

Mentat

User's Guide

Version 3.3







Copyright © 1999 MARC Analysis Research Corporation Printed in U. S. A. This notice shall be marked on any reproduction of this data, in whole or in part.

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

Phone: (650) 329-6800 FAX: (650) 323-5892

Document Title: **Mentat User's Guide, Version 3.3** Part Number: UG-3009-03.3 Revision Date: March 1999

Proprietary Notice

MARC Analysis Research Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice.

ALTHOUGH DUE CARE HAS BEEN TAKEN TO PRESENT ACCURATE INFORMATION, MARC ANALYSIS RESEARCH CORPORATION DISCLAIMS ALL WARRANTIES WITH RESPECT TO THE CONTENTS OF THIS DOCUMENT (INCLUDING, WITHOUT LIMITATION, WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE) EITHER EXPRESSED OR IMPLIED. MARC ANALYSIS RESEARCH CORPORATION SHALL NOT BE LIABLE FOR DAMAGES RESULTING FROM ANY ERROR CONTAINED HEREIN, INCLUDING, BUT NOT LIMITED TO, FOR ANY SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF, OR IN CONNECTION WITH, THE USE OF THIS DOCUMENT.

This software product and its documentation set are copyrighted and all rights are reserved by MARC Analysis Research Corporation. Usage of this product is only allowed under the terms set forth in the MARC Analysis Research Corporation License Agreement. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MARC Analysis Research Corporation is prohibited.

Restricted Rights Notice

This computer software is commercial computer software submitted with "restricted rights." Use, duplication, or disclosure by the government is subject to restrictions as set forth in subparagraph (c)(i)(ii) or the Rights in technical Data and Computer Software clause at DFARS 252.227-7013, NASA FAR Supp. Clause 1852.227-86, or FAR 52.227-19. Unpublished rights reserved under the Copyright Laws of the United States.

Trademarks

All products mentioned are the trademarks, service marks, or registered trademarks of their respective holders.



Contents

Preface

Organization of this Manual.	i
Contents of the Chapters	i
PART I	i
PART II SAMPLE SESSIONS	
Documentation Conventions	Х

1 Introduction

Dhei Look at the Finne Element Analysis Flocess \ldots \ldots \ldots \ldots \ldots \ldots $1-2$	Brief Look at the Finite Element	Analysis Process .			1-2
---	----------------------------------	--------------------	--	--	-----

2 Mechanics of Mentat

Chapter Overview 2-1
Mentat Window Layout 2-2
How Mentat Communicates with You.2-4
How You Communicate with Mentat 2-5
The Mouse
Using the Mouse to Select a Menu Item
Using the Mouse to Point
Keyboard Input
Numerical Data
Literal Data
Menu Structure
Menu System
Menu Components
The Submenu Button 2-13
The Cycle Button 2-13
The Toggle Button 2-14
The Tabular Button
The One-Only Button Group



List Specification	2-19
Identifiers	2-28

3 Background Information

Chapter Overview	·1
Mesh Generation	-2
Mesh Entities	.3
Geometric Entities	.5
The Direct Meshing Technique	.7
The Geometric Meshing Technique	.9
What Constitutes a Good Mesh?	2
Boundary Conditions, Initial Conditions, and Links	6
Material and Geometric Properties	9
Contact	:1
Loadcases and Jobs	2
Results Interpretation	:4
Scalar Plots	25
Vector Plots	:5
Tensor Plots 3-2	:5
Deformed Shape Plots	25
Path Plots	6
History Plots	6

4 Getting Started

Chapter Overview	4-1
Starting the Mentat Program	4-2
Before you start the Mentat program	4-2
Procedure Files	4-4
Stopping the Mentat Program	4-7
Normal Stop	4-7
Emergency Stop	4-7
Following a Sample Session.	4-8

5 A Simple Example

Chapter Overview	5-1
Background Information	5-2
Overview of Steps	5-3
Detailed Session Description	5-4
Procedure File	5-21

6 Solid Modeling and Automatic Meshing

Chapter Overview	6-1
Background Information	6-2
Overview of Steps	6-3
Detailed Session Description	6-4
Procedure File	6-22

7 Tube Flaring

Chapter Overview	7-1
Background Information	7-2
Description	7-2
Idealization	7-3
Requirements for a Successful Analysis	7-3
Full Disclosure.	7-4
Overview of Steps	7-4
Detailed Session Description	7-5
Conclusion	7-33
Procedure File	7-34

8 Container

Chapter Overview	8-1
Background Information	8-2
Description	8-2
Idealization	8-3

Requirements for a Successful Analysis	•			 	•		•	•	•		•	•	•	8-4
Full Disclosure.				 										8-5
Overview of Steps	•		•	 		 •						•		8-7
Detailed Session Description				 										8-8
Conclusion.				 										8-43
Procedure File				 										8-44

9 Manhole

Chapter Overview	9-1
Background Information	9-2
Description	9-2
Idealization	9-4
Requirements for a Successful Analysis	9-6
Full Disclosure.	9-6
Overview of Steps	9-7
Detailed Session Description	9-8
Conclusion	9-37
Procedure File	9-38

10 Tire

Chapter Overview
Background Information
Description
Idealization
Level of Analysis Detail
Analysis
Idealization
Requirements for a Successful Analysis
Full Disclosure. 10-5
Overview of Steps
Detailed Session Description



Conclusion	 •	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	10-32
Procedure File	 •		•			•	•	•		•	•	•	•	•	•	•	•	•	•	•	•	•	•	•		•	•	•	•	•	•		•	•	•	•		10-33

11 Transmission Tower

Chapter Overview
Background Information
Tower Description
Idealization
Requirements for a Successful Analysis
Full Disclosure. 11-3
Overview of Steps
Detailed Session Description
Conclusion
Procedure File

12 Importing a Model

Chapter Overview	12-1
Background Information	12-2
Description	12-2
Detailed Session Description	12-3
Procedure File	12-18

13 Punch

Chapter Overview	-1
Background Information	2
Description	2
Idealization	2
Requirements for a Successful Analysis 13-	3
Full Disclosure. 13-	3
Overview of Steps	6
Detailed Session Description	7
Procedure File	0



14 Bracket

Chapter Overview	14-1
Background Information	14-2
Description	14-2
Idealization	14-2
Requirements for a Successful Analysis	14-2
Full Disclosure.	14-3
Overview of Steps	14-5
Detailed Session Description of the Linear Static Case	14-6
Conclusion	4-30
Dynamic Modal Shape Analysis	4-31
Overview of Steps	4-31
Detailed Session Description of the Modal Shape Analysis	4-32
Dynamic Transient Analysis	4-37
Overview of Steps	4-37
Detailed Session Description of Dynamic Transient Analysis	4-38
Conclusion	4-48
Pressure Table	4-49
Procedure File	4-50

15 Cooling Fin

Chapter Overview	5-1
Background Information	5-2
Description	5-2
Idealization	5-2
Requirements for a Successful Analysis 1	5-2
Full Disclosure 1	5-4
Overview of Steps	5-5
Detailed Session Description	5-6
Procedure File	-17



The purpose of this manual is to introduce the first time user to the Mentat program. The User's Guide covers the basics of the program and helps the novice user in becoming comfortable with Mentat through a number of examples.

Organization of this Manual

This manual is divided into two parts:

Part I introduces the user to the basics of the program and provides information that helps the user interact with Mentat.

Part II consists of a series of *sample sessions* that provide the user with hands-on experience with the functionality of the Mentat program.

Contents of the Chapters

PART I

Chapter 1	Introduction , provides information on the basic steps of the finite element analysis cycle and on how Mentat is used as a tool to accomplish these steps.
Chapter 2	Mechanics of Mentat, describes the user interface aspects of the program
Chapter 3	Background Information , expands on the common features of Mentat and describes some of the underlying philosophies of the program.
Chapter 4	Getting Started , introduces you to Mentat with a simple example of how to create a finite element model.
Chapter 5	A Simple Example, introduces you to use Mentat and MARC to perform a complete linear elastic analysis of a rectangular strip with a hole subjected to tensile loading. Both the preprocessing, analysis, and postprocessing steps will be demonstrated.
Chapter 6	Solid Modeling and Automatic Meshing , introduces you to use Mentat to perform solid modeling and subsequent automatic meshing.



PART II SAMPLE SESSIONS

Chapter 7 **Tube Flaring**

A simple deformable to deformable body contact problem is the focus of this chapter. Emphasis is placed on specification of the contact bodies and the interpretation of the results by means of a history diagram.

Chapter 8 Container

An aluminum container under internal pressure is used to demonstrate the snap through behavior of its arched bottom. Animation of the deformation and the development of plastic strain during the snap through process is featured in the postprocessing section of this chapter.

Chapter 9 Manhole

A cylindrical tank, penetrated by another cylindrical tank of smaller radius that is positioned perpendicular to the first cylinder, is put under internal pressure. Contouring the von Mises stress on the structure is shown in the postprocessing section of this chapter.

Chapter 10 Tire

A simple two-dimensional model is used for an exploratory analysis of an automobile tire coming into contact with the rim of a wheel. Subsequently, a three-dimensional model is used to determine the foot print of the tire as it comes into contact with the road.

Chapter 11 Transmission Tower

A beam structure is modeled and subjected to various loads. A dynamic analysis is used to extract the modal shapes of the structure.

Chapter 12 Importing a Model

An IGES format model is read and meshed using the automatic mesh generator capability.

Chapter 13 Punch

A punching process is simulated. The two-dimensional model features contact and large plastic strain and residual stresses.

Chapter 14 Bracket

A supporting bracket is modeled. The bracket is subjected to several static loads. The natural frequencies are calculated and the bracket is subjected to an earthquake history.

Chapter 15 Cooling Fin

A cooling fin is analyzed to determine the effectiveness of the design. The heat loss in the structure with the fin is compared with the heat loss in the structure without the fin.

Preface

Documentation Conventions

Listed below are some font and syntax conventions that are used to make the information in this manual easier to understand:

- Names of buttons that appear on the Mentat screen are in UPPER CASE.
- Literal user input and program prompts are in courier font.
- Names of processors are indicated in **BOLD UPPER CASE**.
- A carriage return keystroke is indicated by <**CR**>.
- The left mouse button is indicated by <**ML**>.
- The middle mouse button is indicated by <**MM**>.
- The right mouse button is indicated by **<MR>**.
- The mouse cursor is indicated by $<\uparrow>$.
- A filename implies a concatenation of pathname and filename. The pathname may be omitted if the filename is in your current directory.





Welcome to Mentat - a graphical user interface program that allows you to execute a finite element analysis process from start to finish.



Figure 1-1 The Analysis Cycle

Brief Look at the Finite Element Analysis Process

In order to enhance your understanding of Mentat, we will review the finite element analysis cycle before introducing you to the mechanics of the program. The finite element analysis cycle involves five distinct steps as is shown in Figure 1-1. This process may be traversed more than once for a particular design; that is, if the results do no meet the design criteria, you can return to either the conceptualization (Step 1) or modeling (Step 2) phase to redefine or modify the process.

The five distinct steps of the finite element analysis cycle provide the foundation for this guide. In order to improve your productivity, this guide has been designed to focus your attention on two steps of the finite element analysis cycle: Step 2, the model generation phase, and Step 4, results interpretation phase.

Typical engineering problems are used in this guide as a vehicle to demonstrate the key features of Mentat. Steps 2 and 4 of the analysis cycle, modeling and results interpretation, were the pacing parameters for the selection of the example problems. The engineering problems were further selected to meet two criteria:

- **1.** to introduce you to the intricacies of generating a model in a variety of ways,
- 2. to demonstrate a diversity of analysis types so that you become familiar with as many different capabilities of Mentat as possible. The analysis will be performed with the general purpose finite element program MARC. Within the graphical user interface the complete input requirements can be specified.

The dimensionality of each object prescribes the technique used to generate a model and display the results. Accordingly, the finite element models have been grouped by dimensionality and in many cases the complexity of the model corresponds to dimensionality. From this you could conclude that once you know how to solve three-dimensional problems, you also know how to solve one-dimensional problems. However, unique features of one-dimensional objects require that we cover examples of that particular topology. For example, it is difficult to contour a quantity on line elements. With this in mind, the geometry and analysis types, such as heat transfer, statics, or dynamics, have been selected to minimize duplicity in this guide.

The material covered in this tutorial is very basic and should be easy to access and understand for the first time user. Once you have worked through the sample sessions in Part II you should feel comfortable enough to do a complete analysis simply by extrapolating from what you have learned from the example problems.



Mechanics of Mentat

Chapter Overview

Before you get started with Mentat you need to know how to communicate with the program. The goal of this chapter is to give you an overview of how Mentat works and to provide you with the basic information to interact comfortably with the program. Upon completion of this chapter you should have a clearer understanding of the following areas:

- The basic window layout
- How Mentat communicates with you
- How you communicate with Mentat
- The menu system

Mentat Window Layout

The starting point for all communication with Mentat is the window shown in Figure 2-1 that appears at the start of the program.



Figure 2-1 Basic Mentat Window

The Mentat window that appears on your screen is divided into three major areas:

GraphicsMenuDialogue

The **graphics** area is used to display the current state of the database. When you start Mentat, the graphics area is blank to indicate that the database is empty.

The **menu** area is reserved to show the selectable menu-items and is divided into two subareas, the static and dynamic menus. The contents of the dynamic menu area change as the menu-items are selected. In contrast, the static menu is always present and contains items that are applicable and selectable at all times.

The **dialogue** area is a scrollable area of about five visible lines where all program prompts, warnings, and responses appear, and where the user can input data or commands. Within the dialogue area is the status area which is reserved to communicate the state of the program to the user. Either working or ready appears in the status area to reflect the current state of the program.

How Mentat Communicates with You

Mentat communicates with you via prompts and messages and other visual queues.

Mentat's prompts urge you to take action through the input of data or commands. These prompts have 3 types of trailing punctuation marks to indicate the required type of input:

- : enter numeric data;
- > enter a character string, typically a command, file name or set name;
- ? enter a YES or NO answer.

If you misspell a keyword or enter an incorrect response, Mentat warns you through a message posted in the dialogue area. Mentat does not require that you complete every action you initiate. For example, if you are prompted for a filename, and you change your mind, entering a $\langle \mathbf{CR} \rangle$ instead of typing in the filename will tell Mentat to abort the action. If the program is waiting for a list of items to operate on, and instead you enter a command that also requires a list of items or any additional data, Mentat will ignore your original request and process the command. If the command you enter does *not* request additional data, you are returned to the original data request from before the interrupt.

The program assumes at all times that you want to repeat the previous operation on a new set of items and will prompt you for a new list to operate on. This process repeats itself until you indicate otherwise, typically by entering a new command or a $\langle CR \rangle$.

How You Communicate with Mentat

All interaction with Mentat is done through the mouse, keyboard, or a combination of both. This section first discusses the usage of the mouse, followed by a discussion on how to use the keyboard as a means to enter commands and data.

The Mouse

The mouse is used to select items from the menu area or to point at items in the graphics area. It is important to make a distinction between using the mouse in the menu area versus the graphics area because the three mouse buttons have very different functions in each area. Figure 2-2 is a graphical representation of the mouse, mouse buttons, and corresponding cursor.



Figure 2-2 The Mouse, Mouse Buttons, and Corresponding Cursor

As mentioned under Documentation Conventions in the Preface, we have developed name conventions for each mouse button. The left button is represented by $\langle ML \rangle$, the middle button by $\langle MM \rangle$, and the right button by $\langle MR \rangle$. The movements and location of the mouse are visually communicated to the user by the mouse cursor which is represented by a $\langle \uparrow \rangle$ in this guide.

Depress refers to the action of pressing a mouse button and keeping it depressed until we explicitly ask you to *release* it. *Click* refers to a quick single depress-release action.

Using the Mouse to Select a Menu Item

To select a menu item with the mouse, move the $<\uparrow>$ over the item that you want to select and click the $<\mathbf{ML}>$. To return to the previous menu, move the $<\uparrow>$ over the menu area, and click the $<\mathbf{MR}>$. Alternatively, you can click on the RETURN button in the menu area using $<\mathbf{ML}>$. Clicking on the MAIN button takes you to the main menu.

On-line Help

An important companion for novice users in Mentat is the **on-line help** feature. Each menu item has a help panel with a short description and explanation of the function of that menu item. To activate the help feature, position the $<\uparrow>$ over the menu item on which you require help, followed by a click of the <**MM**>. The help panel disappears the moment you select another menu item.

Using the Mouse to Point

The mouse is used in two ways to operate in the graphics area: to point to, or pick, existing items as well as to point to, or pick, the location of yet to be created items.

