The maximum nodal temperature along the hold edge is 815.20°F (or a temperature change of +315.20°F) at nodes 2 and 4. The outer edges are maintained at 500.0°F. As expected for such a symmetrical problem with symmetrical loads and boundary conditions, the temperature distribution is symmetrical about the 45° diagonal line from the center of the channel to the top right corner of the model.

Figure 8-3 is a temperature history plot of the total nodal temperatures at five nodes along the 45° diagonal line – nodes 3, 9, 14, 18, and 25. (Node 30 at the upper right corner of the model in maintained at 500.0°F throughout the thermal analysis and is thus not included.)

We are now ready to proceed to Example 9, and use these temperatures to calculate the resulting thermal stresses in the model.

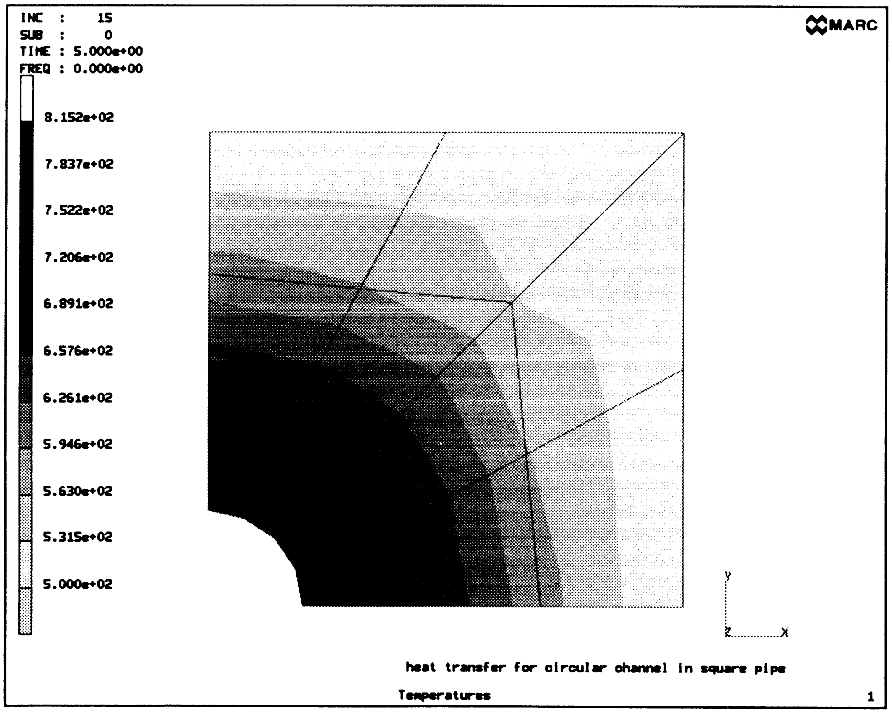


Figure 8.2 **Contour Plot of Temperatures**

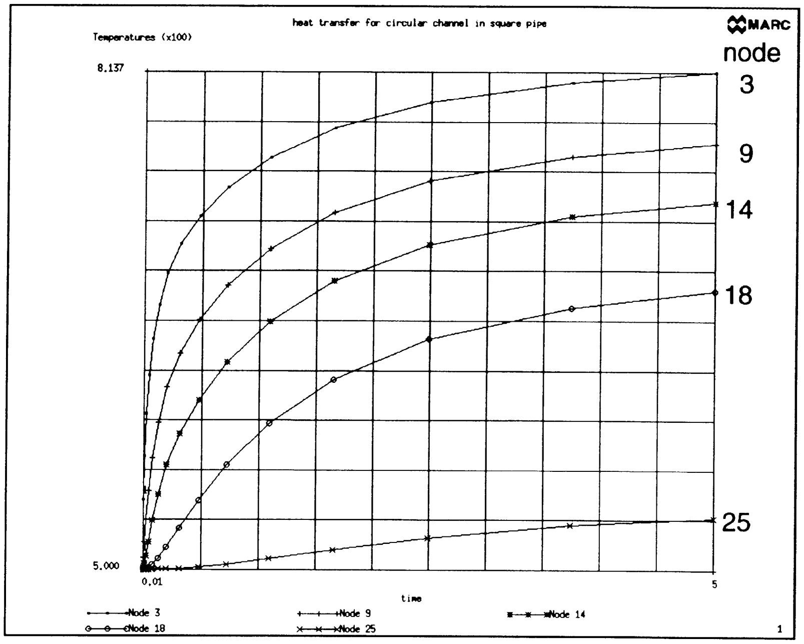


Figure 8.3 Time History of Nodal Temperatures

Example 9

Thermal Stress Analysis of Square Pipe with Circular Channel

The aim of this example is to illustrate thermal stress analysis using MARC, using temperatures previously calculated in a heat transfer analysis. This analysis is thus the second step in a typical two-step analysis process. The FE model is 2-D and plane strain (using MARC Element 11). New options which are demonstrated include ALIAS, INITIAL STATE, AUTO THERM, and CHANGE STATE. This thermal stress analysis is nonlinear, because in the middle of the analysis, the part of the pipe adjacent to the channel will yield.

Sketch

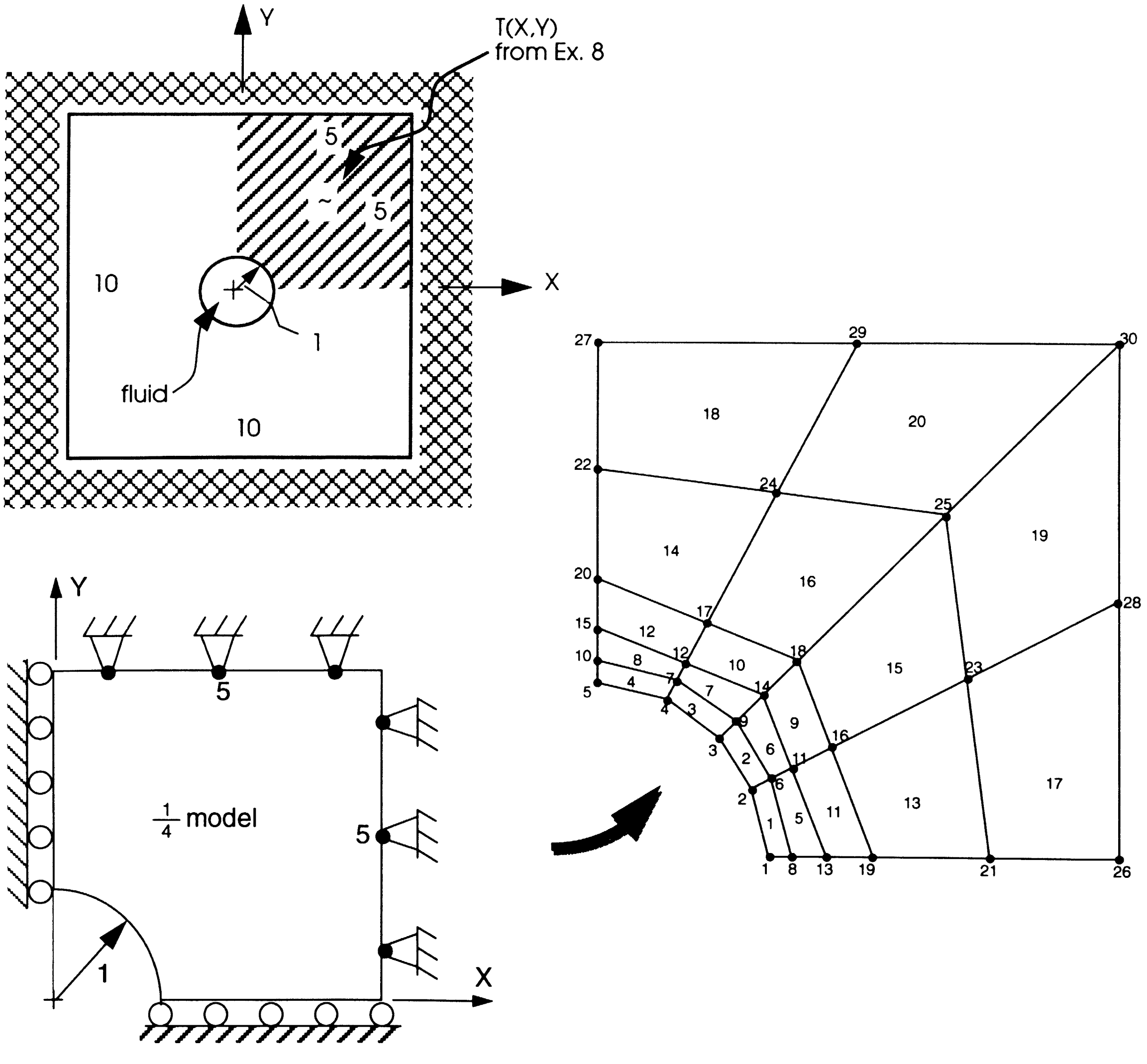


Figure 9.1 **Square Plate with Circular Hole**

Model

The idealized FE one-quarter model is the 2-D cross section of a slice of the square pipe. Plane strain conditions are assumed: the strains and displacements in the Z-directions are zero. Such an assumption is correct for a long structure such as this pipe.

The mesh and boundary conditions are similar to those of Example 1. Symmetry conditions are imposed on the X = 0 and Y = 0 edges: no displacements across the plane of symmetry and no rotations. The quarter model has dimensions of 5 x 5 inches, with a 1-inch channel radius. It is 1.0 inch thick.

MARC Element 11 (see MARC Volume B) is used. This is a 4-noded quadrilateral plane strain element, with two DOFs (X- and Y-displacements) at each node.

Properties

The material properties of the model are identical to those of the beam in Example 5 (Young's modulus, Poisson's ratio, and tensile yield stress), except this time we need to input a value for the coefficient of thermal expansion because this example is a thermal stress problem. Young's modulus is 30E6 psi; Poisson's ratio is 0.3; coefficient of thermal expansion is 4.0E-6 in./in.-°F; the tensile yield stress is 20,000 psi. The initial stress-free temperature is 500.0°F. (Note that this example *does not* consider the effects of material properties varying with temperature, a situation which would have required the use of the TEMPERATURE EFFECTS option.)

Loads

The only loads on the model are thermal; that is, the temperatures previously calculated in Example 8. No mechanical loads are present.

Boundary Conditions

Symmetry boundary conditions are to be imposed on the left (X = 0) and bottom edges (Y = 0); there shall be no displacements across the planes of symmetry and no rotations. In addition, we assume that the pipe is embedded in and bonded to a very stiff soil, so that the outer walls cannot move in-plane.

Special Features

Three special features are illustrated in this example: AUTO THERM for automatic static thermal stress analysis; and CHANGE STATE for changing the temperatures throughout the model – after they have been initialized by the INITIAL STATE option.

Input

A complete input list is included.

PARAMETER Section

The only new PARAMETER option you have not seen before is the ALIAS option which allows you to enter a different element type than the one given in the CON-

NECTIVITY Model Definition block. This option is especially convenient for the case where the same mesh is to be used for both the heat transfer and stress analyses. On the "ALIAS,1,39,11" line, the "1" says there is one ALIAS to be entered, the "39" is the MARC element type of the existing mesh being read in, and "11" is the desired MARC element type to be used for the stress analysis.

MODEL DEFINITION Section

The MODEL DEFINITION options consist of:

a. FE mesh topology – CONNECTIVITY, COORDINATES, DEFINE blocks
b. Geometric properties
c. Material properties
d. Initial temperature definition
e. Boundary conditions
f. Bandwidth minimization
g. Nonlinear analysis controls
h. Output controls

FE Mesh Topology

The FE mesh is identical to that of Examples 1 and 8, except that this time we want to use Element 11 for the thermal stress analysis. We have retained the same three node and element sets as Example 8, except we are adding another element set named ALLE, which consists of all 20 elements in the model.

Geometric Properties

The GEOMETRY block is the same as that for Example 8, indicating that we are assuming unit thickness for all 20 elements (ALLE).

Material Properties

The ISOTROPIC block is similar to that of Example 5, except for two items: we are specifying 4.0E-6 in the fourth field of the data line to denote the coefficient of thermal expansion; and the fact we did not name "VON MISES, ISOTR HARD" on the line before is immaterial – these are the default values anyway.

Initial Temperatures

The INITIAL STATE option provides us with means to initialize various state variables (in this case, temperature) in the model. Initial temperatures read in by this option are assumed to define the stress-free temperature field; that is temperatures which cause zero thermal strains and thermal stresses. Recall that there are no thermal stresses in an isotropic structure unless you have prescribed boundary conditions to restrain it. (For example, an unrestrained bar or plate will undergo uniform thermal strains due to a homogeneous temperature change, but there will be no thermal stresses!)

The "1,4" line means: 1 is temperature, and 4 instructs MARC to initialize the temperature using "line series 5, 6, 7, and 8" which are coming next. The "500.," line says this is the temperature to be used at the start of the zeroth increment, for the elements to be named. The "ALLE" line indicates that this temperature applies to element set ALLE (all 20 elements). And finally, the "1 TO 4" line tells MARC to apply this temperature to all four integration points in each element.

Boundary Conditions

The FIXED DISP option is used to prescribe the displacement boundary conditions of the model. Before discussing the symmetry BCs, we would like to restrain any X-Y motion of the cross section since we are assuming the pipe is imbedded in and bonded to a very stiff and compacted soil. After the blank line, you first see three lines used to restrain the model against X- and Y-displacements. The "0.,0.," line shows the zero values corresponding to the first two DOFs named on the next line ("1 TO 2"). Then the "OUTEDGE" line tells MARC to apply these BCs along the five nodes in the node set OUTEDGE; that is, nodes 26 to 30.

The next six lines in the block first fix the nodes along the Y-axis (nodes 5, 10, 15, 20, 22, 27) against X-displacements, and then fix the nodes along the X-axis (nodes 1, 8, 13, 19, 21, 26) against Y-displacements. These symmetry boundary conditions are identical to those imposed for Example 1.

Bandwidth Minimization

The OPTIMIZE option turns on the bandwidth optimization procedures in MARC. Since we did not flag any particular algorithm, the default Cuthill-McKee scheme is used. The "5," line means we want MARC to try a maximum of five different node numbering schemes, then pick the one which results in the lowest bandwidth.

Nonlinear Analysis Controls

The CONTROL option lets you input parameters to control the convergence and accuracy of the nonlinear stress analysis. The "20," line means we want the maximum number of load steps to be 20. The fact we did not specify any other values means we will get the MARC default values: the maximum number of recycles during an increment is 3, and the tolerance (maximum allowed relative error in residual forces) is 10%, entered as 0.1.

Output Controls

The two output control options in this example are POST and PRINT ELEM. As usual, the POST option creates a post-processor file for later post-processing by Mentat II. The "5," line tells MARC that five element variables are to be written to the file at each increment. Since we did not put in anything in the fourth field of this line, MARC will assume the file is binary. (Unit 16). The next five lines refer to our selection of the post codes: "9" is total temperature; "11" is normal stress in the X-direction; "12" is normal Y-stress; "13" is normal Z-stress; and "17" is equivalent von Mises stress.

The PRINT ELEM option allows you to specify which element variables and for what elements you want printed out. After the usual blank line, the next line indicates that we want to print out total STRESS, total STRAIN, STATE variable (i.e., temperature), and PLASTIC strain. The "FLUID" line means we want the element printout for element set FLUID; that is, elements 1 to 4 along the circular channel. And the "1 TO 4" line says we want all four integration points printed out.

The "END OPTION" line terminated the MODEL DEFINITION section.

LOAD INCREMENTATION Section

This example has two LOAD INCREMENTATION options: AUTO THERM and CHANGE STATE.

The AUTO THERM option allows you to perform automatic, static, elastic-plastic thermally loaded stress analysis, which is based on a set of temperatures defined throughout the mesh as a function of time. The "50.,20," line means that the maximum temperature change to be used per step of stress analysis of 50°F, and the maximum number of increments (steps) to be allowed is 20. The choice of maximum allowable temperature change is obtained by making sure the increment of thermal strain should be restricted to 20-50% of the strain to cause yield. Therefore, the maximum allowable temperature change should be 20-50% of the yield stress divided by the product of Young's modulus and coefficient of thermal expansion, or (2E4)/(30E6)/(4E6) – which in this example is 167°F. Our choice of 50°F for the maximum allowable temperature change happens to be about 30% of 167°F.

The CHANGE STATE option is required to go hand-in-hand with AUTO THERM, and allows you to change the state variable (temperature) throughout the model. (The temperature distribution was previously initialized in the INITIAL STATE MODEL DEFINITION option.) On the "1,3,,25,1,15" line: the first "1" is temperature; the "3" tells MARC to read the new values of the temperatures from a post file written by a previous heat transfer analysis (Example 8); the "25" in the fourth field refers to the unit number from which the post file data for the previous heat transfer run will be read; the second "1" tells MARC to read step 1 of the heat transfer post file, and the "15" refers to the number of sets of temperature history to be read. This corresponds to the fact that in the previous example, 15 increments were performed.

The "CONTINUE" line terminates the input file.

Output

The input echo is included first. The results of the Cuthill-McKee bandwidth minimization are the same as those from Example 1 and therefore are not shown here. Increment 0 is also not included, since it is a null increment with the initial temperature of 500°F everywhere; there are no displacements, nodal forces, or reactions anywhere.

The page showing the start of increment 1 is the first page of interest. The parameters we have selected for the AUTO THERM and CHANGE STATE options are displayed between the outputs of increments 0 and 1, and we can then see these two options going into action. MARC first informs you that AUTO THERM is being invoked for 15 sets of temperature input (recall that Example 8 was completed after 15 increments), and then asks for step 1 data from the temperature file. Then, the AUTO THERM option finds that the *maximum temperature change* from step 1 of the thermal analysis is 38.55°F, which is less than the maximum allowed temperature change of 50°F specified in the AUTO THERM option.

Increment 1 thermal stress analysis results are included here. They show: a maximum von Mises stress of 4,137 psi (element 2 integration point 2, or element 3 point 1); a maximum temperature of 538.6°F; and a maximum nodal displacement value of 3.5126E-5 inches (the X-displacement of node 8 as well as the Y-displacement of node 10). Note that with a *positive* temperature change of 38.55°F, we should check to see that the model does indeed *expand* and also expect nodes 1 to 5 at the circular channel to move radially *inward* (because we have restrained all four edges against translations normal to the edge), which is the case here. Also, we should check for symmetry of the displacement field (and the stresses and strains), since the problem is symmetrical about a 45° diagonal line from the origin. This symmetry of the displacement results (considering roundoff errors) means: node 1 X-displacement being the same as the node 5 Y-displacement; node 3 X- and Y-displacements being identical, and so forth. We see that this symmetry is indeed borne out (to the fifth significant figure). Symmetry of the stresses and strains also exists; results of element 1 point 1 being identical to element 4 point 2; element 2 point 2 and element 3 point 1, etc.

The thermal stress analysis then proceeds to increment 2. Upon asking for step 2 temperature data, MARC checks that the maximum temperature change of 24.89°F is less than 60% of the allowable value of 50.0°F specified in the AUTO THERM option, and proceeds to read in step 3, whose maximum temperature change is 49.40°F. By the end of increment 3, the maximum von Mises stress has reached 17,910 psi in elements 2 and 3. Yielding (in elements 1 to 4) first occurs in increment 4. After increment 6, MARC reads in steps 12 and 13 from the temperature file; the latter has a maximum temperature change of 54.41°F. This value is greater than the maximum allowable value of 50.00°F in the AUTO THERM option. Thus, MARC will subdivide this set into one step of 50.00°F and another step of 4.41°F.

At the end of increment 7, MARC proceeds to ask for step 14 and step 15 temperature data from the temperature file. It sees that the maximum temperature supplied this time is 33.85°F, which is between 60 and 100 percent of the maximum allowed temperature change for both increments. These steps are then merged and the final thermal stress increment is performed. We can now verify that the final temperatures at increment 8 in the stress analysis are indeed identical to the final temperatures at increment 15 in the heat transfer analysis. The amount of plastic strain is approximately 0.00342.

```
                              i n p u t   d a t a

    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
--------------------------------------------------------------------------------
TITLE, THERMAL STRESS GIVEN TEMPERATURE TIME HISTORY
SIZING        100000   20   30
ELEMENTS      11
ALIAS,1,39,11
END
CONNECTIVITY
   20    0    0
    1   39    2    1    8    6
    2   39    3    2    6    9
    3   39    4    3    9    7
    4   39    5    4    7   10
    5   39    6    8   13   11
    6   39    9    6   11   14
    7   39    7    9   14   12
    8   39   10    7   12   15
    9   39   14   11   16   18
   10   39   12   14   18   17
   11   39   11   13   19   16
   12   39   15   12   17   20
   13   39   16   19   21   23
   14   39   20   17   24   22
   15   39   18   16   23   25
   16   39   17   18   25   24
   17   39   23   21   26   28
   18   39   22   24   29   27
   19   39   25   23   28   30
   20   39   24   25   30   29
COORDINATES
    2   30    0    0
    1   1.00000   0.00000
    2   0.92381   0.38247
    3   0.70700   0.70700
    4   0.38247   0.92381
    5-0.25219-6   1.00000
    6   1.10190   0.45623
    7   0.45623   1.10190
    8   1.25000   0.00000
    9   0.88350   0.88350
   10-0.32459-6   1.25000
   11   1.28000   0.53000
   12   0.53000   1.28000
   13   1.50000   0.00000
   14   1.06000   1.06000
   15-0.31869-6   1.50000
   16   1.70000   0.70000
--------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

Alias Element Type 39 to Element Type 11

Can use CONNECTIVITY from the previous analysis in Example 8

```
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
--------------------------------------------------------------------------------
   17   0.70000   1.70000
   18   1.40000   1.40000
   19   2.00000   0.00000
   20-0.50439-6   2.00000
   21   3.50000   0.00000
   22-0.10206-5   3.50000
   23   3.35000   1.60000
   24   1.60000   3.35000
   25   3.20000   3.20000
   26   5.00000   0.00000
   27-0.15777-5   5.00000
   28   5.00000   2.50000
   29   2.50000   5.00000
   30   5.00000   5.00000
DEFINE    ELEMENT   SET       FLUID
1 TO 4
DEFINE    ELEMENT   SET       ALLE
1 TO 20
DEFINE    NODE      SET       OUTEDGE
   26 TO     30
DEFINE    NODE      SET       ALLN
    1 TO     30
GEOMETRY

1.0,
ALLE
ISOTROPIC

    1
30.E6,.3,,4.0E-6,20000.
ALLE
INITIAL STATE
1,4
500.,
ALLE
1 TO 4
FIXED DISP

0.,0.,
1 TO 2
OUTEDGE
0.,
1,
5 10 15 20 22 27
--------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

Coefficient of Thermal Expansion

INITIAL Stress-Free Temperatures

```
                5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
          --------------------------------------------------------------------------------
card   90   0.,
            2,
            1 8 13 19 21 26
            OPTIMIZE
            5,
card   95   CONTROL
            20,

            POST
            5,
card  100   9,
            11,
            12,
            13,
            17,
card  105   PRINT ELEM

            STRESS STRAIN STATE PLASTIC
            FLUID
            1 TO 4
card  110   END OPTION
            AUTO THERM
            50.,20,
            CHANGE STATE
            1,3,,25,1,15
card  115   CONTINUE
          --------------------------------------------------------------------------------
                5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
          --------------------------------------------------------------------------------
          --------------------------------------------------------------------------------
```

Controls reading in file containing temperatures generated in previous analysis.

⋮

```
auto therm
----------

maximum temperature change per step = 5.000E+01  maximum steps =   20 reassembly interval =    0
total transient time = 0.000E+00

change state
----------

    1    3    0   25    1   15    0    0

auto therm is invoked for   15  sets of temp. input
now asking for step    1  from temperature tape

auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 3.855E+01
maximum allowed on auto therm option is 5.000E+01
```

Data is read from Fortran Unit 17

```
    maximum temp. change is between 60p.c. and 100p.c. of allowable - this set will be analysed in one increment


                            s t a r t   o f   i n c r e m e n t   1
```

NOTE

The AUTO THERM option controls the reading of the temperature file.

```
                            load increments associated with each degree of freedom
                            summed over the whole model


                            distributed loads
    0.000E+00 0.000E+00



                            point loads
    0.000E+00 0.000E+00


                            end of matrix back substitution
                             time =       0.90


                           maximum residual force  at node    7 degree of freedom    2 is equal to   0.995E-12
                           maximum reaction force  at node    1 degree of freedom    2 is equal to   0.778E+03
                           convergence ratio                                                         0.128E-14
```

Purely Elastic Increment - effectively insures no error

NOTE

Since there are no temperature dependent material properties, the program will not reassemble the stiffness matrix until the occurrence of plasticity

```
MARC          output for increment    1.    thermal stress given temperature time history


         tresca     mises    mean  p r i n c i p a l   v a l u e s          p h y s i c a l   c o m p o n e n t s
      intensity intensity  normal  minimum intermediate maximum    1       2        3        4        5        6
                                   intensity


element    1  point    1       integration pt. coordinate=      0.981E+00    0.314E+00
stress  4.166E+03 4.132E+03-6.976E+03-8.387E+03-8.318E+03-4.222E+03-4.510E+03-8.030E+03-8.387E+03 1.049E+03
strain  1.805E-04 1.474E-04 6.117E-05 0.000E+00 2.985E-06 1.805E-04 1.680E-04 1.549E-05 0.000E+00 9.087E-05
state vars    538.5

element    1  point    2       integration pt. coordinate=      0.103E+01    0.841E-01
stress  3.870E+03 3.842E+03-6.758E+03-8.068E+03-8.010E+03-4.197E+03-4.288E+03-7.920E+03-8.068E+03 5.811E+02
strain  1.677E-04 1.370E-04 5.673E-05 0.000E+00 2.486E-06 1.677E-04 1.638E-04 6.416E-06 0.000E+00 5.036E-05
state vars    536.7
```

```
element    1  point    3       integration pt. coordinate=     0.109E+01    0.348E+00
stress  4.164E+03 3.934E+03-1.036E+02-1.661E+03-1.152E+03 2.502E+03 2.201E+03-8.510E+02-1.661E+03 1.005E+03
strain  1.804E-04 1.484E-04 6.749E-05 0.000E+00 2.207E-05 1.804E-04 1.674E-04 3.511E-05 0.000E+00 8.707E-05
state vars   517.2

element    1  point    4       integration pt. coordinate=     0.117E+01    0.931E-01
stress  3.881E+03 3.672E+03-9.938E+00-1.458E+03-9.953E+02 2.423E+03 2.319E+03-8.904E+02-1.458E+03 5.895E+02
strain  1.682E-04 1.383E-04 6.275E-05 0.000E+00 2.005E-05 1.682E-04 1.637E-04 2.460E-05 0.000E+00 5.109E-05
state vars   515.7

element    2  point    1       integration pt. coordinate=     0.790E+00    0.671E+00
stress  3.877E+03 3.851E+03-6.763E+03-8.073E+03-8.021E+03-4.196E+03-5.527E+03-6.689E+03-8.073E+03 1.822E+03
strain  1.680E-04 1.372E-04 5.676E-05 0.000E+00 2.270E-06 1.680E-04 1.103E-04 5.997E-05 0.000E+00 1.579E-04
state vars   536.7

element    2  point    2       integration pt. coordinate=     0.916E+00    0.471E+00
stress  4.169E+03 4.137E+03-6.980E+03-8.391E+03-8.326E+03-4.222E+03-5.227E+03-7.322E+03-8.391E+03 1.765E+03
strain  1.806E-04 1.475E-04 6.114E-05 0.000E+00 2.781E-06 1.806E-04 1.371E-04 4.632E-05 0.000E+00 1.529E-04
state vars   538.6

element    2  point    3       integration pt. coordinate=     0.892E+00    0.760E+00
stress  3.888E+03 3.681E+03-1.421E+01-1.463E+03-1.005E+03 2.425E+03 1.299E+03 1.216E+02-1.463E+03 1.611E+03
strain  1.685E-04 1.385E-04 6.277E-05 0.000E+00 1.982E-05 1.685E-04 1.197E-04 6.865E-05 0.000E+00 1.396E-04
state vars   515.7

element    2  point    4       integration pt. coordinate=     0.102E+01    0.526E+00
stress  4.166E+03 3.938E+03-1.079E+02-1.665E+03-1.161E+03 2.501E+03 1.673E+03-3.324E+02-1.665E+03 1.532E+03
strain  1.805E-04 1.485E-04 6.746E-05 0.000E+00 2.184E-05 1.805E-04 1.446E-04 5.773E-05 0.000E+00 1.328E-04
state vars   517.2

element    3  point    1       integration pt. coordinate=     0.471E+00    0.916E+00
stress  4.169E+03 4.137E+03-6.980E+03-8.391E+03-8.326E+03-4.222E+03-7.322E+03-5.227E+03-8.391E+03 1.765E+03
strain  1.806E-04 1.475E-04 6.114E-05 0.000E+00 2.781E-06 1.806E-04 4.632E-05 1.371E-04 0.000E+00 1.529E-04
state vars   538.6

element    3  point    2       integration pt. coordinate=     0.671E+00    0.790E+00
stress  3.877E+03 3.851E+03-6.763E+03-8.073E+03-8.021E+03-4.196E+03-6.689E+03-5.527E+03-8.073E+03 1.822E+03
strain  1.680E-04 1.372E-04 5.676E-05 0.000E+00 2.270E-06 1.680E-04 5.997E-05 1.103E-04 0.000E+00 1.579E-04
state vars   536.7

element    3  point    3       integration pt. coordinate=     0.526E+00    0.102E+01
stress  4.166E+03 3.938E+03-1.079E+02-1.665E+03-1.161E+03 2.501E+03-3.324E+02 1.673E+03-1.665E+03 1.532E+03
strain  1.805E-04 1.485E-04 6.746E-05 0.000E+00 2.184E-05 1.805E-04 5.773E-05 1.446E-04 0.000E+00 1.328E-04
state vars   517.2

element    3  point    4       integration pt. coordinate=     0.760E+00    0.892E+00
stress  3.888E+03 3.681E+03-1.421E+01-1.463E+03-1.005E+03 2.425E+03 1.216E+02 1.299E+03-1.463E+03 1.611E+03
strain  1.685E-04 1.385E-04 6.277E-05 0.000E+00 1.982E-05 1.685E-04 6.865E-05 1.197E-04 0.000E+00 1.396E-04
state vars   515.7
```

```
element    4  point   1       integration pt. coordinate=     0.841E-01    0.103E+01
stress  3.870E+03 3.842E+03-6.758E+03-8.068E+03-8.010E+03-4.197E+03-7.920E+03-4.288E+03-8.068E+03 5.811E+02
strain  1.677E-04 1.370E-04 5.673E-05 0.000E+00 2.486E-06 1.677E-04 6.416E-06 1.638E-04 0.000E+00 5.036E-05
state vars   536.7

element    4  point   2       integration pt. coordinate=     0.314E+00    0.981E+00
stress  4.166E+03 4.132E+03-6.976E+03-8.387E+03-8.318E+03-4.222E+03-8.030E+03-4.510E+03-8.387E+03 1.049E+03
strain  1.805E-04 1.474E-04 6.117E-05 0.000E+00 2.985E-06 1.805E-04 1.549E-05 1.680E-04 0.000E+00 9.087E-05
state vars   538.5

element    4  point   3       integration pt. coordinate=     0.931E-01    0.117E+01
stress  3.881E+03 3.672E+03-9.937E+00-1.458E+03-9.953E+02 2.423E+03-8.904E+02 2.319E+03-1.458E+03 5.895E+02
strain  1.682E-04 1.383E-04 6.275E-05 0.000E+00 2.005E-05 1.682E-04 2.460E-05 1.637E-04 0.000E+00 5.109E-05
state vars   515.7

element    4  point   4       integration pt. coordinate=     0.348E+00    0.109E+01
stress  4.164E+03 3.934E+03-1.036E+02-1.661E+03-1.152E+03 2.502E+03-8.510E+02 2.201E+03-1.661E+03 1.005E+03
strain  1.804E-04 1.484E-04 6.749E-05 0.000E+00 2.207E-05 1.804E-04 3.511E-05 1.674E-04 0.000E+00 8.707E-05
state vars   517.2


                          n o d a l   p o i n t   d a t a


                          i n c r e m e n t a l   d i s p l a c e m e n t s


         1 -5.50126E-06       0.                 2 -3.61479E-06 -1.44858E-06             3 -3.86016E-06 -3.86018E-06
         4 -1.44857E-06 -3.61481E-06             5       0.      -5.50129E-06             6  2.95779E-05  1.23180E-05
         7  1.23180E-05  2.95778E-05             8  3.51247E-05       0.                  9  2.48856E-05  2.48856E-05
        10       0.       3.51247E-05            11  3.20203E-05  1.34064E-05            12  1.34064E-05  3.20203E-05
        13  3.45140E-05       0.                 14  2.45257E-05  2.45257E-05            15       0.       3.45140E-05
        16  2.45399E-05  1.11782E-05            17  1.11782E-05  2.45399E-05            18  1.81682E-05  1.81682E-05
        19  2.46233E-05       0.                 20       0.       2.46232E-05            21  8.89279E-06       0.
        22       0.       8.89278E-06            23  7.85712E-06  4.83331E-06            24  4.83331E-06  7.85712E-06
        25  3.94760E-06  3.94760E-06            26       0.            0.                 27       0.            0.
        28       0.            0.                 29       0.            0.                 30       0.            0.


                          t o t a l   d i s p l a c e m e n t s


         1 -5.50126E-06       0.                 2 -3.61479E-06 -1.44858E-06             3 -3.86016E-06 -3.86018E-06
         4 -1.44857E-06 -3.61481E-06             5       0.      -5.50129E-06             6  2.95779E-05  1.23180E-05
         7  1.23180E-05  2.95778E-05             8  3.51247E-05       0.                  9  2.48856E-05  2.48856E-05
        10       0.       3.51247E-05            11  3.20203E-05  1.34064E-05            12  1.34064E-05  3.20203E-05
        13  3.45140E-05       0.                 14  2.45257E-05  2.45257E-05            15       0.       3.45140E-05
        16  2.45399E-05  1.11782E-05            17  1.11782E-05  2.45399E-05            18  1.81682E-05  1.81682E-05
        19  2.46233E-05       0.                 20       0.       2.46232E-05            21  8.89279E-06       0.
        22       0.       8.89278E-06            23  7.85712E-06  4.83331E-06            24  4.83331E-06  7.85712E-06
        25  3.94760E-06  3.94760E-06            26       0.            0.                 27       0.            0.
        28       0.            0.                 29       0.            0.                 30       0.            0.
```

```
          total equivalent nodal forces (distributed plus point loads)


  1      0.         0.              2      0.         0.              3      0.         0.
  4      0.         0.              5      0.         0.              6      0.         0.
  7      0.         0.              8      0.         0.              9      0.         0.
 10      0.         0.             11      0.         0.             12      0.         0.
 13      0.         0.             14      0.         0.             15      0.         0.
 16      0.         0.             17      0.         0.             18      0.         0.
 19      0.         0.             20      0.         0.             21      0.         0.
 22      0.         0.             23      0.         0.             24      0.         0.
 25      0.         0.             26      0.         0.             27      0.         0.
 28      0.         0.             29      0.         0.             30      0.         0.



          reaction forces at fixed boundary conditions, residual load correction elsewhere


  1 -2.77112E-13   778.29          2  1.70530E-13 -3.41061E-13      3  1.13687E-13 -1.13687E-13
  4  1.13687E-13  1.13687E-13      5   778.29      2.06057E-13      6 -3.97904E-13 -1.42109E-13
  7 -1.74083E-13 -9.94760E-13      8 -4.54747E-13   328.49          9 -1.56319E-13  5.40012E-13
 10   328.49     -3.97904E-13     11  3.41061E-13 -3.97904E-13     12 -1.42109E-14  2.27374E-13
 13  3.69482E-13  -92.568         14 -2.27374E-13  1.42109E-13     15  -92.568      2.84217E-14
 16  6.82121E-13 -1.98952E-13     17 -5.68434E-14  1.70530E-13     18  2.27374E-13  3.26850E-13
 19 -2.27374E-13  -214.32         20  -214.32     -2.27374E-13     21 -8.52651E-14  -35.086
 22  -35.086     -5.68434E-14     23  1.42109E-13 -1.27898E-13     24 -2.48690E-13 -1.84741E-13
 25  3.55271E-15 -1.84741E-13     26  -230.11      19.921          27   19.921      -230.11
 28  -346.54      -92.869         29  -92.869      -346.54         30  -115.21      -115.21

                    summary of externally applied loads

0.00000E+00    0.00000E+00

                    summary of reaction/residual forces

0.31974E-13   -0.22737E-12

                    end of increment 1

                    binary post data at increment   1.   0    on tape 16
                     time =       1.13

     now asking for step    2  from temperature tape


    auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 2.489E+01
      maximum allowed on auto therm option is 5.000E+01
    maximum temp.change is less than 60p.c. of allowable - next temp. input will be read and merged into this set
    now asking for step    3  from temperature tape
```

```
auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 4.940E+01
  maximum allowed on auto therm option is 5.000E+01
maximum temp. change is between 60p.c. and 100p.c. of allowable - this set will be analysed in one increment



                          s t a r t   o f   i n c r e m e n t   2

                          ●                                              ●
                          ●                                              ●
                          ●                                              ●
                          ●                                              ●

                          e n d   o f   i n c r e m e n t   2

now asking for step    4  from temperature tape


auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 2.317E+01
  maximum allowed on auto therm option is 5.000E+01
maximum temp.change is less than 60p.c. of allowable - next temp. input will be read and merged into this set
now asking for step    5  from temperature tape


auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 4.547E+01
  maximum allowed on auto therm option is 5.000E+01
maximum temp. change is between 60p.c. and 100p.c. of allowable - this set will be analysed in one increment



                          s t a r t   o f   i n c r e m e n t   3

                          ●                                              ●
                          ●                                              ●
                          ●                                              ●
                          ●                                              ●

                          e n d   o f   i n c r e m e n t   3


  now asking for step    6  from temperature tape


auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 2.281E+01
  maximum allowed on auto therm option is 5.000E+01
maximum temp.change is less than 60p.c. of allowable - next temp. input will be read and merged into this set
now asking for step    7  from temperature tape


auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 4.510E+01
  maximum allowed on auto therm option is 5.000E+01
maximum temp. change is between 60p.c. and 100p.c. of allowable - this set will be analysed in one increment
```

```
                    start of increment 4




                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
0.000E+00 0.000E+00


                    point loads
0.000E+00 0.000E+00






                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
0.000E+00 0.000E+00


                    point loads
0.000E+00 0.000E+00

                    start of assembly
                     time =       1.81



                    start of matrix solution
                     time =       1.91



                    singularity ratio    3.6546E-01


                    end of matrix solution
                     time =       1.92


                  maximum residual force  at node    4 degree of freedom    2 is equal to    0.380E+03
                  maximum reaction force  at node   29 degree of freedom    2 is equal to    0.588E+04
                  convergence ratio                                                          0.646E-01
```

Material nonlinearity occurs for the first time. Reassembles the stiffness matrix.

```
 MARC          output for increment    4.     thermal stress given temperature time history



        tresca     mises    mean  p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
      intensity intensity  normal   minimum intermediate maximum     1         2         3         4         5         6
                           intensity

element    1  point   1       integration pt. coordinate=     0.981E+00    0.314E+00
stress  2.067E+04 2.000E+04-2.456E+04-3.193E+04-3.050E+04-1.125E+04-1.256E+04-3.062E+04-3.050E+04 5.028E+03
strain  1.287E-03 9.868E-04 3.751E-04-8.065E-05 0.000E+00 1.206E-03 1.124E-03 8.748E-07 0.000E+00 6.269E-04
plas.st 3.906E-04 2.543E-04 4.518E-20-1.365E-04-1.176E-04 2.541E-04 2.291E-04-1.116E-04-1.176E-04 1.911E-04
state vars   675.6

element    1  point   2       integration pt. coordinate=     0.103E+01    0.841E-01
stress  2.075E+04 2.000E+04-2.560E+04-3.305E+04-3.146E+04-1.230E+04-1.271E+04-3.265E+04-3.146E+04 2.861E+03
strain  1.225E-03 9.305E-04 3.492E-04-8.854E-05 0.000E+00 1.136E-03 1.112E-03-6.468E-05 0.000E+00 3.386E-04
plas.st 3.255E-04 2.108E-04-5.873E-20-1.149E-04-9.564E-05 2.106E-04 2.041E-04-1.085E-04-9.564E-05 9.059E-05
state vars   672.6

element    1  point   3       integration pt. coordinate=     0.109E+01    0.348E+00
stress  2.045E+04 2.000E+04-1.047E+04-1.760E+04-1.667E+04 2.848E+03 1.452E+03-1.527E+04-1.760E+04 5.029E+03
strain  1.205E-03 9.847E-04 4.217E-04 0.000E+00 6.049E-05 1.205E-03 1.123E-03 1.425E-04 0.000E+00 5.902E-04
plas.st 3.183E-04 2.059E-04 1.355E-20-1.127E-04-9.286E-05 2.056E-04 1.840E-04-7.134E-05-1.127E-04 1.544E-04
state vars   640.3

element    1  point   4       integration pt. coordinate=     0.117E+01    0.931E-01
stress  2.032E+04 2.000E+04-1.037E+04-1.736E+04-1.670E+04 2.960E+03 2.491E+03-1.624E+04-1.736E+04 3.000E+03
strain  1.138E-03 9.299E-04 3.928E-04 0.000E+00 4.035E-05 1.138E-03 1.112E-03 6.659E-05 0.000E+00 3.354E-04
plas.st 2.577E-04 1.679E-04 4.518E-21-8.989E-05-7.789E-05 1.678E-04 1.618E-04-7.196E-05-8.989E-05 7.540E-05
state vars   632.8

element    2  point   1       integration pt. coordinate=     0.790E+00    0.671E+00
stress  2.080E+04 2.000E+04-2.559E+04-3.310E+04-3.139E+04-1.230E+04-1.986E+04-2.553E+04-3.139E+04 1.001E+04
strain  1.238E-03 9.361E-04 3.491E-04-9.535E-05 0.000E+00 1.143E-03 6.926E-04 3.546E-04 0.000E+00 1.191E-03
plas.st 3.366E-04 2.176E-04 4.518E-21-1.193E-04-9.801E-05 2.173E-04 9.532E-05 2.687E-06-9.801E-05 3.236E-04
state vars   672.6

element    2  point   2       integration pt. coordinate=     0.916E+00    0.471E+00
stress  2.072E+04 2.000E+04-2.466E+04-3.207E+04-3.055E+04-1.135E+04-1.671E+04-2.672E+04-3.055E+04 9.071E+03
strain  1.295E-03 9.890E-04 3.737E-04-8.699E-05 0.000E+00 1.208E-03 8.740E-04 2.471E-04 0.000E+00 1.133E-03
plas.st 3.972E-04 2.582E-04 5.421E-20-1.393E-04-1.186E-04 2.579E-04 1.558E-04-3.725E-05-1.186E-04 3.471E-04
state vars   675.6

element    2  point   3       integration pt. coordinate=     0.892E+00    0.760E+00
stress  2.026E+04 2.000E+04-1.034E+04-1.727E+04-1.673E+04 2.992E+03-3.917E+03-9.826E+03-1.727E+04 9.411E+03
strain  1.144E-03 9.348E-04 3.929E-04 0.000E+00 3.436E-05 1.144E-03 7.559E-04 4.228E-04 0.000E+00 1.059E-03
plas.st 2.663E-04 1.739E-04 0.000E+00-9.249E-05-8.133E-05 1.738E-04 8.476E-05 7.728E-06-9.249E-05 2.433E-04
state vars   632.7

element    2  point   4       integration pt. coordinate=     0.102E+01    0.526E+00
stress  2.041E+04 2.000E+04-1.054E+04-1.763E+04-1.679E+04 2.781E+03-2.012E+03-1.199E+04-1.763E+04 8.415E+03
strain  1.207E-03 9.863E-04 4.207E-04 0.000E+00 5.521E-05 1.207E-03 9.251E-04 3.369E-04 0.000E+00 9.900E-04
plas.st 3.225E-04 2.090E-04 1.355E-20-1.138E-04-9.492E-05 2.087E-04 1.347E-04-2.092E-05-1.138E-04 2.607E-04
state vars   640.3
```

```
element    3  point   1       integration pt. coordinate=     0.471E+00    0.916E+00
stress  2.072E+04 2.000E+04-2.466E+04-3.207E+04-3.055E+04-1.135E+04-2.672E+04-1.671E+04-3.055E+04 9.071E+03
strain  1.295E-03 9.890E-04 3.737E-04-8.699E-05 0.000E+00 1.208E-03 2.471E-04 8.740E-04 0.000E+00 1.133E-03
plas.st 3.972E-04 2.582E-04 0.000E+00-1.393E-04-1.186E-04 2.579E-04-3.725E-05 1.558E-04-1.186E-04 3.471E-04
state vars   675.6

element    3  point   2       integration pt. coordinate=     0.671E+00    0.790E+00
stress  2.080E+04 2.000E+04-2.559E+04-3.310E+04-3.139E+04-1.230E+04-2.553E+04-1.986E+04-3.139E+04 1.001E+04
strain  1.238E-03 9.361E-04 3.491E-04-9.535E-05 0.000E+00 1.143E-03 3.546E-04 6.926E-04 0.000E+00 1.191E-03
plas.st 3.366E-04 2.176E-04-1.355E-20-1.193E-04-9.801E-05 2.173E-04 2.687E-06 9.532E-05-9.801E-05 3.236E-04
state vars   672.6

element    3  point   3       integration pt. coordinate=     0.526E+00    0.102E+01
stress  2.041E+04 2.000E+04-1.054E+04-1.763E+04-1.679E+04 2.781E+03-1.199E+04-2.012E+03-1.763E+04 8.415E+03
strain  1.207E-03 9.863E-04 4.207E-04 0.000E+00 5.521E-05 1.207E-03 3.369E-04 9.251E-04 0.000E+00 9.900E-04
plas.st 3.225E-04 2.090E-04 0.000E+00-1.138E-04-9.492E-05 2.087E-04-2.092E-05 1.347E-04-1.138E-04 2.607E-04
state vars   640.3

element    3  point   4       integration pt. coordinate=     0.760E+00    0.892E+00
stress  2.026E+04 2.000E+04-1.034E+04-1.727E+04-1.673E+04 2.992E+03-9.826E+03-3.917E+03-1.727E+04 9.411E+03
strain  1.144E-03 9.348E-04 3.929E-04 0.000E+00 3.436E-05 1.144E-03 4.228E-04 7.559E-04 0.000E+00 1.059E-03
plas.st 2.663E-04 1.739E-04-9.035E-21-9.249E-05-8.133E-05 1.738E-04 7.728E-06 8.476E-05-9.249E-05 2.433E-04
state vars   632.7

element    4  point   1       integration pt. coordinate=     0.841E-01    0.103E+01
stress  2.075E+04 2.000E+04-2.560E+04-3.305E+04-3.146E+04-1.230E+04-3.265E+04-1.271E+04-3.146E+04 2.861E+03
strain  1.225E-03 9.305E-04 3.492E-04-8.854E-05 0.000E+00 1.136E-03-6.468E-05 1.112E-03 0.000E+00 3.386E-04
plas.st 3.255E-04 2.108E-04 9.035E-21-1.149E-04-9.564E-05 2.106E-04-1.085E-04 2.041E-04-9.564E-05 9.059E-05
state vars   672.6

element    4  point   2       integration pt. coordinate=     0.314E+00    0.981E+00
stress  2.067E+04 2.000E+04-2.456E+04-3.193E+04-3.050E+04-1.125E+04-3.062E+04-1.256E+04-3.050E+04 5.028E+03
strain  1.287E-03 9.868E-04 3.751E-04-8.065E-05 0.000E+00 1.206E-03 8.750E-07 1.124E-03 0.000E+00 6.269E-04
plas.st 3.906E-04 2.543E-04 1.807E-20-1.365E-04-1.176E-04 2.541E-04-1.116E-04 2.291E-04-1.176E-04 1.911E-04
state vars   675.6

element    4  point   3       integration pt. coordinate=     0.931E-01    0.117E+01
stress  2.032E+04 2.000E+04-1.037E+04-1.736E+04-1.670E+04 2.960E+03-1.624E+04 2.491E+03-1.736E+04 3.000E+03
strain  1.138E-03 9.299E-04 3.928E-04 0.000E+00 4.035E-05 1.138E-03 6.659E-05 1.112E-03 0.000E+00 3.354E-04
plas.st 2.577E-04 1.679E-04 4.518E-21-8.989E-05-7.789E-05 1.678E-04-7.196E-05 1.618E-04-8.989E-05 7.540E-05
state vars   632.8

element    4  point   4       integration pt. coordinate=     0.348E+00    0.109E+01
stress  2.045E+04 2.000E+04-1.047E+04-1.760E+04-1.667E+04 2.848E+03-1.527E+04 1.452E+03-1.760E+04 5.029E+03
strain  1.205E-03 9.847E-04 4.217E-04 0.000E+00 6.049E-05 1.205E-03 1.425E-04 1.123E-03 0.000E+00 5.902E-04
plas.st 3.183E-04 2.059E-04-4.518E-21-1.127E-04-9.286E-05 2.056E-04-7.134E-05 1.840E-04-1.127E-04 1.544E-04
state vars   640.3
```

```
                    n o d a l   p o i n t   d a t a



                    i n c r e m e n t a l   d i s p l a c e m e n t s


   1 -9.47993E-05       0.               2 -9.00126E-05 -3.60978E-05           3 -6.63410E-05 -6.63411E-05
   4 -3.60978E-05 -9.00126E-05           5       0.     -9.47994E-05           6 -1.00565E-05 -3.04365E-06
   7 -3.04364E-06 -1.00566E-05           8  4.40939E-06       0.               9  4.71488E-06  4.71483E-06
  10       0.      4.40929E-06          11  4.14762E-05  1.82004E-05          12  1.82004E-05  4.14761E-05
  13  6.53441E-05       0.              14  4.77635E-05  4.77635E-05          15       0.      6.53440E-05
  16  9.64292E-05  4.03346E-05          17  4.03346E-05  9.64291E-05          18  7.62164E-05  7.62164E-05
  19  1.10368E-04       0.              20       0.      1.10367E-04          21  6.83232E-05       0.
  22       0.      6.83232E-05          23  6.42853E-05  3.80048E-05          24  3.80048E-05  6.42853E-05
  25  4.08700E-05  4.08700E-05          26       0.           0.              27       0.           0.
  28       0.           0.              29       0.           0.              30       0.           0.



                    t o t a l   d i s p l a c e m e n t s


   1 -1.42626E-04       0.               2 -1.31828E-04 -5.29149E-05           3 -9.96519E-05 -9.96521E-05
   4 -5.29148E-05 -1.31829E-04           5       0.     -1.42626E-04           6  8.82593E-05  3.82970E-05
   7  3.82971E-05  8.82591E-05           8  1.34533E-04       0.               9  9.74179E-05  9.74178E-05
  10       0.      1.34533E-04          11  1.97875E-04  8.42161E-05          12  8.42162E-05  1.97875E-04
  13  2.48580E-04       0.              14  1.78565E-04  1.78565E-04          15       0.      2.48580E-04
  16  2.59431E-04  1.13737E-04          17  1.13737E-04  2.59431E-04          18  1.99007E-04  1.99007E-04
  19  2.79354E-04       0.              20       0.      2.79354E-04          21  1.36245E-04       0.
  22       0.      1.36245E-04          23  1.25401E-04  7.50319E-05          24  7.50319E-05  1.25401E-04
  25  7.35447E-05  7.35447E-05          26       0.           0.              27       0.           0.
  28       0.           0.              29       0.           0.              30       0.           0.



              total equivalent nodal forces (distributed plus point loads)


   1       0.           0.               2       0.           0.               3       0.           0.
   4       0.           0.               5       0.           0.               6       0.           0.
   7       0.           0.               8       0.           0.               9       0.           0.
  10       0.           0.              11       0.           0.              12       0.           0.
  13       0.           0.              14       0.           0.              15       0.           0.
  16       0.           0.              17       0.           0.              18       0.           0.
  19       0.           0.              20       0.           0.              21       0.           0.
  22       0.           0.              23       0.           0.              24       0.           0.
  25       0.           0.              26       0.           0.              27       0.           0.
  28       0.           0.              29       0.           0.              30       0.           0.
```

```
        reaction forces at fixed boundary conditions, residual load correction elsewhere


     1   232.09        3681.3                 2   379.53       157.09                3   332.52       332.52
     4   157.09        379.53                 5   3681.3       232.09                6  -342.80      -144.98
     7  -144.98       -342.80                 8  -201.09       4392.1                9  -289.31      -289.31
    10   4392.1       -201.09                11 -9.09495E-13 -1.13687E-12           12 -1.36424E-12  1.81899E-12
    13 -2.27374E-13   3057.0                 14  1.36424E-12  4.54747E-13           15   3057.0          0.
    16 -1.36424E-12      0.                  17 -1.59162E-12 -3.41061E-12           18 -1.64846E-12  4.54747E-13
    19 -9.09495E-13   1012.0                 20   1012.0     -4.54747E-13           21  4.54747E-13   516.71
    22   516.71     -9.09495E-13             23  9.09495E-13 -9.09495E-13           24 -1.47793E-12 -1.36424E-12
    25 -5.68434E-13 -1.36424E-12             26  -3743.5       395.83               27   395.83      -3743.5
    28  -5879.2      -1519.1                 29  -1519.1      -5879.2               30  -2036.1      -2036.1

                        summary of externally applied loads

  0.00000E+00    0.00000E+00

                        summary of reaction/residual forces

 -0.18190E-11   -0.22737E-11

                        e n d   o f   i n c r e m e n t   4

                        binary post data at increment   4.   0   on tape 16
                         time =       2.17

     now asking for step    8  from temperature tape


    auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 2.185E+01
      maximum allowed on auto therm option is 5.000E+01
    maximum temp.change is less than 60p.c. of allowable - next temp. input will be read and merged into this set
      now asking for step    9  from temperature tape
    auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 4.384E+01
      maximum allowed on auto therm option is 5.000E+01
    maximum temp. change is between 60p.c. and 100p.c. of allowable - this set will be analysed in one increment



                        s t a r t   o f   i n c r e m e n t   5

                       ●                         ●
                       ●                         ●
                       ●                         ●
                       ●                         ●

                        e n d   o f   i n c r e m e n t   5
```

```
  now asking for step   10  from temperature tape


auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 2.455E+01
  maximum allowed on auto therm option is 5.000E+01
maximum temp.change is less than 60p.c. of allowable - next temp. input will be read and merged into this set
now asking for step   11  from temperature tape


auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 5.187E+01
  maximum allowed on auto therm option is 5.000E+01
maximum temperature change exceeds tolerance.
  this set will be subdivided into    1  equal steps which meet the maximum temp. change tolerance.
the remainder of the set has a maximum temp. change of 1.873E+00
and will be considered after the equal steps are analysed


                    s t a r t   o f   i n c r e m e n t   6

                  •                                           •
                  •                                           •
                  •                                           •
                  •                                           •

                    e n d   o f   i n c r e m e n t   6

                    binary post data at increment   6.   0   on tape 16
                     time =        2.95


  remainder of this temp. input set has maximum temp. change equal to0.037  of allowable
  this is less than 60p.c. of allowable and so will be merged with the next set
  now asking for step   12  from temperature tape


auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 2.956E+01
  maximum allowed on auto therm option is 5.000E+01
maximum temp.change is less than 60p.c. of allowable - next temp. input will be read and merged into this set
now asking for step   13  from temperature tape


auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 5.441E+01
  maximum allowed on auto therm option is 5.000E+01
maximum temperature change exceeds tolerance.
  this set will be subdivided into    1  equal steps which meet the maximum temp. change tolerance.
the remainder of the set has a maximum temp. change of 4.409E+00
and will be considered after the equal steps are analysed
```

```
                    s t a r t   o f   i n c r e m e n t   7

                    ●                                       ●
                    ●                                       ●
                    ●                                       ●
                    ●                                       ●


                    e n d   o f   i n c r e m e n t   7

                    binary post data at increment   7.   0   on tape 16
                     time =        3.35


   remainder of this temp. input set has maximum temp. change equal to0.088  of allowable
   this is less than 60p.c. of allowable and so will be merged with the next set
   now asking for step   14  from temperature tape


 auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 2.372E+01
   maximum allowed on auto therm option is 5.000E+01
 maximum temp.change is less than 60p.c. of allowable - next temp. input will be read and merged into this set
 now asking for step   15  from temperature tape

   continue
   ----------


 auto therm temperature change calculation, based on last temp. input, maximum temp. change supplied is 3.385E+01
   maximum allowed on auto therm option is 5.000E+01
 maximum temp. change is between 60p.c. and 100p.c. of allowable - this set will be analysed in one increment


                    s t a r t   o f   i n c r e m e n t   8


                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
0.000E+00 0.000E+00


                    point loads
0.000E+00 0.000E+00

                    start of assembly
                     time =        3.37


                    start of matrix solution
```

```
                               time =        3.48



                              singularity ratio    2.6073E-01


                              end of matrix solution
                               time =        3.49


                             maximum residual force  at node   14 degree of freedom    2 is equal to    0.268E+03
                             maximum reaction force  at node   29 degree of freedom    2 is equal to    0.439E+05
                             convergence ratio                                                           0.610E-02

 MARC           output for increment    8.     thermal stress given temperature time history

         tresca     mises    mean   p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
       intensity intensity  normal   minimum intermediate maximum     1         2         3         4         5         6
                            intensity

element    1  point   1       integration pt. coordinate=     0.981E+00    0.314E+00
stress  2.304E+04 2.000E+04-2.810E+04-3.914E+04-2.904E+04-1.611E+04-1.786E+04-3.739E+04-2.904E+04 6.109E+03
strain  6.646E-03 4.112E-03 8.534E-04-2.043E-03 0.000E+00 4.603E-03 4.134E-03-1.574E-03 0.000E+00 3.403E-03
plas.st 5.648E-03 3.360E-03-3.614E-20-2.418E-03-8.124E-04 3.230E-03 2.837E-03-2.025E-03-8.124E-04 2.873E-03
state vars   807.0

element    1  point   2       integration pt. coordinate=     0.103E+01    0.841E-01
stress  2.303E+04 2.000E+04-2.880E+04-3.984E+04-2.977E+04-1.680E+04-1.746E+04-3.918E+04-2.977E+04 3.829E+03
strain  6.634E-03 4.095E-03 8.367E-04-2.062E-03 0.000E+00 4.572E-03 4.410E-03-1.900E-03 0.000E+00 2.046E-03
plas.st 5.636E-03 3.350E-03-4.698E-19-2.421E-03-7.950E-04 3.215E-03 3.082E-03-2.287E-03-7.950E-04 1.714E-03
state vars   805.2

element    1  point   3       integration pt. coordinate=     0.109E+01    0.348E+00
stress  2.281E+04 2.000E+04-2.951E+03-1.332E+04-5.031E+03 9.495E+03 7.744E+03-1.156E+04-5.031E+03 6.073E+03
strain  5.897E-03 3.903E-03 1.102E-03-1.296E-03 0.000E+00 4.602E-03 4.176E-03-8.696E-04 0.000E+00 3.054E-03
plas.st 4.909E-03 3.009E-03 0.000E+00-1.949E-03-1.012E-03 2.960E-03 2.610E-03-1.598E-03-1.012E-03 2.527E-03
state vars   785.3

element    1  point   4       integration pt. coordinate=     0.117E+01    0.931E-01
stress  2.285E+04 2.000E+04-4.829E+03-1.528E+04-6.774E+03 7.567E+03 6.907E+03-1.462E+04-6.774E+03 3.825E+03
strain  5.968E-03 3.902E-03 1.057E-03-1.398E-03 0.000E+00 4.570E-03 4.419E-03-1.247E-03 0.000E+00 1.875E-03
plas.st 4.978E-03 3.035E-03 3.614E-20-2.003E-03-9.729E-04 2.976E-03 2.853E-03-1.880E-03-9.729E-04 1.544E-03
state vars   780.4

element    2  point   1       integration pt. coordinate=     0.790E+00    0.671E+00
stress  2.304E+04 2.000E+04-2.880E+04-3.987E+04-2.971E+04-1.683E+04-2.476E+04-3.193E+04-2.971E+04 1.095E+04
strain  6.764E-03 4.165E-03 8.355E-04-2.129E-03 0.000E+00 4.635E-03 2.260E-03 2.462E-04 0.000E+00 6.457E-03
plas.st 5.766E-03 3.423E-03 2.168E-19-2.485E-03-7.961E-04 3.281E-03 1.250E-03-4.536E-04-7.961E-04 5.509E-03
state vars   804.9

element    2  point   2       integration pt. coordinate=     0.916E+00    0.471E+00
stress  2.305E+04 2.000E+04-2.909E+04-4.020E+04-2.992E+04-1.715E+04-2.267E+04-3.469E+04-2.992E+04 9.834E+03
```

```
strain  6.725E-03 4.146E-03 8.398E-04-2.103E-03 0.000E+00 4.622E-03 2.962E-03-4.422E-04 0.000E+00 5.799E-03
plas.st 5.726E-03 3.402E-03 1.446E-19-2.461E-03-8.039E-04 3.265E-03 1.843E-03-1.040E-03-8.039E-04 4.947E-03
state vars   806.9

element    2  point   3       integration pt. coordinate=     0.892E+00    0.760E+00
stress  2.285E+04 2.000E+04-4.281E+03-1.474E+04-6.207E+03 8.108E+03 1.962E+02-6.832E+03-6.207E+03 1.087E+04
strain  6.075E-03 3.961E-03 1.063E-03-1.443E-03 0.000E+00 4.632E-03 2.505E-03 6.835E-04 0.000E+00 5.795E-03
plas.st 5.084E-03 3.095E-03 0.000E+00-2.053E-03-9.794E-04 3.032E-03 1.248E-03-2.688E-04-9.794E-04 4.853E-03
state vars   780.0

element    2  point   4       integration pt. coordinate=     0.102E+01    0.526E+00
stress  2.282E+04 2.000E+04-3.264E+03-1.366E+04-5.294E+03 9.164E+03 3.660E+03-8.157E+03-5.294E+03 9.764E+03
strain  5.950E-03 3.926E-03 1.097E-03-1.329E-03 0.000E+00 4.621E-03 3.159E-03 1.330E-04 0.000E+00 5.123E-03
plas.st 4.961E-03 3.037E-03 0.000E+00-1.976E-03-1.009E-03 2.985E-03 1.762E-03-7.523E-04-1.009E-03 4.277E-03
state vars   785.2

element    3  point   1       integration pt. coordinate=     0.471E+00    0.916E+00
stress  2.305E+04 2.000E+04-2.909E+04-4.020E+04-2.992E+04-1.715E+04-3.469E+04-2.267E+04-2.992E+04 9.834E+03
strain  6.725E-03 4.146E-03 8.398E-04-2.103E-03 0.000E+00 4.622E-03-4.422E-04 2.962E-03 0.000E+00 5.799E-03
plas.st 5.726E-03 3.402E-03 7.228E-20-2.461E-03-8.039E-04 3.265E-03-1.040E-03 1.843E-03-8.039E-04 4.947E-03
state vars   806.9

element    3  point   2       integration pt. coordinate=     0.671E+00    0.790E+00
stress  2.304E+04 2.000E+04-2.880E+04-3.987E+04-2.971E+04-1.683E+04-3.193E+04-2.476E+04-2.971E+04 1.095E+04
strain  6.764E-03 4.165E-03 8.355E-04-2.129E-03 0.000E+00 4.635E-03 2.462E-04 2.260E-03 0.000E+00 6.457E-03
plas.st 5.766E-03 3.423E-03 2.168E-19-2.485E-03-7.961E-04 3.281E-03-4.536E-04 1.250E-03-7.961E-04 5.509E-03
state vars   804.9

element    3  point   3       integration pt. coordinate=     0.526E+00    0.102E+01
stress  2.282E+04 2.000E+04-3.264E+03-1.366E+04-5.294E+03 9.164E+03-8.157E+03 3.660E+03-5.294E+03 9.764E+03
strain  5.950E-03 3.926E-03 1.097E-03-1.329E-03 0.000E+00 4.621E-03 1.330E-04 3.159E-03 0.000E+00 5.123E-03
plas.st 4.961E-03 3.037E-03-1.446E-19-1.976E-03-1.009E-03 2.985E-03-7.523E-04 1.762E-03-1.009E-03 4.277E-03
state vars   785.2

element    3  point   4       integration pt. coordinate=     0.760E+00    0.892E+00
stress  2.285E+04 2.000E+04-4.281E+03-1.474E+04-6.207E+03 8.108E+03-6.832E+03 1.962E+02-6.207E+03 1.087E+04
strain  6.075E-03 3.961E-03 1.063E-03-1.443E-03 0.000E+00 4.632E-03 6.835E-04 2.505E-03 0.000E+00 5.795E-03
plas.st 5.084E-03 3.095E-03 7.228E-20-2.053E-03-9.794E-04 3.032E-03-2.688E-04 1.248E-03-9.794E-04 4.853E-03
state vars   780.0

element    4  point   1       integration pt. coordinate=     0.841E-01    0.103E+01
stress  2.303E+04 2.000E+04-2.880E+04-3.984E+04-2.977E+04-1.680E+04-3.918E+04-1.746E+04-2.977E+04 3.829E+03
strain  6.634E-03 4.095E-03 8.367E-04-2.062E-03 0.000E+00 4.572E-03-1.900E-03 4.410E-03 0.000E+00 2.046E-03
plas.st 5.636E-03 3.350E-03 1.446E-19-2.421E-03-7.950E-04 3.215E-03-2.287E-03 3.082E-03-7.950E-04 1.714E-03
state vars   805.2

element    4  point   2       integration pt. coordinate=     0.314E+00    0.981E+00
stress  2.304E+04 2.000E+04-2.810E+04-3.914E+04-2.904E+04-1.611E+04-3.739E+04-1.786E+04-2.904E+04 6.109E+03
strain  6.646E-03 4.112E-03 8.534E-04-2.043E-03 0.000E+00 4.603E-03-1.574E-03 4.134E-03 0.000E+00 3.403E-03
plas.st 5.648E-03 3.360E-03-2.891E-19-2.418E-03-8.124E-04 3.230E-03-2.025E-03 2.837E-03-8.124E-04 2.873E-03
state vars   807.0

element    4  point   3       integration pt. coordinate=     0.931E-01    0.117E+01
stress  2.285E+04 2.000E+04-4.829E+03-1.528E+04-6.774E+03 7.567E+03-1.462E+04 6.907E+03-6.774E+03 3.825E+03
```

```
strain  5.968E-03 3.902E-03 1.057E-03-1.398E-03 0.000E+00 4.570E-03-1.247E-03 4.419E-03 0.000E+00 1.875E-03
plas.st 4.978E-03 3.035E-03 1.446E-19-2.003E-03-9.729E-04 2.976E-03-1.880E-03 2.853E-03-9.729E-04 1.544E-03
state vars   780.4

element    4  point   4       integration pt. coordinate=      0.348E+00    0.109E+01
stress  2.281E+04 2.000E+04-2.951E+03-1.332E+04-5.031E+03 9.495E+03-1.156E+04 7.744E+03-5.031E+03 6.073E+03
strain  5.897E-03 3.903E-03 1.102E-03-1.296E-03 0.000E+00 4.602E-03-8.696E-04 4.176E-03 0.000E+00 3.054E-03
plas.st 4.909E-03 3.009E-03 7.228E-20-1.949E-03-1.012E-03 2.960E-03-1.598E-03 2.610E-03-1.012E-03 2.527E-03
state vars   785.3


                          n o d a l   p o i n t   d a t a


                          i n c r e m e n t a l   d i s p l a c e m e n t s


   1 -9.11297E-04       0.                  2 -7.91868E-04 -3.17663E-04           3 -6.31546E-04 -6.31546E-04
   4 -3.17663E-04 -7.91868E-04              5       0.     -9.11298E-04           6 -6.13576E-04 -2.47628E-04
   7 -2.47628E-04 -6.13576E-04              8 -6.68174E-04       0.               9 -4.51748E-04 -4.51748E-04
  10       0.     -6.68174E-04             11 -4.76601E-04 -1.98942E-04          12 -1.98942E-04 -4.76601E-04
  13 -4.67231E-04       0.                 14 -3.06865E-04 -3.06865E-04          15       0.     -4.67232E-04
  16 -1.94071E-04 -1.11768E-04             17 -1.11768E-04 -1.94071E-04          18 -1.27671E-04 -1.27671E-04
  19 -1.57079E-04       0.                 20       0.     -1.57080E-04          21  3.77721E-05       0.
  22       0.      3.77720E-05             23  3.57779E-05  2.99258E-05          24  2.99258E-05  3.57778E-05
  25  5.27782E-05  5.27782E-05             26       0.           0.              27       0.           0.
  28       0.           0.                 29       0.           0.              30       0.           0.
                          t o t a l   d i s p l a c e m e n t s
```