The need to identify existing items displayed in the graphics area occurs frequently during a user's interaction with the Mentat program. The mouse is used for this by moving the <1> over the item to be identified followed by a click of the <ML>. In the remainder of the manual we describe this action by *clicking on an item*. If you inadvertently click on an item, you can undo that action by clicking the <MM> anywhere in the graphics area.



Figure 2-3 Using the Mouse in the Menu Area

At times, you will need to identify more than a single item. A list of items must be terminated by a click of the $\langle MR \rangle$ with the $\langle \uparrow \rangle$ positioned anywhere in the graphics area. Alternatively, you can click on the END LIST button in the menu area using $\langle ML \rangle$.

2. In order to use the mouse for creating a new item you need to define the relation between the position on the graphics screen and its location in global coordinates. In Mentat, it is possible to define a grid that is positioned in space and where the grid consists of points that can be pointed to. If you click in the vicinity of a grid point, the coordinates of the item that you created will be snapped to that grid point. In addition, you can also pick an existing node, point, or surface-grid-point to specify a location.



Figure 2-4 Using the Mouse in the Graphics Area

Keyboard Input

Not all data can be entered through the mouse; **numerical** and **literal data** *must* be entered via the keyboard. The program mode prescribes the specific requirements for proper entry of each type of data. The program can be in **data mode** or in **command/literal data mode** and is described under the following two headings.

Numerical Data

You must use the keyboard for numerical data entry. The program interprets the data entry according to the context in which it is used. If the program expects a real number and you enter an integer, Mentat will automatically convert the number to its floating point value. Conversely, if a floating point format number is entered where an integer is expected, the program will convert the real number to an integer.

Scientific notation for real numbers is allowed in the following formats:

.12345e01 .12345e01 -0.12345e-01 The interpreter does not allow imbedded blanks in the format. Whenever the program encounters an illegal format, the message bad float! will appear in the dialogue area. The prompt for numerical data is a colon (:).

Literal Data

Literal data is used for file, set and macro names. A literal data string may *not* be abbreviated. Commands as introduced in the beginning on page 2-4 are considered **string data** (as opposed to **literal string data**) and *can* be abbreviated as long as the character string is unique within the Mentat command library. For example, *add_elements cannot be abbreviated to *add because of the other commands that start with the same characters such as *add_nodes and *add_curves. The program checks the input for validity against the internal library of valid responses. For example, if you enter an ambiguous or misspelled command, Mentat responds by listing all the valid entries that start with the same first letter of the command. The prompt for literal data is a greater-than symbol (>).

If the program is in data mode which is identified by the : prompt, you must enter a command preceded by an * (asterisk) to instruct the program that you are entering a command.

For example: Enter node (1): *add_nodes

If you enter a command without the asterisk when the program is in data mode, Mentat responds with an error message in the dialogue area.

The asterisk *can* be omitted when the program is in command or literal data mode which is indicated by the greater-than symbol (>).

For example: Command > add_nodes

Editing the Input Line

The experienced user can enter a sequence of commands or requests in a single 160- column input line. Note that anything typed beyond the input line limit is lost! Use $\langle CR \rangle$ to avoid this. You must use a blank space to separate entries when you are entering multiple responses on a single input line. All entries in the buffer are processed sequentially.

Mentat maintains a history of lines that are entered and offers limited recall and editing capabilities for the command line. The arrow keys \land and \lor on the keyboard can be used to scroll up and down in the dialogue area to make these lines visible. Use CTRL-p (that is, hold down the CTRL key and press the p key) to recall a previously entered input line. Repeat the CTRL-p sequence to recall as many lines as you need. Use CTRL-n to move to the next line in the history of command lines. (By the way, *p* and *n* stand for *previous* and *next* respectively in these control sequences.)

Edit functions for the current line are: backspace for character delete and CTRL-u for line delete. The left and right arrow keys are used to position the cursor at the desired location to overwrite or insert characters. The TAB key is used as a toggle to switch from insert to overwrite mode and vice versa. For example, if you type *view_viewpont 0.0 0.0 1.0, the program responds with the message unknown command in the dialogue area. To correct the entry, recall the line using CTRL-p, use the left arrow key to move the cursor to the letter n of view_viewpont, press the TAB key, type i, and press <**CR**> to enter the line. The command will now be *view_viewpoint.

Menu Structure

This section focuses on the menu system as a means to communicate with Mentat. The first sub-section discusses the structure of menus that constitute the program. The second sub-section analyzes the components of each menu.

Menu System

The kernel of the Mentat program consists of a set of processors in a parallel configuration that operate on the database. The database is the most compact, yet complete, description of the current state of the model you are analyzing. Typical examples of processors are **SUBDIVIDE** and **PATH PLOT**.

Every processor may depend on a number of parameters that influence the process. The combined number of processors and parameters in Mentat is too large to show in one menu. To help you in the scheduling of tasks, we have structured menus around the processors that lead you through the steps from top down. Figure 2-1 shows you the organization of the main menu that appears when you start Mentat and how it corresponds to the main tasks of the analysis cycle depicted in Figure 1-1 of the first chapter.

For your convenience, the menu items have been grouped in panels by the four main tasks: preprocessing, analysis, postprocessing, and configuration. The menu items and sub-tasks on each of these panels represent yet another group of corresponding tasks. It is important to realize that most of the menus for the global tasks do not contain processors; these menus are for navigation purposes only and are not part of the kernel of the program!

MAIN MENU														5
PREPROCESSING														
MESH GENERATION	1													
BOUNDARY CONDITIONS	1													
INITIAL CONDITIONS	1													
LINK3	-													
MATERIAL PROPERTIES	<i>r</i>													
GEOMETRIC PROPERTIES	-													
CONTACT	-													
DESIGN WARABLES	-													
DESIGN CONSTRAINTS	+													
UNAL YS13	_													
LOADCASES	1													
J083	-													
POSTPROCESSING	_													
RESULTS	1													
CONFIGURATION	_												1	
DEMOS	1												4	_
VISUALIZATION	· ·													
TRUE	UNDO	SAVE	DRAW	FILL	RESETVIEW	Dis.	T¥+	TZ+	RX4	RYs	RZ+	ZOOM	IN	U
	UTL S	FILES 7	PLOT	MEW.	F DVN, MODEL	TK-	TY-	12-	RX-	BY-	87-	BOX	OUT	104

Figure 2-5 Organization of Main Menu

A task and corresponding sub-task is selected by clicking on a menu item of that menu. After the (sub-)task is accomplished, it is necessary to traverse the menus in the opposite direction. There are two ways to do this:

- 1. Click on the RETURN or MAIN menu items in the static menu area. RETURN takes you to the previous menu and MAIN takes you to the main menu.
- 2. Move the <1> over the menu area and click the <**MR**>. The result of this sequence is equivalent to clicking on the **RETURN** menu item.

Menu Components

This section describes the anatomy or different components of the Mentat menu and the meaning of each component.

Understanding the definition of components will help you to use the menu system in a proficient manner. We have already mentioned that the menu items are grouped into panels by task and related sub-tasks. Each menu has a title that describes its task.

Positioned on the panel are flat and raised rectangles. The raised rectangles in the released state suggest a light shining directly from above. The task is printed on the raised rectangle and is selectable by clicking on it with the \langle ML \rangle . Flat rectangles are not selectable; they convey the setting of parameters. The program does not respond to clicking the \langle ML \rangle or \langle MM \rangle on the flat rectangles.

Mentat contains 5 types of raised rectangles. Throughout the remainder of this document we will use the term **button** for raised rectangle. Below is a list of the different types of buttons and their functions.

The Submenu Button

As mentioned before, this button represents a gateway to a submenu. It is recognized by a > symbol on the right hand side of the button.

The Cycle Button

A cycle button is used to set a parameter to a value when there is a choice of three or more alternatives. The parameter is set to the value that is currently displayed on the button. Clicking on this button will change the displayed value to the next consecutive value in the list of alternatives. If the list is exhausted, the process will start over again with the first alternative. This button is identified by a ∇ symbol. Note that the symbol is indicative of the unidirectional way the list of alternatives is traversed.

The Toggle Button

A special type of cycle button is the toggle button where the number of alternatives is limited to two. It is a switch that connotates a state of *on* or *off*; a button is *depressed* to flag on or active, and *released* (or *raised*) to flag that the listed parameter is off or inactive. This button is identified by a **o** symbol.

The Tabular Button

A tabular button represents a combination of a parameter button and a flat rectangle. They show one or more numerical or alpha-numerical values that are associated with the parameter represented by the button. Clicking on this button type usually implies that you have to enter data through the keyboard, which is then displayed in the rectangular fields after the keyboard input is completed.

Tabular buttons may contain a large number of numerical data fields. There are instances where the tabular buttons pop-up over the graphics area. If this is the case, you need to confirm that all entries have been completed by clicking on the OK button. Before returning to regular menu selection, you can clear all entries by clicking on the RESET button which usually appears in the lower left hand side of the panel. The pop-up table then disappears from the graphics area and the original graphics area is restored. Typical examples of these compounded tabular buttons can be found in the boundary conditions and material properties menus.

The One-Only Button Group

The alternative values of cycle buttons are also represented as individual toggles under a one-only button group. In a cycle, only one value can be selected, hence if a button in a one-only group of buttons is depressed, another is released. The one-only button sequence is identified by a \equiv symbol shown on each button of that sequence.

As a typical example of a menu, the Coordinate System panel of the Mesh Generation menu as shown in Figure 2-6 will be discussed. These buttons are also summarized in Figure 2-10.



Figure 2-6 Coordinate System Panel

The GRID button is a toggle; it can be switched *on* or *off*. The default position for this button is the raised or released state which means that the grid is off. Clicking it will turn the grid *on* and leave the button in a depressed state.



Figure 2-7 Released and Depressed States of a Toggle Button

The button next to it displays RECTANGULAR and has the ∇ symbol which implies a cycle. In contrast to the toggle, a cycle button has more than two values. In this example the button is an adjective to grid and specifies the type of grid to be used. Again, in contrast to the toggle, this button will not stay depressed. A click on this button changes the value of the parameter displayed on the button. The default value, RECTANGULAR, is changed into CYLINDRICAL. Clicking on it again changes the value to SPHERICAL, to be followed by RECTANGULAR again if this is repeated.

For three items in the cycle list this is still a viable way of setting the value of a parameter. If there are more than three, it becomes a tedious task to cycle through the alternatives. Therefore you will often find a submenu button combined with a cycle button. For example, the SET button is a gateway to a submenu as can be seen by the > symbol. By clicking on the SET button you are taken to a submenu where the cycle that describes the type of grid to be used is represented by a one-only group of buttons.

COORDINATE SYSTEM			
GRID	ON O		
SPACING	0.1		
	0.1	-	
SIZING	0.1	-	
	0.1	-	
ТҮРЕ		_	Examples of Simple
RECTANGULAR			labular Buttons
CYLINDRICAL			
SPHERICAL			
ORIGIN			
SET XYZ	0	-	
SET XY	0	-	
SET Z	0		
ALIGN	RESET		
TRANSLATE	ROTATE		

Figure 2-8 Example of Simple Tabular Buttons

Figure 2-8 gives you examples of tabular buttons that are found in the SET submenu.



The following table summarizes the different types of buttons found in the Mentat menu.

Figure 2-9 Summary of Mentat Menu Buttons

List Specification

Menu Structure on page 2-11 discussed the difference between menu buttons that are used to navigate through the menus and buttons that represent processors. Processors generally require two types of data:

- Parameters associated with the process
- A list of items to operate on.

If the list to operate on consists of only one item, you can use the mouse to point to that item on the graphics screen (see page 2-6 on pointing). If the list of items contains twenty items, pointing to each item individually becomes a cumbersome task and; if the list contains a hundred items, pointing becomes an impossible task. This section concentrates on the capabilities in Mentat to specify a list of items.

The Mentat program recognizes the following items:

- Points
- Curves
- Surfaces
- Solids
- Vertices of solids
- Edges of solids
- Faces of solids
- Nodes
- Elements
- Edges of elements
- Faces of elements.

A simple example of how to generate a list follows. You can then extrapolate from what you have learned in this section to do more intricate examples. Assume you want to subdivide an existing element that is already displayed in the graphics area of the Mentat window. The processor to use is **SUBDIVIDE**, and it operates on elements only.

MESH GENERATION	DYNAMIC MENU AREA
COORDINATE SYSTEM	GRAPHICS AREA
ALL	

Figure 2-10 Locating the SUBDIVIDE Processor in the Mesh Generation Menu

After you activate the **SUBDIVIDE** processor and click on the **ELEMENTS** button in the subdivide submenu, the following program prompt appears in the dialogue area:

```
Enter subdivide element list:
```

Chances are that you don't know the element number (nor should you care at this point). For this reason, answering this question by typing a number in the dialogue area may be possible but is not necessarily a viable option. Instead, move the mouse over the graphics area and use the \langle ML \rangle to click on the center of the element. You have now entered the first element in the list. The program keeps prompting you for more elements; if this is the only one you want to subdivide, you must let the program know that this is the end of the list. This can be done in one of three ways:

- **1.** Press the END LIST button in the menu area,
- 2. Type a '#' sign in the dialogue area, or
- **3.** Click **<MR>** with **<↑>** anywhere over the graphics area.

The most convenient way of ending the list is of course to click $\langle MR \rangle$ since the $\langle \uparrow \rangle$ is most likely already over the graphics area and saves you a keystroke from the keyboard.

Using a Box to Specify a List

Suppose the number of subdivisions was set to 20 by 20, creating 400 elements. Assume you want to enter the left 200 elements in a list by creating a rectangle to fence in those elements. Position the $<\uparrow>$ at one of the corners of the box. Depress the $<\mathbf{ML}>$ and move the $<\uparrow>$ to the opposite corner of the box you want to create. The rectangle that appears tells you exactly which elements are included in the box. Once you have reached the desired position, release $<\mathbf{ML}>$. Every element that is *completely* inside the box is included in the list specification.



Step 2: Hold down the <ML> and drag the cursor to the desired position



Step 3: Release the <ML> button



Figure 2-11 Selecting an Element Using the Box Pick Method
There are times when you may need the guidance of cross hairs to help you determine what is to be included in your selection. To activate the cross hairs, press the SHIFT key while moving the $<\uparrow>$ in the graphics area.

Note: You can relax the completely inside constraint mentioned previously by using the PARTIAL button on the picking panel under DEVICE.

Using a Polygon to Specify a List (CTRL Key + <ML>)

An alternative to using the box pick for list specification is to use a polygon around the elements that you want to include in the list. As with the box pick method, only those elements that are completely inside the polygon are entered into the list. To use the polygon pick, move the < \uparrow > to the first corner point of the polygon. Click <**ML**> while holding down the CTRL key on the keyboard. Move to the next vertex of the polygon and click <**ML**> again, continue to hold the CTRL key down. Repeat this process until a closed loop is formed. The last point needs to be in the vicinity of the starting point and must be clicked on to end the selection. A variation on this polygon pick is the **lasso pick**. This is done by holding down the CTRL key and the <**ML**> down simultaneously while slowly moving the mouse, until the elements to be selected are surrounded by the lasso. With either approach, a final click on <**ML**> is required at the polygon or lasso.

Note: The PARTIAL and COMPLETE buttons mentioned under the Box Pick Method also apply to the Polygon Pick Method.

Table 2-1 at the end of this chapter summarizes the mouse selection options in both the graphics and menu areas.





The LIST Buttons

For your ease of use, we have pre-programmed some of the more common list options and assigned them buttons which are located in the lower left hand side of the static menu.

The LIST buttons are:

all: EXISTING	
all: SELECTED	
all: UNSELECTED	
all: VISIBLE	
all: INVISIBLE	
all: OUTLINE	(all nodes and edges on the outline)
all: SURFACE	(all nodes, edges and faces on the outer surface)



Figure 2-13 Selecting an Element Using the Lasso Pick Method

All: EXISTING

Perhaps the most used list button is all EXISTING. It specifies all existing elements, nodes, curves, points, or surfaces (whichever is applicable), to be operated on by the processor that requested the list.

The contents of the selected/unselected, visible/invisible list are determined by the two operators: SELECT and VISIBLE. The meaning of each and their connection is explained in the next paragraphs.

All: SELECTED/UNSELECTED

The SELECT operator is a very powerful way to separate specific items from others. The methods by which items are selected range from a single item to a path of nodes, a box of items, or all items on a plane, and are connected by Boolean operators such as *and*, *except*, *invert*, and *intersect*. An example of this syntax is:

```
(use) single [items] and (a) box (of) [items] except single [items]
```

where the words *use*, *a*, *of*, and *item* are implied because they do not appear as buttons. A powerful feature of the **SELECT** processor is the ability to name a group of items, and refer to them by that name in list specifications. The **STORE** command facilitates this process.