Note symmetry

```
   1 -2.44333E-03       0.                  2 -2.14041E-03 -8.66905E-04           3 -1.70474E-03 -1.70474E-03
   4 -8.66905E-04 -2.14041E-03              5       0.     -2.44333E-03           6 -1.31367E-03 -5.29610E-04
   7 -5.29610E-04 -1.31367E-03              8 -1.31988E-03       0.               9 -8.96442E-04 -8.96442E-04
  10       0.     -1.31988E-03             11 -7.45904E-04 -3.06356E-04          12 -3.06356E-04 -7.45905E-04
  13 -5.89009E-04       0.                 14 -3.74196E-04 -3.74197E-04          15       0.     -5.89011E-04
  16  3.12180E-05 -5.42125E-05             17 -5.42123E-05  3.12172E-05          18  9.33517E-05  9.33514E-05
  19  1.71719E-04       0.                 20       0.      1.71717E-04          21  5.14651E-04       0.
  22       0.      5.14651E-04             23  4.94295E-04  3.30244E-04          24  3.30244E-04  4.94294E-04
  25  4.15607E-04  4.15607E-04             26       0.           0.              27       0.           0.
  28       0.           0.                 29       0.           0.              30       0.           0.



                 total equivalent nodal forces (distributed plus point loads)


   1        0.          0.                  2        0.          0.               3        0.          0.
   4        0.          0.                  5        0.          0.               6        0.          0.
   7        0.          0.                  8        0.          0.               9        0.          0.
  10        0.          0.                 11        0.          0.              12        0.          0.
  13        0.          0.                 14        0.          0.              15        0.          0.
  16        0.          0.                 17        0.          0.              18        0.          0.
  19        0.          0.                 20        0.          0.              21        0.          0.
  22        0.          0.                 23        0.          0.              24        0.          0.
  25        0.          0.                 26        0.          0.              27        0.          0.
```

```
         28      0.         0.                  29      0.         0.                 30      0.          0.



              reaction forces at fixed boundary conditions, residual load correction elsewhere



          1  -24.418       4090.1               2  -100.75      -27.670               3  -48.840      -48.840
          4  -27.670      -100.75               5   4090.1      -24.418               6   76.791       20.320
          7   20.320       76.791               8   90.974       7195.6               9   101.48       101.48
         10   7195.6       90.974              11   185.42       101.44              12   101.44       185.42
         13   190.96       14375.              14   267.93       267.93              15   14375.       190.96
         16  -86.438       13.701              17   13.701      -86.437              18  -47.525      -47.525
         19  -15.802       35592.              20   35592.      -15.802              21  3.63798E-12   26947.
         22   26947.     -3.63798E-12          23 -3.63798E-12  1.81899E-12          24       0.     1.27329E-11
         25 -2.72848E-12  3.63798E-12          26  -25351.       6014.3              27   6014.3      -25351.
         28  -43889.      -8199.4              29  -8199.3      -43889.              30  -17473.      -17473.

                          summary of externally applied loads

  0.00000E+00    0.00000E+00

                          summary of reaction/residual forces

 -0.46043E-11    0.16371E-10

                          e n d   o f   i n c r e m e n t   8

                          binary post data at increment   8.   0   on tape 16
                           time =       3.75



                          *** end of input deck - job ends




                                       marc exit number 3004
```

Results

Let us look at the displacement field after the completion of thermal stress analysis in increment 9. For nodes 1 to 5 next to the circular channel:

Node	X-displacement (in.)	Y-displacement (in.)
1 (at X-axis)	-0.002443	0.000000
2	-0.002140	-0.000867
3 (at 45° line)	-0.001705	-0.001705
4	-0.000867	-0.002140
5 (at Y-axis)	0.000000	-0.002443

The displacement field is indeed symmetrical about a 45° diagonal, as it should be. (You should develop a habit to always check the computer results, to see if they make sense.)

The figure shows the final equivalent von Mises tensile stress distribution in the model. Note that the stress distribution is also symmetrical about the 45° diagonal. The first two rows of elements (and part of the third row) closest to the circular channel have yielded due to the temperature increase. Therefore, this example demonstrates that a structure can yield due to thermal stresses also.

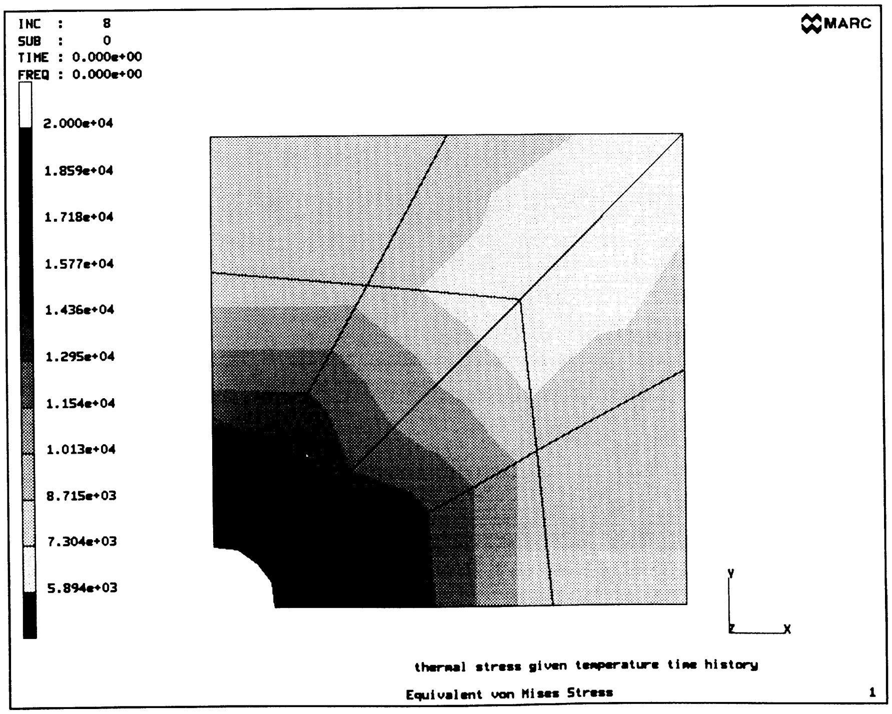


Figure 9.2 **Equivalent von Mises Stress**

Exercises

Try Example 9 with the following changes (applied only one at a time):

1. Remove the displacement constraints along the two outer edges (node set OUTEDGE), rerun the stress analysis, and compare the resulting displacements and stresses with those from Example 9.
2. Tighten the CONTROL tolerance from 0.1 to 0.05.
3. Reduce the maximum allowed temperature change in AUTO THERM from 50°F to 25°F.

Do you think this problem can be analyzed using a one-eighth model?

CHAPTER 5: Contact and Rubber Analyses

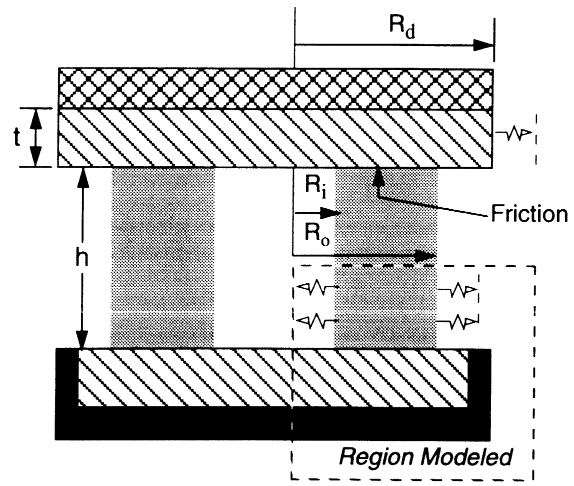
This final chapter discusses two additional nonlinear examples, both of which involve 2-D contact analysis using the automated contact/friction capability (without the use of gap elements. Example 10 describes the large deformation analysis of an aluminum ring being deformed by a steel disk, featuring thermal-mechanical coupling and automated contact. And Example 11 illustrates the side pressing of a solid rubber cylinder, where the rubber is idealized as a Mooney-Rivlin material and undergoes large deformations.

Example 10

Coupled Analysis of Ring Compression

The purpose of Example 10 is to demonstrate the large deformation, coupled thermal-mechanical, contact analysis of a ring under compression. An aluminum ring is deformed by a steel disk such that there is a 50% reduction in height of the ring. Both are deformable bodies, and can slide with respect to each other. Friction is considered. MARC options seen here for the first time are: PRINT,8; FINITE; UPDATE; COUPLE; CONVERT; WORK HARD DATA; DIST FLUXES; CONTACT; TRANSIENT NON AUTO; and DISP CHANGE.

Sketch



R_i = 13.5 mm h = 18 mm
R_o - 27.0 mm t = 6 mm
R_d = 42 mm

Convective film coefficient: Aluminum ring to air	0.01 W/mm°C
Convective film coefficient: Steel disk to air	0.01 W/mm°C
Heat transfer coefficient: Aluminum ring to steel disk	35.00 W/mm°C
Friction (shear) coefficient	1.00 W/mm°C

▨ Steel – Initial Temperature 20°C
▒ Aluminum – Initial Temperature 427°C
☐ Air – Temperature 20°C
■ Fixed Temperature of 20°C
▩ Prescribed displacement in axial direction

Material Properties		
	Aluminum	**Steel**
Young's Modulus, E	10000 N/mm^2	100000 N/mm^2
Poisson's ratio, ν	.33	.33
density, ρ	1 g/mm^3	1 g/mm^3
coefficient of therm expansion (α)	1.3-5 mm/mm°C	0.
yield stress, $\bar{\sigma}$	3.4 N/mm^2	elastic
$\partial\bar{\sigma}/\partial T$	-0.007 N/mm^2/°C	--
conductivity κ	242 N/s°C	19 N/s°C
specific heat c	2.4255 N-mm/g°C	3.77 N-mm/g°C
initial temperature	427°C	20°C

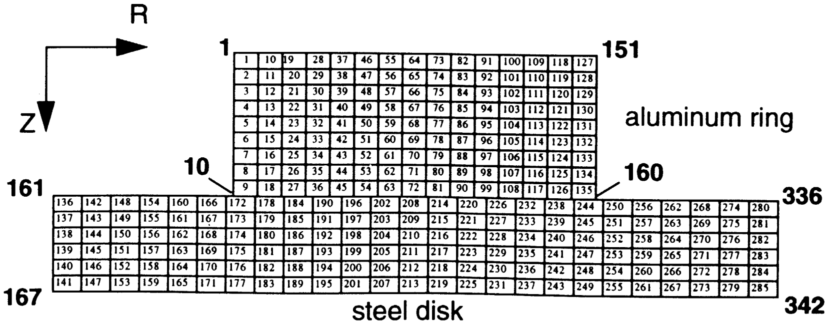


Figure 10.1 Ring Model

Model

The aluminum ring is deformed by the steel disk. Both bodies are deformable. Separate nodes exist along both sides of the contact surfaces so that sliding is possible. No gap elements are used. The interface between the two is assumed to have cohesive friction with a friction factor of 1.0 (the resisting friction stress will be equal to the flow stress in the adjacent material). Free surfaces have convective heat transfer to the environment. As soon as contact is detected, a contact thermal barrier, defined using a film coefficient, starts operating between the two bodies.

Mentat II was used to generate the CONNECTIVITY and COORDINATES data. The steel disk has a radius of 42.0 mm and a Z-thickness of 6.0 mm; it is modeled with 150 quadrilateral elements (numbered 136 to 285). The aluminum annular ring has an inner radius of 13.5 mm, an outer radius of 27.0 mm, and a Z-thickness of 9.0 mm. It is modeled with 135 quadrilateral elements, numbered 1 to 135. The entire model has 342 nodes.

MARC Element 10 (*see MARC Volume B*) is used for the axisymmetric analysis. It is a four-noded linear axisymmetric element, with two translational DOFs at each node and four Gaussian integration points. In a coupled analysis, a displacement element automatically produces the coupled (displacement-temperature) formulation to be used.

Properties

The aluminum ring is assumed to be an elastic-plastic material (material-id number of 1), with the following properties: Young's modulus of 10,000 N/mm^2; Poisson's ratio of 0.33; mass density of 1.0 g/mm^3; coefficient of thermal expansion of 1.3 x 10^{-5} mm/mm°C; initial (stress-free) temperature of 200°C; and equivalent tensile yield stress of 3.4 N/mm^2. The material work hardens from the initial yield stress to a yield stress of 5.78 N/mm^2 for strains above 70%, according to a piecewise linear function entered using the WORK HARD DATA option. The flow stress decreases with temperature at a rate of 0.007 N/mm^2 per degree. The thermal properties are: thermal conductivity of 242 N/s°C; and specific heat of 2.4255 N-mm/g°C.

The steel disk is treated as an elastic material (material-id number of 2), with the following properties: Young's modulus of 100,000 N/mm^2; Poisson's ratio of 0.33; and mass density of 1.0 g/mm^3. Its thermal properties are: thermal conductivity of 19.0 n/s°C; and specific heat of 3.77 N-mm/g°C.

Loads

Mechanical loads on the model occur because the steel disk compresses the aluminum ring. Distributed flux loads occur due to the fact that internal heat is generated due to the plastic deformation and friction.

Boundary Conditions

Symmetry displacement boundary conditions are imposed on the ring meridian plane and on the disk axis (using the FIXED DISPLACEMENT option). The disk is moved down by application of displacement boundary conditions on the face opposite to the contact face. Space for such boundary conditions is reserved in the MODEL DEFINITION section. On the outside surface of the disk, the temperature is constrained to 20°C to simulate a much larger size disk (using the FIXED TEMPERATURE option). This is illustrated in the sketch of the first page of Example 10.

Special Features

The first important special feature in this problem is the use of the COUPLE option for coupled thermal-mechanical analysis. There are at least five sources of coupling in this analysis.

1. As the temperature changes, thermal stresses are developed due to a non-zero coefficient of thermal expansion (and the presence of boundary constraints).
2. As the temperature changes, the mechanical properties change. In this example, this change is caused by 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.

To perform the finite deformation analysis, we're introducing, for the first time, the use of these features (in the PARAMETER section): FINITE and UPDATE, in addition to the LARGE DISP option. These flag the finite (large) strain plasticity option and the updated Lagrange option, respectively. These three options should always be used for metal forming simulation. The PRINT,8 option was also requested in order to obtain additional printout information regarding nodes which acquire or lose contact.

The key new feature illustrated in this example is CONTACT, the automated contact analysis option without the use of gap elements. Here, the application is frictional contact between two deformable metallic bodies. (In Example 11, CONTACT will also be used to illustrate contact between a deformable rubber body and a rigid surface.) To control the contact analysis, we'll use the TRANSIENT NON AUTO option (which suppresses automatic time stepping) and the DISP CHANGE option (which allows us to add new displacement boundary conditions or modify old displacement conditions).

Other miscellaneous options include CONVERT (input of a heat generation conversion factor between inelastic mechanical energy and heat transfer flux in a coupled thermal-mechanical analysis); WORK HARD DATA (direct input of equivalent stress and equivalent plastic strain data lying on the stress-strain curve); NO PRINT (which suppresses element and nodal output, thus avoiding the voluminous printout this example would have generated); and DIST FLUXES (specifying distributed surface or volume fluxes).

Input

A complete input echo is included.

PARAMETER Section

The "TITLE" lines are self-explanatory. The "SIZING" line sets a workspace of 300,000 words. This large workspace is needed because of the model size. The "ELEMENTS" line says we'll use MARC Element 10.

The PRINT,8 option is a special PARAMETER printing option which allows us to print out additional contact analysis information regarding nodes touching or separating from surfaces.

The FINITE option flags the large strain option, which accounts for the effects of the change in metric due to large inelastic deformations. The use of this option insures a correct formulation for the elastic-plastic relationship. This option may only be used in combination with the UPDATE option.

The LARGE DISP option is used for all large displacement problems. You've seen its previous applications in Examples 6 and 7.

The UPDATE option flags the updated Lagrange procedure for those elements with this capability (such as Element 10). Its use has two consequences. First, the element stiffnesses will be assembled in the current configuration of the element. Second, the stress and strain output will be given in the coordinate system which is applicable to the updated configuration of the element. In this case, where continuum elements are used, it is the global Z-R system. This option may be used with or without the LARGE DISP option. In this case, since we're using the LARGE DISP option, the effect of the internal stresses on the stiffness of the structure is taken into account. Also, the strain increment will be calculated to second order accuracy, thus allowing for large rotation increments. Since this is also a coupled thermal-stress analysis, the element conductivity and specific heat will be assembled based on the current configuration of the element.

The COUPLE option provides for a coupled thermal-stress analysis. It is a very powerful capability found in very few FE codes which solves, within each increment, both the heat transfer problem and the structural problem, achieving both thermal and mechanical equilibrium. Therefore, the independent variables are temperature and displacement. Note that in the MODEL DEFINITION section, you must include a "DIST FLUXES" line (with a flux type of 101) to obtain the coupling between plastic work and internal heat generated, as well as the CONVERT option to properly convert inelastic mechanical energy to thermal energy.

The "END" line terminates the PARAMETER section.

MODEL DEFINITION section

This section consists of:

a. FE mesh topology

b. Boundary conditions

c. Nonlinear analysis controls
d. Output controls
e. Fixed and initial temperatures
f. Material properties
g. Conversion of inelastic mechanical energy to thermal energy
h. Work hardening data
i. Temperature dependence of properties
j. Suppression of element and nodal printout
k. Geometric properties
l. Distributed fluxes
m. Contact analysis controls

FE Mesh Topology

The axisymmetric FE model has 285 elements and 342 nodes, and consists of two rectangular parts: a 135-element aluminum ring and a 150-element steel disk. It was generated using Mentat II. The "CONNECTIVITY" and "COORDINATES" lines are self-explanatory. The elements in the aluminum ring are numbered 1 TO 135, and in the steel disk 136 to 285. Note that in the "COORDINATES" lines, the first column is the node number, the second column is the Z-coordinate (mm), and in the third column is the R-coordinate (mm). Element type 10 is used in the analysis; in a coupled analysis, heat transfer element type 40 is automatically generated. If element type 40 was used for a specific part of the mesh, then that region would be rigid.

Boundary Conditions

The FIXED DISP option is used to prescribe displacement boundary conditions. (Note, however, that these are later changed by use of the DISP CHANGE option.) After the blank line, the "0.0" line is the value of the displacements which are to be defined. The "1" line refers to the DOF (Z-displacement) we want to prescribe these zero-valued displacements. And the next line (naming nodes 1 TO 151 BY 10 AND 167 TO 342 BY 7) gives the node numbers for the 42 nodes we want these zero displacements to apply to, which are the bottom and top edges of the model.

The second "0.0" line followed by the "2" line again indicates that zero displacements will be prescribed for the second DOF (R-displacement). The "161 TO 167" line names the seven nodes on the axis of symmetry which are affected.

Nonlinear Analysis Controls

The CONTROL option lets you input parameters which control the convergence and accuracy of the nonlinear analysis. The "161,15,0,1,0,," line means: the maximum number of load steps is 161; the maximum number of recycles during an increment is 15; the minimum number of recycles is zero; the flag of 1 indicates the convergence testing is done on displacements; and the final zero is the flag for relative error testing. Leaving the sixth field blank implies we will use the default full Newton-Raphson iterative scheme. The ".15," line means a relative error of 15%, which is the maximum allowable value of the change in displacement in an iteration divided by the displacement in the increment. If displacement testing is used, then there will be at least one iteration per increment. And, the ",,10.," line means the maximum error in temperature estimate (for property evaluation) is 10°C. It is only necessary to specify this if there are temperature dependent material properties. The first tolerance (default of 20°C) controls the automatic time stepping, which is not used. The second tolerance (default 100°C) governs reassembly. In a coupled analysis, the heat transfer matrices are reassembled every increment.

Output Controls

The POST block creates a post processing file for Mentat II. The "6,,,1,1,,,,50," line means: there are six element variables to be written to the file; the 1 in the fourth field flags a formatted post file; the 1 in the fifth field tells MARC to write connectivity and coordinates on the post file; and the 50 in the ninth field says to write the post data every 50 increments. The next six lines identify the element variables: 7 is equivalent plastic strain; 11, 12, 13, 14 are the four stress components in the Z-, R-, and hoop directions plus the shear stress in the Z-R plane; and 17 is equivalent Mises stress.

Fixed and Initial Temperatures

The FIXED TEMPERATURE option defines the fixed temperature that each node must take during the first and subsequent increments. The "1," line indicates there is one such set of boundary conditions. The "20.," line means the prescribed temperature is 20°C. The "167 TO 342 BY 7 AND 336 TO 341" line tells MARC this temperature is to be applied to these nodes along the bottom and right edges of the steel disk.

The INITIAL TEMPERATURE option defines initial temperatures in the heat transfer analysis. The "2," line indicates there are two such sets of initial temperatures to be defined. The "427.," and "1 TO 160" lines are the first set, which applies an initial temperature of 427°C to nodes 1 to 160 (all the nodes in the aluminum ring). The "20.," and "161 TO 342" lines prescribe an initial temperature of 20°C to nodes 161 to 342 (all the nodes in the steel disk).

Notice that MARC does not know per se that the units for temperature are in degrees Celsius (as in this example). It is up to you, the user, to be consistent, and to make sure that these temperature units are consistent with those you use for thermal properties, such as the coefficient of thermal expansion, thermal conductivity, specific heat, etc.

Material Properties

Instead of the ISOTROPIC option used in previous examples, here we'll use the PROPERTY option to input properties for the coupled thermal-stress analysis. The "2," line indicates two sets of element properties will be defined.

First, the aluminum ring properties are input. The next line shows that Young's modulus is 10,000., Poisson's ratio is 0.33, coefficient of thermal expansion is 1.3E-5, reference temperature is 200., initial yield stress is 3.4, and the material-id number is 1. The "242.,2.4255,1.," line gives the thermal conductivity, specific heat, and mass density, respectively. The "1 TO 135" line names the 135 aluminum elements to which the above properties apply.