Figure 2-14 Location of LIST Buttons

All: VISIBLE/INVISIBLE

Sometimes the model may be so complex that it takes an unacceptably long time to update the graphics screen every time the database changes. It is advantageous to focus on the items that you are working on. By activating and deactivating items from the display list, you can minimize the items that are displayed. Note that activating or deactivating does not imply that they are removed from the database. The **PLOT** processor facilitates this activation and deactivation process by using the VISIBLE and INVISIBLE commands.

Table 2-1 and Table 2-2 summarize the functions of the three-mouse buttons in the graphics and menu areas.

	<ml></ml>	<mm></mm>	<mr></mr>
	single pick or box pick	unpick	end of list
SHIFT	single pick or box pick with cross hairs	unpick	end of list
CTRL	polygon pick or lasso pick	unpick	end of list

Table 2-1	Mouse Button	Functions in	Graphics Area
-----------	--------------	--------------	----------------------

Table 2-2	Mouse Button	Functions	in	Menu A	rea
-----------	--------------	-----------	----	--------	-----

<ml></ml>	<mm></mm>	<mr></mr>
command selection	on-line help	return

Identifiers

In many applications, an identifier is associated with a group of data. These applications include material properties, link properties, geometric properties, boundary conditions, initial conditions, tables, transformations, beam sections, loadcases, and jobs. The identifier can be any name, if none is given a default name is given. These id names are then referenced in other commands. The use of ids is detailed below.

When using many of the menus, the following buttons will appear.

PREV			NEXT
NEW	NAME	REM	EDIT

NEW

creates a new entry in the list of applications and makes it the current application.

NAME

allows you to provide a name to the current application.



removes the current application id and the associated data.



displays a list of the id's and allows you to select a particular id. The selected id becomes the current one.



selects the previous id and makes it the current application.



selects the next id and makes it the current application.



Background Information

Chapter Overview

The purpose of this chapter is to give you an overview of the most common Mentat features. These features recur throughout the sample sessions in the last part of this guide. For example, all sample sessions contain a mesh generation step.

You may find it helpful to use the information in this chapter as a supplement to Part II as you work through the example problems. The order in which the common Mentat features are described in this chapter is based on the preprocessing, analysis, and postprocessing sequence of the finite element analysis process.

The key features discussed in this chapter are listed below.

- Mesh generation
 - mesh entities
 - geometric entities
 - direct meshing technique
 - geometric meshing technique
- Boundary conditions, initial conditions and links
- Material and geometric properties
- Contact
- Loadcases and jobs
- Results interpretation

Mesh Generation

The preprocessing task is considered a significant part of the finite element analysis process. In fact, at times it may be the most complex and time consuming part of the entire job. For this reason it is important that you use the conceptualization phase as indicated in Figure 1-1 to determine in advance what the objective of your analysis is and what answers you are seeking since both factors strongly influence the choice of your model.

Mentat distinguishes two techniques to build a mesh. The first is the **direct** or manual approach where you generate finite elements from bottom up. The second is the **geometric** approach where the model is first generated using *geometric entities* followed by a conversion step in which these entities are converted to finite elements. The two techniques are by no means mutually exclusive and often the best results are obtained by alternating between the two.

The following guidelines will simplify the task of generating a mesh using either one of the available methods.

Plan your model.

Plan your model carefully and take the time to formulate a strategy. This will save you time and resources.

Look for symmetry and duplication.

Many structures exhibit some form of symmetry. Look for the simplest component of your model. Also look for duplicates (or close duplicates) of another portion, and use the **SYMMETRY** or **DUPLICATE** processor.

Select a logical origin.

The lower left corner of your model or drawing is not necessarily the best location for the origin. Take some time to examine the model for an origin that makes the creation process easier. For example, if a model is symmetrical about a hole in a plate, consider placing the origin at the center of the hole. You may change the location of the origin within the same session while generating your model.

Choose a logical coordinate system.

The default coordinate system in Mentat is rectangular Cartesian. A cylindrical or spherical coordinate system may be more suitable for a particular model.

Create 1D and 2D before 3D.

For many structures often the best mesh generation strategy is to create first a 1D and/or 2D mesh and to drag it into a 3D mesh using the **EXPAND** processor.

Mesh Entities



Figure 3-1 Element Classes

Two types of mesh entities can be distinguished: nodes and elements.

Nodes

Nodes are characterized by three coordinates and symbolized by a \Box on the screen. When nodes are attached to a geometric entity, the symbol \circ is used instead.

Elements

Elements consist of element edges and element faces and are defined by a sequence of nodes. The number of edges, faces and nodes depend on the element class. Mentat employs a wide variety of element classes which are identified in Figure 3-1 and Figure 3-2. The **CHANGE CLASS** processor allows you to change the class of existing elements. The button CLEAR MESH in the mesh generation menu enables you to remove all mesh entities from the database. When elements are drawn in wireframe mode, the faces are indicated with a cross and the first edge is indicated with a half-arrowhead.



Figure 3-2 Element Classes

During the mesh generation phase it is not required to make a decision on the element type to be used. Only the element class is important at this stage. For instance: planar elements can be used to model a planar or an axisymmetric structure. In the phase of the analysis type definition it has to be decided if the element is axisymmetric, plane stress or plane strain.

Geometric Entities

The building blocks of the geometric mesh technique are points, curves, surfaces and solids.

Points

Points are characterized by three coordinates and symbolized by a '+' on the screen.

Curves

The following curve types can be used to define curves: line, polyline, tangent, arc, fillet, circle, cubic spline, interpolate, Bezier curve, NURB curve, and composite curve.

The line curve is a straight line segment between two points, the polyline is a concatenation of linear line segments through a series of points. The tangent is a straight line tangent to an existing curve. The arc curve is part of a circle and 5 methods are available to specify this circle segment. The fillet curve creates an arc between two curves. The circle curve is a complete circle and can be specified in two ways. The cubic spline and the interpolate curve pass through a series of points (a curve fitting technique). The Bezier curve and NURB curve can be used to define more general curves. The composite curve enables joining several previously defined curves.

Surfaces

Currently Mentat recognizes the following surface types which can be specified directly: quad, ruled surface, driven surface, cylinder, Bezier surface, NURB surface, sphere, swept surface, interpolate, coons, and skin surface.

The *quad* surface is the most simple surface definition as it is defined by 4 (non-collinear) points. The *cylinder* surface is a cone which is defined by the coordinates of two points on the axis and two radii. The *sphere* surface is defined by the center and the radius. The *ruled* surface is spanned by a family of straight lines between two curves. Both the *driven* and the *swept* surface are generated by dragging a curve along another curve. The *interpolate*,

Bezier and *NURB* surfaces are logical extensions of the interpolate, Bezier and NURB curves. The *coons* surface is created from a closed boundary consisting of 4 curves. The *skin* surface is created through a list of curves.

You can also generate axisymmetric surfaces by revolving a curve about the local y-axis using the **REVOLVE** processor.

In addition, the CAD interfaces allow Mentat to read in trimmed surfaces.

When you display a curve or surface, you often see a crude representation of that entity on your screen. We emphasize the word *representation* here. By default, the resolution of a curve is set to 10. The curve is represented by 10 straight line segments. For small curves, this may be sufficient to give the impression of a smooth curve. For larger curves, however, 10 subdivisions may not be sufficient. You can change the resolution by increasing the number of divisions in the DIVISIONS submenu of the continued part of the PLOT menu. Be aware that increased resolution requires more time for the program to draw the curve.

Solids

A solid is a volume which is bounded by a number of faces. Solid faces are bounded by edges and solid edges are bounded by solid vertices. Mentat offers five basic solid types: block, cylinder, sphere, torus, and prism.

The *block* entity is a rectangular block which is defined by the coordinates of a corner point and three dimensions. The *cylinder* entity is a solid cone which is defined by two points on the axis of revolution and two radii. The *sphere* is defined by the center point and the radius and the *torus* is defined by the coordinates of the center and the two radii. The *prism* is defined by two axis points, a radius, and the number or prism faces.

The basic solids can be manipulated through the **SOLIDS** processor. First a series of boolean operations such as UNITE, SUBTRACT, and INTERSECT can be used to modify the basic solid entities. In addition, the BLEND and CHAMFER operations exist to make smooth transitions between various faces.

Geometric entities can be converted to other geometric entities using the **CONVERT** or **SOLIDS** processor. This allows the following conversions:

curve	polyline curve
curve	interpolated curve
surface	polyquad surface
surface	interpolated surface
solid vertex	point
solid edge	curve
solid face	trimmed surface
trimmed surface	solid face

The button CLEAR GEOM in the mesh generation menu allows you to remove all geometric entities from the database.

The Direct Meshing Technique

Elements are used as the basic building blocks to generate a coarse mesh that can be refined later using the tools provided by Mentat specifically for this purpose. This approach is particularly suitable for a domain with a simple geometry. The direct meshing technique is not based on an algorithm but consists of the enumeration, by you, of the most coarse mesh that still represents the desired geometry. Use the ADD button of the element and node panels in the mesh generation menu to define the building blocks.

Once you have generated a coarse model, you can refine it (locally) to the desired level using the **SUBDIVIDE** processor. You can expand the model to a higher dimension using the **EXPAND** processors. The processors **DUPLICATE**, **SYMMETRY**, and **MOVE** allow scaling, translation, rotation and duplication of part of the model. The processors **RELAX** and **STRETCH** are available to relocate nodal points based on a given element connectivity. Removal of duplicate nodes or elements is achieved through the **SWEEP** processor and renumbering of the mesh can be performed with the **RENUMBER** option. CHECK allows specific checks on the correct definition of the mesh.



Figure 3-3 Example of a Coarse Mesh

The direct mesh generation process in Mentat is a three step procedure:

- **Step 1** Generate nearly correct coordinates and fully correct connectivity. Subdivide and refine the initially specified elements where necessary.
- Step 2 Modify the boundary nodes for exact boundary coordinates.
- **Step 3** Redistribute the internal coordinates to create reasonably shaped or relaxed elements.

The Geometric Meshing Technique

The basic building blocks for this technique are geometric entities rather than mesh entities. The geometric entities available in Mentat are points, curves, surfaces, and solids. They may be converted to mesh entities after you have completed the geometric model. This approach is more complex than the direct meshing technique as it involves the extra layer of geometric entities. However, the advantage of the geometric meshing technique is that increased complexity is offset by increased flexibility in generating geometries of complex shape. It is important to differentiate mesh entities from geometric entities; for example, a two-noded line element is not the same as a line curve, and a node is not the same as a point.

Convert

To change the geometric model to a finite element mesh, you may *convert* the geometric entities to finite elements. For instance, curves can be converted into line elements and surfaces into quadrilateral elements. The following conversions are possible:

point	node
curve	line elements
surface	quadrilateral elements

The **ATTACH** processor is a very powerful tool to put nodes on a curve or surface. Please note that after a CONVERT operation the resulting nodes have been attached to the geometric entity.

Automesh

Mentat contains as optional products automatic mesh generators which generate finite element meshes on solids, on trimmed surfaces and within curves i

Typically, 3 steps can be considered.

- **1.** Clean up and repair of the curves (coming from a CAD tool)
- 2. Set the curve divisions which basically controls the mesh density
- 3. Automatic generation of the mesh

Three classes can be distinguished for the automatic meshers:

- **1.** 2D PLANAR MESHING
- **2.** SURFACE MESHING
- 3. SOLID MESHING

Both the 2D Planar and the Surface meshing have several alternatives for creation of a mesh. All meshers with exception of the OVERLAY mesher use the seed points defined in the CURVE DIVISIONS menu.

The **OVERLAY** processor allows you to describe the geometry by its boundary instead of a surface. This can be applied either to a planar structure or a trimmed surface. You may use any curve type available in Mentat to specify the boundary. (This also implies that a combination of curve types is permitted.)

The TRI MESH! mesh generator creates triangular elements, on a trimmed surface or within curves in a plane. Either the Delaunay technique or the Advancing Front technique can be used.



The QUAD MESH! mesh generator creates quadrilateral elements on the faces of a solid, on a trimmed surface or within curves in a plane based on an Advancing Front technique.



Figure 3-4 Overlay Mesh



Quadrilateral Mesh

In addition, mixed triangular and quad elements can be generated using the Advancing Front technique.

The two solid meshers use the surface mesh created with the above mentioned meshers as input. The TETRA MESH! **mesh generator** creates tetrahedral elements in a solid volume or within a volume bounded by triangular elements. The HEXMESH! generator creates hexahedral solid elements in a volume spanned by the surface mesher.



What Constitutes a Good Mesh?

Unfortunately, this question can only be answered a posteriori. Only when the analysis is complete, and a convergence study conducted, is it possible to quantify the answer to this question. A priori qualifications, although often necessary, are generally not sufficient.

Elements have ideal shapes when there is little or no error in the numerical computation of individual stiffness matrices. It would be convenient if triangles were always equilateral, quadrilaterals always squares, and hexahedra always cubes. However, it is almost impossible to model complex systems with a mesh of ideally shaped elements. Therefore, it is advisable to match the mesh density to stress gradients and deformation patterns which imply that elements vary in size, have unequal side lengths and are warped or tapered.

With the above in mind, the remainder of this section will concentrate on a few guidelines you can use to determine the quality of a mesh. These guidelines are aspect ratio, distortions, and transitioning.

Aspect Ratio

The element aspect ratio is the quotient between the longest and the shortest element dimensions. This ratio is by definition greater than or equal to one. If the aspect ratio is 1, the element is considered to be ideal with respect to this measure. Acceptable ranges for the aspect ratio are element and problem dependent, but a rule of thumb is:

 $AR \le 3$ for linear elements

 $AR \le 10$ for quadratic elements.

Elements with higher-order displacement functions and higher-order numerical quadrature for a given displacement function are less sensitive to large aspect ratios than linear elements. Elements in regions of material nonlinearities are more sensitive to changes in the aspect ratio than those in linear regions. If a problem has a deflection or stress gradient dominant in a single direction, elements may have relatively large (10) aspect ratios, provided that the shortest element dimension is in the direction of the maximum gradient.

Distortions

Skewing of elements and their out-of-plane warping are important considerations. Skewness is defined as the variation of element vertex angles from 90 degrees for quadrilaterals and from 60 degrees for triangles. Warping occurs when all the nodes of three-dimensional plates or shells do not lie on the same plane, or when the nodes on a single face of a solid deviate from a single plane.

Transitioning

Two types of transitioning exist. The first type is the change in element density in the direction of the stress gradient. The greatest refinement is then in the region with the highest gradient. A good tool to apply to this type of transitioning is biased subdivision.



Figure 3-5 A Biased Mesh

The second type is transverse transitioning, which is used between element patterns with different densities across a transverse plane.



Figure 3-6 A Transition Mesh

If a model requires transverse transition regions, they should only be used in low-stress gradient regions, never near regions of maximum stress, deflection, or other regions of interest. The REFINE option in Mentat allows you to create a transition region.

Note: Within the framework of the CONTACT option, MARC and Mentat allow the automatic connection of two different parts which do not have common nodal points. Thus, various parts in the structure can be modelled with different mesh densities without the need for transition regions.

Boundary Conditions, Initial Conditions, and Links

The **BOUNDARY CONDITIONS** processor is used to define the boundary conditions applied to the model in order to perform the analysis.

Mentat distinguishes the following groups of boundary conditions:

- Mechanical
- Thermal
- Joule
- Acoustic
- Bearing
- Electrostatic
- Magnetostatic
- Electromagnetic

Depending on the analysis class, boundary conditions must be taken from one of these groups. An exception to this rule is the coupled analysis, for which both mechanical and thermal boundary conditions may be defined. The following boundary condition types can be found in the Mechanical submenu:

- Fixed displacement
- Fixed acceleration
- Point load
- Edge load
- Face load
- Global load
- Gravity load
- Centrifugal load
- Fluid drag
- Edge foundation
- Face foundation

- State variable
- Nodal temperature
- Release nodes

The specifications of the boundary conditions and associated parameters, along with the location, are grouped in one menu. The application of boundary conditions can best be thought of as an answer to the question: "Apply *what, where, and when*".

Every *what* requires a list specification for *where* and possibly *when*. It will be clear that fixed displacements are applied to nodes as are point loads. Edge loads are applied to edges of elements, while face loads to faces of elements, etc. For specification of the *where* part we refer back to the beginning of Chapter 2 on *List Specification*. If nodes have been attached to a curve or surface, it is also possible to apply the boundary conditions to the curve or surface. The associated nodes, element edges or element faces will inherit this boundary condition.

An important consideration of the *when* part is that one is defining potential boundary conditions, based upon a unique boundary conditions id. The boundary conditions are not applied in an analysis, unless they are selected in the **LOADCASE** processor, and the loadcase is selected in the **JOBS** processor or unless they are selected as INITIAL LOADS in the **JOBS** processor. Note that boundary conditions can also be specified as a function of time through the TABLE option.

Note: It is important to apply the correct number of boundary conditions. Too many will cause the system of equations to become over constrained; too few will cause a rigid body mode.

In addition to the boundary conditions, often a set of initial conditions can be present. Examples of these are the initial velocity in a dynamic analysis, and the initial temperature in a heat transfer analysis. The initial conditions can be defined in the **INITIAL CONDITIONS** processor.

Similar to boundary conditions, one defines here only potential initial conditions. They become active only if they are selected as INITIAL LOADS in the **JOBS** processor.

For specific analyses, it can be required to set up constraint equations between various components of the boundary conditions. Also springs can be present between two nodes. The **LINKS** processor allows the definition of constraint equations and links or dashpots. (Note that springs are not associated with element behavior.)

3 Background Information

Material and Geometric Properties

Virtually all of the required material data for an analysis with MARC may be entered through Mentat. The program recognizes the following material data:

- Isotropic
- Orthotropic
- Anisotropic
- Hypoelastic
- Mooney
- Ogden
- Foam
- Soil
- Powder
- Heat transfer
- Joule heating
- Acoustic
- Bearing
- Electrostatic
- Magnetostatic
- Electromagnetic

Note that for a coupled analysis the heat transfer material type must be combined with one of the mechanical material types.

In Chapter 2 on *List Specification* explains how to apply material data to elements. The **MATERIAL PROPERTIES** processor in the main menu facilitates the application of material constants and functions to elements.

Both in the Orthotropic and the Anisotropic material type, direction dependent material constants have to be defined. These material properties are usually defined in a local material axis system. The **ORIENTATION** processor allows specification of the material axis system. In addition, the **COMPOSITE** processor is available to define layered shell structures with different (direction dependent) properties and thicknesses.

Truss, beam, plane stress, plane strain, axisymmetric, membrane, plate, and shell elements are based on theories that are limiting cases of the general continuum theory. Shell theory, for instance, requires the shell element to have a thickness. This thickness (although strictly speaking a part of the geometry) does not enter into the mesh generation phase. This data is entered through the **GEOMETRIC PROPERTIES** processor. Other element types have similar properties such as area for truss elements and moments of inertia and local axis systems for beam elements.

For some element types, special options may be flagged in order to get more accurate results. For instance, the classical 4-node plane strain element is known to give a too stiff behavior if the element is subjected to bending. By selecting the assumed strain formulation, the element type is modified into a description with improved bending behavior. If for the same element the material behavior is nearly incompressible, also the constant dilatation formulation has to be selected. These special options are also defined in the **GEOMETRIC PROPERTIES** processor.

Furthermore, the data for the MARC gap/friction elements can be entered here.

Contact

A very powerful analysis capability in the MARC program is the automatic contact analysis. The boundary nodes and segments for a given set of elements will be determined and when the analysis requires it, automatically the boundary conditions to be applied will be adapted. Mentat supports this analysis capability completely. It allows definition of both deformable and rigid bodies, friction and thermal contact.

A deformable contact body is defined by a list of elements. A rigid contact body is defined by curves for 2D applications and surfaces for 3D applications.



The **CONTACT** processor in the MAIN menu allows the definition of the following tasks:

- CONTACT BODIES: defining the contact bodies, the properties of the contact body and allowing a graphical verification if the bodies are defined correctly.
- CONTACT TABLES: defining for which bodies contact will be checked, local friction coefficients, local separation forces, and heat transfer coefficients. Also here it can be specified that the so-called glued contact will occur, which implies automatic coupling of different parts.
- CONTACT AREAS: defining for which subset of the nodes in a contact body contact will be checked.

Note that similar to Boundary and Initial Conditions, both the CONTACT TABLES and CONTACT AREAS only define potential different applications of these options. They will only be applied if they are selected in the LOADCASE or the JOBS processor.

Loadcases and Jobs

Linear finite element analysis is characterized by a force-displacement relationship that only contains linear terms. The system of equations always produces a unique solution. In contrast, nonlinear analysis does not guarantee a unique solution. In fact, there may be multiple solutions or no solution at all. The task of providing analysis directives (i.e. controls by which the program will come to a solution) is far from simple. Solving nonlinear equations is an incremental and iterative process.

A linear static mechanical analysis with a known external load can be performed in one step. If nonlinearities are expected, it may be necessary to apply the load in increments and let each load increment iterate to the equilibrium state, within a specified tolerance, using a particular iteration scheme such as Newton-Rhapson. Also the complete load history might consider of a number of load vectors, each applied at a specific time in the load history. Each (set of) loads to be applied in a specific time period can be considered as a loadcase. A job is then the subsequent performance of various loadcases. In this way, the complete loading history can be defined. Note that a loadcase is not necessarily identical to a load step. A loadcase may consist of 10 load steps to reach the total load of the loadcase. In a loadcase, multiple boundary condition id's can be present.

A dynamic transient analysis of a beam structure with pre-load P1 and dynamic load P2 using the modal superposition technique consists of the following loadcases:

- Loadcase 1: Apply pre-load P1.
- Loadcase 2: Perform eigen frequency analysis based on pre-stressed structure
- Loadcase 3: Perform transient analysis using superposition of eigen modes. The load P2 is defined as a function of time through the TABLE option. Each loadcase can have different control values for the iterative processes used.

Depending on the analysis type (e.g. mechanical, heat transfer), the **LOADCASE** processor on the analysis panel of the main menu allows you to specify the following:

Load incrementation i.e selecting the boundary conditions, the number of steps, automatic versus fixed stepping, and the controls for this loadcase.

The **JOBS** processor is used to control the overall flow of the analysis process. This includes the analysis class, the selection of the loadcases, the analysis options, the results which are required, the initial loads, contact control, and other parameters. Also the element type specification, the check on integrity of the job, and the actual submitting of the job is done in this processor.

Typically, the finite element analysis produces an enormous volume of numerical data. Before you submit the job for analysis, use the **JOB RESULTS** processor to control which variables are to appear in the results file beyond the default parameters associated with the analysis type.

Before you submit the job, it is advisable to perform an integrity verification to check for inconsistencies in geometrical and material properties. The program will automatically verify the determinant of the Jacobian for all elements in the mesh. Errors found during this process are reported and corrective action should be taken before the job is submitted.

Once the data is verified by the program and passes the validity test, the job may be submitted. The SUBMIT button initiates the job in the background and leaves the terminal free to do other tasks. Use the UPDATE or MONITOR button to monitor the progress of the job during execution.

Results Interpretation

Once you have completed the analysis, you need to analyze the results and verify the criteria for acceptance. For each increment, the requested results are stored in a sequential file. Use the following 3 basic steps to gain access to the results.

- **Step 1** Open the results file.
- **Step 2** Select the desired information.
- **Step 3** Select an appropriate display technique and display the results.

The **RESULTS** processor on the postprocessing panel gives you access to the various plot options available in Mentat.

As we have already mentioned, a typical nonlinear finite element analysis consists of several steps called *increments*. The results for an increment can be accessed through the OPEN, NEXT, or SKIP sequence of commands. OPEN accesses the file and opens it for reading. The results file name is a concatenation of the job name and the suffix .*t19* or .*t16*. NEXT forwards the file pointer to the next increment. The results data for the increment that was read by the NEXT command is available for processing.

The solution of the finite element analysis involves a geometrical discretization of the object, and if applicable, also a temporal discretization. The geometrical discretization is obtained by creating the finite element mesh that consists primarily of nodes and elements. The results (depending on their nature) are supplied at either the nodes or the integration points of the elements. We make the distinction by referring to one as *data at nodes*, and the other as *data from elements at integration points*.

Data at nodes is a vector where the number of degrees of freedom of the quantity indicates the number of components in the vector. Data from elements at integration points is either scalar, vector, or tensor data.

The data from elements at integration points are not in a form that can be used directly in a graphics program. Data from elements at integration points is extrapolated to the nodes thus creating *data at nodes from elements*. The values at the nodes are calculated by a linear extrapolation of the average centroidal value and the integration point closest to the node.

A node may be shared by several elements. Each element contributes a potentially different value to that shared node. The values are summed and averaged by the number of contributing elements.

If a node is shared by elements of different materials, the averaging process may not be appropriate. To prevent the program from averaging values, use the ISOLATE option.

Scalar Plots

Scalar data may be represented graphically by means of contour bands, contour lines, symbols, numerics, iso-surfaces, cutting planes, beam contours, or beam values. A legend to the left of the drawing shows the correspondence between the colors used and the numeric interval they represent. *Contour plots* are lines or bands of equal value drawn over the elements. This display technique is applicable to two-dimensional elements, such as shells and plates, or to faces of three-dimensional elements, such as bricks. The three-dimensional counterpart to contour plot is the *iso-surfaces* plot, where the surfaces of constant value are displayed.

Vector Plots

Vector data may be represented graphically by arrows that are displayed at the nodes.

Tensor Plots

Tensor data may be represented graphically by arrows that are displayed at the centroid of the elements.

Deformed Shape Plots

The deformations found in a mechanical analysis can be shown in what is known to Mentat as a *deformed shape plot*. The mesh is deformed by an amount that is proportional to the actual displacement at the node.



Path Plots

Path plots are snapshots created by freezing time or an increment. The variables for the abscissa and ordinate are selected from the list of available variable names. For path plots, the position where the quantity is evaluated is the most likely candidate for the abscissa.

History Plots

As the name indicates, history plots capture phenomena over time or increments. The abscissa variable is very likely to be time or an increment number. As Mentat keeps only one increment of data in memory, it is necessary to collect data by scanning over the range of increments or time that is of interest before the history can be displayed.



Getting Started

Chapter Overview

This chapter describes the routine interactions with Mentat listed below.

- Starting Mentat,
- Using the PROCEDURE option,
- Stopping a Mentat session.

Chapter 4 concludes with a simple example to acquaint you with the program. It is best to focus on the overall session and not to dwell on the details. Once you have mastered the basic steps described in this chapter, you should feel comfortable enough with Mentat to venture on to the sample sessions in the remaining of this manual.

Getting Started

Starting the Mentat Program

Before you start the Mentat program. . .

- 1. You will need an account so you can use Mentat on your system.
- 2. If you don't know how to invoke Mentat, ask a current Mentat user or call MARC customer support. Although the starting command is system dependent, it most likely is mentat.

On machines supporting OpenGL graphics, one would type: mentatOGL. For the example session in Chapter 6 which uses the solid modeling capability type: mentatS or mentatSOGL.

3. The Mentat program is based on X-WindowsTM; you must start the program in a window environment.

Assuming you are already logged in on your computer, type mentat at the prompt of your operating system. Provided your version has been installed correctly, once Mentat is loaded into memory, the program should start by opening a window on your X-terminal in which the MARC logo appears. The MARC logo is then replaced by a window that displays the basic Mentat screen which consists of a main menu, a blank graphics area and a dialogue area. Figure 4-1 shows you the initial Mentat display.

If the Mentat script does not invoke the program, or does not invoke it correctly, ask your system administrator or call your nearest MARC office for support. Our telephone numbers are on the first page of this manual.
MAIN MENU														2014
PREPROCESSING														W 100
MESH GENERATION	1													
BOUNDARY CONDITIONS	F													
INITIAL CONDITIONS	1-1													
UNKS	-													
MATERIAL PROPERTIES	-													
GEOMETRIC PROPERTIES	-													
CONTACT	-													
DESIGN WARDARLES	-													
DESIGN CONSTRAINTS	+-													
UNALYSIS	_													
LOADCASES	1													
3089	-													
OSTPROCESSING	_													
RESULTS	1													
CONFIGURATION	-												- í	
DEMOL	1												×	
VISUALIZATION	· ·													
put	UNDO	SAVE	DRAW	FILL	RESET NEW	TK+	T¥+	TZ+	RX+	Rife	RZ+	2004	IN	USER
	UTLS	FILES '	PLOT 1	VIEW 1	F DWN, MODEL	TK-	TY-	12-	RX-	RY-	82-	808	OUT	DEFINE

Figure 4-1 Initial Mentat Display

Procedure Files

A procedure file is a record of all commands issued during a session and is useful for the tasks listed below.

- Protecting your work.
- Performing repetitive operations.
- Doing parametric design.
- Demonstrating your work.
- Reporting errors.

The PROCEDURES command has two modes of operation:

- 1. Record mode: creates a new procedure file or appends an existing procedure file
- 2. Playback mode: partial or complete execution of a procedure file

In both modes of operation, a choice can be made if the procedure file should reflect the changes in the menus. If the MENU RECORD button is activated all changes in the menu are recorded while creating procedure files. If the MENU EXECUTE button is activated and if a procedure file is used in which the menu changes have been recorded the menus modify while playing back the procedure file. Upon clicking on any of the CREATE, APPEND, LOAD, or EXECUTE buttons, a file browser appears. The FILTER block indicates the file extensions for the file type being used in the current application (here a .proc extension). Either click an existing file in the FILES block or type a new file name in the SELECTION blockls followed by an OK.

In playback mode, the LOAD button followed by the STEP mode allows stepwise playback of the procedure file. (Observe that the Mentat Procedure Control window can be moved to any position of the screen). START/CONT will start the execution of the procedure file until the STOP button is clicked.

All sessions listed in Part II of this manual are procedure files that are included on the Mentat installation tape in the *examples/userguide* directory. You can play these sessions back by executing the procedure file using the following button sequence:

MAIN	
UTILS	
PR	OCEDURES
	LOAD
	path/filename
	OK
	STEP or START/CONT

(main menu) (located at the bottom of the static menu)

(located on the PROCEDURE window)

Remember to use <**ML**> to click on a button. After you click on the LOAD button, enter the file name of the procedure file you want to execute. Once you have done this STEP and observe the changes as the information stored in the procedure file is executed. START/ CONT will automatically continue until the either the STOP button is activated or until in the procedure file the *stop_procedure command is present. Continue with the remaining information with either STEP or START/CONT.

An excellent way to learn more about the program is to make changes to the procedure file or to mimic it and to try to predict the results.

On the previous page, you were introduced to the concept of a **button sequence diagram**. A button sequence diagram is a way of prescribing a sequence of mouse clicks and corresponding data entry. An indent indicates a new menu. Aligned options indicate they are available from the same menu. A button sequence diagram starts at the main menu and works its way to the desired option. Buttons in the static menu do not require you to start with the **main** menu. The button sequence diagram is used frequently throughout the remainder of this chapter and in the sample sessions.

If there is any ambiguity as to which button you must click on, the button will be preceded by the specific panel or menu title. For example, if "elems ADD" appears in a button sequence, the idea is to click on the "ADD" button next to "elems" rather than nodes or curves on a particular panel. Another example is "all: EXIST." which indicates that you should click on the "EXIST." button of the "all:" panel. If you are not at the main level before you execute the button sequence diagram, you can click on the $\langle MR \rangle$ with $\langle \uparrow \rangle$ anywhere over the menu area until you reach the main menu. You can also click on the MAIN button in the lower left hand corner of the menu area to return immediately to the main menu.

The initial state of the program prescribes, wherever possible, a default for every setting. These settings are chosen because they are applicable to most cases. For example, the default number of divisions for SUBDIVIDE is set to 2, 2, 2. You can return to this default state at any time during the execution of the program by clicking on the RESET PROGRAM button. Use the following button sequence:

UTILS RESET PROGRAM (in the static menu)

When you create a procedure file, you are only recording commands that are issued from the time the procedure was started. The procedure file does not contain information on the state, or settings, of the program at the time it was started.

Stopping the Mentat Program

Always make sure to save your work before you stop the Mentat program. Use the SAVE button to write a copy of the database in Mentat format. The SAVE button is located in the static menu directly under the graphics area. This way you are assured all data is saved. Using other formats such as the MARC format does not guarantee all information is saved.

Normal Stop

Use the QUIT button located on the main menu to end a Mentat session. You have now the choice between SAVE & EXIT, EXIT, and CANCEL. SAVE & EXIT implies that the current changes to the database are stored and that the session will be terminated. EXIT will terminate the session directly and CANCEL will return to the main menu. Alternatively, you can type *quit in the dialogue area followed by a y for yes at the Exit program? prompt at any time during a session.