Then, the next three lines define the steel disk properties. The first of these shows that Young's modulus is 100,000.0, Poisson's ratio is 0.33, mass density is 1., and the material-id number is 2. Thermal expansion was not taken into consideration, and no reference temperature was necessary because none of the steel material properties were made temperature-independent. No yield stress is specified, so a large default value is chosen. This insures that the steel disk will remain elastic. The "19.,3.77,1.," line gives the thermal conductivity, specific heat, and mass density, respectively. The "136 TO 285" line assigns these properties to all the elements in the steel disk.

Conversion of Inelastic Mechanical Energy to Thermal Energy

The CONVERT option allows us to input a conversion factor between the mechanical energy and thermal energy in a coupled thermal-stress analysis. The "1.," line shows the conversion factor for this problem, which also happens to be the default value. If all of the inelastic energy will not be converted into heat, a factor can also be included through this option. For most problems involving ductile metals, a factor between 80% and 90% is appropriate. In addition, this option can be used if inconsistent units are used between the mechanical system and the thermal system, such as in the English system. The CONVERT option is used in conjunction with the DIST FLUXES option.

Work Hardening Data

The WORK HARD option enables us to input work hardening data, in this case for the aluminum material. Use of the word DATA after WORK HARD flags the input option of directly entering the stress and plastic strain data points from the stress-strain curve. The "4,0,1" line tells MARC there are four data points to be entered, and these are for material-id number 1. The next four lines give four pairs of equivalent stress and the associated equivalent plastic strain. The "3.4,0.," line give the initial yield stress value of 3.4, where the equivalent plastic strain is of course zero. (This initial yield stress must be identical to that given on the ISOTROPIC or PROPERTY option.) The "5.1,0.15," line now defines the second data point, where the equivalent stress is 5.1 and the equivalent plastic strain is 0.15. Above strains of 70%, the equivalent stress is 5.78. This behavior is defined by the third and fourth data lines, "5.78,0.7," and "5.78,5.,".

NOTE

As the UPDATED LAGRANGE and FINITE STRAIN options are used, the stress/plastic strain data should be obtained from the Cauchy (true) stress vs. logarithmic strain derived from a uniaxial test.

Temperature Dependence of Properties

The TEMPERATURE EFFECTS option defines the variation of element properties with temperature. The yield stress decreases with temperature increases at a rate of 0.007 N/mm^2. In the "1,,,,,,,,,1," line, the 1 in the first field indicates that there is one slope in the yield stress versus temperature curve, and the 1 in the last field is the material-id type. The "-0.007,200.," line gives the negative slope of the yield stress versus temperature curve, and the reference temperature of 200.0 at which the slope becomes operative.

Suppression of Element and Nodal Printout

The NO PRINT option suppresses element and nodal printout, thus limiting the printout to a minimum.

Geometric Properties

The GEOMETRY option is for inputting element geometry. The "1," line says one set of element geometries will be input. The "0.,1.," line indicates that the second field (EGEOM2) is "1.,"; since this value is non-zero, the volume strain will be constant throughout the element. This "constant dilatation" option should always be used for the analysis of structures in the fully plastic range when lower order elements are used. The "1 TO 285" line means this geometric property applies to all 285 elements in the model.

Distributed Fluxes

The DIST FLUXES option lets you specify distributed (surface and volumetric) fluxes. Flux type 101 is used to indicate that internal heat is generated due to plastic deformation. The magnitude is ignored for this load type. The "1," line says one set of distributed fluxed will be given. The "1 TO 285" line applies these fluxes to all 285 elements.

Contact Analysis Controls

The final block in the MODEL DEFINITION section is the CONTACT option. This option allows you to perform automated contact analysis without use of gap elements for rigid-to-deformable contact as well as deformable-to-deformable contact. In this example, the option calls for deformable-to-deformable contact of two bodies with cohesive (shear) friction between them. That is, the frictional stress is proportional to the flow stress in the material. This model is often more appropriate than the adhesive model (Coulomb) where large normal stresses are developed. MARC calculates both the contact tolerance and the sticking tolerance. The first two data lines in the block give the overall control parameters of the contact problem. The "2,65,65,1" line indicates: two surfaces (bodies) will be defined; there are a maximum of 65 entities to be created for any body; the maximum number of nodes that lie on the periphery of any deformable surface is also 65; and the friction type is 1 (shear or cohesive friction). In a contact problem, the boundary of a body is defined by "entities". The number of entities for a deformable body is the number of element edges on the boundary. The ",," line says you want MARC to use the default value of the relative sliding velocity between surfaces below which friction forces drop ("sticking"), and calculate the distance below which a node is considered touching a surface. This value is calculated based upon the model geometry.

The first deformable body to be specified in the aluminum ring. No motion of this body needs to be specified. The free surfaces of the ring have convective heat transfer defined by a film coefficient of 0.01, and a sink temperature of 20°C. Four data lines describe the aluminum ring. The "1,0," line says we are naming the aluminum ring body 1, and the zero means the body is deformable. The next line with five commas says we are not inputting the six quantities which describe the initial position and velocity of this body or the friction coefficient. It is not possible to prescribe the velocity of a deformable body. The "0.01,20.,,," line gives a film coefficient (to the environment) of 0.01 and a sink (environment) temperature of 20°C. The fields for the contact film coefficient and the surface temperature are left blank because these two values are used only for rigid bodies. In this case, since the body is deformable, the program will ignore any values given. The "1 TO 135" line gives the list of elements for deformable body 1.

The second deformable body is the steel disk. A reference point and an axial velocity are given. Although these were not used in the calculation, this was done as a reminder of what the imposed boundary conditions are simulating. The "2,0," line says the steel disk is body 2, and that it is deformable. The "9.,0.,0.,-150.,0.,0.,1.," line means the initial Z-value of the center of rotation of the reference point is 9.0; the axial component of velocity is -150.0; and the cohesive friction factor is 1.0.

NOTE

For a deformable body, the program internally sets the first six values of this line to zero regardless of the values that are entered.

The "0.01,20.,35.,0.," line means: the film coefficient (to the environment) is 0.01: the sink temperature is 20.0; the contact film coefficient is 35.0. Finally, the "136 to 285" line defines the elements which make up the steel disk.

As it is not anticipated that either body will contact itself, the CONTACT TABLE option could be used to reduce the amount of computations

Once again, the OPTIMIZE option is used to reduce the bandwidth of the problem. Additionally, when large sliding occurs in contact analysis between multiple deformable bodies, the bandwidth of the problem is continuously being changed. The inclusion of the OPTIMIZE option insures that a minimum computation time will be required.

All contact analyses are time-driven even if they are quasi-static in nature. If rigid surfaces are included, a velocity is prescribed and the time step is used to determine the motion.

The "END OPTION" line terminated the MODEL DEFINITION section.

LOAD INCREMENTATION Section

The TRANSIENT option controls the time step for the heat transfer analysis and the motion of any rigid surfaces present. The time step is also used to calculate strain rates which are necessary if rate effects are used in the material model. The NON AUTO parameter on this title line means we are suppressing the automatic time-stepping. The "0.0003,.03," data line tells MARC to use a fixed time step of 0.0003 seconds and a total time period of 0.03 seconds. The increment size is fixed. The TRANSIENT option cannot be used to request that the adaptive time stepping procedure to used in a coupled analysis; use the AUTO TIME option instead.

In each increment, we wish to impose a displacement of 0.045 mm to the bottom edge nodes of the steel disk (opposite to the contact surface). Since the CONTACT option *always bypasses* increment zero, this displacement increment is prescribed in the "LOAD INCREMENTATION" lines using the DISP CHANGE option and not in the original boundary conditions.

The DISP CHANGE option allows new displacement boundary conditions to be specified, or old ones to be changed. The zero on the following line indicates a complete replacement of the previously defined boundary condition set, as opposed to just modifying the previously defined boundary condition set. The "0.0" line gives the value of the displacement change, and the "1" line indicates the first DOF (Z-displacement) is to be constrained. The next line "1 TO 151 BY 10" names the 16 nodes along the top (symmetry) edge of the aluminum ring. These nodes have been placed on rollers and cannot move in the Z-direction.

Similarly, the next three lines inform MARC to zero out the R-displacement of these seven nodes on the steel axis of symmetry: 161 to 167. Finally, the last three lines impose a displacement change of -0.045 mm (per increment) in the Z-direction to the 26 nodes ("167 TO 342 BY 7") along the bottom edge of the steel disk.

The "CONTINUE" line ends the LOAD INCREMENTATION section and the input file.

Output

Except for the echo of the input listing, the printout for this example is not included. It is voluminous! If you do elect to print out the element quantities, remember that the stresses are the Cauchy (or true) stresses and the strains are the logarithmic strains, as opposed to the engineering stresses and strains.

```
                                        input  data

    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
TITLE       COUPLED ANALYSIS OF RING COMPRESSION
SIZING,300000,,,,
ELEMENT,10,
PRINT,8,
FINITE
LARGE DISP
UPDATE
COUPLE________________________Flag coupled analysis
END
CONNECTIVITY
  285    0    1
    1   10    1    2   12   11
    2   10    2    3   13   12
    3   10    3    4   14   13
    4   10    4    5   15   14
    5   10    5    6   16   15
    6   10    6    7   17   16
    7   10    7    8   18   17
    8   10    8    9   19   18
    9   10    9   10   20   19
   10   10   11   12   22   21
   11   10   12   13   23   22
   12   10   13   14   24   23
   13   10   14   15   25   24
   14   10   15   16   26   25
   15   10   16   17   27   26
   16   10   17   18   28   27
   17   10   18   19   29   28
   18   10   19   20   30   29
   19   10   21   22   32   31
   20   10   22   23   33   32
   21   10   23   24   34   33
   22   10   24   25   35   34
   23   10   25   26   36   35
   24   10   26   27   37   36
   25   10   27   28   38   37
   26   10   28   29   39   38
   27   10   29   30   40   39
   28   10   31   32   42   41
   29   10   32   33   43   42
   30   10   33   34   44   43
   31   10   34   35   45   44
   32   10   35   36   46   45
   33   10   36   37   47   46
   34   10   37   38   48   47
--------------------------------------------------------------------------------
    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                      5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                  --------------------------------------------------------------------------------
                     35   10   38   39   49   48
                     36   10   39   40   50   49
                     37   10   41   42   52   51
                     38   10   42   43   53   52
card     50          39   10   43   44   54   53
                     40   10   44   45   55   54
                     41   10   45   46   56   55
                     42   10   46   47   57   56
                     43   10   47   48   58   57
card     55          44   10   48   49   59   58
                     45   10   49   50   60   59
                     46   10   51   52   62   61
                     47   10   52   53   63   62
                     48   10   53   54   64   63
card     60          49   10   54   55   65   64
                     50   10   55   56   66   65
                     51   10   56   57   67   66
                     52   10   57   58   68   67
                     53   10   58   59   69   68
card     65          54   10   59   60   70   69
                     55   10   61   62   72   71
                     56   10   62   63   73   72
                     57   10   63   64   74   73
                     58   10   64   65   75   74
card     70          59   10   65   66   76   75
                     60   10   66   67   77   76
                     61   10   67   68   78   77
                     62   10   68   69   79   78
                     63   10   69   70   80   79
card     75          64   10   71   72   82   81
                     65   10   72   73   83   82
                     66   10   73   74   84   83
                     67   10   74   75   85   84
                     68   10   75   76   86   85
card     80          69   10   76   77   87   86
                     70   10   77   78   88   87
                     71   10   78   79   89   88
                     72   10   79   80   90   89
                     73   10   81   82   92   91
card     85          74   10   82   83   93   92
                     75   10   83   84   94   93
                     76   10   84   85   95   94
                     77   10   85   86   96   95
                     78   10   86   87   97   96
card     90          79   10   87   88   98   97
                     80   10   88   89   99   98
                  --------------------------------------------------------------------------------
                      5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                --------------------------------------------------------------------------------
                   81   10   89   90  100   99
                   82   10   91   92  102  101
                   83   10   92   93  103  102
card   95          84   10   93   94  104  103
                   85   10   94   95  105  104
                   86   10   95   96  106  105
                   87   10   96   97  107  106
                   88   10   97   98  108  107
card  100          89   10   98   99  109  108
                   90   10   99  100  110  109
                   91   10  101  102  112  111
                   92   10  102  103  113  112
                   93   10  103  104  114  113
card  105          94   10  104  105  115  114
                   95   10  105  106  116  115
                   96   10  106  107  117  116
                   97   10  107  108  118  117
                   98   10  108  109  119  118
card  110          99   10  109  110  120  119
                  100   10  111  112  122  121
                  101   10  112  113  123  122
                  102   10  113  114  124  123
                  103   10  114  115  125  124
card  115         104   10  115  116  126  125
                  105   10  116  117  127  126
                  106   10  117  118  128  127
                  107   10  118  119  129  128
                  108   10  119  120  130  129
card  120         109   10  121  122  132  131
                  110   10  122  123  133  132
                  111   10  123  124  134  133
                  112   10  124  125  135  134
                  113   10  125  126  136  135
card  125         114   10  126  127  137  136
                  115   10  127  128  138  137
                  116   10  128  129  139  138
                  117   10  129  130  140  139
                  118   10  131  132  142  141
card  130         119   10  132  133  143  142
                  120   10  133  134  144  143
                  121   10  134  135  145  144
                  122   10  135  136  146  145
                  123   10  136  137  147  146
card  135         124   10  137  138  148  147
                  125   10  138  139  149  148
                  126   10  139  140  150  149
                --------------------------------------------------------------------------------
                    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                   5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
               --------------------------------------------------------------------------------
                 127   10  141  142  152  151
                 128   10  142  143  153  152
card   140       129   10  143  144  154  153
                 130   10  144  145  155  154
                 131   10  145  146  156  155
                 132   10  146  147  157  156
                 133   10  147  148  158  157
card   145       134   10  148  149  159  158
                 135   10  149  150  160  159
                 136   10  161  162  169  168
                 137   10  162  163  170  169
                 138   10  163  164  171  170
card   150       139   10  164  165  172  171
                 140   10  165  166  173  172
                 141   10  166  167  174  173
                 142   10  168  169  176  175
                 143   10  169  170  177  176
card   155       144   10  170  171  178  177
                 145   10  171  172  179  178
                 146   10  172  173  180  179
                 147   10  173  174  181  180
                 148   10  175  176  183  182
card   160       149   10  176  177  184  183
                 150   10  177  178  185  184
                 151   10  178  179  186  185
                 152   10  179  180  187  186
                 153   10  180  181  188  187
card   165       154   10  182  183  190  189
                 155   10  183  184  191  190
                 156   10  184  185  192  191
                 157   10  185  186  193  192
                 158   10  186  187  194  193
card   170       159   10  187  188  195  194
                 160   10  189  190  197  196
                 161   10  190  191  198  197
                 162   10  191  192  199  198
                 163   10  192  193  200  199
card   175       164   10  193  194  201  200
                 165   10  194  195  202  201
                 166   10  196  197  204  203
                 167   10  197  198  205  204
                 168   10  198  199  206  205
card   180       169   10  199  200  207  206
                 170   10  200  201  208  207
                 171   10  201  202  209  208
                 172   10  203  204  211  210
                 173   10  204  205  212  211
               --------------------------------------------------------------------------------
                   5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                       5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                  --------------------------------------------------------------------------------
card   185          174   10  205  206  213  212
                    175   10  206  207  214  213
                    176   10  207  208  215  214
                    177   10  208  209  216  215
                    178   10  210  211  218  217
card   190          179   10  211  212  219  218
                    180   10  212  213  220  219
                    181   10  213  214  221  220
                    182   10  214  215  222  221
                    183   10  215  216  223  222
card   195          184   10  217  218  225  224
                    185   10  218  219  226  225
                    186   10  219  220  227  226
                    187   10  220  221  228  227
                    188   10  221  222  229  228
card   200          189   10  222  223  230  229
                    190   10  224  225  232  231
                    191   10  225  226  233  232
                    192   10  226  227  234  233
                    193   10  227  228  235  234
card   205          194   10  228  229  236  235
                    195   10  229  230  237  236
                    196   10  231  232  239  238
                    197   10  232  233  240  239
                    198   10  233  234  241  240
card   210          199   10  234  235  242  241
                    200   10  235  236  243  242
                    201   10  236  237  244  243
                    202   10  238  239  246  245
                    203   10  239  240  247  246
card   215          204   10  240  241  248  247
                    205   10  241  242  249  248
                    206   10  242  243  250  249
                    207   10  243  244  251  250
                    208   10  245  246  253  252
card   220          209   10  246  247  254  253
                    210   10  247  248  255  254
                    211   10  248  249  256  255
                    212   10  249  250  257  256
                    213   10  250  251  258  257
card   225          214   10  252  253  260  259
                    215   10  253  254  261  260
                    216   10  254  255  262  261
                    217   10  255  256  263  262
                    218   10  256  257  264  263
card   230          219   10  257  258  265  264
                    220   10  259  260  267  266
                  --------------------------------------------------------------------------------
                       5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                        5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                  --------------------------------------------------------------------------------
                      221   10  260  261  268  267
                      222   10  261  262  269  268
                      223   10  262  263  270  269
card   235            224   10  263  264  271  270
                      225   10  264  265  272  271
                      226   10  266  267  274  273
                      227   10  267  268  275  274
                      228   10  268  269  276  275
card   240            229   10  269  270  277  276
                      230   10  270  271  278  277
                      231   10  271  272  279  278
                      232   10  273  274  281  280
                      233   10  274  275  282  281
card   245            234   10  275  276  283  282
                      235   10  276  277  284  283
                      236   10  277  278  285  284
                      237   10  278  279  286  285
                      238   10  280  281  288  287
card   250            239   10  281  282  289  288
                      240   10  282  283  290  289
                      241   10  283  284  291  290
                      242   10  284  285  292  291
                      243   10  285  286  293  292
card   255            244   10  287  288  295  294
                      245   10  288  289  296  295
                      246   10  289  290  297  296
                      247   10  290  291  298  297
                      248   10  291  292  299  298
card   260            249   10  292  293  300  299
                      250   10  294  295  302  301
                      251   10  295  296  303  302
                      252   10  296  297  304  303
                      253   10  297  298  305  304
card   265            254   10  298  299  306  305
                      255   10  299  300  307  306
                      256   10  301  302  309  308
                      257   10  302  303  310  309
                      258   10  303  304  311  310
card   270            259   10  304  305  312  311
                      260   10  305  306  313  312
                      261   10  306  307  314  313
                      262   10  308  309  316  315
                      263   10  309  310  317  316
card   275            264   10  310  311  318  317
                      265   10  311  312  319  318
                      266   10  312  313  320  319
                      267   10  313  314  321  320
                      268   10  315  316  323  322
                  --------------------------------------------------------------------------------
                        5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                  --------------------------------------------------------------------------------
card   280        269   10  316  317  324  323
                  270   10  317  318  325  324
                  271   10  318  319  326  325
                  272   10  319  320  327  326
                  273   10  320  321  328  327
card   285        274   10  322  323  330  329
                  275   10  323  324  331  330
                  276   10  324  325  332  331
                  277   10  325  326  333  332
                  278   10  326  327  334  333
card   290        279   10  327  328  335  334
                  280   10  329  330  337  336
                  281   10  330  331  338  337
                  282   10  331  332  339  338
                  283   10  332  333  340  339
card   295        284   10  333  334  341  340
                  285   10  334  335  342  341
                  COORDINATES
                    2  342    0    1
                    1 0.0        1.35000+1 0.0
card   300          2 1.0000007 1.35000+1 0.0
                    3 2.0000014 1.35000+1 0.0
                    4 3.0000019 1.35000+1 0.0
                    5 4.0000028 1.35000+1 0.0
                    6 5.0000028 1.35000+1 0.0
card   305          7 6.0000038 1.35000+1 0.0
                    8 7.0000047 1.35000+1 0.0
                    9 8.0000057 1.35000+1 0.0
                   10 9.0000076 1.35000+1 0.0
                   11 0.0        1.44000+1 0.0
card   310         12 1.0000004 1.44000+1 0.0
                   13 2.0000009 1.44000+1 0.0
                   14 3.0000014 1.44000+1 0.0
                   15 4.0000019 1.44000+1 0.0
                   16 5.0000028 1.44000+1 0.0
card   315         17 6.0000028 1.44000+1 0.0
                   18 7.0000038 1.44000+1 0.0
                   19 8.0000038 1.44000+1 0.0
                   20 9.0000038 1.44000+1 0.0
                   21 0.0        1.53000+1 0.0
card   320         22 1.0000004 1.53000+1 0.0
                   23 2.0000009 1.53000+1 0.0
                   24 3.0000014 1.53000+1 0.0
                   25 4.0000019 1.53000+1 0.0
                   26 5.0000019 1.53000+1 0.0
card   325         27 6.0000028 1.53000+1 0.0
                   28 7.0000028 1.53000+1 0.0
                  --------------------------------------------------------------------------------
                    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
             --------------------------------------------------------------------------------
               29 8.0000038 1.53000+1 0.0
               30 9.0000057 1.53000+1 0.0
               31 0.0       1.62000+1 0.0
card   330     32 1.0000004 1.62000+1 0.0
               33 2.0000009 1.62000+1 0.0
               34 3.0000009 1.62000+1 0.0
               35 4.0000019 1.62000+1 0.0
               36 5.0000019 1.62000+1 0.0
card   335     37 6.0000019 1.62000+1 0.0
               38 7.0000038 1.62000+1 0.0
               39 8.0000038 1.62000+1 0.0
               40 9.0000057 1.62000+1 0.0
               41 0.0       1.71000+1 0.0
card   340     42 1.0000004 1.71000+1 0.0
               43 2.0000009 1.71000+1 0.0
               44 3.0000019 1.71000+1 0.0
               45 4.0000019 1.71000+1 0.0
               46 5.0000019 1.71000+1 0.0
card   345     47 6.0000038 1.71000+1 0.0
               48 7.0000038 1.71000+1 0.0
               49 8.0000038 1.71000+1 0.0
               50 9.0000057 1.71000+1 0.0
               51 0.0       1.80000+1 0.0
card   350     52 1.0000004 1.80000+1 0.0
               53 2.0000009 1.80000+1 0.0
               54 3.0000014 1.80000+1 0.0
               55 4.0000019 1.80000+1 0.0
               56 5.0000019 1.80000+1 0.0
card   355     57 6.0000028 1.80000+1 0.0
               58 7.0000038 1.80000+1 0.0
               59 8.0000038 1.80000+1 0.0
               60 9.0000057 1.80000+1 0.0
               61 0.0       1.89000+1 0.0
card   360     62 1.0000004 1.89000+1 0.0
               63 2.0000009 1.89000+1 0.0
               64 3.0000014 1.89000+1 0.0
               65 4.0000019 1.89000+1 0.0
               66 5.0000019 1.89000+1 0.0
card   365     67 6.0000028 1.89000+1 0.0
               68 7.0000038 1.89000+1 0.0
               69 8.0000038 1.89000+1 0.0
               70 9.0000057 1.89000+1 0.0
               71 0.0       1.98000+1 0.0
card   370     72 1.0000004 1.98000+1 0.0
               73 2.0000009 1.98000+1 0.0
               74 3.0000014 1.98000+1 0.0
               75 4.0000019 1.98000+1 0.0
             --------------------------------------------------------------------------------
                5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
--------------------------------------------------------------------------------
   76 5.0000019 1.98000+1 0.0
   77 6.0000028 1.98000+1 0.0
   78 7.0000028 1.98000+1 0.0
   79 8.0000038 1.98000+1 0.0
   80 9.0000057 1.98000+1 0.0
   81 0.0       2.07000+1 0.0
   82 1.0000004 2.07000+1 0.0
   83 2.0000009 2.07000+1 0.0
   84 3.0000014 2.07000+1 0.0
   85 4.0000019 2.07000+1 0.0
   86 5.0000019 2.07000+1 0.0
   87 6.0000028 2.07000+1 0.0
   88 7.0000038 2.07000+1 0.0
   89 8.0000038 2.07000+1 0.0
   90 9.0000057 2.07000+1 0.0
   91 0.0       2.16000+1 0.0
   92 1.0000004 2.16000+1 0.0
   93 2.0000009 2.16000+1 0.0
   94 3.0000014 2.16000+1 0.0
   95 4.0000019 2.16000+1 0.0
   96 5.0000028 2.16000+1 0.0
   97 6.0000028 2.16000+1 0.0
   98 7.0000038 2.16000+1 0.0
   99 8.0000038 2.16000+1 0.0
  100 9.0000057 2.16000+1 0.0
  101 0.0       2.25000+1 0.0
  102 1.0000004 2.25000+1 0.0
  103 2.0000009 2.25000+1 0.0
  104 3.0000014 2.25000+1 0.0
  105 4.0000019 2.25000+1 0.0
  106 5.0000028 2.25000+1 0.0
  107 6.0000028 2.25000+1 0.0
  108 7.0000028 2.25000+1 0.0
  109 8.0000038 2.25000+1 0.0
  110 9.0000057 2.25000+1 0.0
  111 0.0       2.34000+1 0.0
  112 1.0000007 2.34000+1 0.0
  113 2.0000014 2.34000+1 0.0
  114 3.0000019 2.34000+1 0.0
  115 4.0000028 2.34000+1 0.0
  116 5.0000019 2.34000+1 0.0
  117 6.0000038 2.34000+1 0.0
  118 7.0000038 2.34000+1 0.0
  119 8.0000057 2.34000+1 0.0
  120 9.0000057 2.34000+1 0.0
  121 0.0       2.43000+1 0.0
--------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