Emergency Stop

An emergency stop can be made at any time by using CTRL-C (that is, hold down the CTRL key and press C) from the parent window. Typing CTRL-C in the dialogue area does not stop the program. A host-induced stop usually does not offer you much of an option as you lose some or all of the data in memory.

Following a Sample Session

At this point, you may begin duplicating the first sample session on your computer. Do not try to understand everything at once: all concepts will be explained as you progress through the subsequent chapters. For now, concentrate only on becoming comfortable with the Mentat user interface.

The structure you are going to model has the dimensions shown in Figure 4-2.



Figure 4-2 Dimensions of Structure to be Modeled

The first step is to type mentat. The MARC logo appears on your screen and is immediately replaced with a window that displays the main menu.

Use the *<***ML***>* to click on the MESH GENERATION button of the PREPROCESSING panel. In Figure 4-3 the MESH GENERATION button is marked by a small arrow. Throughout the rest of this chapter, a small arrow is used to mark the button you need to click on to execute an operation.

MAIN MEMU														Maapo
PREPROCESSING														ACCORD.
MESH GENERATION	-													
BOUNDARY CONDITIONS	-													
INTRAL CONDITIONS	1													
LINKS	1													
MATERIAL PROPERTIES	-													
GEOMETRIC PROPERTIES	-													
CONTACT	-													
DESIGN VARIABLES	-													
DESIGN CONSTRAINTS	1													
ANALYSIS	_													
LONDCASES	1													
J085	-													
POSTPROCESSING	_													
RESULTS	-													
CONFIGURATION													1	
DEVICE	1												z	×
VISUALIZATION	r													
our	UNDO	SAVE	DRAW	PILL	RESETVEW	TK+	T¥+	TZ+	RX4	RYA	RZ+	ZOOM	IN	USER
	UTLS	FILES "	PLOT 1	MEM.	DYN. MODEL	TX-	TY-	1Z-	RX-	RY-	82-	BOX	OUT	DELINED

Figure 4-3 Accessing the Mesh Generation Menu

Figure 4-4 shows you that the dynamic portion of the main menu has been replaced by the mesh generation menu. Notice how the lower rows of buttons that make up the static menu do not change.

The next step is to establish an input grid to help you specify the nodes of your model. Click on the SET button of the COORDINATE SYSTEM panel. The dynamic portion of the menu is replaced by the set coordinate system menu where the grid settings are located.

Į													
	4	4	4	4	4	4	4	4	4	4	4	4	4
ĺ													
4													
1													
1													
1													
		1											
	<u> </u>												
		1	1	1									
					-								
1													1
1	1												
1													z
1	UNDO	UNDO SAVE	UNDO SAVE DRAW	UNDO SAVE DRAW FILL	UNDO SAVE DRAW FILL RESETVIEW	UNDO SAVE DRAW FILL RESETVIEW TX+	UNDO SAVE DRAW FEL RESETVIEW TX+	UNDO SAVE DRAW FEL RESETVEW TX+ TY+ TZ+	UNDO SAVE DRAW FILL RESETVIEW TX+ TX+ RX+	UNDO SAVE DRAW FEL RESETVIEW TX+ TX+ RX+ RY+	UNDO SAVE DRAW FILL RESETVIEW TX+ TX+ RX+ RY+ RZ+	UNDO SAVE DRAW FEL RESETVIEW TX+ TZ+ RX+ RY+ RZ+ ZOOM	UNDO SAVE DRAW FEL RESETVEW TX+ TX+ RX+ RY+ RZ+ Z00M IN
	UTLS'	UTLS' FILES'	UTLS FILES PLOT	UTLS' FILES' PLOT ' VIEW '	UTILS FILES PLOT WEW COMM. MODEL	UTILS FILES PLOT WEW " DVNL MODEL TX-	UTILS FILES PLOT WEW FOWN MODEL TX- TY-	UTILS FILES PLOT WEW FOWN MODEL TX- TY- TZ-	UTILS FILES FLOT WEW FOWN MODEL TX- TY- TZ- RX-	UTILS FILES PLOT WEW FORM MODEL TX- TY- TZ- RX- RY-	UTILS FILES FLOT WEW TOWN MODEL TX TY TZ RX RY RZ	UTILS FILES PLOT WEW FOWLMODEL TX- TY- TZ- RX- RY- RZ-	UTILS' FILES' PLOT' VIEW ' DVNL MODEL TX- TY- TZ- RX- RY- RZ- HXM OUT

Figure 4-4 Mesh Generation Menu

The object you want to model has maximum dimensions 8×6 units. Click on the U DOMAIN button and use the keyboard to enter $0 \quad 10$ to set the grid size in x-direction.



Figure 4-5 Coordinate System Menu

Figure 4-6 shows you the updated coordinate system menu with the numerals 0 and 10 appearing in the flat fields next to the U DOMAIN button.

The spacing between the grid points does not need to be finer than 1 unit since all the corner points are at integer distances from the origin.

Click the U SPACING button and type in 1. The program updates the menu accordingly as is shown in Figure 4-7. Repeat the steps for the V DOMAIN and V SPACING to set the values in y-direction.



Figure 4-6 Setting the Grid U Spacing

In Chapter 2, we mentioned that the "GRID" button was a toggle button that can be switched *on* or *off*. Click on the "GRID" button to turn the grid *on*.



Figure 4-7 Activating the Grid Display

COORDINATE SYSTEM 20MARC GRID ANES SET WES' UDOMAIN 0 10 . π U SPACING V DOMAIN 0 10 V SPACING ī w ō TYPE FIX * RECTANGULAR °FRU *FRV * CYLINDRICAL * SPHERICAL +FRW + DOTS CLINES MAX POINTS 10000 SET ORIGIN 2072 14 х 2 0 0 0 ALIGN RESET TRANSLATE ROTATE 8AVE LOAD
 TX+
 TX+
 RX+
 RY+
 RZ+
 ZOOM
 IM

 TX+
 TX+
 RX+
 RY+
 RZ+
 BOX
 OU
 RETURN MAIN UNDO SAVE DRAW FILL RESETVIEW USER DEFINED UTLS' FILES' PLOT' VIEW ' DVN. MODEL TX- TY- TZ- RX- RY- RZ-OUT

Figure 4-8 depicts the graphics area with the input grid displayed.

Figure 4-8 Input Grid Activated

Click on the FILL button to scale the picture to fit the screen. The FILL button is located in the static menu area directly under the graphics area.

COORDINATE SYS	TEM		y=10	-	-								Maason
GRID AN	S SET AVES	1											Conserve.
UDOMAIN	0												
	10	1											
U SPWCING	1	1											
U.	0			-			,						
V DOMAIN	0	1											
	10	1											
V SPACING	1			-									
v	0												
WIDOWNN		1											
	1												
WISPACING	0.1												
w	0												
TYPE	FIX												
* RECTANGULAR	°FRU	1			-								
° CYLINDRICAL	° FIXV	11											
SPHERICAL	*FKW]											
+ DOTS	CUNES	1											
MAXPOINTS	10000	1											
SET ORIGIN	202	il -											
X Y	2	1										- 2	r
0 0	0	1		\backslash								1	
ALIGN	RESET	1										· 4	x
TRANSLATE	ROTATE	1											
SAVE '	LOAD	1											.u-10
RETURN	MAIN	UNDO	SAVE	DRAW	FILL	RESET NEW	TK+	T¥+	TZ+ RX+	RY+ B	Z+ Z00M	IN .	USER
		UTLS	FILES '	PLOT 1	VIEW (C DWN, MODE	L TX-	TY-	TZ- RX-	RY- R	5- B00	OUT	DEFINED

Figure 4-9 Scaling the Picture

Return to the mesh generation menu by clicking on the RETURN button located in the bottom left corner of the menu area or by clicking $\langle MR \rangle$ with the $\langle \uparrow \rangle$ over the menu area.

COONDINATE \$15	STEM		y=10	-	-				-						MAANDO
• GRID /* AV	ES SET MES'	1													Comparison C
UDOMAIN	0	-													
	10														
U SPWCING	1														
U.	Ū.			-											
VIDOMAIN	0	1													
	10														
V SPACING	1	1		-											
v	0	1													
WIDOWNN	-1	r .													
	1	1													
W SPACEG	0.1	1													
w	0														
TYPE	FBC														
* RECTANGULAR	FRU	1		-					-						
[^] CYLINDRICAL	° FRV	11													
* SPHERICAL	+FKW	1													
+ DOTS	CUNES														
MAKPOINTS	10000	1													
SET ORIGIN	20/2	1													
X Y	2	11												2	1
0 0	0	1												- î	
ALIGN	RESET	1												· 2	
TRANSLATE	ROTATE	1													
SAVE	LOND	F 1							-						.u-30
RETURN	MAIN	UNDO	SAVE	DRAW	FILL	RESET VIE	w	TK+	TY+	1Z+ /	534 RY-	RZ+	2004	IN	USER
		UTLS'	FILES '	PLOT 1	VIEW 1	DYN. MC	IBORL	TX-	TY-	12- 1	K- RY-	RZ-	80%	OUT	DEFINED

Figure 4-10 Returning to the Mesh Generation Menu

Below follows a button sequence diagram of the steps required to set the coordinate system that we discussed in the previous pages. A comparison of this button sequence to a detailed step description shows you that the button sequence step format is condensed and easy to follow.

```
MAIN
MESH GENERATION
SET
U DOMAIN
0 10
U SPACING
1
```

```
V DOMAIN
0 10
V SPACING
1
grid ON
FILL
RETURN
```

(on)

To help you keep track of the elements and nodes that you are going to create, you must label them. Click on the PLOT button located in the static menu area to access the plot settings menu.

MESH CE	NERAI	ON				y=10	-												10144 DO
NODES	ADD	REM	EDIT	SHOW	1														Winner()
ELEMS	ADD	REM	EDIT	SHOW															
PTS	ADD	REM	EDIT	SHOW	1														
CRVS	ADD	REM	EDIT	SHOW	1														
SRES	ADD	REM	EDIT	SHOW	1														
SOUDS	ADD	REM	1	SHOW															
BETWEEP	NODE	86	WEEN P	TIMO															
ELEMENT	CLASS	° 00	(4) GM	T															
CURVE T	MPE	" UN	e	Ť															
SURFAC	E TYPE	* OU	MD	7	1														
SOUD TY	/PE	· 81.	оск.	+	l '														
COORDE	NATE 51	STEM			1														
SET '	RECTA	NGUL	NR " *0	GIRG	- I				,								,		
CLEAR M	6.51	(a.	EAR GEO	M	1														
ATTACH		* AU	TOMES	۲ I	I .														
CHANGE	CLASS	CH	ECK	۲															
CONVER	T	1 DU	PLICATE																
EXPAND		* INT	ERSECT														1		
MOVE		* RE	LAK	1	1														
RENUM	ER	' RE	VOLVE																
80UD8		1 817	RETCH																
SUBDIVE	DE	544	eer															- 7	
SYMMET	HY	+							/										
ALL:	SELEC	I. VB	1015	KITLINE														×	X
EX45/1	UNSE	0.00	15. 1	URFAC															
SELECT		" EN	o List (i	1			-	1-				-					•		.0-30
RETURN		MA	an	1	UNDO	SAVE	DRAW	FILL	RESETVE	w	TK+	T¥+	TZ+	POS+	RYs.	RZ+	ZOOM	IN	USER
					UTLS?	FILES '	PLOT	VIEW 1	F DVN, MC	DEL	TK-	TY-	TZ-	RX-	RY-	82-	BOX	OUT	DEFINED

Figure 4-11 Accessing the Plot Menu

The DRAW panel determines whether or not an entity will be drawn. With the LABEL panel it can be indicated if the entity will be labeled. Click on the NODES button of the LABEL panel. The NODES button is a toggle button; as long as it is depressed, every node you create will be labeled by its respective node number.

PLOT SETTINGS			v=10											
DR/W	LABEL	1												COMMIC:
* NODES	" HODES													
* ELEMENTS	" ELEMENTS	<u>ا</u>												
EDGE8	C _ EDGES													
_FACES	_FACES													
* POINTS	POINTS	I '												
CURVES	CURVES													
 SURFACES 	" SURFACES	Ι.												
SOLDS	C SOLIDS													
·_VERDCES	-VERDCES													
-EDGES	-EDGES	<u>ا</u>												
-FACES	C _FACES													
BOUND CONDS	F BOUND CONDS													
EINIT COMDS	E NIT CONDS	I '												
*LINKS	^{//} LINKS													
LOADCASES	LOADCASES	<u>ا</u>			-								-	
ELEMENT SIZE (%)		1												
^50 ^60 ^70	00 090 + 100													
BLEMENTS	SOLIDS	- I												
+WIREFILME	+WREFILAME													
SOLID	^ SOLID													
MORE		1											2	1
RESET DRAW	REDRAW REGEN	i .											. Î	
ALL: SELECT.	ARIETE OOUTINE	1											4	X
EXIST. UNSEL	INVES. SURFAC	1												
SELECT '	END LIST (#)	· 1			-									Ju-10
RETURN	MAIN	UNDO	SAVE	DRAW	FILL	RESET NEW	TX	+ TY+	TZ+ R	X+ RY+	RZ+	20044	IN	UBER
		UTLS'	FILES '	PLOT 1	VIEW 1	F DVN, MOD	BL TX	- TY-	TZ- R	N- RY-	RZ-	806	OUT	DEFINED

Figure 4-12 Activating the Node Labeling

Similarly, click on the ELEMENTS button of the LABEL panel. The ELEMENTS button is also a toggle and stays depressed indicating that every element you create will be labeled by the corresponding element number.

PLOT SETTINGS		1	v=10	-	-										
DRAW	LABEL.	1													COMMIC:
* NODES	*NODES														
* ELEMENTS	" ELEMENTS	<u>ا</u>													
·_EDGES	CEDGES														
·_FACES	C_FACES														
* POINTS	C POINTS	I '													
* CURVES	CURVES														
• SURFACES	" SURFACES	Ι.													
* SOLIDS	7 SOLDS														
·_VERDCES	-VERICES														
-EDGES	"_EDGES	<u>ا</u>													
-FACES	C.FACES														
BOUND CONDS	F BOUND CONDS														
F INIT COMDS	E NIT COMDS	· ·													
*LINKS	⁷ LINKS														
LOADCASES	LOADCASES				-									-	
ELEMENT SIZE (%)		1													
^50 ^60 ^70	100 100 + 100														
BLEMENTS	SOUDS														
+ WIREFILAME	+WIREFILME														
SOLID	SOLID	. ·	,												
MORE	1	1												2	,
RESET DRAW	REDRAW REGEN	i .												. Î	
ALL: SELECT	VERELE OUTLINE	1												4	X
EXIST UNSEL	HAVES. SURFAC														
SELECT	END LIST (#)			-					-				•		,0-10
RETURN	MAIN	UNDO	SAVE	DRAW	FILL	RESETVE	w	TK+	TY+	TZ+	ROGA PTYL	RZ+	2004	IN	USER
		UTLS'	FILES '	PLOT 1	VIEW	C DVN, MC	DEL	TK-	TY-	1Z-	RX- RY-	RZ-	808	OUT	DEFINED

Figure 4-13 Activating the Element Labeling

Return to the mesh generation menu by clicking on the RETURN button located in the bottom left corner of the menu area or by clicking $\langle MR \rangle$ with the $\langle \uparrow \rangle$ over the menu area.

PLOT SETTINGS			v=10	-	-				-							2014 A 1017
DRAW	LABEL	1														COMMIC:
* NODES	*NODES	I														
ELEMENTS	* ELEMENTS															
EDGE8	C.EDGES	I														
 _FACES 	FFACES	I														
* POINTS	POINTS	I 1		-												
CURVES	CURVES	I														
• SURFACES	" SURFACES															
* SOLIDS	7 SOLDS															
·_VERDCES	-VERICES	I														
·_EDGES	-EDGES															
FACES	- FACES	I														
· BOUND CONDS	F BOUND CONDS	I														
EINIT COMDS	E NIT COMDS	L 1														
*LINKS	LINKS	I														
LOADCASES	LOADCASES															
ELEMENT SIZE (%)		1														
^50 00 ^70	001 100 000															
BLEMENTS	SOUDS	L .												1		
*WREFRAME	+WIREFILAME															
SOLID	SOLID															
MORE		i													- 2	,
RESET DRAW	REDRAW REGEN	1													. î	
ALL: SELECT	VERELE OUTLINE														4	X
EXIST. UNSEL	INVIS. SURFAC															
SELECT	END LIST (#)	1		-					-							_u=10
RETURN	MAIN	UNDO	SAVE	DRAW	PILL	RESET VIEW	V	TK+	TY+	TZ+	RX4	RY-	RZ+	ZOOM	IN	USER
7		UTLS'	FILES '	PLOT 1	MEM.	DVN. MOE	HEL.	TX-	TY-	1Z-	RX-	RY-	RZ-	BOW	OUT	DELINED

Figure 4-14 Returning to the Mesh Generation Menu

To enter an element, click on the ADD button of the ELEMS panel. The default element class is QUAD(4), a four-noded quadrilateral, which is the element type you are going to use for your model.

MESH GE	NERAI	ON				v-10		-				-							Maaape
NODES	ADD	REM	EDIT	SHOW	1														warmon o
ELEMS	ADD	REM	EDIT	SHOW															
PTS	1400	REM	EDIT	SHOW															
CHVS	ADD	REM	EDIT	SHOW															
SPES	ADD	REM	EDIT	SHOW															
SOUDS	ADD	REM		SHOW															
BETWEEP	4 NODE	88	WEEN	TRIOP															
BLEMENT	CLASS	1 00	4D (4)	Ť															
CURVE T	YPE	" UN	t	T															
SURFACE	ETYPE	1 00	WD	*															
SOUDTY	PE	· 81.0	оск	7															
COORDE	NATE 51	STEM																	
SET *	RECTA	NGUL/	va ' •	CBR													,		
CLEAR M	131	a	LAR GEO	M															
ATTACH		* AU	TOMES	۲ I			-												
CHANGE	CLASS	CH	ECK	-															
CONVER	r	. 00	PLICATE																
EXPAND		° INT	ERSECT																
MOVE		" RE	LAK	-															
RENUMB	ER	' RE	VOLVE																
\$0UD8		511	LETCH															÷.,	,
SUBDIVE	Æ	54	6614	^														- 6	
SYMMET	RΥ	<u></u>																	
ALL:	28150	1. 115	181.5 4	FUILINE														*	X
1,931	CRIDEI		10. I	100 40															11-30
SELECT		254	o List 0	1		•	•		•	•			•		·		•		70-00
RETURN		MA	a 14	1	UNDO	SAVE	DRYM	PILL	RESETVIE	w	TK+	T¥+	1Z+	RX+	RYA	RZ+	ZOOM BOX	IN	DEFINED
					OILS.	FILES '	PLOT	NEM.	· DVN, MC	OFF.	18-	14-	12-	RX-	RY-	R2-		001	

Figure 4-15 Adding an Element

The program prompts you to enter four nodes. Look for the prompt in the dialogue area.

Pick the grid point at the origin of the u-v system shown on the screen for the first node of a quadrilateral element. Click the <ML> with the <?> close to that grid point. The program confirms the location of the first node with a small square at the grid point and the node number 1 in the graphics area. The entry is confirmed in the dialogue area with node(0,0,0) at the Enter element node (1) : prompt.

MESH CE	NERAI	ON				y=10						-							Million .
NODES	ADD	REM	EDIT	SHOW															Annual C
ELEMS	ADD	REM	EDIT	SHOW															
PTS	ADD	REM	EDIT	SHOW															
CHVS	ADD	REM	EDIT	SHOW															
SRES	ADD	REM	EDIT	SHOW															
SOUDS	ADD	REM		SHOW		-													-
BETWEET	NODE	84	TWEEN P	OINT															
BLEMEN	CLASS	0	(4) GAU	Ť															
CURVE 1	YPE	* U	ME	T															
SURFAC	E TYPE	10	UND	+															
SOUDTY	/PE	· 81	OCK.	7															
COORDE	NATE 51	STEM																	
SET '	RECTA	NGUL	AR []*0	CERC															
CLEAR M	6.51	a	EAR GEO	M	1														
ATTACH		* AI	TOMES	1 1															
CHANGE	CLASS	10	4ECK																
CONVER	T	1 DI	JPLICATE																
EXPAND		* IN	TERSECT																
MOVE		* R	XALE	-															
RENUM	ER	" (R	EVOLVE	-															
SOUDS		1 81	RETCH															÷.,	
SUBDIVE	DE	59	WEEP*															- i	
SYMMET	RY	-																	
ALL:	58150	1. ¥8	5815 0	RUTUINE														¥	X
5351	UNISE	111	VT5. 5	URLE AC															
SELECT		10	ID LIST (R	1		*1	-	•	•	•		-	· ·	_	•		•		-u-10
RETURN		M	AIN	1	0.0	SAVE	DRAW	PILL	RESETVIE	W	TK+	T¥+	TZ+	FOG+	RYA	RZ+	ZOOM	IN	DEENED
				/	UILS.	FILES '	PLOT '	NEM.	DVN, MO	OEL	TK-	TY-	1Z-	RX-	RY-	R2-	Jun	OUT	ounes

Figure 4-16 First Node of Element 1

To create Node 2, repeat the steps for Node 1 six units, or grid points, to the right of the first node.

MESH GENERATION						y=10	-											Million 1
NODES	ADD	REM	EDIT	SHOW	1													Werner C
BLEMS	ADD	REM	EDIT	SHOW														
PTS	ADD	REM	EDIT	SHOW														
CHVS	ADD	REM	EDIT	SHOW														
SRES	ADD	REM	EDIT	SHOW														
SOUDS	ADD	REM		SHOW														
BETWEEN NODE BETWEEN POINT																		
ELEMENT CLASS QUAD (4)				-														
CURVE TYPE UNE																		
SURFAC	E TYPE	1 0	UND	1														
SOUD TYPE SLOCK		1																
COORD	NATE 51	STEM			-													
SET "RECTANGULAR "GRID		- I																
CLEAR MESH CLEAR GEOM		1																
ATTACH		^ AL	TOMES	- 1														
CHANGE CLASS		10	4ECK	-														
CONVERT		· D4	DUPLICATE															
EXPAND		* IN	TERSECT													1		
MOVE		- R	LAK	-														
RENUM	ER	- RE	EVOLVE															
SOUDS		81	RETCH			-											÷.,	
SUBDIVI	DE	59	WEEP	^													- 1	
SYMMET	нү	<u> </u>															. [•
ALL:	28150		3818 4	FUILINE:														X
120101	CHORT SUBLY SUBLY AC															11-30		
SELECT END LIST (M			*	-	-	•	•		-		•		•	•	70-10			
RETURN		1.00	AIN	1	UNDO	SAVE	DRYM	PILL	RESETVEN	Ŷ	TK#	T¥4	1Z+ R	X+ RTF+	RZ+	ZOOM BOX	IN	DEFINED
	_				OUTR.	FILES '	PLOT	MEM .	DANY WOR	XEL.	TK-	19-	TZ- R	X- RY-	R2-		OUT	

Figure 4-17 Second Node of Element 1



Repeat this for Node 3 at a location two units above Node 2.

Figure 4-18 Third Node of Element 1

Finally, pick a location two units above Node 1 for the fourth node.

The program draws the entire element. It includes a cross at the center of the element and a half-arrowhead on the first side of the element in the direction of the connectivity. The cross in the middle of the element is the *handle* of the face of the element; in 2D, the face is the element itself. If you need to pick this element, click the $<\uparrow>$ in the center of the element.



Figure 4-19 Element 1 Completed

You do not need to click on the ADD button on the ELEMS panel again to enter a second element. As discussed in Chapter 2, page 2-4, until you explicitly instruct it otherwise, the program assumes you want to continue the previous action: in this case, adding elements.

Pick Node 2 of Element 1 for the first node of the second element. You will see this node light up on your screen.



Figure 4-20 First Node of Element 2

The second node of Element 2 is positioned two units to the right of the first node. Again, the program confirms this by drawing a square and the node number, Node 5, at that location.

MESH GENERATION						y=10	-										MAADO
NODES	ADD	REM	EDIT	SHOW													0011010
BLEMS	ADD	REM	EDIT	SHOW													
PTS	ADD	REM	EDIT	SHOW													
CRVS	ADD	REM	EDIT	SHOW													
SRES	ADD	REM	EDIT	SHOW													
SOUDS	ADD	REM		SHOW													
BETWEEN NODE BETWEEN POINT																	
ELEMENT CLASS GUND (4)					-												
CURVE TYPE UNE																	
SURFAC	E TYPE	1 01	MD	+													
SOUD TYPE BLOCK																	
COORDINATE SYSTEM																	
SET RECTANGULAR GRID																	
CLEAR N	6.51	a	EAR GEO	M													
ATTACH		* AU	TOMES	۲ P		4	_				_	3					
CHANGE CLASS C		° 0	IECK.	٢		I						Ī					
CONVER	IT	1 DU	IPLICATE														
EXPAND		1 IN	TERSECT			•											
MOVE		* RE	1AX	1													
RENUM	ER	' RE	VOLVE						~								
\$0UD8		1 81	RETCH			1			<u>~</u> '			1				· .	
SUBDIVE	DE	59	ALT N											\backslash		- 7	
SYMMET	RV	+												\backslash			
ALL:	SELEC	3. VS	1111	KITLINE										\.		¥	XX
EX851	UNSER	. 100	119. 1	URFAC													
SELECT		* EN	D LIST (1		<u></u>	-	-			-	-2			e S		,0-30
RETURN		1.00	AIN	1	UNDO	SAVE	DRAW	FILL	RESET NEW	TK+	T¥+	TZ+ P	SA RYA	RZ+	20044	IN .	USER
					UTLS'	FILES '	PLOT "	VIEW.	F DWN, MODEL	TK-	TY-	TZ- P	K- RY-	RZ-	808	OUT	DEFINED

Figure 4-21 Second Node of Element 2

The third node is positioned four units above Node 5. Click the *<***ML***>* on that particular grid point to create Node 6.



Figure 4-22 Third Node of Element 2

Pick the last node of Element 2 so that it coincides with the third node of Element 1. The program confirms this pick by highlighting the existing node. The connectivity for this element is complete and is confirmed by the display of the entire element.



Figure 4-23 Element 2 Completed

Add the third element by repeating the same sequence of steps. Pick Node 1 of Element 3 to coincide with Node 4 of Element 1.



Figure 4-24 First Node of Element 3

To create the second node of Element 3, pick Node 3 of Element 1 which also coincides with Node 3 of Element 2. Once again, the node will light up to confirm it has been picked.



Figure 4-25 Second Node of Element 3

The third node of Element 3 is positioned above the second node (Node 3). Use *<***ML***>* to pick this node by clicking on the grid point that is two units above the previous one.



Figure 4-26 Third Node of Element 3



Complete the element by picking a grid point two units above the first node of this element.

Figure 4-27 Element 3 Completed

Turn the grid off by clicking on the GRID button located on the COORDINATE SYSTEM panel. The toggle returns to the default released state.



Figure 4-28 Turning Off the Input Grid

Click on the FILL button to scale the picture to fit the screen. The FILL button is located in the static menu area. The picture was previously scaled by the object and the grid (see Figure 4-7). Now that the grid is turned off, the object occupies the entire graphics area.



Figure 4-29 Picture Scaled to Fill Screen; Locating Subdivide

Assume you want to subdivide the elements. Click on the SUBDIVIDE button to update the dynamic portion of the menu.

The DIVISIONS button shows the default values for subdivisions in the first, second, and third direction. The first direction is defined by the half-arrowhead on the first side of each individual element. The element type you are using in this model is a two-dimensional QUAD(4) element. Even though a two-dimensional element does not have a third direction, its coordinate must still be entered. Click on the DIVISIONS button (using the <ML>) and type in 2 2 1.



Figure 4-30 Setting the Subdivide Parameters

Click on the ELEMENTS button to indicate that you are ready to subdivide elements using the current settings. The program prompts you for the element list to be subdivided with the following string:





Figure 4-31 Activating the Subdivide Processor

Use the *<***ML***>* to click on the handle of each element to indicate that you want to subdivide it. Each element that you click on will light up.

Once you have picked every element on the screen, click the $\langle MR \rangle$ with $\langle \uparrow \rangle$ anywhere over the graphics area to indicate an *end of list* to the program. Alternatively, you can click on the END LIST (#) button in the static menu area. All elements are now subdivided. Instead of picking all the individual elements, you can click on the all: EXIST. button to subdivide all elements.



Figure 4-32 Indicating End of List
Notice the double nodes at the corners of the original elements. The **SWEEP** processor eliminates duplicate nodes. To access the **SWEEP** processor, you must first return to the mesh generation menu by clicking on the **RETURN** button and subsequently click the **SWEEP** button.



Figure 4-33 Returning to the Mesh Generation Menu



Figure 4-34 Accessing the Sweep Processor

Note that there are 35 nodes, some of which are duplicate nodes. Click on the NODES button on the SWEEP panel to eliminate the duplicate nodes. Use the default value 0.0001, for the tolerance.



Figure 4-35 Activating the Sweep Processor

The program prompts you for the list of nodes to sweep with the following string:

```
Enter sweep node list:
```

Click on the all: EXIST. button to indicate that you want to sweep all nodes. The program removes the duplicate nodes and displays the mesh with only 21 nodes remaining.



Figure 4-36 Identifying the Nodes to Sweep

Save the mesh by clicking on the SAVE button. The mesh is saved in Mentat format in a binary file called *model1.mud*. If *model1.mud* already exists on the disk, the numeral in the file name is automatically incremented by one, and the file name is thus called *model2.mud*.

Stop the session with the following button sequence:

MAIN QUIT EXIT

You should now feel comfortable interacting with Mentat. We encourage you to practice with the example detailed in this chapter, and to build on the experience you gained through this session.

Before you read through the first detailed session description, a small note needs to be made on the button sequence diagram introduced above. All button sequence diagrams used in this guide assume you are starting from the main menu. This approach is used to avoid any possible ambiguity as to where buttons are located. As you become more confident in using the program, you will note that it is not necessary to return to the main menu for each operation as outlined in the button sequence diagram.





A Simple Example

Chapter Overview

In this section, it will be demonstrated how to set up the basic requirements for a linear elastic stress analysis. For this purpose, a flat square plate with a circular hole subjected to a tensile load will be analyzed. It is generally known that around the hole a stress concentration exists. Both the deformed structure and the stress distribution need to be determined. The goal of the analysis is to demonstrate:

- a simple mesh generation technique, using the geometric meshing approach
- how to apply boundary conditions
- how to set material properties
- how to set geometric properties
- selecting quantities to be calculated in the analysis for subsequent postprocessing
- how to submit a job using the MARC finite element program
- how to generate deformed structure plots, contour plots, and path plots

Background Information

A square plate with dimensions 20 * 20 mm and a thickness of 1 mm contains a circular hole with radius 1 mm at the center of the plate. The material behavior is assumed to be linear elastic with Young's modulus $E = 200000 \text{ N/mm}^2$ and Poisson's ratio v = 0.3. A tensile load with magnitude $p = 10 \text{ N/mm}^2$ will be applied both at the top and the bottom of the plate.

Calculate the deformed structure and determine the yy-component of stress along the cross section near the hole.



Figure 5-1 Plate with a Hole Subjected to Tension

Due to the symmetry of the problem, it is sufficient to analyze only a quarter of the problem. At the line x = 0 and y = 0 symmetry boundary conditions have to be applied.





Overview of Steps

- Step 1 Mesh generation
- Step 2 Boundary conditions
- Step 3 Material behavior
- **Step 4** Geometric properties
- Step 5 Job definition
- Step 6 Postprocessing

Detailed Session Description

Step 1

The applied approach for generating the model is to use the geometrical technique to specify the boundary curves and the surface spanned by these curves. Subsequently, the surface will be converted into finite elements.