```
    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
  122 1.0000004 2.43000+1 0.0
  123 2.0000009 2.43000+1 0.0
  124 3.0000014 2.43000+1 0.0
  125 4.0000019 2.43000+1 0.0
  126 5.0000019 2.43000+1 0.0
  127 6.0000028 2.43000+1 0.0
  128 7.0000038 2.43000+1 0.0
  129 8.0000038 2.43000+1 0.0
  130 9.0000076 2.43000+1 0.0
  131 0.0       2.52000+1 0.0
  132 1.0000007 2.52000+1 0.0
  133 2.0000014 2.52000+1 0.0
  134 3.0000014 2.52000+1 0.0
  135 4.0000028 2.52000+1 0.0
  136 5.0000019 2.52000+1 0.0
  137 6.0000028 2.52000+1 0.0
  138 7.0000047 2.52000+1 0.0
  139 8.0000057 2.52000+1 0.0
  140 9.0000057 2.52000+1 0.0
  141 0.0       2.61000+1 0.0
  142 1.0000004 2.61000+1 0.0
  143 2.0000009 2.61000+1 0.0
  144 3.0000014 2.61000+1 0.0
  145 4.0000019 2.61000+1 0.0
  146 5.0000019 2.61000+1 0.0
  147 6.0000028 2.61000+1 0.0
  148 7.0000038 2.61000+1 0.0
  149 8.0000038 2.61000+1 0.0
  150 9.0000076 2.61000+1 0.0
  151 0.0       2.70000+1 0.0
  152 1.0000007 2.70000+1 0.0
  153 2.0000014 2.70000+1 0.0
  154 3.0000019 2.70000+1 0.0
  155 4.0000028 2.70000+1 0.0
  156 5.0000028 2.70000+1 0.0
  157 6.0000038 2.70000+1 0.0
  158 7.0000047 2.70000+1 0.0
  159 8.0000057 2.70000+1 0.0
  160 9.0000076 2.70000+1 0.0
  161 9.0000076 0.0       0.0
  162 1.00000+1 0.0       0.0
  163 1.10000+1 0.0       0.0
  164 1.20000+1 0.0       0.0
  165 1.30000+1 0.0       0.0
  166 1.40000+1 0.0       0.0
  167 1.50000+1 0.0       0.0
  168 9.0000057 1.6799998 0.0
--------------------------------------------------------------------------------
    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
  169 1.00000+1 1.6799995 0.0
  170 1.10000+1 1.6799993 0.0
  171 1.20000+1 1.6799995 0.0
  172 1.30000+1 1.6799993 0.0
  173 1.40000+1 1.6799993 0.0
  174 1.50000+1 1.6799998 0.0
  175 9.0000076 3.3599996 0.0
  176 1.00000+1 3.3599991 0.0
  177 1.10000+1 3.3599987 0.0
  178 1.20000+1 3.3599991 0.0
  179 1.30000+1 3.3599987 0.0
  180 1.40000+1 3.3599987 0.0
  181 1.50000+1 3.3599996 0.0
  182 9.0000076 5.0399999 0.0
  183 1.00000+1 5.0399990 0.0
  184 1.10000+1 5.0399980 0.0
  185 1.20000+1 5.0399990 0.0
  186 1.30000+1 5.0399980 0.0
  187 1.40000+1 5.0399990 0.0
  188 1.50000+1 5.0399999 0.0
  189 9.0000076 6.7199993 0.0
  190 1.00000+1 6.7199983 0.0
  191 1.10000+1 6.7199974 0.0
  192 1.20000+1 6.7199983 0.0
  193 1.30000+1 6.7199974 0.0
  194 1.40000+1 6.7199974 0.0
  195 1.50000+1 6.7199993 0.0
  196 9.0000076 8.3999996 0.0
  197 1.00000+1 8.3999977 0.0
  198 1.10000+1 8.3999958 0.0
  199 1.20000+1 8.3999977 0.0
  200 1.30000+1 8.3999958 0.0
  201 1.40000+1 8.3999977 0.0
  202 1.50000+1 8.3999996 0.0
  203 9.0000076 1.00799+1 0.0
  204 1.00000+1 1.00799+1 0.0
  205 1.10000+1 1.00799+1 0.0
  206 1.20000+1 1.00799+1 0.0
  207 1.30000+1 1.00799+1 0.0
  208 1.40000+1 1.00799+1 0.0
  209 1.50000+1 1.00799+1 0.0
  210 9.0000057 1.17599+1 0.0
  211 1.00000+1 1.17599+1 0.0
  212 1.10000+1 1.17599+1 0.0
  213 1.20000+1 1.17599+1 0.0
  214 1.30000+1 1.17599+1 0.0
  215 1.40000+1 1.17599+1 0.0
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
                    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                 --------------------------------------------------------------------------------
                  216 1.50000+1 1.17599+1 0.0
card   515        217 9.0000057 1.34399+1 0.0
                  218 1.00000+1 1.34399+1 0.0
                  219 1.10000+1 1.34399+1 0.0
                  220 1.20000+1 1.34399+1 0.0
                  221 1.30000+1 1.34399+1 0.0
card   520        222 1.40000+1 1.34399+1 0.0
                  223 1.50000+1 1.34399+1 0.0
                  224 9.0000057 1.51200+1 0.0
                  225 1.00000+1 1.51200+1 0.0
                  226 1.10000+1 1.51199+1 0.0
card   525        227 1.20000+1 1.51199+1 0.0
                  228 1.30000+1 1.51199+1 0.0
                  229 1.40000+1 1.51199+1 0.0
                  230 1.50000+1 1.51200+1 0.0
                  231 9.0000057 1.68000+1 0.0
card   530        232 1.00000+1 1.67999+1 0.0
                  233 1.10000+1 1.67999+1 0.0
                  234 1.20000+1 1.67999+1 0.0
                  235 1.30000+1 1.67999+1 0.0
                  236 1.40000+1 1.67999+1 0.0
card   535        237 1.50000+1 1.68000+1 0.0
                  238 9.0000057 1.84799+1 0.0
                  239 1.00000+1 1.84799+1 0.0
                  240 1.10000+1 1.84799+1 0.0
                  241 1.20000+1 1.84799+1 0.0
card   540        242 1.30000+1 1.84799+1 0.0
                  243 1.40000+1 1.84799+1 0.0
                  244 1.50000+1 1.84799+1 0.0
                  245 9.0000057 2.01600+1 0.0
                  246 1.00000+1 2.01600+1 0.0
card   545        247 1.10000+1 2.01599+1 0.0
                  248 1.20000+1 2.01600+1 0.0
                  249 1.30000+1 2.01599+1 0.0
                  250 1.40000+1 2.01600+1 0.0
                  251 1.50000+1 2.01600+1 0.0
card   550        252 9.0000057 2.18400+1 0.0
                  253 1.00000+1 2.18400+1 0.0
                  254 1.10000+1 2.18399+1 0.0
                  255 1.20000+1 2.18400+1 0.0
                  256 1.30000+1 2.18400+1 0.0
card   555        257 1.40000+1 2.18400+1 0.0
                  258 1.50000+1 2.18400+1 0.0
                  259 9.0000057 2.35200+1 0.0
                  260 1.00000+1 2.35200+1 0.0
                  261 1.10000+1 2.35199+1 0.0
                 --------------------------------------------------------------------------------
                    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
  262 1.20000+1 2.35199+1 0.0
  263 1.30000+1 2.35199+1 0.0
  264 1.40000+1 2.35200+1 0.0
  265 1.50000+1 2.35200+1 0.0
  266 9.0000057 2.52000+1 0.0
  267 1.00000+1 2.52000+1 0.0
  268 1.10000+1 2.51999+1 0.0
  269 1.20000+1 2.52000+1 0.0
  270 1.30000+1 2.51999+1 0.0
  271 1.40000+1 2.52000+1 0.0
  272 1.50000+1 2.52000+1 0.0
  273 9.0000057 2.68800+1 0.0
  274 1.00000+1 2.68799+1 0.0
  275 1.10000+1 2.68799+1 0.0
  276 1.20000+1 2.68800+1 0.0
  277 1.30000+1 2.68799+1 0.0
  278 1.40000+1 2.68799+1 0.0
  279 1.50000+1 2.68800+1 0.0
  280 9.0000057 2.85600+1 0.0
  281 1.00000+1 2.85600+1 0.0
  282 1.10000+1 2.85600+1 0.0
  283 1.20000+1 2.85600+1 0.0
  284 1.30000+1 2.85600+1 0.0
  285 1.40000+1 2.85600+1 0.0
  286 1.50000+1 2.85600+1 0.0
  287 9.0000057 3.02400+1 0.0
  288 1.00000+1 3.02400+1 0.0
  289 1.10000+1 3.02400+1 0.0
  290 1.20000+1 3.02400+1 0.0
  291 1.30000+1 3.02400+1 0.0
  292 1.40000+1 3.02400+1 0.0
  293 1.50000+1 3.02400+1 0.0
  294 9.0000057 3.19200+1 0.0
  295 1.00000+1 3.19200+1 0.0
  296 1.10000+1 3.19199+1 0.0
  297 1.20000+1 3.19200+1 0.0
  298 1.30000+1 3.19199+1 0.0
  299 1.40000+1 3.19200+1 0.0
  300 1.50000+1 3.19200+1 0.0
  301 9.0000057 3.36000+1 0.0
  302 1.00000+1 3.36000+1 0.0
  303 1.10000+1 3.35999+1 0.0
  304 1.20000+1 3.36000+1 0.0
  305 1.30000+1 3.36000+1 0.0
  306 1.40000+1 3.36000+1 0.0
  307 1.50000+1 3.36000+1 0.0
  308 9.0000057 3.52800+1 0.0
--------------------------------------------------------------------------------
    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
--------------------------------------------------------------------------------
  309 1.00000+1 3.52800+1 0.0
  310 1.10000+1 3.52799+1 0.0
  311 1.20000+1 3.52800+1 0.0
  312 1.30000+1 3.52799+1 0.0
  313 1.40000+1 3.52800+1 0.0
  314 1.50000+1 3.52800+1 0.0
  315 9.0000057 3.69600+1 0.0
  316 1.00000+1 3.69599+1 0.0
  317 1.10000+1 3.69599+1 0.0
  318 1.20000+1 3.69599+1 0.0
  319 1.30000+1 3.69599+1 0.0
  320 1.40000+1 3.69600+1 0.0
  321 1.50000+1 3.69600+1 0.0
  322 9.0000057 3.86400+1 0.0
  323 1.00000+1 3.86399+1 0.0
  324 1.10000+1 3.86399+1 0.0
  325 1.20000+1 3.86399+1 0.0
  326 1.30000+1 3.86399+1 0.0
  327 1.40000+1 3.86399+1 0.0
  328 1.50000+1 3.86400+1 0.0
  329 9.0000057 4.03200+1 0.0
  330 1.00000+1 4.03200+1 0.0
  331 1.10000+1 4.03199+1 0.0
  332 1.20000+1 4.03200+1 0.0
  333 1.30000+1 4.03200+1 0.0
  334 1.40000+1 4.03200+1 0.0
  335 1.50000+1 4.03200+1 0.0
  336 9.0000076 4.20000+1 0.0
  337 1.00000+1 4.20000+1 0.0
  338 1.10000+1 4.20000+1 0.0
  339 1.20000+1 4.20000+1 0.0
  340 1.30000+1 4.20000+1 0.0
  341 1.40000+1 4.20000+1 0.0
  342 1.50000+1 4.20000+1 0.0
FIXED DISPLACEMENT

  0.0
    1
 1 TO 151 BY 10 AND 167 TO 342 BY 7
  0.0
    2
 161 TO 167
CONTROL
161,15,0,1,0,,
.15,
,,10.,
POST
--------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

Tolerances given for both stress and heat transfer analyses

```
     5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
-------------------------------------------------------------------------------------------------
6,,,1,1,,,,50,
7,
11,
12,
13,
14,
17,
FIXED TEMPERATURE
1,
20.,
167 TO 342 BY 7 AND 336 TO 341
INITIAL TEMPERATURE
2,
427.,
1 TO 160
20.,
161 TO 342
PROPERTY
2,
10000.,.33,1.,1.3E-5,200.,3.4,,1,
242.,2.4255,1.,
```

Mechanical and thermal properties

```
1 TO 135
100000.,.33,1.,,,,,2,
19.,3.77,1.,
136 TO 285
CONVERT
1.,
WORK HARD DATA
4,0,1,
3.4,0.,
5.1,0.15,
5.78,0.7,
5.78,5.,
TEMPERATURE EFFECTS
1,,,,,,,,,,1,
-0.007,200.,
NO PRINT
GEOMETRY
1,
0.,1.,
```

Constant Dilatation

```
1 TO 285
DIST FLUXES
1,
101,1.,
```

Include heat generated due to plasticity

```
1 TO 285
CONTACT
-------------------------------------------------------------------------------------------------
     5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

```
                  5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
              --------------------------------------------------------------------------------
card   700    2,65,65,1,
              ,,
              1,0,
              ,,,,,
              0.01,20.,
card   705    1 TO 135_________________________Define aluminum ring as Body 1
              2,0,
              9.,0.,0.,-150.,0.,0.,1.,
              0.01,20.,35.,0.,
              136 TO 285_______________________Define steel disk as Body 2
card   710    OPTIMIZE,2,
              5,
              END OPTION
              TRANSIENT NON AUTO
              0.0003,.03,
card   715    DISP CHANGE
              0,
              3,
                0.0
                  1
card   720    1 TO 151 BY 10
                0.0
                  2
                161 TO 167
               -.045,
card   725        1
               167 TO 342 BY 7
              CONTINUE
              --------------------------------------------------------------------------------
                  5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
              --------------------------------------------------------------------------------
              --------------------------------------------------------------------------------
                                   ************************************************
                                   ************************************************

                                    program sizing and options requested as follows



                                    element type requested************************     10
                                    element type requested************************     40
                                    number of elements in mesh********************    285
                                    number of nodes in mesh***********************    342
                                    max number of elements in any dist load list***   285
                                    maximum number of boundary conditions*********     81
                                    thermal stress analysis flagged***************
                                    load correction flagged or set****************
                                    option for debug print out********************      1
                                    stresses stored at all integration points******
```

```
          tape no.for input of coordinates + connectivity    5
          no.of different materials    3 max.no of slopes    5
          heat transfer analysis,  extrapolation flag, **    1
          maximum number of distributed flux lists*******    3
          maximum elements variables per point on post tp   33
          number of points on shell section *************   11
          option for terminal debug**********************
          geometry updated after each load step**********
          formulation for large strain plasticity *******
          new style input format will be used************
          maximum number of set names is*****************   10
          coupled thermal-mechanical analysis flagged****
          number of processors used *********************    1
          vector length used ****************************    1


          end of parameters and sizing
         *************************************************
         *************************************************




         element type   10


4-node isoparametric quadrilateral ring

stresses and strains in global directions
  1=zz
  2=rr
  3=hoop
  4=zr

displacements in global directions
  1=u axial direction
  2=v radial direction


         element type   40


4-node heat transfer  axi-symmetric ring

1 degree of freedom per node - temperature


workspace needed for input and stiffness assembly     67271
```

Notice heat transfer elements are generated automatically.

```
   internal core allocation parameters
   degrees of freedom per node (ndeg)  2
   coords per node (ncrd)  2
   strains per integration point (ngens)  4
   max. nodes per element (nnodmx)  4
   max.stress components per int. point (nstrmx)   4
   max. invariants per int. points (neqst)  1
   max. degrees of freedom per node  2
   max. number of coordinates per node  2
   max. gradients per int. point  4
   max. nodes per element   4
   max. invariants per int. point  1


   flag for element storage (ielsto)  1
   elems out of core, words per elem (nelsto) 1140
   elems per buffer (mxels)    3

out-of-core space needed for element storage =    324900  based on record size of  3420
   vectors in core, total space required     16331


   words per track on disk set to 4096


   internal element variables


   internal element number  1  library code type 10
   number of nodes=  4
   stresses stored per integration point =  4
   direct continuum components stored =  3
   shear continuum components stored =  1
   shell/beam flag = 0
   curvilinear  coord. flag = 0
   int.points for elem. stiffness  4
   number of local inertia directions  2
   int.point for print if all points not flagged  5
   int. points for dist. surface loads (pressure)  2
   library code type = 10
   no local rotation flag = 1
   generalized displ. flag = 0
   large disp. row counts     4    4    2    7
   number of nodes  4
   number of gradient components at each int. point  4
   integration points for conductivity  4
   integration point for print-out  5
   integration points for surface b.c.s  2
   no local rotation flag    1
   generalized variable flag    0
```

```
                internal element number  2  library code type 40
                number of nodes=  4
                stresses stored per integration point =  2
                direct continuum components stored =  2
                shear continuum components stored =  0
                shell/beam flag = 0
                curvilinear  coord. flag = 0
                int.points for elem. stiffness  4
                number of local inertia directions  1
                int.point for print if all points not flagged  5
                int. points for dist. surface loads (pressure)  2
                library code type = 40
                no local rotation flag = 1
                generalized displ. flag = 0
                large disp. row counts     0    0    0    0
                number of nodes  4
                number of gradient components at each int. point  2
                integration points for conductivity  4
                integration point for print-out  5
                integration points for surface b.c.s  2
                no local rotation flag    1
                generalized variable flag    0


                residual load correction is invoked


connectivity
----------

meshr1,iprnt
    5      1
elem no., type,        nodes
coordinates
----------

ncrd1 ,meshr1,iprnt
    2      5      1
 node      coordinates

fixed displacement
----------

fixed displacement =  0.000E+00  0.000E+00
a list of degrees of freedom given below
     1
from node      1 to node   151 by    10
       and
from node   167 to node   342 by     7
fixed displacement =  0.000E+00  0.000E+00
```

```
a list of degrees of freedom given below
     2
from node   161 to node   167 by     1

fixed boundary condition summary.
total fixed degrees of freedom read so far =   49

 b.c.    node   degree of  magnitude            b.c.    node   degree of  magnitude
number            freedom                      number            freedom

    1       1          1    0.000E+00              2      11          1    0.000E+00
    3      21          1    0.000E+00              4      31          1    0.000E+00
    5      41          1    0.000E+00              6      51          1    0.000E+00
    7      61          1    0.000E+00              8      71          1    0.000E+00
    9      81          1    0.000E+00             10      91          1    0.000E+00
   11     101          1    0.000E+00             12     111          1    0.000E+00
   13     121          1    0.000E+00             14     131          1    0.000E+00
   15     141          1    0.000E+00             16     151          1    0.000E+00
   17     167          1    0.000E+00             18     174          1    0.000E+00
   19     181          1    0.000E+00             20     188          1    0.000E+00
   21     195          1    0.000E+00             22     202          1    0.000E+00
   23     209          1    0.000E+00             24     216          1    0.000E+00
   25     223          1    0.000E+00             26     230          1    0.000E+00
   27     237          1    0.000E+00             28     244          1    0.000E+00
   29     251          1    0.000E+00             30     258          1    0.000E+00
   31     265          1    0.000E+00             32     272          1    0.000E+00
   33     279          1    0.000E+00             34     286          1    0.000E+00
   35     293          1    0.000E+00             36     300          1    0.000E+00
   37     307          1    0.000E+00             38     314          1    0.000E+00
   39     321          1    0.000E+00             40     328          1    0.000E+00
   41     335          1    0.000E+00             42     342          1    0.000E+00
   43     161          2    0.000E+00             44     162          2    0.000E+00
   45     163          2    0.000E+00             46     164          2    0.000E+00
   47     165          2    0.000E+00             48     166          2    0.000E+00
   49     167          2    0.000E+00

control
----------

max.     max.     min.
incs   recycles recycles
  162      15       0

maximum allowed relative change in displacement increment     0.15000E+00



full newton-raphson technique chosen



maximum nodal temperature change per time step = 0.20000E+02
```

```
maximum nodal temperature change before reassembly = 0.10000E+03


recycle for properties when temperature estimation error exceeds 0.10000E+02


post
----------



*** note - format of post code cards has changed.
          in k4, enter code in first field and  layer number in second field


elem vars,post tape,prev tape,  type  , conn fl ,post tape, prev tape,  repost ,frequency, k2post
       6      16       17        1        1       19       20         0       50        0
element variables appear on post-processor tape 16 in following order
post variable   1 is post code    7 =
post variable   2 is post code   11 =
post variable   3 is post code   12 =
post variable   4 is post code   13 =
post variable   5 is post code   14 =
post variable   6 is post code   17 =

***maximum record length on formatted post file=   80     approximate no. of records per increment
on file=    1486

fixed temperature
----------

fixed temperature=  0.200E+02
from node   167 to node   342 by     7
       and
from node   336 to node   341 by     1




fixed boundary condition summary.
total fixed degrees of freedom read so far =   32

 b.c.    node   degree of  magnitude              b.c.    node   degree of  magnitude
number            freedom                        number            freedom



    1     167          1    2.000E+01                2     174          1    2.000E+01
    3     181          1    2.000E+01                4     188          1    2.000E+01
    5     195          1    2.000E+01                6     202          1    2.000E+01
    7     209          1    2.000E+01                8     216          1    2.000E+01
    9     223          1    2.000E+01               10     230          1    2.000E+01
   11     237          1    2.000E+01               12     244          1    2.000E+01
   13     251          1    2.000E+01               14     258          1    2.000E+01
   15     265          1    2.000E+01               16     272          1    2.000E+01
   17     279          1    2.000E+01               18     286          1    2.000E+01
```

```
  19      293          1    2.000E+01                   20      300          1    2.000E+01
  21      307          1    2.000E+01                   22      314          1    2.000E+01
  23      321          1    2.000E+01                   24      328          1    2.000E+01
  25      335          1    2.000E+01                   26      336          1    2.000E+01
  27      337          1    2.000E+01                   28      338          1    2.000E+01
  29      339          1    2.000E+01                   30      340          1    2.000E+01
  31      341          1    2.000E+01                   32      342          1    2.000E+01


initial temperature
----------


number of series used for initial temperatures is     2
read from unit     5
initial value  0.4270000E+03
from node     1 to node   160 by     1
initial value  0.2000000E+02
from node   161 to node   342 by     1

isotropic
----------



youngs mod.,poisson r.,density, alpha   ,tot.temp.,   yielp,   yielp2,  mat
 0.100E+05 0.330E+00 0.100E+01 0.130E-04 0.200E+03 0.340E+01 0.000E+00    1
conductivity=  0.242E+03  specific heat =  0.243E+01  density =  0.100E+01 material id=   1
from element     1 to element   135 by     1
*** warning - initial state option is preferred for input of stress-free temperature.
          initial state option should follow property option.
```

Both mechanical and thermal propertie defined.

```
youngs mod.,poisson r.,density, alpha   ,tot.temp.,   yielp,   yielp2,  mat
 0.100E+06 0.330E+00 0.100E+01 0.000E+00 0.000E+00 0.100E+21 0.000E+00    2
conductivity=  0.190E+02  specific heat =  0.377E+01  density =  0.100E+01 material id=   2
from element   136 to element   285 by     1

convert
----------


mechanical-heat conversion factor  fcmech=          1.00

work hard data
----------


 no. of points for primary curve is    4 for secondary curve is    0 material type    1
read from unit     5
   yield stress    equivalent plastic strain
         0.34000E+01       0.00000E+00
         0.51000E+01       0.15000E+00
         0.57800E+01       0.70000E+00
         0.57800E+01       0.50000E+01
```

```
 temperature effects
 ----------

 material id =      1
 number of slopes     for yield      =     1
 yield      curve
 slope          breakpoint
-0.70000E-02  0.20000E+03

 no print
 ----------



 geometry
 ----------

   egeom1     egeom2     egeom3     egeom4     egeom5     egeom6
   0.000E+00  0.100E+01  0.000E+00  0.000E+00  0.000E+00  0.000E+00
 from element     1 to element   285 by     1

 dist fluxes
 ----------

 read from unit     5
 type index distributed flux
   101    0  0.1000000E+01
 from element     1 to element   285 by     1

 contact
 ----------

 number of bodies                   =    2
 max number of entities per body    =   65
 bound on number of boundary nodes  =   65
 friction type(1-m , 2-coulomb)     =    1

 distrib-0 or nodal-1 coul. frict   =    0

 relative velocity below which a
 node is considered sticking        =  0.00000E+00
 distance below which a node is
 considered touching a surface      =  0.00000E+00
 nodal reaction above which a node
 separates from a body              =  0.00000E+00

 body number                        =    1
 number of sets of data             =    0

           body positioning data
 1st coordinate of center of rotation       0.00000E+00
 2nd coordinate of center of rotation       0.00000E+00
```

Shear friction

Deformable Body 1
(aluminum ring)

```
angle rotated                             0.00000E+00
1st component of velocity                 0.00000E+00
2nd component of velocity                 0.00000E+00
angular velocity                          0.00000E+00
friction coefficient                      0.00000E+00

          body heat transfer data
heat transfer coefficient to environment  0.10000E-01
environment sink temperature              0.20000E+02
heat transfer coefficient when contacted  0.00000E+00
temperature if rigid                      0.00000E+00

from element     1 to element   135 by      1
```

Deformable Body 2 (steel disk)

```
body number                       =    2
number of sets of data            =    0

          body positioning data
1st coordinate of center of rotation      0.90000E+01
2nd coordinate of center of rotation      0.00000E+00
angle rotated                             0.00000E+00
1st component of velocity                -0.15000E+03
2nd component of velocity                 0.00000E+00
angular velocity                          0.00000E+00
friction coefficient                      0.10000E+01

          body heat transfer data
heat transfer coefficient to environment  0.10000E-01
environment sink temperature              0.20000E+02
heat transfer coefficient when contacted  0.35000E+02
temperature if rigid                      0.00000E+00

from element   136 to element   285 by      1


optimize,2,
----------



cuthill-mckee algorithm

end option
----------



                 total workspace needed with in-core matrix storage =    110947
```

Initial contact between bodies

```
node   10 of body  1 is touching body   2 segment    54
 the retained nodes are  224  217
node   20 of body  1 is touching body   2 segment    54
 the retained nodes are  224  217
node   30 of body  1 is touching body   2 segment    53
 the retained nodes are  231  224
node   40 of body  1 is touching body   2 segment    53
 the retained nodes are  231  224
node   50 of body  1 is touching body   2 segment    52
 the retained nodes are  238  231
node   60 of body  1 is touching body   2 segment    52
 the retained nodes are  238  231
node   70 of body  1 is touching body   2 segment    51
 the retained nodes are  245  238
node   80 of body  1 is touching body   2 segment    51
 the retained nodes are  245  238
node   90 of body  1 is touching body   2 segment    50
 the retained nodes are  252  245
node  100 of body  1 is touching body   2 segment    50
 the retained nodes are  252  245
node  110 of body  1 is touching body   2 segment    49
 the retained nodes are  259  252
node  120 of body  1 is touching body   2 segment    49
 the retained nodes are  259  252
node  130 of body  1 is touching body   2 segment    48
 the retained nodes are  266  259
node  140 of body  1 is touching body   2 segment    47
 the retained nodes are  273  266
node  150 of body  1 is touching body   2 segment    47
 the retained nodes are  273  266
node  160 of body  1 is touching body   2 segment    46
 the retained nodes are  280  273




          sliding velocity below which sticking is considered   0.15000E+01




                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
  0.000E+00 0.000E+00


                    point loads
```

```
   0.000E+00 0.000E+00



                    increment zero is a null step





          distributed flux    type    current
          list number                 magnitude



                   1          101      1.000

                    e n d   o f   i n c r e m e n t   0

                    formatted post data at increment   0.   0   on tape 19
                     time =       9.34



       transient non auto
       ----------



          time      time     maximum   assembly  max iter
       increment   period     steps    interval  mcreep
        3.000E-04 3.000E-02    100         0      5

       disp change
       ----------


       b.c. changes=    0
       fixed displacement =  0.000E+00  0.000E+00
       a list of degrees of freedom given below
            1
       from node     1 to node   151 by    10
       fixed displacement =  0.000E+00  0.000E+00
       a list of degrees of freedom given below
            2
       from node   161 to node   167 by     1
       fixed displacement = -0.450E-01  0.000E+00
       a list of degrees of freedom given below
            1
       from node   167 to node   342 by     7



       fixed boundary condition summary.
       total fixed degrees of freedom read so far =   49
```

```
   b.c.     node    degree of   magnitude              b.c.     node    degree of   magnitude
  number              freedom                         number              freedom


      1        1          1     0.000E+00                 2       11          1     0.000E+00
      3       21          1     0.000E+00                 4       31          1     0.000E+00
      5       41          1     0.000E+00                 6       51          1     0.000E+00
      7       61          1     0.000E+00                 8       71          1     0.000E+00
      9       81          1     0.000E+00                10       91          1     0.000E+00
     11      101          1     0.000E+00                12      111          1     0.000E+00
     13      121          1     0.000E+00                14      131          1     0.000E+00
     15      141          1     0.000E+00                16      151          1     0.000E+00
     17      161          2     0.000E+00                18      162          2     0.000E+00
     19      163          2     0.000E+00                20      164          2     0.000E+00
     21      165          2     0.000E+00                22      166          2     0.000E+00
     23      167          2     0.000E+00                24      167          1    -4.500E-02
     25      174          1    -4.500E-02                26      181          1    -4.500E-02
     27      188          1    -4.500E-02                28      195          1    -4.500E-02
     29      202          1    -4.500E-02                30      209          1    -4.500E-02
     31      216          1    -4.500E-02                32      223          1    -4.500E-02
     33      230          1    -4.500E-02                34      237          1    -4.500E-02
     35      244          1    -4.500E-02                36      251          1    -4.500E-02
     37      258          1    -4.500E-02                38      265          1    -4.500E-02
     39      272          1    -4.500E-02                40      279          1    -4.500E-02
     41      286          1    -4.500E-02                42      293          1    -4.500E-02
     43      300          1    -4.500E-02                44      307          1    -4.500E-02
     45      314          1    -4.500E-02                46      321          1    -4.500E-02
     47      328          1    -4.500E-02                48      335          1    -4.500E-02
     49      342          1    -4.500E-02

  continue
  ----------


  auto control specified for time of 0.300E-01



                      s t a r t   o f   i n c r e m e n t   1



  fluxes summed over the whole model

  from distributed fluxes
 0.000E+00


  concentrated fluxes
 0.000E+00
```

First solve heat transfer problem

```
                    start of assembly
                     time =      11.16



                    start of matrix solution
                     time =      12.61

                    singularity ratio    7.6978E-01



                    end of matrix solution
                     time =      12.64



                    maximum error between temperature estimate and solution is    0.0000 at node    1
                    this is within tolerance



                    maximum nodal temperature change is  0.517E+01 at node  160

                    this is 0.259E+02  percent of change allowed on control option



                    automatic time stepping is switched off




                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
 0.000E+00 0.000E+00



                    point loads
 0.000E+00 0.000E+00

                    start of assembly
                     time =      13.52




                    start of matrix solution
                     time =      16.82




                    singularity ratio    2.4438E-01



                    end of matrix solution
```

Then solve mechanical problem

```
                     time =      17.00


               maximum displacement  change   at node  192 degree of freedom    1 is equal to    0.450E-01
               maximum displacement increment at node  192 degree of freedom    1 is equal to    0.450E-01
                 convergence ratio                                                                0.100E+01

                    failure to converge to tolerance

                    increment will be recycled


                    maximum connectivity is   16  at node  231


                    workspace needed for optimizing =     67371
                    maximum connectivity is   16  at node  231



                    maximum half-bandwidth is  216  between nodes    9  and  224

                     number of profile entries including fill-in is       5072

                     number of profile entries excluding fill-in is       1641



                    total workspace needed with in-core matrix storage =    125627

                    maximum connectivity is   16  at node  231


                    maximum half-bandwidth is  216  between nodes    9  and  224

                     number of profile entries including fill-in is       5072

                     number of profile entries excluding fill-in is       1641



                    total workspace needed with in-core matrix storage =    125627



                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
0.000E+00 0.000E+00
```

```
                        point loads
      0.000E+00 0.000E+00

                        start of assembly
                         time =      29.14



                        start of matrix solution
                         time =      32.65



                        singularity ratio    2.0027E-01


                        end of matrix solution
                         time =      32.83


                    maximum displacement  change   at node   20 degree of freedom    2 is equal to    0.170E-02
                    maximum displacement increment at node  151 degree of freedom    2 is equal to    0.641E-01
                      convergence ratio                                                               0.266E-01

          separation force required is      0.10511E+03 __________Force required to cause separation


MARC          output for increment    1.      coupled analysis of ring compression


                        dynamic change has reached time of 0.300E-03  of total time period 0.300E-01


                        total transient time = 3.00000E-04



                 distributed flux    type    current
                 list number                 magnitude


                       1          101      1.000

                        e n d   o f   i n c r e m e n t   1
                         time =      36.58
```

Results

The most important heat transfer phenomenon that is occurring in this analysis is the contact of two bodies at different temperatures. The cooling of the aluminum ring results in an increase of the flow stress – effectively hardening the material. Figure 10.2 shows the deformed geometry at the end of 50 increments. Figure 10.3 shows the deformed geometry after 100 increments, corresponding to a 50 percent reduction in height of the aluminum ring. As a consequence of the high friction coefficient used, the ring folds onto the steel disk on both sides. There is an increase of the outer diameter as well as a decrease of the inner diameter. The amount of interface sliding is very small due to the high friction value. Notice the very large deformation in the region where the cylinder folds onto the disk. This results in large distortion of the finite element mesh, which often requires a rezoning of the mesh to insure an accurate analysis. Elastic deformations of the steel disk are not visible; the disk looks like it underwent a rigid body translation.

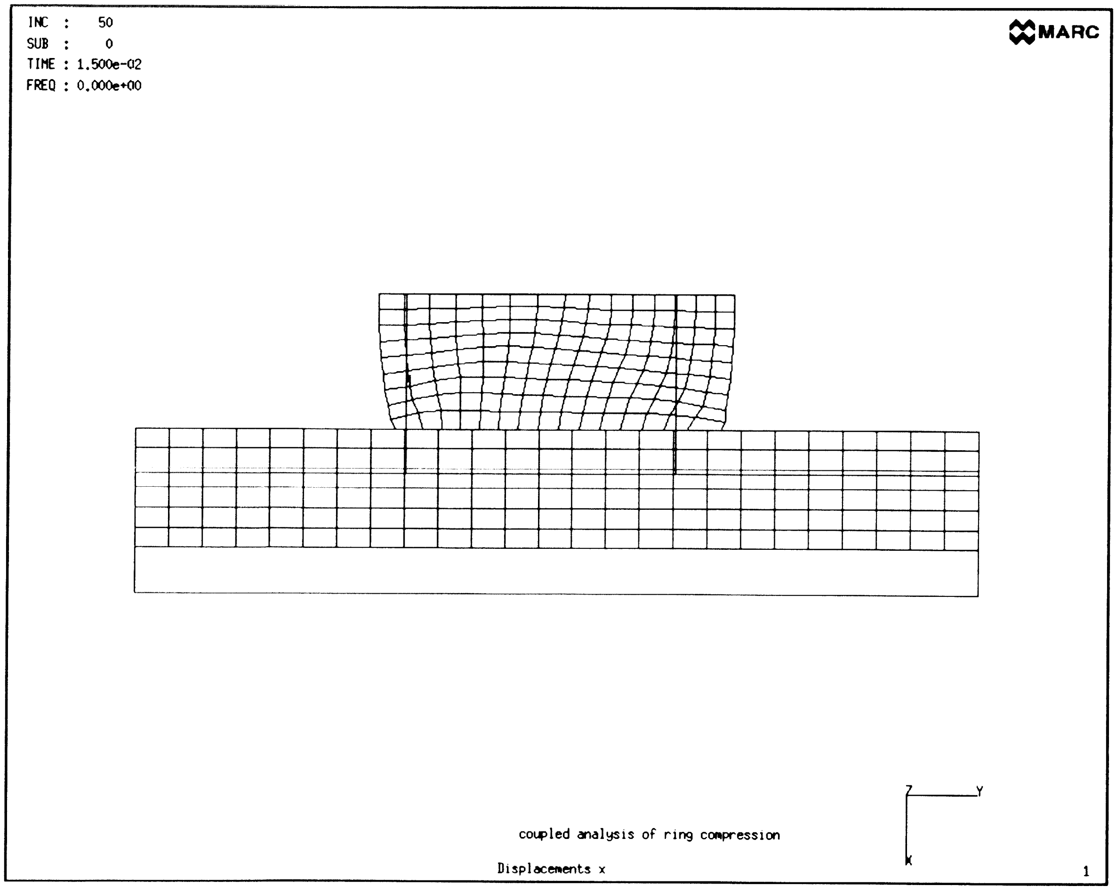


Figure 10.2 **Deformed Geometry After 50 Increments**

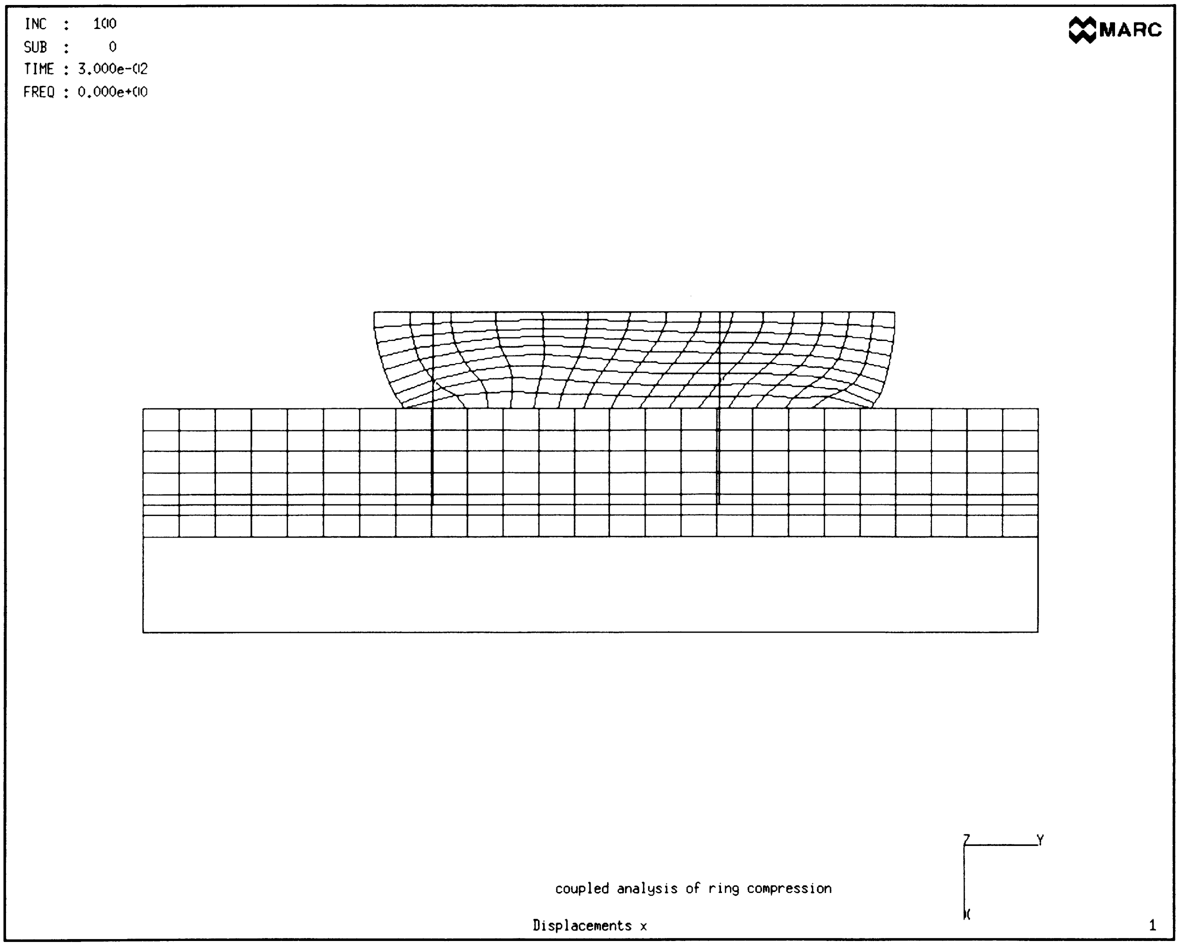


Figure 10.3 **Deformed Geometry after 100 Increments**

Figure 10.4 is a contour plot of the equivalent plastic strains in the ring. They ranged from small amounts in the middle of the contact area (neutral zone) and near the free surface, to very large amounts at the corners where folding took place, and also in the center of the middle plane.

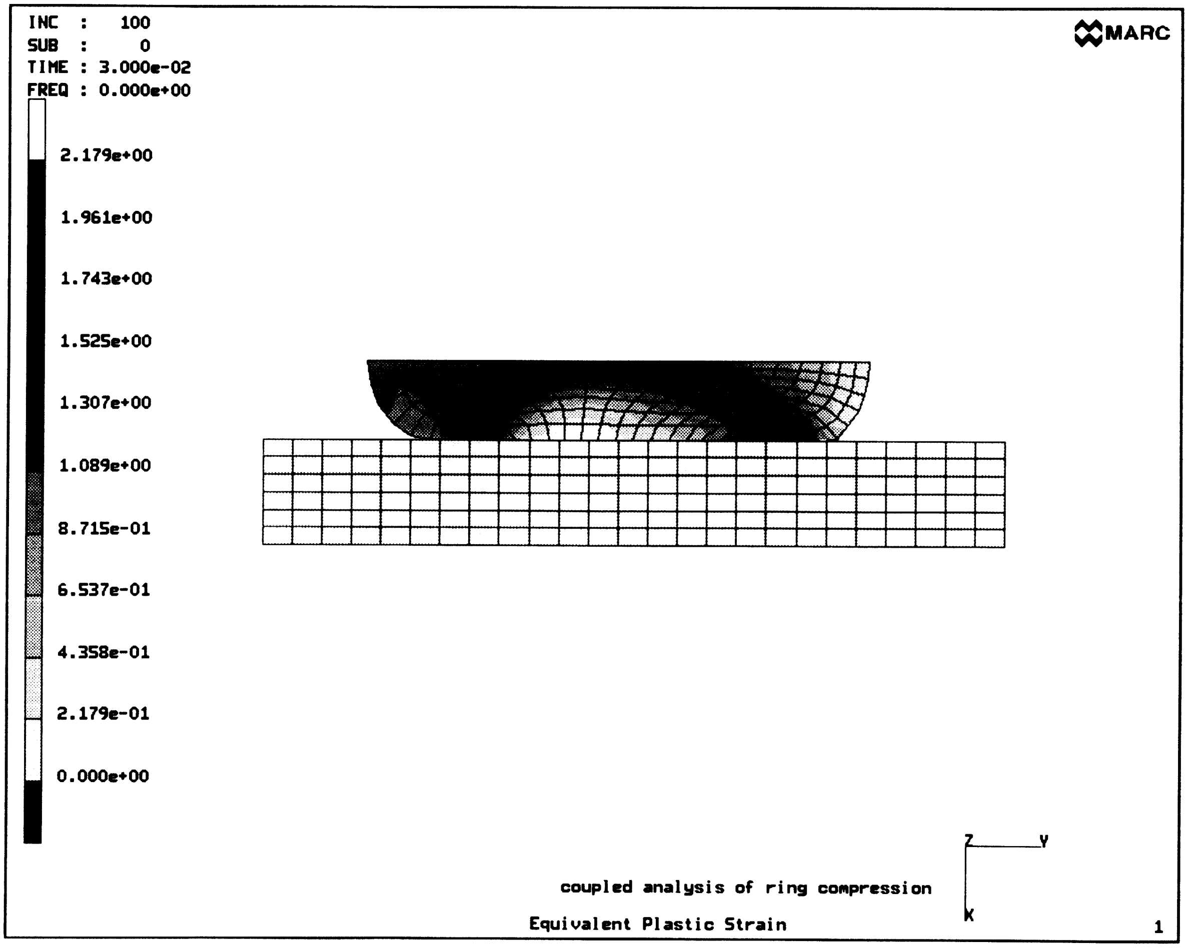


Figure 10.4 **Equivalent Plastic Strains in Ring**

Figure 10.5 shows the equivalent von Mises stress distribution. These stresses are higher in the disk than in the ring, because of the elastic-plastic properties of the aluminum ring. They increase from low values in the free standing areas towards the center. Local peaks in the friction shearing zones are also visible.

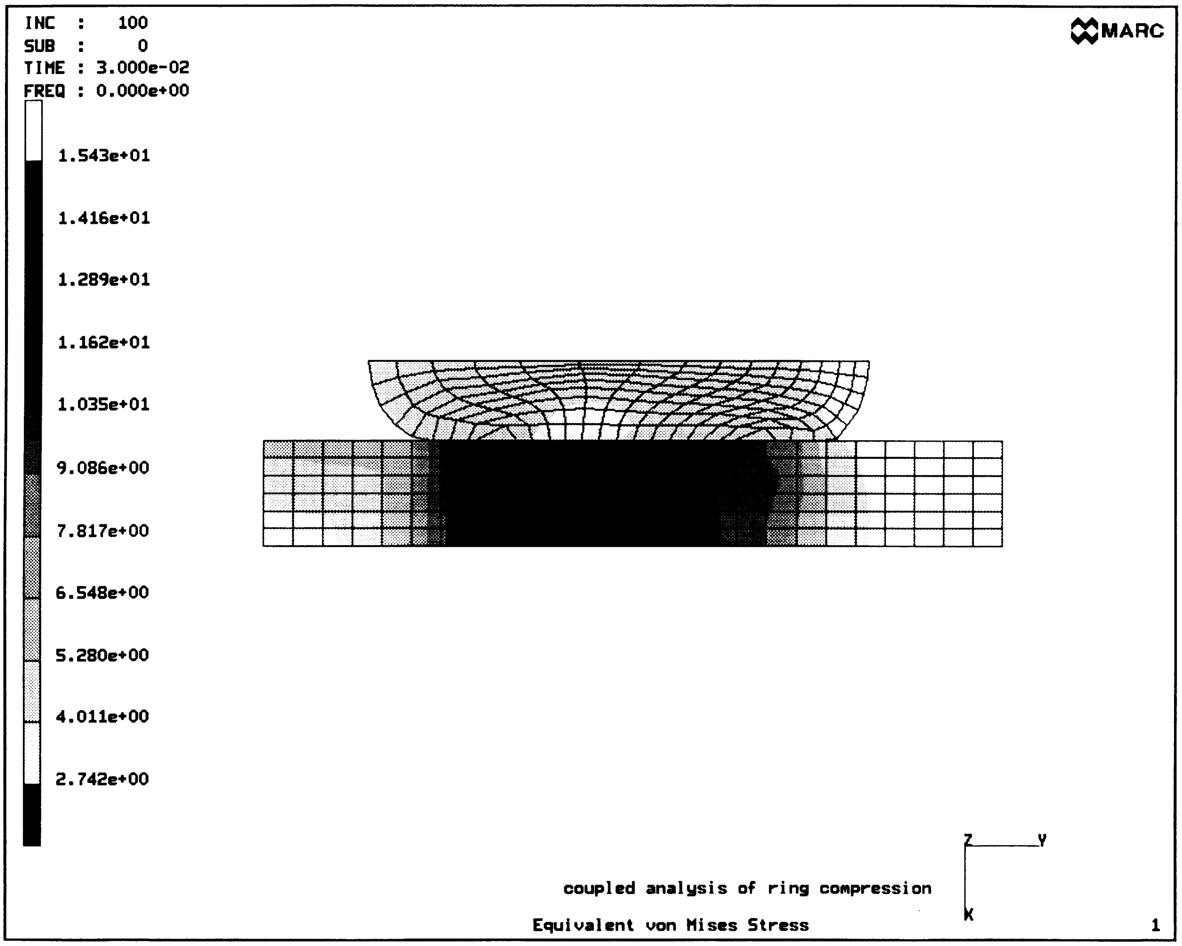


Figure 10.5 **Equivalent Von Mises Stress Distribution in Ring**

Figure 10.6 is the temperature distribution produced from the thermal analysis. The total time of the deformation is only 0.03 seconds. All the thermal effects are confined to the contact region. The high temperature and low flow stress of aluminum produce no noticeable heating due to plastic deformation. On the ring side, the temperature decreases about 55°C at the interface, while the disk heats up to about 140°C. The lower conductivity of the steel and the short time step result in the warmer temperature not fully penetrating the disk.

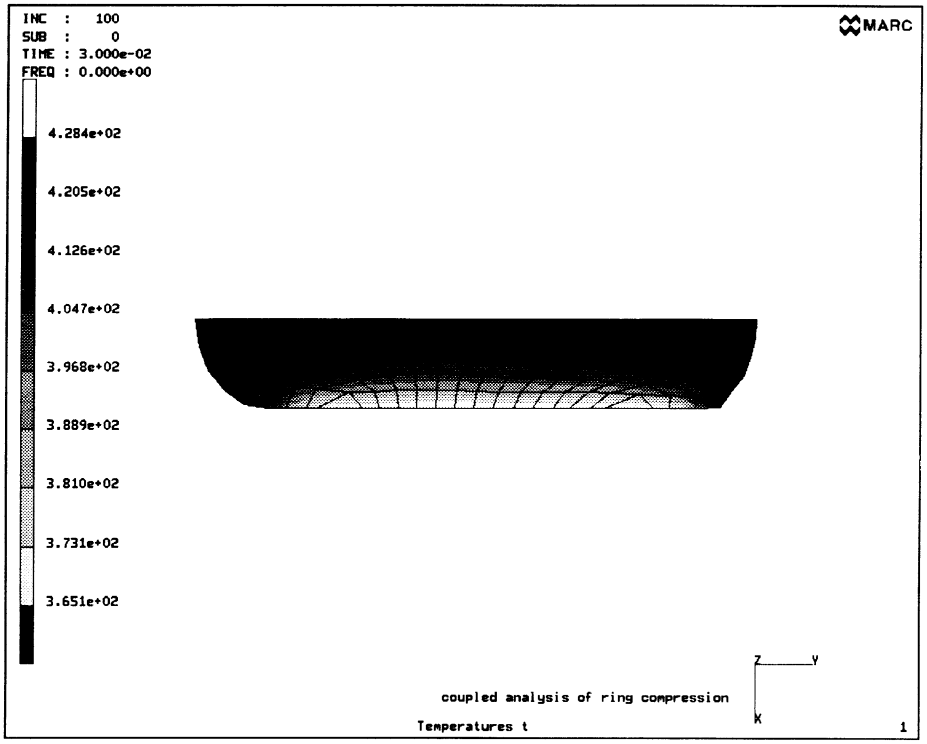


Figure 10.6a **Temperature Distribution in Aluminum Ring**

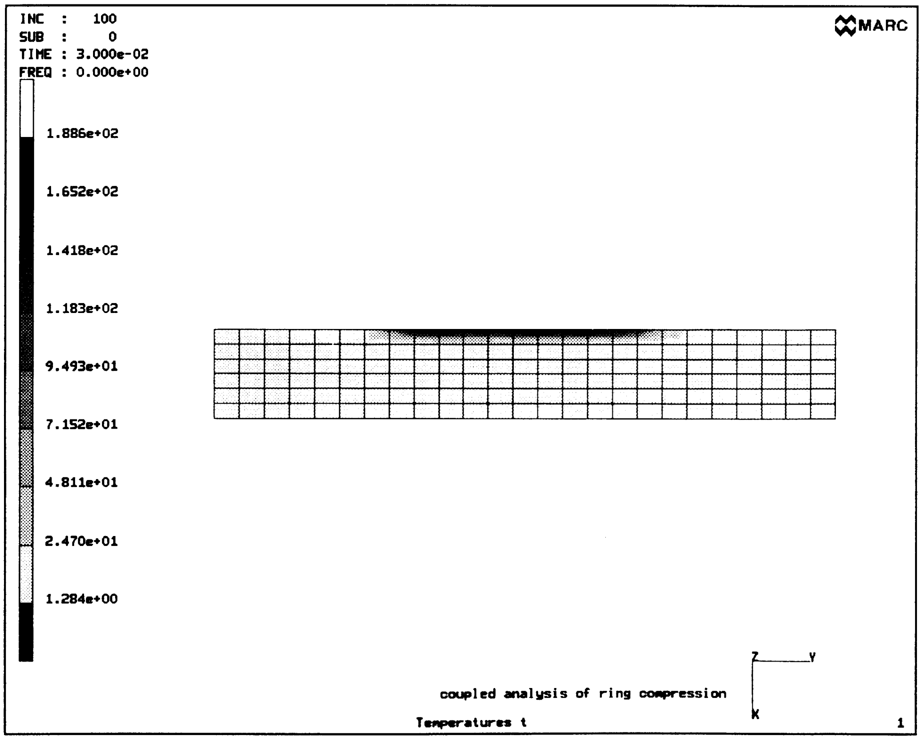


Figure 10.6a **Temperature Distribution in Steel Disk**

Example 11

Side Pressing of a Solid Rubber Cylinder

You have now been exposed to some key concepts in contact analysis. Let us extend these contact analysis ideas to consider an important class of materials commonly found in many manufacturing industries: elastomers or rubber materials. These materials are typically characterized by incompressible or nearly incompressible behavior (Poisson's ratio nearly equal to one-half), nonlinear elastic behavior, and strains of hundreds of percent under load. Therefore, the finite element analysis of these materials requires some special considerations. The aim of Example 11 is to illustrate the modeling and analysis of a rubber material, in this case idealized as a so-called Mooney-Rivlin material (using the MOONEY option). The example treats the side pressing of a solid rubber cylinder by a frictionless rigid surface, which can be modeled as a 2-D plane strain problem (using MARC Element 80). The RESTART option is also illustrated.

Sketch

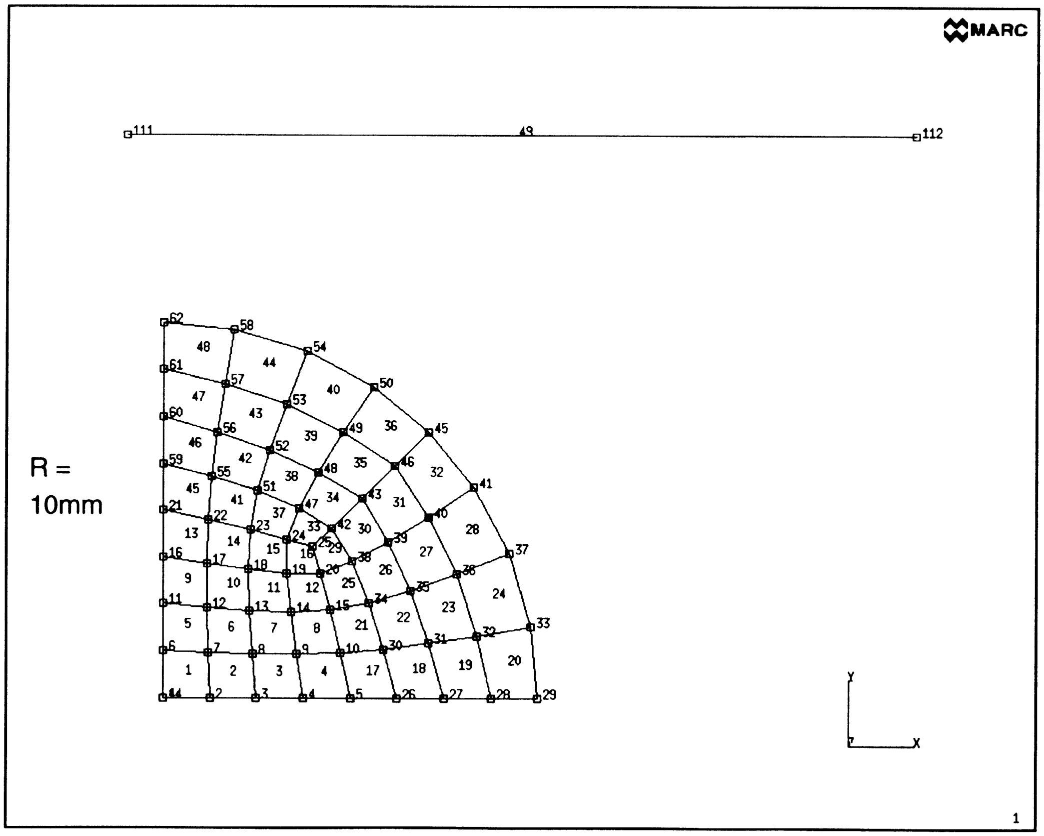


Figure 11.1 Rubber Cylinder and Contact Surface

Model

The idealized FE model of the solid rubber cylinder is a quarter-model of 10.0 mm radius. Symmetry is assumed along the left and bottom edges. (Note that in the plot of the model on the previous page, the rigid body defined by "nodes" 111 and 112 and "element" 49 is shown for display purposes only in Mentat II; these entities do not actually exist internally in the MARC data base for this problem.)

Rubber materials are characterized by four concepts in mechanics; isotropic behavior, incompressible behavior, large strains and visoelastic behavior. Many problems approximate the last two concepts by using an elastic formulation, thus neglecting the rate effects and considering the material nearly incompressible. The LARGE DISP option allows us to accurately model large displacement elastic problems. The Mooney-Rivlin formulation is a reasonably accurate characterization for rubber if the strains are less than 100%. The Herrmann formulation allows the correct treatment of incompressible behavior without the numerical difficulties associated with conventional displacement formulation elements. In particular, conventional elements should not be used in plane strain, axisymmetric, or 3-D continuum analysis if Poisson's ratio is close to .5.

MARC Element 80 (*see MARC Volume B*) is used for the cylinder model. This element has the same formulation as that of Element 11. The only difference is that Element 80 has been modified for the Herrmann variational principle for use in incompressible analysis. The element is a 5-noded isoparametric, quadrilateral, plane strain element. The fifth node is the extra pressure node, meaning it only has a pressure degree of freedom which is not shared with other elements. This extra node also does not require the usual spatial coordinates to be defined.

Properties

The only material properties needed in this problem are the two Mooney-Rivlin constants to characterize the rubber material: C_{10} is given as 8.0 N/mm^2, and C_{01} is 2.0 N/mm^2. (These constants are usually deduced from load-deflection test data.) No thermal effects are considered; the problem is isothermal.

Loads

No mechanical or thermal loads are prescribed. The only loading results from the rubber cylinder being pressed sideways by the rigid surface.

Boundary Conditions

Symmetry conditions are imposed on the bottom and left edges of the model. Along the X-axis, the nine nodes comprising the node set named YFIXME (nodes 1, 2, 3, 4, 5, 26, 27, 28, and 29) are placed on rollers and fixed against displacement in the Y-direction. Along the Y-axis, the nine nodes comprising the node set named XFIXME (nodes 1, 6, 11, 16, 21, 59, 60, 61, and 62) are also placed on rollers and fixed against displacement in the X-direction. (Notice node 1 belongs to both node set YFIXME and node set XFIXME.) The rigid surface will be defined using the CONTACT option.

Special Features

In addition to the MOONEY option for characterizing the rubber behavior, special features described in this example include: output of Cauchy stress (true stress) and total Green-Lagrange strain to examine the stress-strain behavior of the rubber cylinder; the CONTACT option for automated contact analysis without use of gap elements; the RESTART option; and use of the TIME STEP and AUTO LOAD options to control the contact analysis.

Input

A complete input file is included.

PARAMETER Section

The "TITLE" line is self-explanatory. The "SIZING" line tells MARC to set a workspace of 100,000 words. The "ELEMENTS" line indicates Element 80 will be used. The "LARGE DISP" line flags large deformation analysis, and the "END" line terminates the PARAMETER section. LARGE DISP invokes the total Lagrange option, which is appropriate for rubber analysis. The PRINT option is used to obtain additional information regarding the progression of the contact.

MODEL DEFINITION Section

The MODEL DEFINITION options consists of:

a. FE mesh topology – CONNECTIVITY, COORDINATES, and DEFINE blocks
b. Material properties – MOONEY option
c. Boundary conditions
d. Output controls
e. Contact analysis controls
f. RESTART

FE Mesh Topology

The only thing unusual about the "CONNECTIVITY" lines is the presence of a fifth "pressure" node in the last field of each data line. The "COORDINATES" lines are straightforward. It is not necessary to define coordinate positions for the fifth "pressure" node. We are using the DEFINE option to name one element set named ALLE (representing all 48 elements) and four node sets; SURFACE (which corresponds to the nine nodes along the curved edge); XFIXME (the nine nodes along the X-axis); YFIXME (the nine nodes along the Y-axis); and ALLN (all 62 nodes).

Material Properties

The material data to describe the rubber material is entered through the MOONEY option. In this example, we are idealizing the rubber material as a Mooney-Rivlin material, which requires us to input two constants: C_{10} and C_{01}.

NOTE

If we had chosen a neo-Hookian material, we would only need one constant, and if we had chosen a James-Green-Simpson material, we would have to input five constants.

The "1," line merely indicates the material-id number. The '8.,2.," line gives the values of C_{10} and C_{01}, respectively. And the "ALLE" line assigns these values to all 48 elements in the model.

Boundary Conditions

The FIXED DISP option defines the fixed displacement that each specified DOF must take during the first and subsequent increments. After the usual blank line, the "0.,","2", and "YFIXME" lines mean that a zero value shall be assigned to the second DOF (or Y-displacement) for the nine nodes along the X-axis in node set YFIXME. Similarly, the "0.,","1", and "XFIXME" lines mean that a zero value shall be prescribed to the first DOF (or X-displacement) for the nine nodes along the Y-axis in node set XFIXME. As an alternative, two additional contact surfaces could be defined which would represent the symmetry surfaces.

Output Controls

This example has three output control options: POST, PRINT ELEM, and PRINT NODE. The POST option creates a post-processor file for later post-processing by Mentat II. The "6," line tells MARC that six element variables are to be written on the file at each increment. The next four lines ("41" through "44") refer to the four components of Cauchy stress at the four integration points of the element to be output: normal stresses in the global X-, Y-, and Z-directions, and the shear stress in the X-Y plane. The last two lines in the block ("47","48") are the equivalent Cauchy stress and the strain energy density.

The PRINT ELEM option allows us to choose which element quantities to be printed out and for which elements. After the blank line, the "CAUCHY STRAIN" line means we are specifying the Cauchy stress and total strain to be printed out. The next "1" line says we want results for element 1 to be printed, and the last "1" line means we select integration point 1 only.

The PRINT NODE option allows us to select which nodal quantities to be printed out and for which nodes. After the blank line, the "REAC" line indicates we would like to print out reactions/residual forces. The final line in this block, "XFIXME AND YFIXME AND SURFACE", illustrates the use of the union of these three node sets to request a printout of nodal reactions/residual forces for all the nodes contained in the three node sets.

Contact Analysis Controls

The CONTACT option permits you to define surfaces (bodies) in contact problems. You have previously seen this option applied for metallic contact analysis in Example 10. The "2,100,100" line tells MARC that two surfaces are to be defined, that there is a maximum of 100 entities for any surface, and that there is also a maximum of 100 nodes lying on the periphery of any deformable surface. No friction is required in this analysis. The ",,,," line is a simple way to direct MARC to calculate the default value of two of the four parameters: relative sliding velocity between surfaces and the distance below which a node is considered touching a surface. (Of these two parameters, the only one necessary in this problem is the second one – the contact tolerance, which will be calculated automatically by MARC.)

The next three lines constitute the next block in CONTACT and define deformable body 1. The "1," refers to surface (body) 1. All the fields of the next line are blank (",,,") because body 1 is deformable. Since a deformable body cannot have rigid body motion, it is not necessary to give a center of rotation, angular position or velocity. If any values are given, they are ignored by the program. And, the "ALLE" line means this body 1 is comprised of all 48 elements of the quarter-cylinder.

Then, the following five lines make up the next block in the CONTACT option which define rigid surface 2.

NOTE

Rigid bodies are always numbered higher than all deformable bodies.

The "2,1" line refers to body 2, and that there will be one set of geometrical data to be defined for this body. (The non-zero value in the second field of this line informs MARC that this body will be a rigid body.) The "0.,15.,0.,0.,-1.,0.,0.," line gives the seven values which define the body 2 positioning data: zero for the first coordinate of center of rotation; "15." for the second coordinate of the center of rotation, zero for the angle rotated; zero for the first (X) component of velocity; "-1." for the second (Y) component of velocity; zero for the angular velocity; and zero for the friction coefficient. Then, the last three lines in this block define the geometrical data set for rigid body 2. On the "1,2" line, 1 refers to entity type 1 (series of segments); and 2 says two points are to be entered. The "-1.,15.," line gives the coordinates of *point 1* (which is graphically depicted as "node" 111 in the Mentat II plot but does not actually exist in the MARC data base): X-coordinate of -1. and Y-coordinate of 15. And, finally, the "20.,15.," line gives the coordinates of *point 2* (displayed as "node" 112 in the Mentat II plot but does not exist in MARC): X-coordinate of 20., and Y-coordinate of 15. The order of the points of the entities is critical to determine the surface normal. The program uses a right hand rule to determine the outward normal of the surface coming into contact with the rigid surface.

Restart

The RESTART option sets up the flags for the restart files, for either the input of a previous restart file or the output of a restart file from the current analysis. (Here, we will first illustrate the used of the latter. Later, after discussing results from increment 10, we will show you how to set up the restart run.) The "1,10" line after the "RESTART" line is interpreted as follows: the "1" instructs MARC to write out restart data, and the "10" says to do it after every ten increments. This information is written to a file associated with FORTRAN unit 8 by default. See Section 9 in Volume C for more details regarding the machine dependence associated with this file.

The "END OPTION" line terminates the MODEL DEFINITION section.

LOAD INCREMENTATION Section

Two load incrementation options are used together to control the contact analysis: TIME STEP and AUTO LOAD.

The TIME STEP option allows you to prescribe a time step for *static* analysis. The ".5," line means a time step of 0.5 seconds. All contact problems require a time step, even if they are quasi-static in nature.

The AUTO LOAD option is used to describe a number of equal load steps. The "10," line says to use ten equal load increments.

Although the term "time step" is used, remember that this contact analysis is still a static, not dynamic, analysis. The time step is used in conjunction with the velocity prescribed for the rigid surface to control the deformation of the rubber cylinder. Alternatively, instead of using the TIME STEP option with the AUTO LOAD option to control the contact analysis, we could also use either of the two following options: DYNAMIC CHANGE (which will accomplish the same results as this example); for AUTO TIME (in which MARC will perform the adaptive time-stepping).

The "CONTINUE" line terminates the LOAD INCREMENTATION section as well as the entire input file.

Output

Selective portions of the output are included. For reference, the parameter and sizing table is included, as is descriptive information for MARC Element 80. The next page of interest is the interpretation of the input data for the CONTACT option. We see a MARC message (set off by asterisks) that informs us the "distance below which a node is considered touching a surface" is 4.5251E-02, which is considered a reasonable value. Since we had not input a value for this contact tolerance, MARC calculated it to be 1/20 of the smallest element x or y dimension in our model. This contact tolerance value should not be too small (e.g., in the 10^{-4} range); a very small value will lead to potential numerical difficulties.

We then see, after the message "Increment zero is a null step", that the "total transient time" is 5.0 seconds. Why is this so? When CONTACT is used, certain initialization operations are performed during increment zero. These include bringing a rigid body into first contact with a deformable body. Therefore, in our case, rigid body 2 is moved down toward deformable body 1 located 5.0 mm away in the Y-direction, at a Y-velocity of -1.0 mm/seconds. The time consumed was thus 5.0 seconds, which explains the value of the total transient time at the end of increment zero. At this time, increment 1 begins the actual contact analysis.

On the next page, MARC begins to find the first node in contact. We see that automatic subincrementation (also called "increment splitting" in Volume A) is being used in the analysis. After a total transient time of 5.226 seconds (or a time increment of 0.226 seconds), node 62 was found to be in contact. Because we included a PRINT,8 PARAMETER line, MARC proceeds to print out the incremental displacements and contact forces for contact node 62 in a local coordinate system. When a node of a deformable body contacts a rigid surface, the degrees of freedom are transformed into a normal (to the surface) and tangential "local" system. MARC reports the nodal quantities in both the local system and the global system. Then, in the second subincrement (which began with the second "START OF INCREMENT 1" message), MARC found a second node in contact (node 58). The incremental displacements and contact forces for contact nodes 58 and 62 are printed out. At this point in the analysis, the total transient time is now 5.5 seconds after "Increment 1" (with two subincrements of 0.226 seconds and 0.274 seconds) Note that the total time of the subincrements equals the prescribed time step.

Now, MARC prints out the requested data for Increment 1: the Cauchy stresses and total strains (only for element 1, integration point 1). For elements modeled using the MOONEY model, the strain printed out is the Green-Lagrange strain. The second Piola-Kirchhoff stress and Cauchy stresses are normally printed out by default. Then, we see the nodal point data (reaction forces at fixed boundary conditions, residual load correction elsewhere); and the "global die data" (which summarized the status of the two bodies). Increment 1 is now completed, and increment 2 is begun.

During the contact analysis, this automatic increment splitting occurs in Increments 1, 3, 5, and 7. It does not occur in Increments 2, 4, 6, 8, 9, and 10. The solution proceeds through Increment 10, at the conclusion of which the total transient time equals the requested 10.0 seconds. At the successful completion of the analysis, a total of six nodes along the original circular edge are in contact on top with the rigid body. (Appropriate output pages have been included for Increments 1 and 10 only.) The final die data reports the location and velocity of the key point associated with the rigid surface. In addition, the total load on the surface is reported. The total load shown in the X-direction is due to the rubber cylinder contacting the rigid surface at the Y-axis of symmetry.

input data

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
TITLE, SIDE PRESSING OF A SOLID RUBBER CYLINDER
SIZING           100000    48      62
ELEMENTS        80
LARGE DISP
PRINT,8
END
CONNECTIVITY
   48    0    0
    1   80    1    2    7    6   63
    2   80    2    3    8    7   64
    3   80    3    4    9    8   65
    4   80    4    5   10    9   66
    5   80    6    7   12   11   67
    6   80    7    8   13   12   68
    7   80    8    9   14   13   69
    8   80    9   10   15   14   70
    9   80   11   12   17   16   71
   10   80   12   13   18   17   72
   11   80   13   14   19   18   73
   12   80   14   15   20   19   74
   13   80   16   17   22   21   75
   14   80   17   18   23   22   76
   15   80   18   19   24   23   77
   16   80   19   20   25   24   78
   17   80    5   26   30   10   79
   18   80   26   27   31   30   80
   19   80   27   28   32   31   81
   20   80   28   29   33   32   82
   21   80   10   30   34   15   83
   22   80   30   31   35   34   84
   23   80   31   32   36   35   85
   24   80   32   33   37   36   86
   25   80   15   34   38   20   87
   26   80   34   35   39   38   88
   27   80   35   36   40   39   89
   28   80   36   37   41   40   90
   29   80   20   38   42   25   91
   30   80   38   39   43   42   92
   31   80   39   40   46   43   93
   32   80   40   41   45   46   94
   33   80   25   42   47   24   95
   34   80   42   43   48   47   96
   35   80   43   46   49   48   97
   36   80   46   45   50   49   98
   37   80   24   47   51   23   99
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

Additional node for Herrmann (incompressible) formulation