As in the sample session in chapter 4, the first step for building the mesh is to establish an input grid. Click on the MESH GENERATION button of the main menu. Next click on the SET button to access the coordinate system menu where the grid settings are located. Use the following button sequence to set the horizontal and vertical grid spacing to 1 and both the horizontal and vertical grid dimensions to 10.

```
MAIN
MESH GENERATION
SET
U DOMAIN
0 10
U SPACING
1
V DOMAIN
0 10
V SPACING
1
grid ON
FILL
RETURN
```

(on)

Two geometrical entities will be used to describe the boundary contour. First set the curve type to a circular arc and define the arc segment.

```
MAIN

MESH GENERATION

CURVE TYPE

CENTER/POINT/POINT

RETURN

crvs ADD

0 0 0

1 0 0

0 1 0
```

(Pick the following points from the grid) (center point) (starting point) (ending point)

In the graphics window, a circular arc will now be visible. Change the curve type subsequently to a polyline.

```
MAIN

MESH GENERATION

CURVE TYPE

POLYLINE

RETURN

crvs ADD

point(10,0,0)

point(10,10,0)

point(0,10,0)

END LIST (#)
```

(Pick the following points from the grid)

MESH CE	NERAD	ON				y~10													
NODES	ADD	REM	EDIT	SHOW															COMMAC
BLEMS	ADD	REM	EDIT	SHOW															
PTS	ADD	REM	EDIT	SHOW															
CRVS	ADD	REM	EDIT	SHOW															
SRES	ADD	REM	EDIT	SHOW															
SOUDS	ADD	REM	-	SHOW															
BETWEEP	NODE	86	WEEN	POINT															
ELEMENT CLASS / QUND (4)																	+		
CURVE T	YPE	1 PO	LYLINE	T															
SURFAC	ETYPE	* OU	WD	+	1														
SOUDTY	PE	· 81.0	эск	7															
COORDE	NATE 51	STEM			1														
SET -	RECTA	NGUL/	w 11+0	GRID															
CLEAR M	131	1 cu	LAN GEO	DM I															
ATTACH		* AU	TOMES																
CHANGE	CLASS	· CH	ECK	+		•	-	-											T I
CONVER	r	1 DU	PLICATE																
EXPAND		1 INT	ERSECT																
MOVE	_	* RE	LAK	÷															
RENUMB	ER	1 RE	VOLVE	*															
SOUDS		1 819	LETCH																
SUBDIVE	Æ	1 214	CEP															- 2	
SYMMET	RY .	+			1													. Î	
All:	SELEC	1. VS	81.E 4	NUT		_	-											· 4	×
	UNDER	. 1019	19. 1	SURFAC			/												
SELECT		* EN	O LIST 0	6			1	-				-							u-10
RETURN		MA	an	1	UNDO	SAVE	DRAW	FILL	RESETVIE	N	TK+	TY+	TZ+	ROLA	RYA	RZ+	20044	IN	USER
					UTLS?	FILES '	PLOT "	VIEW 1	DVN. MO	DEL	TK-	TY-	TZ-	RX-	RY-	82-	808	OUT	DEFINED

Figure 5-3 Boundary Curves of Quarter of the Plate

The two basic curves will now be used to describe a ruled surface. Set the surface type to ruled and specify the both curves.

```
MAIN
MESH GENERATION
SURFACE TYPE
RULED
RETURN
srfs ADD
1
2
```

(pick the arc) (pick the polyline)



Figure 5-4 Surface Definition

With the **CONVERT** processor, the surface will be converted to finite elements. In the CONVERT menu, it can be observed that by default the mesh division will be set to 10 by 10 elements. The BIAS FACTORS will be used to ensure that the mesh is more refined in the direction of the hole. The first surface direction is along the arc; the second surface direction runs from the arc to the polyline. A negative bias factor will be specified here, indicating that the refinement must be near the hole. Now convert the surface to a finite element mesh.



Figure 5-5 Generated Element Mesh

The arrows near the element edges indicate that the elements are numbered clockwise. MARC requires that planar elements are numbered counter-clockwise. The numbering can be changed using the UPSIDE DOWN and FLIP ELEMENTS options in the CHECK menu. While checking, all elements with incorrect numbering are put in the temporary selection buffer, which is graphically shown by a change of color. Therefore, the list of elements that need to be flipped can easily be specified using the all: SELECTED button. Repeating the check will show that no upside-down elements are found anymore so that the temporary selection buffer will be empty again.

MAIN MESH GENERATION CHECK UPSIDE DOWN FLIP ELEMENTS all: SELECT. UPSIDE DOWN RETURN

Step 2

The symmetry conditions can be applied using the following button sequence:

```
MAIN
```

```
BOUNDARY CONDITIONS
  MECHANICAL
     FIXED DISPLACEMENT
        ON x displace
                                                                       (on)
        OK
     nodes ADD
                                             (box pick the nodes on the line x=0)
        END LIST (#)
     FILL
     NEW apply
     FIXED DISPLACEMENT
        ON y displace
                                                                       (on)
        OK
     nodes ADD
                                             (box pick the nodes on the line y=0)
        END LIST (#)
     FILL
```



Figure 5-6 Applied Boundary Conditions at the line y=0

The applied loading is a tensile edge load with magnitude 10. Mentat allows to prescribe a distributed pressure on element edges. The following button sequence will give the prescribed loading.

```
MAIN
BOUNDARY CONDITIONS
MECHANICAL
NEW
EDGE LOAD
PRESSURE
-10
OK
edges ADD
END LIST (#)
```

(specify pressure)

(box pick the edges on the line y = 10)

A graphical verification of the applied edge loading is now obtained.



Figure 5-7 Edge Loading

Step 3

The material behavior is identical for all elements. Elastic behavior with Young's modulus and Poisson's ratio must be specified for this material. The following button sequence fulfills the requested task. Note that after entering the Young's modulus, Mentat automatically requests for the Poisson's ratio, which can subsequently be entered. After entering this value, the mass density is requested. By entering a $\langle CR \rangle$, this sequence may be stopped.

After specifying the list of elements, the material description is complete.

MAIN MATERIAL PROPERTIES ISOTROPIC YOUNG'S MODULUS 200000. 0.3 OK elements ADD all: EXIST.



Figure 5-8 Material Properties

Step 4

Many elements require geometrical properties such as cross-sectional areas for beams and thickness for plate and shell elements. For this plane stress analysis, the thickness must be specified for all elements.

```
MAIN
GEOMETRIC PROPERTIES
PLANAR
PLANE STRESS
THICKNESS
1
OK
elements ADD
all: EXIST.
RETURN
RETURN
```

Step 5

All ingredients for a linear elastic static analysis are now created. No incremental steps are required, nor does the loading consist of various load vectors. Therefore, entering the LOADCASE menu is not necessary.

In the JOBS menu, first set the analysis class to MECHANICAL, indicating that a stress analysis will be performed. In the pop-up menu first select JOB RESULTS. Here, the analyst has to specify which element quantities have to be written to the post file. For simplicity, the full stress tensor is selected. Alternatively, all requested components of the stress tensor can be selected. (Note that the stress tensor writes 6 components to the post file; three of them are zero for a plane stress element).

In the INITIAL LOADS menu, it can be verified if all boundary conditions (symmetry conditions and edge load) are active as initial loads. The initial loading is the complete loading for a linear elastic analysis. Loading histories or different loading steps require the use of the LOADCASE option. The INITIAL LOADS screen must contain the following on the next page:

MAIN JOBS MECHANICAL JOB RESULTS tensors stress OK INITIAL LOADS OK

(Select stress)



Figure 5-9 Initial Loads

Next, the element type will be set. In this analysis, the 4 noded plane stress element type 3 with full integration will be used for all elements.

MAIN
JOBS
ELEMENT TYPES
PLANE STRESS
3
OK
all: EXIST.

(full integration, QUAD(4))



Figure 5-10 Selecting the Element Type

The job is now ready for submission. The RUN command controls the submission of the job. One of the SUBMIT commands starts the analysis starts and with the MONITOR option the current status of the job can be observed.

MAIN JOBS SAVE RUN SUBMIT 1 MONITOR

2005	+ ~ A	1 -1 1		
NEW REM	INUN JOB			
WWE job1	TITLE ,	CHECK SIZES		
COPY PREV NEXT EDIT	MEMORY ALLOCATION	1000000		
ANALYSIS CLASS	F EXTENDED PRECESION INPUT FILE			
+ MECHANICAL	WRITE INPUT	EDIT INPUT		
CRIGID PLASTIC	USER SUB FILE	CLEAR FILE		
CHEAT TRANSFER				
° COUPLED 7	COMPLETING SAVE	SEL USER SUES		
COUPLED NGID PLASTIC	a Farina and	10		
CJOULE HEATING	a posteraria	la.		
^ ACOUSTIC	RUN DDM	1		
* BEARING	SUDMIT 1	DECUTE 1		
MORE	SUBMIT 2	EXECUTE 2		
ADAPTINE MESHING CRITERIA	SUBMIT 3	EXECUTE 3		
LEMENT TYPES	unner famin	on fear		
ADDITIONAL PRINT	OPOWIE MONEY	ON NEL		
COMAIN DECOMPOSITION	SUBMISSION #	1		
CHECK RENUMBER ALL TABLES	STATUS	Complete		
ADIATION VEWFACTORS	INCREMENT	0		
REACTIVATED ELEMENTS	SUE-NOREMENT	0		
ADD REAL TO	CPU TIME	4.90		
	SINGULARITY RATIO	0.40578		
ALIN .	CONVERGENCE BATIO	0		
ALL: SELECT. VEHELE OUTUNE	EXIT NUMBER	3004		
DRST. UNSEL INVIS. SURFACT	ENT MESSAGE	EDIT FILE		
BAD LIST (8)	EDIT OUTPUT FILE	EDIT LOG FILE		
LETURIN MAAIN U	acset 1	14		

Figure 5-11 Screen After Job Completion

Step 6

Once the job is complete we are able to do postprocessing. The postprocessing tasks are performed on the MARC post file which does not contain all the information from the Mentat database. Therefore, it is always recommended to save the database before doing any postprocessing.

The MARC post file contains an analysis header (containing the mesh topology information) and the results of the various increments. Therefore, NEXT INC must be used after opening the post file, to view the results of the first increment (usually increment 0). Clicking the DEF & ORIG button only does not seem to show any deformation on the screen. The magnitude of the displacement is simply too small for any visual effect. (By default the displacement will be added with multiplication factor 1 to the original coordinates to get the deformed structure). Automatic scaling will show the requested deformed structure.

MAIN RESULTS OPEN DEFAULT NEXT INC DEF & ORIG deformed shape SETTINGS deformation scaling AUTOMATIC FILL



Figure 5-12 Deformed Structure Plot

Contour plots of stress or displacement components can be made with the continuous contours or the contour bands option. First, the quantity to be contoured has to be selected followed by clicking the CONTOUR BANDS option.





Figure 5-13 Contour Plot of yy Component of Stress

A plot of the yy component of stress along the cross-section near the hole can be obtained with the path plot option. First, a node path has to be selected (followed by an end list) and then the curve to be plotted can be specified. With the SHOW MODEL option, the screen can be changed to display the model again.

MAIN RESULTS PATH PLOT NODE PATH END LIST (#) SET VARIABLES ADD CURVE Arc Length Comp 22 of Stress FIT RETURN SHOW MODEL RETURN

(pick the first and last node at the line y = 0)



Figure 5-14 Path Plot

The results show that the plate has been analyzed correctly. Around the hole a stress concentration factor of about 3 is present and the deformed structure plot shows an to be expected deformation field.

Close the post file and leave Mentat.

Procedure File

```
| Version : Mentat 3.3
*alias action
*stop_procedure
*exec_p gen_image.proc
@top()
*action | image 0
*action | figure 5.1 - dummy
*action | figure 5.2 - dummy
@push(meshgen)
@push(coord system)
*grid_u_domain
0 10
*grid_u_spacing
1
*grid_v_domain
0 10
*grid_v_spacing
1
*set grid on
*fill_view
@pop(1)
@push(curve_type)
*set_curve_type arc_cpp
@pop(1)
*add curves
0 0 0
1 0 0
0 1 0
@push(curve_type)
*set_curve_type polyline
@pop(1)
*add_curves
point( 1.0000000000e+001, 0.0000000000e+000,
0.00000000000e+000)
 point( 1.0000000000e+001, 1.0000000000e+001,
point( 0.00000000000e+000, 1.0000000000e+001,
0.00000000000e+000)
```

```
# | End of List
*action | figure 5.3
@push(surface_type)
*set_surface_type ruled
@pop(1)
*add surfaces
 1
 2
*action | figure 5.4
@push(convert)
*set_convert_bias_factors
0 -0.5
*convert_surfaces
 1
# | End of List
@pop(1)
*set_grid off
*fill view
*action | figure 5.5
@push(check elements)
*check_upside_down
*flip_elements
@push(select_screen)
all selected
@pop(1)
*check_upside_down
@pop(1)
@pop(1)
 step 2 - boundary conditions
@push(apply)
@push(apply_mechanical)
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed_disp_pm)
*apply_dof x
@popdown(fixed_disp_pm)
*add_apply_nodes
 11 22 33 44 55 66 77 88 99 110 121
# | End of List
```

```
*fill view
*new_apply
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed disp pm)
*apply_dof y
@popdown(fixed_disp_pm)
*add apply nodes
 1 12 23 34 45 56 67 78 89 100 111
# | End of List
*fill view
*action | figure 5.6
*new_apply
*apply_type edge_load
@set($usersubfv,FORCEM)
@popup(apply_edge_load_popmenu)
*apply value p
-10
@popdown(apply_edge_load_popmenu)
*add_apply_edges
 96:0 97:0 98:0 99:0 100:0
#
*action | figure 5.7
@pop(1)
@pop(1)
   step 3 - material behaviour
@push(mat_props)
*material_type mechanical:isotropic
@popup(mat isotropic)
*material value isotropic:youngs modulus
200000
.3
*action | figure 5.8
@popdown(mat isotropic)
*add_material_elements
all_existing
all_existing
```

```
@pop(1)
```

```
step 4 - geometrical properties
@push(geom props)
@push(geometry_mech_planar)
*geometry type mech planar pstress
@popup(geom_mech_pl_stress_pm)
*geometry_value thick
1
@popdown(geom mech pl stress pm)
*add_geometry_elements
all existing
all existing
@pop(1)
@pop(1)
 step 5 - job specification
@push(jobs)
*job_class mechanical
@popup(job mechanical)
@list(job_add_lcase_list,0)
@list(job_rem_lcase_list,0)
@popup(tensor_scalar_popmenu)
@list(post var list1,0)
*add_post_tensor stress
@list(post tensor list1,0)
@popdown(tensor_scalar_popmenu)
@list(job_add_lcase_list,0)
@list(job_rem_lcase_list,0)
@list(job add lcase list,0)
@list(job_rem_lcase_list,0)
@popup(init loads popmenu)
*action | figure 5.9
@popdown(job_mechanical)
@popdown(init_loads_popmenu)
@push(element_types)
@popup(eltype_pl_stress_pm)
*action | figure 5.10
```

```
*element_type 3
@popdown(eltype_pl_stress_pm)
all_existing
@pop(1)
@push(file screen)
@set($file browser label,SAVE AS FILE)
@set($file_browser_command,*save_as_model)
@popup(save file browser popmenu)
@list(save_file_browser_filelist,0)
@list(save_file_browser_dirlist,0)
*save as model c05.mud yes
@popdown(save file browser popmenu)
@pop(1)
*update job
@popup(job_run_popmenu)
*submit_job 1
*monitor_job
*action | figure 5.11
@popdown(job_run_popmenu)
@pop(1)
 step 6 - post processing
@push(results)
*post_open_default
*post next
*set deformed both
@push(deformed_settings)
*set automag on
*fill view
*action | figure 5.12
@pop(1)
*post value
@popup(post_scalar_pm)
@list(post scalar list,0)
Comp 22 of Stress
@popdown(post_scalar_pm)
*post contour bands
*zoom out
*zoom_out
*action | figure 5.13
```

```
@push(post_pathplot)
*set_pathplot_path
1
111
# | End of List
@push(pathplot_vars)
@list(path_var_list,0)
*pathplot_add
Arc Length
Comp 22 of Stress
@list(path_var_list,0)
*pathplot_fit
*action | figure 5.14
*show_model
@top()
```



Solid Modeling and Automatic Meshing

Chapter Overview

The sample session described in this chapter demonstrates the process of solid modeling and automatic meshing. A simple bolt structure will be modeled. The goal of the analysis is to demonstrate:

- Solid modeling, entering simple building blocks.
- Using boolean operations and blending techniques to complete the solid model.
- Use of symmetry to reduce the solid model.
- Convert solid faces into surfaces and use of automatic surface meshers.
- Use of the automatic tetrahedral mesh generator to generate the mesh.
- Use of the symmetry and duplicate options to complete the model.



Background Information

In this example, it will be demonstrated how to generate an element mesh for a simple bolt structure as shown in Figure 6-1.



Figure 6-1 Simple Bolt Structure

As can be seen from this figure, the model globally consists of three simple geometrical components: two cylinders with different radii and a 6-sided prism. A solid model of the structure can be created using two boolean operations. Two cylinders must be united and from the resulting solid a prism must be subtracted. After these operations, a complex solid is obtained. Some of the edges of this solid must be given a specific curvature. This is achieved with the BLEND operator.

Before using the mesh generators however, the solid model will be reduced. The model is symmetrical with respect to a segment of 30 degrees. By subtracting two solid blocks from the solid model, the 30 degrees segment will be obtained.