```
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
------------------------------------------------------------------------------------------------
   38    80    47    48    52    51   100
   39    80    48    49    53    52   101
   40    80    49    50    54    53   102
   41    80    23    51    55    22   103
   42    80    51    52    56    55   104
   43    80    52    53    57    56   105
   44    80    53    54    58    57   106
   45    80    22    55    59    21   107
   46    80    55    56    60    59   108
   47    80    56    57    61    60   109
   48    80    57    58    62    61   110
COORDINATES
    2    62     0     0
    1   0.00000   0.00000
    2   1.25000   0.00000
    3   2.50000   0.00000
    4   3.75000   0.00000
    5   5.00000   0.00000
    6   0.00000   1.25000
    7   1.20239   1.20242
    8   2.39481   1.16635
    9   3.57014   1.15612
   10   4.73025   1.19078
   11   0.00000   2.50000
   12   1.16634   2.39491
   13   2.30892   2.30901
   14   3.40778   2.26961
   15   4.45948   2.32059
   16   0.00000   3.75000
   17   1.15620   3.57034
   18   2.26964   3.40800
   19   3.29552   3.29563
   20   4.20054   3.30699
   21   0.00000   5.00000
   22   1.19103   4.73060
   23   2.32079   4.45986
   24   3.30701   4.20076
   25   3.99943   3.99952
   26   6.25000   0.00000
   27   7.50000   0.00000
   28   8.75000   0.00000
   29  10.00000   0.00000
   30   5.89332   1.28833
   31   7.09258   1.44651
   32   8.37183   1.65085
   33   9.82290   1.87366
   34   5.50180   2.51737
   35   6.60588   2.84754
   36   7.82207   3.28346
   37   9.23879   3.82683
------------------------------------------------------------------------------------------------
    5    10    15    20    25    30    35    40    45    50    55    60    65    70    75    80
```

```
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
   38   5.05010   3.61456
   39   6.00776   4.14350
   40   7.07193   4.80878
   41   8.27071   5.62097
   42   4.49162   4.49168
   43   5.30330   5.30330
   45   7.07107   7.07107
   46   6.18718   6.18718
   47   3.61464   5.05032
   48   4.14346   6.00778
   49   4.80875   7.07192
   50   5.62097   8.27071
   51   2.51778   5.50234
   52   2.84781   6.60622
   53   3.28357   7.82221
   54   3.82683   9.23879
   55   1.28886   5.89392
   56   1.44689   7.09301
   57   1.65103   8.37203
   58   1.87366   9.82290
   59   0.00000   6.25000
   60   0.00000   7.50000
   61   0.00000   8.75000
   62   0.00000  10.00000
DEFINE    NODE      SET       SURFACE
   29   33   37   41   45   50   54   58   62
DEFINE    NODE      SET       YFIXME
    1    2    3    4    5   26   27   28   29
DEFINE    NODE      SET       XFIXME
    1    6   11   16   21   59   60   61   62
DEFINE    NODE      SET       ALLN
    1 TO     62
DEFINE    ELEMENT   SET       ALLE
    1 TO     48
COMMENT,  THE MOONEY CARD IS USED TO INPUT FIVE MOONEY-RIVLIN COEFFS
COMMENT,  HERE, ONLY THE FIRST TWO ARE USED.
MOONEY

1,
8.,2.,
ALLE
FIXED DISP

0.,
2
YFIXME
0.,
1
XFIXME
POST
--------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
```

```
    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
6,
41,
42,
43,
44,
47,
48,
PRINT ELEM

CAUCHY STRAIN
1
1
PRINT NODE

REAC
XFIXME AND YFIXME AND SURFACE
CONTACT
2,100,100
,,,,
1,
,,,
ALLE
2,1
0.,15.,0.,0.,-1.,0.,0.,
1,2
-1.,15.,
20.,15.,
END OPTION
TIME STEP
.5,
AUTO LOAD
10,
CONTINUE
--------------------------------------------------------------------------------
    5    10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
--------------------------------------------------------------------------------
--------------------------------------------------------------------------------
                          ********************************************************
                          ********************************************************


                           program sizing and options requested as follows




                           element type requested**************************    80
                           number of elements in mesh**********************    48
                           number of nodes in mesh*************************   110
                           max number of elements in any dist load list***      0
                           maximum number of boundary conditions**********     18
                           large displacement analysis flagged************
                           load correction flagged or set*****************
```

```
      number of lists of distributed loads***********    3
      option for debug print out*********************    1
      stresses stored at all integration points******
      tape no.for input of coordinates + connectivity    5
      no.of different materials     1 max.no of slopes   5
      mooney material in hybrid elements*************
      maximum elements variables per point on post tp   33
      number of points on shell section *************   11
      option for terminal debug**********************
      new style input format will be used************
      maximum number of set names is*****************   10
      number of processors used *********************    1
      vector length used ****************************    1


      end of parameters and sizing
     *************************************************
     *************************************************



key to stress, strain and displacement output



          element type   80


5-node isoparametric quadrilateral plane strain
with extra pressure node
herrmann formulation

stresses and strains in global directions
  1=xx
  2=yy
  3=zz
  4=xy

displacements in global directions at corner nodes
  1=u global x direction
  2=v global y direction

deg. freedom at fifth node
  1=mean pressure variable


workspace needed for input and stiffness assembly      79467


internal core allocation parameters
```

```
                degrees of freedom per node (ndeg)  2
                coords per node (ncrd)  2
                strains per integration point (ngens)  5
                max. nodes per element (nnodmx)  5
                max.stress components per int. point (nstrmx)   5
                max. invariants per int. points (neqst)  1


                flag for element storage (ielsto)  0
                elements in core, words per element (nelsto)      1144
                                total space required           54912
                vectors in core, total space required      3493


                words per track on disk set to 4096


                internal element variables



                internal element number  1  library code type 80
                number of nodes=  5
                stresses stored per integration point =  5
                direct continuum components stored =  3
                shear continuum components stored =  1
                shell/beam flag = 0
                curvilinear  coord. flag = 0
                int.points for elem. stiffness  4
                number of local inertia directions  2
                int.point for print if all points not flagged  5
                int. points for dist. surface loads (pressure)  2
                library code type = 80
                no local rotation flag = 1
                generalized displ. flag = 0
                large disp. row counts     4    4    0    7    0


                residual load correction is invoked

                  ⋮                                      ⋮

comment,  the mooney card is used to input five mooney-rivlin coeffs
comment,  here, only the first two are used.

mooney
----------

mooney material   material id =     1
  c10        c01        rho        a3        c11        c20        c30
```

```
  0.800E+01  0.200E+01  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
 name of element set is alle
                 •                                       •
                 •                                       •
                 •                                       •
                 •                                       •

 contact
 ----------


 number of bodies                    =    2
```

Two bodies

```
 max number of entities per body   =  100
 bound on number of boundary nodes =  100
 friction type(1-m , 2-coulomb)    =    0
 distrib-0 or nodal-1 coul. frict  =    0
```

No friction

```
 relative velocity below which a
 node is considered sticking       =  0.00000E+00
 distance below which a node is
 considered touching a surface     =  0.00000E+00
 nodal reaction above which a node
 separates from a body             =  0.00000E+00
```

MARC will calculate its own tolerances

```
 body number                       =    1
 number of sets of data            =    0
```

First body is a deformable body consisting of elements in set ALLE (zero indicates deformable body)

```
            body positioning data
 1st coordinate of center of rotation       0.00000E+00
 2nd coordinate of center of rotation       0.00000E+00
 angle rotated                              0.00000E+00
 1st component of velocity                  0.00000E+00
 2nd component of velocity                  0.00000E+00
 angular velocity                           0.00000E+00
 friction coefficient                       0.00000E+00

 name of element set is alle

 body number                       =    2
 number of sets of data            =    1
```

Second body is a rigid surface defined by one set.

```
            body positioning data
 1st coordinate of center of rotation       0.00000E+00
 2nd coordinate of center of rotation       0.15000E+02
 angle rotated                              0.00000E+00
 1st component of velocity                  0.00000E+00
 2nd component of velocity                 -0.10000E+01
 angular velocity                           0.00000E+00
 friction coefficient                       0.00000E+00

 data set type                     = 1
 number of points/method to be read= 2
```

Rigid surface composed of a straight line

```
   point                  coordinates
       1   -0.10000E+01  0.15000E+02
       2    0.20000E+02  0.15000E+02

   data internally reduced to     1 segments



   end option
   ----------
```

⋮

MARC calculates contact tolerances

```
   * * * * * * * * * * * *

    distance below which a node is considered
    touching a surface is                      4.52510E-02

   * * * * * * * * * * * *



                    total workspace needed with in-core matrix storage =    108077



node   62 of body  1 is touching body   2 segment     1
     it will be fixed to the body because it has a b.c. applied









                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
     0.000E+00 0.000E+00


                    point loads
     0.000E+00 0.000E+00
```

```
                 reaction forces/residuals at transformed shell nodes in transformed system

 node            residuals and reactions

   62  0.000E+00  0.000E+00



                 increment zero is a null step



                 total transient time = 5.00000E+00




                 e n d   o f   i n c r e m e n t   0


                 binary post data at increment   0.   0   on tape 16
                  time =      1.25



  time step
  ----------
                                  Desired Time Step

  time increment =     0.50000

  auto load
  ----------



  iotnum,incasm
     10    0

  continue
  ----------



  equal load incs specified for  10  increments



                 s t a r t   o f   i n c r e m e n t   1



                 load increments associated with each degree of freedom
                 summed over the whole model

                 distributed loads
 0.000E+00 0.000E+00



                 point loads
 0.000E+00 0.000E+00
```

```
                  start of assembly
                   time =       1.29




                  start of matrix solution
                   time =       1.92
```

NOTE

Increment 1 has been subdivided to satisfy contact condition. Time step has been subdivided.

```
                  singularity ratio    5.9829E-01


                  end of matrix solution
                   time =       2.04
```

Displacements of nodes in contact in local system.

```
                  incremental displacements at transformed shell nodes in transformed system

  node            incremental displacements

   62   0.000E+00  -5.000E-01

                 maximum residual force  at node   58 degree of freedom    1 is equal to    0.401E+0C
                 maximum reaction force  at node   62 degree of freedom    2 is equal to    0.807E+01
                 convergence ratio                                                          0.497E-01



   separation force required is     0.40080E+00
```

Reaction forces of nodes in contact in local system.

```
                  reaction forces/residuals at transformed shell nodes in transformed system

  node            residuals and reactions

   62   4.537E+00  -8.066E+00


MARC          output for increment   1.    side pressing of a solid rubber cylinder



                  total transient time = 5.22611E+00
```

```
                    e n d   o f   i n c r e m e n t   1
                     time =       2.45




                    s t a r t   o f   i n c r e m e n t   1



node   58 of body  1 is touching body   2 segment     1


                    load increments associated with each degree of freedom
                    summed over the whole model

                    distributed loads
   0.000E+00 0.000E+00




                    point loads
   0.000E+00 0.000E+00

                    start of assembly
                     time =       2.46


                    start of matrix solution
                     time =       3.03
```

Remainder of Increment One is applied

```
                    singularity ratio    5.9829E-01


                    end of matrix solution
                     time =       3.15



                    incremental displacements at transformed shell nodes in transformed system

   node             incremental displacements

     58   2.447E-02  -2.739E-01
     62   0.000E+00  -2.739E-01

                maximum residual force  at node   57 degree of freedom    1 is equal to    0.150E+00
                maximum reaction force  at node   62 degree of freedom    2 is equal to    0.645E+01
                 convergence ratio                                                          0.233E-01
```

```
        separation force required is      0.15047E+00




                        reaction forces/residuals at transformed shell nodes in transformed system

      node              residuals and reactions

        58   4.610E-01  -6.220E+00
        62   2.941E+00  -6.446E+00



 MARC          output for increment    1.     side pressing of a solid rubber cylinder



                        total transient time = 5.50000E+00


       tresca     mises    mean   p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
     intensity intensity  normal   minimum intermediate maximum      1          2          3          4          5          6
                         intensity



element    1  point   1        integration pt. coordinate=      0.262E+00     0.262E+00
cauchy  2.996E+00 5.732E-02-7.721E-01-2.272E+00-7.690E-01 7.244E-01 7.244E-01-2.272E+00-7.690E-01 7.300E-03
strain  7.465E-02 4.310E-02 2.933E-04-3.688E-02 0.000E+00 3.776E-02 3.776E-02-3.688E-02 0.000E+00 3.492E-04


                        n o d a l   p o i n t   d a t a



                 reaction forces at fixed boundary conditions, residual load correction elsewhere


          1 -0.44351        1.4619                  2 -3.35490E-03   2.8238                  3 -5.44087E-03   2.5319
          4 -5.61666E-03   2.0970                   5 -4.42654E-03   1.5862                  6 -0.87960      2.44223E-03
         11 -0.85730       5.25744E-03             16 -0.81277       9.19447E-03            21 -0.73129      1.62683E-02
         26 -2.70468E-03   1.0891                  27 -1.31770E-03   0.64634                28 -4.39023E-04  0.29192
         29 -6.66644E-05   5.50456E-02             33  1.59373E-04  -6.16683E-04            37  1.10974E-03 -1.39849E-03
         41  3.30971E-03  -1.05534E-03             45  6.98017E-03  -2.51094E-03            50  1.74476E-02  1.86569E-03
         54  0.10016      -5.17264E-02             58  0.46101      -6.2199                 59 -0.55411      2.99837E-02
         60 -0.25296       7.15146E-02             61  1.6323        6.43309E-02            62  2.9413      -6.4461
```

```
                contact forces between bodies

       1      0.          0.           2      0.          0.           3      0.          0.
       4      0.          0.           5      0.          0.           6      0.          0.
      11      0.          0.          16      0.          0.          21      0.          0.
      26      0.          0.          27      0.          0.          28      0.          0.
      29      0.          0.          33      0.          0.          37      0.          0.
      41      0.          0.          45      0.          0.          50      0.          0.
      54      0.          0.          58  0.46101     -6.2199         59      0.          0.
      60      0.          0.          61      0.          0.          62  2.9413      -6.4461

                          summary of externally applied loads

  0.00000E+00    0.00000E+00

                          summary of reaction/residual forces

 -0.31655E-01    0.17764E-14


                               g l o b a l   d i e   d a t a


       body number                           1

       1st coord. of center of rotation      0.000
       2nd coord. of center of rotation      0.000
       total angle rotated                   0.000

       1st component of velocity             0.000
       2nd component of velocity             0.000
       angular velocity                      0.000

       1st component of total load       0.000E+00
       2nd component of total load       0.000E+00
       moment w.r.t. center of rotation  0.000E+00


       body number                           2

       1st coord. of center of rotation      0.000
       2nd coord. of center of rotation      9.500
       total angle rotated                   0.000

       1st component of velocity             0.000
       2nd component of velocity            -1.000
       angular velocity                      0.000

       1st component of total load      -0.461E+00
       2nd component of total load       0.127E+02
       moment w.r.t. center of rotation  0.119E+02
```

Prints out forces on rigid surface

```
                  e n d   o f   i n c r e m e n t   1

                  binary post data at increment   1.   0   on tape 16
                   time =        3.61
                   ●                                           ●
                   ●                                           ●
                   ●                                           ●
                   ●                                           ●
```

Last Increment

```
                  s t a r t   o f   i n c r e m e n t  10



node   41 of body  1 is touching body   2 segment      1







                  load increments associated with each degree of freedom
                  summed over the whole model

                  distributed loads
   0.000E+00 0.000E+00


                  point loads
   0.000E+00 0.000E+00

                  start of assembly
                   time =       16.21




                  start of matrix solution
                   time =       16.79



                  singularity ratio    5.8923E-01



                  end of matrix solution
                   time =       16.90
```

```
                    incremental displacements at transformed shell nodes in transformed system

         node             incremental displacements

          41   9.898E-01  -4.748E-01
          45   8.242E-01  -5.000E-01
          50   6.596E-01  -5.000E-01
          54   4.626E-01  -5.000E-01
          58   2.269E-01  -5.000E-01
          62   0.000E+00  -5.000E-01

                   maximum residual force  at node   40 degree of freedom    2 is equal to    0.879E+00
                   maximum reaction force  at node   58 degree of freedom    2 is equal to    0.204E+03
                   convergence ratio                                                           0.432E-02


          separation force required is     0.87871E+00


                    reaction forces/residuals at transformed shell nodes in transformed system

         node             residuals and reactions

          41   1.039E+00  -2.189E+01
          45   2.922E-01  -8.129E+01
          50   2.644E-02  -1.364E+02
          54   2.303E-02  -1.830E+02
          58   1.382E-02  -2.036E+02
          62   1.526E+01  -1.017E+02

MARC          output for increment   10.     side pressing of a solid rubber cylinder


                   total transient time = 1.00000E+01


      tresca     mises    mean  p r i n c i p a l   v a l u e s        p h y s i c a l   c o m p o n e n t s
    intensity intensity  normal  minimum intermediate maximum     1         2         3         4         5         6
                          intensity


element    1  point   1       integration pt. coordinate=     0.262E+00    0.262E+00
cauchy  1.029E+02 3.497E+01-2.908E+01-7.365E+01-4.284E+01 2.925E+01 2.925E+01-7.365E+01-4.284E+01 4.701E-02
```

```
strain  2.508E+00 1.747E+00 5.649E-01-4.066E-01 0.000E+00 2.101E+00 2.101E+00-4.066E-01 0.000E+00-4.010E-03



                              n o d a l   p o i n t   d a t a




                 reaction forces at fixed boundary conditions, residual load correction elsewhere


          1  -7.9483        104.79                 2  1.59811E-02   198.63                 3  2.93374E-02   167.39
          4  3.41780E-02    123.71                 5  2.93969E-02   78.932                 6  -15.293      -6.12351E-02
         11  -13.467       -0.10415               16  -10.272      -0.13465               21  -5.3896      -0.15265
         26  1.88558E-02    42.603                27 -5.48082E-03   17.566                28 -1.18754E-02   2.8949
         29 -1.08488E-02   -1.5750                33  1.55076E-02 -1.81560E-02            37  0.31573      -0.55992
         41   1.0388       -21.892                45  0.29216      -81.287                50  2.64372E-02  -136.42
         54  2.30280E-02   -183.01                58  1.38206E-02  -203.63                59   1.2882      -0.13450
         60   10.320       -0.15347               61   21.842      -0.13276               62   15.265      -101.75



                 contact forces between bodies


          1       0.            0.                 2       0.            0.                 3       0.            0.
          4       0.            0.                 5       0.            0.                 6       0.            0.
         11       0.            0.                16       0.            0.                21       0.            0.
         26       0.            0.                27       0.            0.                28       0.            0.
         29       0.            0.                33       0.            0.                37       0.            0.
         41   1.0388       -21.892                45  0.29216      -81.287                50  2.64372E-02  -136.42
         54  2.30280E-02   -183.01                58  1.38206E-02  -203.63                59       0.            0.
         60       0.            0.                61       0.            0.                62   15.265      -101.75

                              summary of externally applied loads

    0.00000E+00     0.00000E+00

                              summary of reaction/residual forces

   -0.56518E+00     0.21316E-12



                                   g l o b a l   d i e   d a t a



        body number                                1

        1st coord. of center of rotation           0.000
```

```
2nd coord. of center of rotation        0.000
total angle rotated                     0.000

1st component of velocity               0.000
2nd component of velocity               0.000
angular velocity                        0.000

1st component of total load         0.000E+00
2nd component of total load         0.000E+00
moment w.r.t. center of rotation    0.000E+00




body number                             2

1st coord. of center of rotation        0.000
2nd coord. of center of rotation        5.000
total angle rotated                     0.000

1st component of velocity               0.000
2nd component of velocity              -1.000
angular velocity                        0.000

1st component of total load        -0.139E+01
2nd component of total load         0.728E+03
moment w.r.t. center of rotation    0.391E+04

                          e n d   o f   i n c r e m e n t  10

                          binary post data at increment  10.    0    on tape 16
                           time =       17.36



                          *** end of input deck - job ends




                                     marc exit number 3004
```

Results

Figure 11.2 shows the final deformed geometry (Increment 10) of the rubber cylinder quarter-model. Six nodes (41, 45, 50, 54, 58, 62) along the original circular edge are in contact on top with the rigid body. The height reduction due to the cylinder side compression is approximately half the radius.

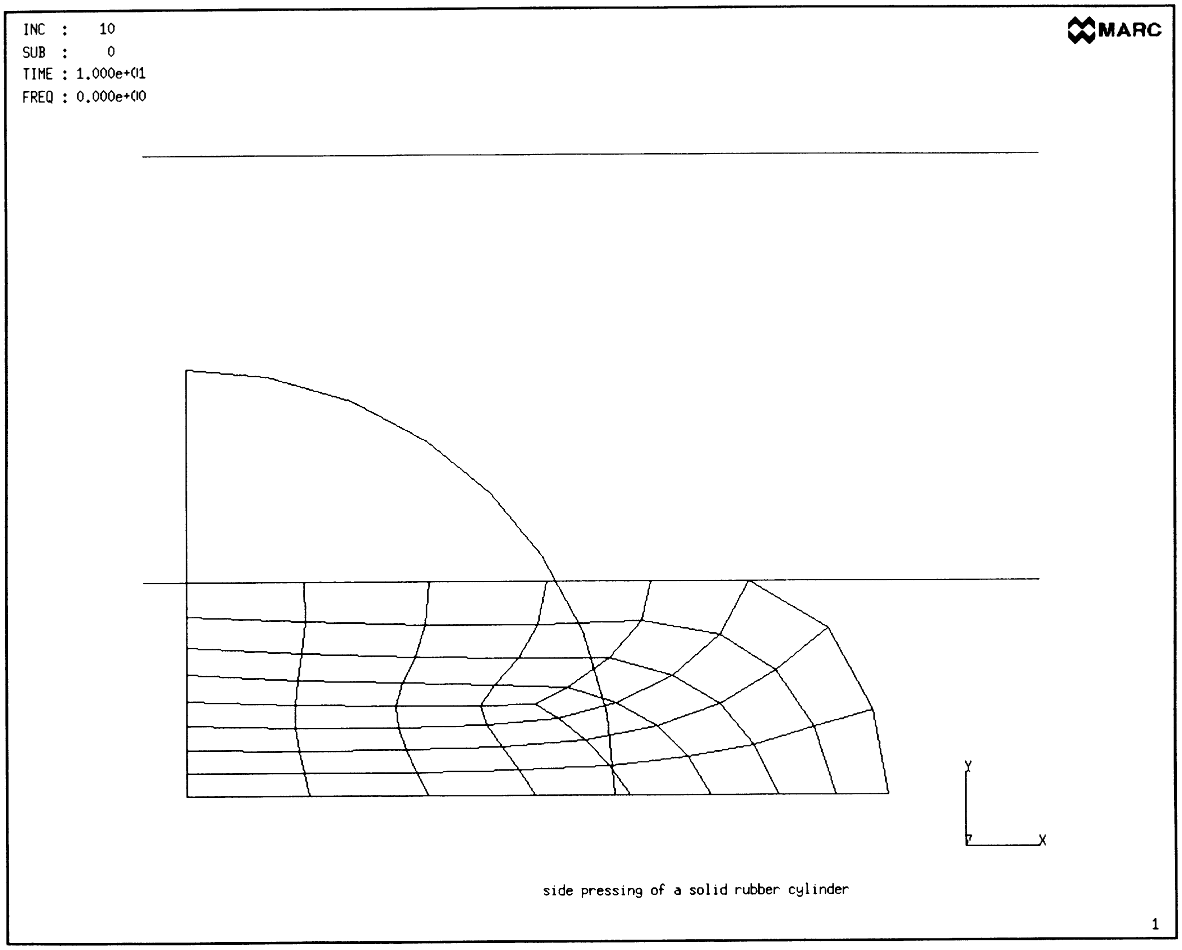


Figure 11.2 **Deformed Geometry**

Since we included a PRINT,8 option in the run, additional printout in the output shows how the contact surface grew in size and the total contact force in the Y-direction increased in magnitude from increment to increment.

Increment	Time (seconds)	Contact Nodes	Total Contact Force (N)
1	5.50	58, 62	12.7
2	6.00	58, 62	29.9
3	6.50	54, 58, 62	56.4
4	7.00	54, 58, 62	87.2
5	7.50	50, 54, 58, 62	131.2
6	8.00	50, 54, 58, 62	187.0
7	8.50	45, 50, 54, 58, 62	264.4
8	9.00	45, 50, 54, 58, 62	369.6
9	9.50	45, 50, 54, 58, 62	509.7
10	10.00	41, 45, 50, 54, 58, 62	727.8

Figure 11.3 shows the variation of the equivalent Cauchy stress plotted on the deformed geometry. At element 1 integration point 1, the equivalent Cauchy stress is 92.36 N/mm^2, while the maximum X-strain is 2.1 (or 210%).

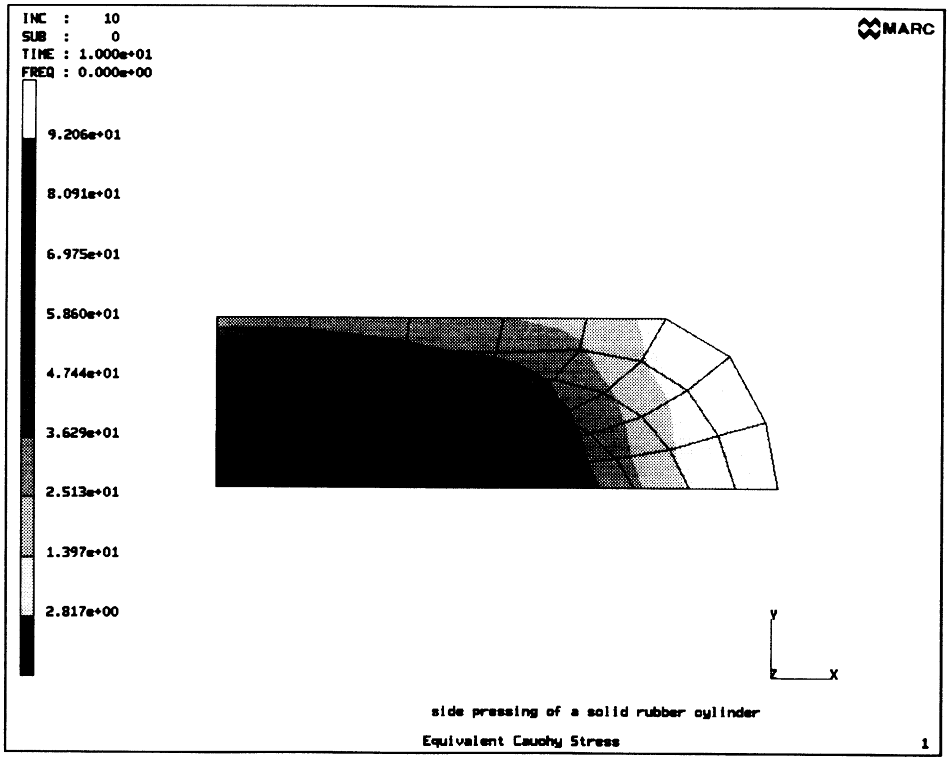


Figure 11.3 **Equivalent Cauchy Stress Distribution**

Figure 11.4 is a plot of the strain energy distribution in the model, showing maximum values in elements 1 and 5 at the center of the cylinder.

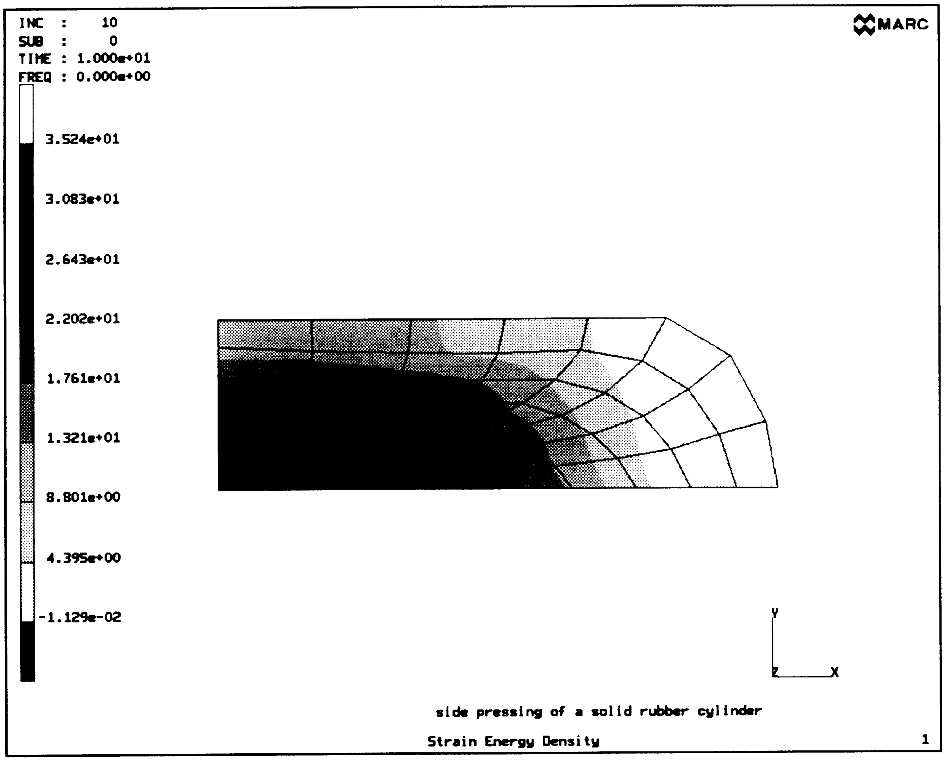


Figure 11.4 **Strain Energy Density Distribution**

The Restart Run

Let us suppose you wish to use the restart run file you have generated previously, and make a second run for another five increments using a finer time step of 0.2 seconds.

The input file for the restarted run is quite simple, and is considerably shorter than before because most of the mesh information (CONNECTIVITY, COORDINATES, material property, boundary conditions) has already been stored in the restart file.

```
TITLE, SIDE PRESSING OF A SOLID RUBBER CYLINDER--RESTART
RUN
SIZING,100000
ELEMENTS, 80
LARGE DISP
END
CONTACT
2,100,100
,,,,
1,
,,,
ALLE
2,1
0.,15.,0.,0.,-1.,0.,0.,
1,2
-1.,15.,
20.,15.,
POST
6,
41,
42,
43,
44,
47,
48,
RESTART
3,10,10
END OPTION
TIME STEP
.2,
AUTO LOAD
5,
CONTINUE
```

The five lines in the PARAMETER section are the same as before, except for the TITLE option which indicates this is a restarted run. (In fact, most of the PARAMETER options *cannot* be changed in a restart run.) In the MODEL DEFINITION section, the CONTACT block and the POST block remain the same.

The RESTART block is changed. The "3,10,10" line is interpreted thus: "3" means to restart a problem (and continue writing restart data for subsequent restart); the first "10" is the number of increments between writing of restart data; and the second "10" is the increment at which the restarted run will be begun. Although the RESTART block can be placed anywhere in the MODEL DEFINITION section, its location relative to the location of the POST block is important and determines the exact contents in the POST file. If the RESTART block appears *before* the POST block, the POST file will contain information from the increments of the restart file as well as the increments on the continuing analysis. If the RESTART block appears *after* the POST block (such as in our example), the POST file will contain only information from the increments of the continuing analysis.

The previously generated restart information would be read from unit 9 (by default) and subsequent restart information would be stored on unit 8. See Section 9 of Volume C regarding the machine-dependent nature in the use of restart files.

The TIME STEP and AUTO LOAD options in the LOAD INCREMENTATION section perform the same functions as previously described. The ".2," line means a time step of 0.2 seconds. And the "5," line says to use five equal load increments.

Exercises

Try the same problem using the AUTO TIME option, instead of using TIME STEP with AUTO LOAD options. Compare the answers. What do you think the results would be like if the cylinder were hollow instead of solid? Try different wall thicknesses for the cylinder and see if rubber-to-rubber contact occurs.

Appendix A: Keyword Index

Table A-1 Parameter Cards

Keyword	MARC PRIMER Example
ALIAS	9
BEAM SECT	4
COMMENT	3B, 3C, 4, 5, 6, 7
COUPLE	10
DYNAMIC	3A, 3B, 3C
ELEMENTS	All
END	All
FINITE	10
HEAT	8
LARGE DISP	6, 7, 10, 11
LUMP	8
PRINT	10, 11
SCALE	5
SHELL SECT	3B, 7
SIZING	All
TITLE	All
UPDATE	10

Table A-2 Model Definition

Keyword	MARC PRIMER Example
CHANGE STATE	9
COMMENT	All
COMPOSITE	4
CONNECTIVITY	All
CONTACT	10, 11
CONTROL (*for stress analysis*)	5, 7, 9, 10
CONTROL (*for heat transfer analysis*)	8
CONVERT	10
COORDINATES	All
DAMPING	3C
DEFINE	2A, 2B, 3A, 3B, 3C, 4, 6, 7, 8, 9, 11
DIST FLUXES	10
DIST LOADS	1, 2A, 2B, 3B, 4, 6
ELEM SORT	2B
END OPTION	All
FILMS	8
FIXED DISP	1, 2A, 2B, 3A, 3B, 3C, 4, 5, 6, 7, 9, 10,11
FIXED TEMPERATURE	8, 10
GEOMETRY	1, 3A, 3B, 3C, 4, 5, 6, 7, 8, 9, 10
INITIAL DISP	3C
INITIAL STATE	9
INITIAL TEMP	8, 10
ISOTROPIC (*for stress analysis*)	1, 2A, 2B, 3A, 3B, 3C, 4, 5, 6, 7, 9
ISOTROPIC (*for heat transfer analysis*)	8
MOONEY	11

Table A-2 Model Definition

Keyword	**MARC PRIMER Example**
NODE SORT	2B
NO PRINT	2B, 3C, 10
OPTIMIZE	1, 8, 9
ORIENTATION	2A, 4
ORTHOTROPIC	4
POINT LOAD	5, 7
POST	All
PRINT ELEMENT	1, 2A, 3B, 4, 5, 6, 9, 11
PRINT NODE	2A, 3B, 4, 6, 11
PROPERTY	10
RESTART	11
SUMMARY	2B
TEMPERATURE EFFECTS (*for couples thermal-stress analysis*)	10
TEMPERATURE EFFECTS (*for heat transfer analysis*)	8
TRANSFORMATION	2A
WORK HARD	10

Table A-3 History Definition

Keyword	MARC PRIMER Example
AUTO INCREMENT	7
AUTO LOAD	5, 6, 11
AUTO THERM	9
CHANGE STATE	9
COMMENT	All
CONTINUE	3A, 3B, 3C, 5, 6, 7, 8, 10, 11
DISP CHANGE	10
DIST LOADS	3B, 6
DYNAMIC CHANGE	3B, 3C
MODAL SHAPE	3A, 3C
POINT LOAD	7
PRINT ELEM	7
PRINT NODE	7
PROPORTIONAL INCREMENT	5
RECOVER	3A, 3C
TIME STEP	11
TRANSIENT	8, 10

INDEX

2-D contact analysis 5-1

3-D beam element 2-62, 2-76, 2-115

3-D elastic beam element 2-62

3-D solid element 1-9, 1-12

A

ALIAS 4-25, 4-27

ALL 2-10

ALL POINTS 1-19

alphabetic keyword 1-18

analysis messages 1-22

AND 1-19

ASME code applications 2-12

assumed strain elements 1-8

AUTO INCREMENT 3-72, 3-74, 3-75, 3-76

AUTO LOAD 3-3, 3-7, 3-44, 5-51, 5-54, 5-78

AUTO THERM 4-25, 4-26, 4-29, 4-30

AUTO TIME 5-12, 5-54

axisymmetric elements 1-9, 1-10, 1-11, 1-12, 1-13, 1-15, 3-72
  nonaxisymmetric loads 1-11

axisymmetric solution 2-47

B

Background Papers (MARC Volume F) 1-1

bandwidth minimization 2-9, 2-119, 4-7, 4-28

Bathe, K.J. and E.L. Wilson 2-111

beam elements 1-9, 1-10, 1-12, 1-15, 1-20, 1-23, 3-3, 3-5

BEAM SECT 1-19, 2-113, 2-115, 2-116, 2-117, 2-119

beam stiffeners 2-116

binary post file 1-24

boundary conditions 1-3, 2-8, 2-33, 2-35, 2-48, 2-62, 2-63, 2-115, 2-118, 3-4, 3-6, 3-40, 3-42, 3-72, 3-74, 4-4, 4-26, 4-28, 5-4, 5-7, 5-50, 5-52
  DIST LOADS 1-20
  FIXED DISP 1-21
  POINT LOAD 1-20

boundary nonlinearity 3-1

Burnett, David S. 3-75

C

cantilevered beam 2-62

Cartesian 1-21

CAUCHY 2-10

Cauchy stress 5-51, 5-52, 5-55, 5-75

CENTROID 1-19, 1-23

change of state analysis 1-5

CHANGE STATE 1-20, 4-25, 4-26, 4-29, 4-30

closed section beam element 3-4

COMPOSITE 2-113, 2-117, 2-118, 2-119

composite layup 2-118

composite structure 2-113

composites 1-5

concrete 1-6

CONNECTIVITY 1-20, 2-8, 2-10, 2-34, 2-63, 2-80, 2-116, 2-117, 3-5, 3-41, 3-73, 4-5, 4-27, 5-4, 5-7, 5-51, 5-77

constant dilatation 1-11, 5-10

constraints 1-19

CONTACT 5-3, 5-5, 5-10, 5-12, 5-50, 5-51, 5-53, 5-54, 5-55, 5-77

contact analysis 5-1

contact analysis controls 5-10, 5-53

CONTACT TABLE 5-11

CONTINUE 1-17, 2-2, 2-64, 2-65, 2-83, 2-96, 2-97, 3-7, 3-44, 3-76, 4-8, 4-29, 5-12, 5-54

continuum elements 1-23

CONTROL 1-19, 1-21, 3-3, 3-6, 3-7, 3-8, 3-9, 3-74, 3-75, 3-77, 4-7, 4-8, 4-28, 5-7
- convergence tolerance 3-77
- coupled analysis 5-7
- thermal analysis 4-7

convergence ratio 3-8, 3-36

CONVERT 5-3, 5-5, 5-6, 5-9

COORDINATES 1-20, 2-8, 2-11, 2-34, 2-63, 2-116, 2-117, 3-5, 3-41, 3-73, 4-5, 4-27, 5-4, 5-7, 5-51, 5-77

Coulomb 5-10

COUPLE 5-3, 5-5, 5-6

coupled thermal-mechanical analysis 5-3, 5-5

CRACK 2-9

CREEP 1-20, 2-9

creep 1-6

cross-sectional area of the beam 2-63

Cuthill-McKee 1-21, 2-9, 2-11, 2-119, 4-7, 4-28, 4-29

cyclic plasticity 1-6

D

damped modal superposition of a beam 2-95

DAMPING 2-96

damping factor 2-111

data interpretation 3-76

DEFINE 1-19, 2-34, 2-35, 2-48, 2-63, 2-116, 2-117, 3-41, 3-73, 4-5, 4-27, 5-51
- element 2-34
- incs 2-34
- int 2-34
- layer 2-34
- node 2-34

deformable bodies 5-53

degrees of freedom (DOF) 1-8, 1-20, 1-21, 2-6, 2-11, 2-32, 2-34, 2-47, 2-62, 2-118, 3-4, 3-6, 3-8, 3-42, 3-72, 3-74, 4-1, 4-4, 5-4, 5-7, 5-12, 5-52

Demonstration Problems (MARC Volume E) 1-1

DISP CHANGE 1-21, 5-3, 5-5, 5-7, 5-12

DIST FLUXES 5-3, 5-5, 5-6, 5-9, 5-10

DIST LOADS 1-20, 2-6, 2-8, 2-11, 2-33, 2-35, 2-80, 2-83, 2-119, 3-42, 3-44
- incremental 2-83

distributed loads 1-10

distributed fluxes 5-10

DYNAMIC 2-62, 2-64, 2-80, 2-95, 2-96

dynamic analysis 1-4
- direct integration 2-79
- eigenvalue (modal) extraction 2-64
- modal superposition 2-95
- output 1-23

DYNAMIC CHANGE 2-83, 2-96, 2-97, 5-54

E

eigenvalue extraction 2-61, 2-62, 2-65, 2-95, 2-96

ELASTIC 1-19

elastic-perfectly plastic 3-4, 3-6

elastic-plastic 1-6, 3-3

elastomers 1-6, 5-49

electromagnetic analysis 1-5

ELEM SORT 2-31, 2-48

element output 1-23

element summary table 1-13

element types 1-9

element usage 1-10

ELEMENTS 1-19, 2-7, 2-33, 2-62, 2-116, 3-5, 3-41, 3-72, 4-5, 5-6, 5-51
- Element 1 2-28, 2-45, 3-72
- Element 2 2-48
- Element 3 1-13, 1-14, 4-4
- Element 6 2-35
- Element 7 1-13, 1-14
- Element 9 1-13, 1-15
- Element 10 1-13, 1-14, 2-47, 5-4, 5-6, 5-7
- Element 11 1-13, 1-14, 2-32, 2-33, 2-35, 4-4, 4-25, 4-26, 4-27, 5-50
- Element 19 4-4
- Element 22 1-13, 1-16, 2-136, 2-137
- Element 25 1-13, 1-15, 3-3, 3-4, 3-5
- Element 26 2-28
- Element 27 2-32, 2-35
- Element 28 2-48
- Element 39 1-13, 1-14, 4-3, 4-4, 4-5
- Element 40 1-13, 1-14, 5-7
- Element 43 1-13, 1-14
- Element 52 1-13, 1-15, 2-62, 2-76, 2-80
- Element 72 1-13, 1-16, 2-115, 2-116, 2-117
- Element 75 1-13, 1-16, 2-137, 3-40, 3-41
- Element 77 1-13, 2-115, 2-116, 2-117
- Element 80 1-13, 1-14, 5-49, 5-50, 5-51, 5-54
- Element 82 1-13, 1-14
- Element 84 1-13, 1-14
- Element 89 1-13, 1-15, 3-72
- Element 98 1-13, 1-15

END 1-17, 1-19, 2-2, 2-7, 2-33, 2-62, 2-116, 3-5, 3-41, 3-73, 4-5, 5-6, 5-51

END OPTION 1-17, 2-2, 2-10, 2-11, 2-64, 2-80, 2-119, 3-7, 3-43, 3-75, 4-7, 4-29, 5-11, 5-54

ENERGY 2-10

EXCEPT 1-19

F

failure to converge to tolerance 3-8

FILMS 4-6

FINITE 3-44, 5-3, 5-5, 5-6, 5-9

FIXED DISP 1-21, 2-9, 2-11, 2-36, 2-63, 2-118, 3-6, 3-42, 3-74, 4-28, 5-4, 5-7, 5-52

fixed field input format 1-18

FIXED TEMPERATURE 5-5, 5-8

fixed temperatures 4-6, 5-8

fluid/structure interaction analysis 1-5

FOLLOW FOR 3-70

formatted post file 1-24

FORTRAN 1-18

Fourier elements 1-11

fracture mechanic analysis 1-4

free field input format 1-18

full integration (quadrature) 1-7

full Newton-Raphson technique 3-8, 3-74

G

GAP DATA 1-20

gap/friction element 1-9

Gauss points 1-7, 1-13, 1-14, 1-16, 1-23, 2-28, 2-32, 2-47, 2-62, 2-115, 3-4

generalized plane strain element 1-2, 1-9

geometric nonlinearity 3-1

geometric properties 2-63, 2-115, 2-117, 3-5, 3-41, 3-74, 4-5, 4-27, 5-10

GEOMETRY 1-20, 2-8, 2-11, 2-63, 2-116, 2-117, 2-118, 3-5, 3-41, 3-44, 3-74, 4-5, 4-27, 5-10
  for element 3 2-8
  for element 39 4-5
  for element 52 2-63
  for element 7 3-41
  for element 77 2-117

Goodier, J.N. 2-5

Green-Lagrange strain 3-44, 5-51, 5-55

H

HEAT 4-5

heat conductions 1-12

heat transfer analysis 1-4, 4-1, 4-3
  nodal output 1-23

heat transfer elements 1-10

Herrmann elements 1-14

Herrmann formulation 5-50

hexahedra 1-13, 1-14

higher-order elements 1-10

HISTORY DEFINITION options 1-17, 2-64, 2-96, 4-7

Holman, J.P. 4-1

Huber-Von Mises yield criterion 3-4

Huebner, K.H. 4-1

hydrodynamic bearing analysis 1-4

hypoelastic 1-5

I

IBODY 1-20

incompressible elements 1-8, 1-11, 1-12
  constant dilatational strain 1-11

incremental values 4-8

INITIAL DISP 2-96

INITIAL STATE 1-20, 4-25, 4-26, 4-27, 4-29

INITIAL TEMPERATURE 5-8

initial temperatures 4-6, 4-27, 5-8

input 1-17, 2-7, 2-33, 2-48, 2-62, 2-80, 2-95, 2-116, 3-5, 3-41, 3-72, 4-4, 4-26, 5-6, 5-51

input echo and interpration 1-22

input format
  fixed field 1-18
  free field 1-18

integration points 1-7, 1-13, 1-16, 1-19, 1-23, 1-24
  full integration 1-7
  reduced integration 1-7

internal pressure 2-31, 2-47

interpolation (shape) function 1-8

INTERSECT 1-19

inverse power sweep 2-64, 2-65

isoparametric 1-7

ISOTROPIC 1-20, 2-8, 2-11, 2-35, 2-63, 2-117, 3-6, 3-42, 3-73, 4-6, 4-27, 5-8, 5-9
  heat transfer analysis 4-6

J

Johnson, W. 3-4

joule heating analysis 1-4

K

kinematic constraints 1-21

L

Lagrange 5-5, 5-6, 5-9

Lanczos method 2-95, 2-96, 2-110

LARGE DISP 3-39, 3-40, 3-41, 3-44, 3-71, 3-72, 3-76, 5-5, 5-6, 5-50, 5-51

large displacement and plasticity analysis 3-39

LAST 2-116, 2-119

libraries 1-3
  element library 1-5, 1-7
  function library 1-6
  material library 1-5
  procedure library 1-3

linear analysis 1-10, 1-13

linear axisymmetric element 2-47, 5-4

linear dynamic analysis using direct integration 2-79

linear elastic analysis 1-3, 1-5, 1-11

lists 1-18

load displacement 3-118

LOAD INCREMENTATION options 1-17, 3-7, 3-44, 3-75, 4-29, 5-12, 5-54

loads 2-6, 2-33, 2-62, 2-79, 2-115, 2-119, 3-4, 3-6, 3-40, 3-42, 3-72, 3-74, 4-26, 5-4, 5-50
  mechanical 1-20
  thermal loads 1-20

loads and constraints 1-7

lower- order elements 1-10

LUMP 4-5

M

MARC Element Library (MARC Volume B) 1-1

material nonlinearity 3-1

material properties 2-8, 2-35, 2-62, 2-63, 2-115, 2-117, 3-6, 3-42, 3-73, 4-27, 5-8, 5-52
  ISOTROPIC 2-6, 2-115
  linear elastic 2-6
  Mooney-Rivlin material 5-52
  ORTHOTROPIC 2-115
  PROPERTY 1-20, 5-8

materials 1-19

MEAN NORMAL INTENSITY 2-12

mechanical loads 1-20

Mellor, P. B. 3-4

membrane element 1-9

mesh 1-19, 1-20

MISES INTENSITY 2-12

modal analysis 2-61

modal analysis of a cantilevered beam 2-61

MODAL SHAPE 2-64, 2-96

modal stresses 2-64

modal superposition 2-95, 2-96, 2-111

MODEL DEFINITION options 1-17, 1-19, 1-20, 1-24, 2-7, 2-33, 2-48, 2-63, 2-80, 2-96, 2-116, 3-5, 3-41, 3-73, 4-5, 4-27, 5-6, 5-51

MOONEY 1-20, 5-49, 5-51, 5-52, 5-55
  Mooney-Rivlin material 5-1, 5-49, 5-52

N

Newmark-beta method 2-80

Newton-Raphson 3-8, 3-42, 3-74

NO PRINT 2-31, 2-48, 2-81, 5-5, 5-10

nodal output 1-23
  dynamic 1-23

nodal quantities 2-37

NODAL THICKNESS 2-118

NODE SORT 2-31, 2-48

nonlinear analysis 1-3, 1-10, 1-13, 1-17, 1-20, 1-21, 1-24, 3-1

nonlinear analysis controls 3-6, 3-74, 4-28, 5-7

non-positive definite 3-74

numerical data 1-18

numerical integration 1-7

O

OGDEN 1-20

OPTIMIZE 1-21, 1-22, 2-6, 2-9, 2-11, 2-119, 4-4, 4-7, 4-28, 5-11

ORIENTATION 2-31, 2-32, 2-36, 2-37, 2-113, 2-118, 2-119

orientation angle 2-118

orthotropc material properties 2-115

ORTHOTROPIC 1-20, 2-117, 2-118

output 1-22, 2-10, 2-37, 2-49, 2-65, 2-97, 2-119, 3-8, 3-44, 3-76, 4-8, 4-29, 5-12, 5-54
  plotted output 1-22
  post file for Mentat post-processing 1-22
  printed output 1-22
  restart file 1-22
  selective printed output 1-22

output controls 2-9, 2-36, 2-64, 2-119, 3-7, 3-43, 4-7, 4-28, 5-8, 5-52

output of analysis results 1-23

P

PARAMETER options 1-17, 1-19, 2-7, 2-33, 2-62, 2-80, 2-95, 2-116, 3-5, 3-41, 3-72, 4-4, 4-27, 5-6, 5-51

Peterson, R. E. 2-28

Piola-Kirchhoff stress 3-44, 5-55

pipe-bend element 1-9

plane strain elements 1-9, 1-12, 5-50

plane strain solution 2-31

plane stress elements 1-9, 1-12

PLASTIC 2-9

plastic strain 3-66

plasticity 3-66

plasticity, onset of 3-66

plate elements 1-9

plotted output 1-22

POINT LOAD 1-20, 3-3, 3-6, 3-74, 3-75

Poisson's ratio 2-6, 2-7, 2-8, 2-32, 2-62, 2-115, 2-117, 3-3, 3-6, 3-40, 3-42, 3-72, 3-73, 4-1, 4-26, 5-4, 5-9, 5-49

POST 1-24, 2-9, 2-11, 2-36, 2-49, 2-64, 2-96, 2-119, 3-7, 3-43, 3-75, 4-7, 4-28, 5-8, 5-52, 5-77, 5-78
  thermal stress analysis 4-7

post file 1-24
  binary 1-24
  formatted 1-24

post file for Mentat post-processing 1-22

postbuckling analysis 3-71

post-processing 2-9

pressure loading 2-8, 2-35

principal values
  intermediate 2-12
  maximum 2-12
  minimum 2-12

PRINT CHOICE 2-81

PRINT ELEM 1-24, 2-9, 2-11, 2-36, 2-80, 2-81, 2-119, 3-7, 3-43, 3-75, 4-28, 4-29, 5-52

PRINT NODE 1-24, 2-31, 2-37, 2-80, 2-81, 2-119, 3-43, 5-52

PRINT,8 5-3, 5-5, 5-6, 5-55, 5-74

printed output 1-22

Program Input (MARC Volume C) 1-1

properties 2-6, 2-32, 3-4, 3-40, 3-72, 4-4, 4-26, 5-4, 5-50

PROPERTY 1-20, 5-8, 5-9

PROPORTIONAL INCREMENT 3-3, 3-7

Q

quadrilateral axisymmetric element 2-48

quadrilateral plane-strain elements 2-32

R

radial displacement 2-45

radial stress 2-45

RATE EFFECTS 1-20

rebar elements 1-9

recommended elements 1-13

RECOVER 2-64, 2-65, 2-96, 2-97, 2-111

reduced integration 1-7

RESTART 1-24, 5-49, 5-51, 5-54, 5-77, 5-78

restart file 1-22

restart run 5-77

rigid bodies 5-53

Roark, R. J. 2-45, 2-76

rubber analysis 5-1

rubber materials 5-49, 5-50

S

SCALE 3-3, 3-5, 3-8

selective output 1-24

selective printed output 1-22

semi-infinite elements 1-9

sequence 1-19

set name 1-19

shape function 1-8

shear panel element 1-9

shell elements 1-9, 1-12, 1-23

SHELL SECT 1-19

side pressing of a solid rubber cylinder 5-49

singularity ratio 1-22, 2-11, 3-9

SIZING 1-19, 2-7, 2-11, 2-33, 2-62, 2-116, 3-5, 3-41, 3-72, 4-4, 5-6, 5-51

soils 1-6

special elements 1-9, 1-12

spherical cap 3-71

SPRINGS 1-21

standard exit in a normal MARC run 1-33

STATE 2-10

static 1-23

static analysis
  nodal output 1-23

sticking 5-11

stiffened composite roof under uniform pressure 2-113

STOP 1-19

STRAIN RATE 1-20

STRESS STRAIN 2-9

substructuring analysis 1-3

SUMMARY 2-31, 2-48

symmetry 2-5, 2-6, 2-9, 3-40, 3-41, 3-43, 3-72, 3-74
  of output 3-66
  thermal analysis 4-4

T

tangential stress 2-45

temperature dependence of properties 5-10

TEMPERATURE EFFECTS 1-20, 1-21, 4-6, 4-26, 5-10

THERMAL 1-19, 2-9

thermal analysis 4-1, 4-3
  boundary conditions 4-1

thermal loads 1-20

thermal properties 4-6

thermal stress analysis 4-25

thermo-mechanical analysis 1-5

thick cylinder under internal pressure 2-31, 2-47

thick shell elements 3-40

thin shell element 2-115

Thornton, E. A. 4-1

TIME STEP 5-51, 5-54, 5-78

Timoshenko, S.P. 2-5

TITLE 1-19, 2-7, 2-33, 2-116, 3-41, 5-6, 5-51, 5-77

TRANSFORMATION 1-20, 1-21, 2-33, 2-35, 2-36

TRANSIENT 4-7, 5-12

transient analysis 1-23
  nodal output 1-23

transient analysis of a beam 2-79

transient heat conduction analysis 4-3

TRANSIENT NON AUTO 5-3, 5-5

TRANSORMATION 2-58

transverse shear deformation 3-72

TRESCA INTENSITY 2-12

triangular axisymmetric ring element 2-48

truss element 1-9

TSHEAR 2-137

U

UPDATE 3-44, 5-3, 5-5, 5-6, 5-9

User Information Manual (MARC Volume A) 1-1

User Subroutines and Special Routines (MARC Volume D) 1-1

V

viscoelasticity 1-6

viscoplasticity 1-6

von Mises 3-4, 3-8, 3-36, 3-42, 3-66, 4-30, 4-50, 5-46

W

Wilkinson, J. H. 3-75

WORK HARD 1-20, 5-9

WORK HARD DATA 5-3, 5-4, 5-5

work hardening data 5-9

workspace 2-10

Y

yield stress 3-3, 3-6, 5-9, 5-10

Young's modulus 2-62

Young, W. C. 2-45, 2-76

Young’s modulus 2-6, 2-7, 2-8, 2-32, 2-62, 2-115, 2-117, 3-3, 3-6, 3-40, 3-42, 3-72, 3-73, 4-1, 4-26, 5-4, 5-9

Z

Zienkiewicz, O.C 3-119