Solid meshes are generated with a three step approach. First the faces of a solid are converted to surfaces. These surfaces will be used to generate a surface mesh. This surface mesh is used to create a solid mesh.

After specifying an average edge length for all edges of the solid, the segment will be meshed automatically, creating tetrahedral elements. The resulting mesh will subsequently be expanded to a mesh for the complete bolt by use of the **SYMMETRY** and **DUPLICATE** processors.

Overview of Steps

- Step 1 Input of basic solids
- **Step 2** Refining the solid model
- Step 3 Reducing the solid model to the smallest segment with symmetry
- Step 4 Surface meshing on the reduced solid model
- **Step 5** Meshing of the solid model based on the generated surface mesh
- **Step 6** Use of symmetry and duplication operations to complete the mesh

Detailed Session Description

Step 1

The approach used in generating the solid model is to start with three simple building blocks. The building blocks are two cylinders and one prism.

MAIN	
MESH GENERATION	
SOLID TYPE	
CYLINDER	
RETURN	
solids ADD	
0 0 0	(solid cylinder origin coordinates)
0 0 1	(solid cylinder axis coordinates)
0.4 0.4	(solid cylinder radii)
0 0 1	(solid cylinder origin coordinates)
0 0 2	(solid cylinder axis coordinates)
0.7 0.7	(solid cylinder radii)
VIEW	
activate 4	(on)
activate 1	(off)
PERSPECTIVE	
show 4	
FILL	
RETURN	

In the above VIEW process, view 4 has been activated and set to a perspective projection. View 1 has been deactivated to prevent switching on perspective plotting for this view.


Figure 6-2 Two Cylindrical Solids

The two basic cylinders will now be united.

```
MAIN
MESH GENERATION
SOLIDS
UNITE
1 2
END LIST (#)
RETURN
```

(Pick solid 1 and 2)



Figure 6-3 Result of UNITE Operation

Next, the prism will be defined and subtracted from the current solid.

```
MAIN
  MESH GENERATION
     SOLID TYPE
       PRISM
       RETURN
     solids ADD
       0 0 1.4
             2.1
       0
          0
       0.4
       6
     SOLIDS
       SUBTRACT
          1
             2
          END LIST (#)
```

(solid prism base coordinates) (solid prism axis coordinates) (solid prism radius) (number of solid prism faces)

(Pick solids)

The basic solid model is now completed and the result is shown in Figure 6-4. Note that at this stage only one solid exists. The basic building blocks are no longer present in the database after the boolean operations.



Figure 6-4 Result of SUBTRACT Operation

Step 2

In this step, two blending operations will be applied to the outer edges of top cylinder of the basic solid model. The blending process consists of specifying a radius and indicating to which solid edge the blending operation will be applied. Before performing the blending operations, the solid edges will be labeled. This is not strictly necessary for the process, since all edges will be graphically picked. For describing the process, however, it is useful to indicate the edge labels. Note that after each blending operation the edge numbering changes. The detailed button sequence is shown on the following page:

MAIN MESH GENERATION PLOT label solid EDGES REGENERATE RETURN SOLIDS blend RADIUS 0.1 blend EDGE 1:12



(on)



Figure 6-5 Activating the Labeling of Solid Edges



Figure 6-6 First Edge Blended

The result after the first blending is shown in Figure 6-6. The second edge will now be blended and the labelling of the solid edges will be switched off.

```
MAIN
MESH GENERATION
SOLIDS
blend EDGE
1:20
PLOT
label solid EDGES
REGENERATE
RETURN
```

(off)

The solid modeling phase is now completed as is shown in Figure 6-7.



Figure 6-7 Completed Solid Model

Step 3

By looking at the model, we can observe that the solid model has certain symmetry planes. Since the complete model (360 degrees) has a prism with 6 edges, the model can be considered as a duplication of 6 segments. Furthermore, this 60 degree segment is symmetric. Thus, a 30 degree segment is the smallest section for which a mesh is required. In this step, the solid model is reduced to a 30 degree segment. This is achieved by adding two solid blocks and subtracting them from the solid model. MAIN MESH GENERATION VIEW show 1 FILL RETURN SOLID TYPE BLOCK RETURN solids ADD -1 0 -1 2 2 4

(solid block origin coordinates) (solid block X, Y, and Z dimensions)



Figure 6-8 First Solid Block Added

The second block will be duplicated from the first one.

MAIN MESH GENERATION DUPLICATE ROTATIONS 0 0 150 SOLIDS 2 END LIST (#)

(duplicate rotations in X, Y, and Z)

(*Pick the solid block*)



Figure 6-9 Duplicating the Solid Block

Finally, solid 3 and solid 2 will be subtracted from body 1.

MAIN MESH GENERATION SOLIDS SUBTRACT 1 2 3 END LIST (#) FILL VIEW show 4 RETURN FILL

(Pick solid to subtract from) (Pick solid to be subtracted) (Pick solid to be subtracted)



Figure 6-10 Completed Solid Segment

Step 4

In this step, the surface of the reduced model will be meshed automatically using one of the automatic surface meshers. Therefore, first all faces of the solids will be converted into surfaces, then plotting of surfaces and points will be switched off.

```
MAIN

MESH GENERATION

SOLIDS

convert SOLID FACES TO SURFACES

all: EXIST.

PLOT

draw SOLIDS

draw POINTS

REGEN

RETURN

RETURN
```

```
(off)
(off)
```

Next, a target element edge length will be specified by using the curve division command. Here, for all curves an average element edge length of 0.1 is used. In order to ensure that elements generated for specific surface match, the curve divisions along particular curves will be matched.

```
MAIN

MESH GENERATION

AUTOMESH

CURVE DIVISIONS

AVG LENGTH

0.1

APPLY CURVE DIVISIONS

all: EXIST.

0.005

(enter tolerance for vertex points)

all: EXIST.

(for all curves)
```

In Figure 6-11, the resulting seed points are shown.



Figure 6-11 Seed Points for Automatic Meshing

Next, the automatic surface mesh generator will be used to generate a surface mesh. After the mesh generation, the drawing of the nodes will be switched off for easier verification of the generated elements (the elements will be drawn in solid mode instead of wireframe mode).

```
MAIN

MESH GENERATION

AUTOMESH

SURFACE MESHING

triangles (delaunay) SURFACE TRI MESH!

all: EXIST (for all surfaces)

PLOT

draw NODES (off)

elements SOLID

REGENERATE

RETURN
```



Figure 6-12 Completed Surface Mesh for the Segment

Note that for each surface a surface mesh has been created. Due to the use of match curve divisions, it is ensured that nodes along the curves are close to each other. However, there are still duplicate nodes at these points.

Step 5

The results of the surface mesh generator can be used as input for the solid mesh generators. It is required that a closed surface is present. Duplicate nodes present to the use of the surface mesher for each surface are removed with the **SWEEP** processor. This can be verified by plotting the outline edges only for the structure.

MAIN MESH GENERATION PLOT MORE edges OUTLINE RETURN RETURN SWEEP TOLERANCE .0001 NODES all: EXIST RETURN PLOT MORE edges SURFACE RETURN RETURN

Subsequently, it is specified that the automatic tetrahedral mesher will be used for all triangular elements generated in the previous step.

MAIN MESH GENERATION AUTO MESH SOLID MESHING tetrahedra TET MESH! all: EXIST

(list of triangular elements)



Figure 6-13 Completed Tetrahedral Mesh



Step 6

The mesh for the solid segment has been generated automatically. As discussed before, the complete model consists of 6 identical parts each with a symmetry plane. First, the **SYMMETRY** processor will be used to generate a 60 degrees segment. Note that a formula is used to enter the symmetry plane normal.

```
MAIN

MESH GENERATION

SYMMETRY

NORMAL

sin(30*pi/180)

cos(30*pi/180)

0

ELEMENTS

all: EXIST.
```



Figure 6-14 Mesh after use of the SYMMETRY Operator

The generated 60 degrees segment will now be duplicated 5 times to generate the complete model.





Figure 6-15 Element Mesh after the DUPLICATE Process

Both the **SYMMETRY** and the **DUPLICATE** processor generate new elements but do not check if duplicate nodes are created which have to be removed in order to make the connection between the different parts. This removal of duplicate nodes is achieved with the SWEEP process using the tolerance of 0.001.

The mesh for the bolt structure is now completed. The model is saved in a Mentat model file.

MAIN MESH GENERATION SWEEP sweep NODES all: EXIST. SAVE



Figure 6-16 The SWEEP Processor

Procedure File

```
| Version : Mentat 3.3
*alias action
*exec_p gen_image.proc
*action | figure 6.0 - dummy
*action | figure 6.1 - dummy
@pop(1)
  step 1: add the basic solid building bocks
@push(meshgen)
@push(solid_type)
*set_solid_type cylinder
@pop(1)
*add solids
0 0 0
0 0 1
0.4 0.4
0 0 1
0 0 2
.7.7
@push(viewing_screen)
*activate view 4
*deactivate_view 1
*view_perspective
*show view 4
*fill view
@pop(1)
*action | figure 6.2
@push(solids)
*solids_unite
 1
 2
# | End of List
*action | figure 6.3
@pop(1)
@push(solid_type)
*set_solid_type prism
@pop(1)
@push(solids)
```

```
*solids_subtract
@pop(1)
*add_solids
0 0 1.4
0 0 2.1
0.4
6
@push(solids)
*solids_subtract
 1
 2
# | End of List
*action | figure 6.4
@pop(1)
 step 2: modify the solid model by including blending
@push(plot_screen)
*set_solid_edge_labels on
*regenerate
*action | figure 6.5
@pop(1)
@push(solids)
*solid_blend_radius
.1
*solid_blend_edge
1:12
*action | figure 6.6
*solid_blend_edge
1:20
@push(plot_screen)
*set_solid_edge_labels off
*regenerate
@pop(1)
*action | figure 6.7
@pop(1)
 step 3 prepare for the meshing by reducing the model to
              the smallest symmetrical structure
@push(viewing_screen)
```

*show_view 1 *fill_view @pop(1) @push(solid_type) *set_solid_type block @pop(1) *add_solids -1 0 -1 2 2 4 *action | figure 6.8 @push(duplicate) *set_duplicate_rotations 0 0 150 *duplicate_solids 2 # | End of List *action | figure 6.9 @pop(1) @push(solids) *solids subtract 1 2 3 # | End of List *fill_view @push(viewing_screen) *show view 4 @pop(1)*fill view *action | figure 6.10 step 4 start the meshing on the reduced model *convert_solid_faces_surfaces all existing @push(plot_screen) *set_solids off *set_points off *regenerate @pop(1) @pop(1)

```
@push(automesh)
@push(curve_divisions)
*set_curve_div_type_fix_avgl *set_curve_div_avgl
0.1
*apply_curve_divisions
all existing
*match_curves
.005
all_existing
@push(select_screen)
*select clear
@pop(1)
*action | figure 6.11
@pop(1)
@push(automesh surfaces)
*dt_surface_trimesh
all existing
@push(plot_screen)
*set_nodes off
*set_surfaces off
*set curves off
*elements_solid
@pop(1)
*action | figure 6.12
@pop(1)
@pop(1)
@push(plot_screen)
@push(plot1_screen)
*edges outline
*regenerate
@pop(1)
@pop(1)
@push(sweep)
*set_sweep_tolerance
.001
*sweep nodes
all_existing
@pop(1)
@push(plot_screen)
@push(plot1_screen)
*edges_full
```

```
*regenerate
@push(plot_screen)
@pop(1)
@pop(1)
@push(automesh)
@push(automesh_solids)
*dt_tet_tri_mesh
all existing
*action | figure 6.13
@pop(1)
@pop(1)
@pop(1)
 step 5 use symmetry and duplicate to complete the mesh
            finally sweep to remove double nodes
@push(symmetry)
*set_symmetry_normal
sin(30*pi/180)
cos(30*pi/180)
0
*symmetry_elements
all existing
*action | figure 6.14
@pop(1)
@push(duplicate)
*set_duplicate_rotations
0 0 60
*set_duplicate_repetitions
5
*duplicate_elements
all existing
*fill view
*action | figure 6.15
@pop(1)
@push(sweep)
*sweep_nodes
all existing
*action | figure 6.16
```



Chapter Overview

The sample session described in this chapter analyzes the process of flaring. A cone-shaped flaring tool is pushed into a cylindrical tube to permanently increase the diameter of the tube end. Both the steel tube and flaring tool are modeled as deformable contact bodies. The goal of the quasi-static analysis described in this chapter is threefold:

- to determine whether the final shape of the tube meets the objective of the analysis
- to study whether residual stresses are present in the steel tube and flaring tool
- to determine the magnitude of the residual stresses (if present)

Background Information

Description

This session demonstrates the analysis of a contact problem involving two deformable contact bodies, multiple materials, kinematic constraints and loads. The nonlinear nature of the problem along with the irreversible characteristics make it impossible to determine in advance the load required to drive the tool into the tube. As a result, multiple runs through the analysis cycle are necessary to determine the maximum load required to meet the objective of the analysis.

The diameter of the tube is 8 inches, the thickness is 0.3 inches and the length is 8 inches. The flaring tool is modeled as a hollow cone with an apex angle of 30 degrees, a wall thickness of 0.6 inches, and a length sufficient to model the process.



Figure 7-1 Cylindrical Tube and Flaring Tool



Idealization

The loading and geometry of the structure are symmetrical about the center line of the cylindrical tube. Due to the nature of the analysis, you are only required to analyze an axisymmetric model of the structure. If the appropriate boundary condition is prescribed, the tube is prevented from moving in the axial direction but is free to move in a radial direction at one end. A load is applied to the rim of the flaring tool to push it into the free end of the pipe.



Figure 7-2 Axisymmetric Model of Tube and Flaring Tool

Requirements for a Successful Analysis

The analysis is considered successful when the flaring tool expands the tube diameter by 10%. You can plot the tool load versus the radial displacement at the tube end for several load increments to adjust the maximum load and repeat the analysis cycle until you reach the objective.

Full Disclosure

The steel tube is modeled by four-noded axisymmetric elements with a Young's Modulus of 30.0e6 psi and a Poisson's Ratio of 0.3. It is assumed that the tube material with an initial yield stress of 3.6e4 psi will not harden during the process. The tube diameter is 8.0 inches, and the thickness is 0.3 inches.

The flaring tool is modeled as a hollow cone with an apex angle of 30 degrees and a thickness of twice that of the tube with a suitable length to model the working area. The flaring tool is modeled as a case hardened steel object with a Young's Modulus of 40.0e6 psi, a Poisson's ratio of 0.3, an initial yield stress of 6.0e4 psi. The larger diameter end of the flaring tool is loaded to drive the smaller end of the flaring tool into the steel tube.

Overview of Steps

- **Step 1** Create a model of 2 patches and convert to finite elements. Apply kinematic constraints to the tube and add a low stiffness spring to avoid rigid body motions of the tool.
- **Step 2** Apply material properties to the tube and flaring tool.
- Step 3 Create contact bodies.
- **Step 4** Apply edge loads to the larger diameter end of the tool to push it into the steel tube and create a loadcase.
- **Step 5** Create a job and activate appropriate large strain plasticity procedure.
- **Step 6** Submit the job.
- **Step 7** Postprocess the results by looking at the deformed structure and a history plot of the tip deflection of the tube.

Detailed Session Description

Step 1

The approach used in this session to generate the model is to use the geometric meshing technique which involves converting geometric entities to finite elements. Refer to Chapter 3 for a detailed discussion on mesh generation techniques.

As in the sample session described in Chapter 4, the first step for building a finite element mesh is to establish an input grid. Click on the MESH GENERATION button of the main menu. Next click on the SET button to access the coordinate system menu where the grid settings are located. Use the following button sequence to set a grid in the u domain between 0 and 8 with spacing 1 and in the v domain between 0 and 5 with spacing 1.

```
MAIN
MESH GENERATION
SET
U DOMAIN
0 8
U SPACING
1
V DOMAIN
0 5
V SPACING
1
ON
RETURN
FILL
```

(on)

The geometric entity used in this session is a patch. The surface type used to enter a patch is a QUAD which is the default setting for surface types. Use the following button sequence to define the first patch:

```
MAIN

MESH GENERATION

PLOT

label POINTS (on)

RETURN

srfs ADD (Pick the following corner points from the grid)

point(0,4,0)

point(8,4,0)

point(8,4.5,0)

point(0,4.5,0)
```

These points are the four corners of the first patch defined as the cross section of the cylindrical tube. Next, move the top two points of the patch to their exact location.

```
MAIN
MESH GENERATION
MOVE
TRANSLATIONS
0 -0.2 0
POINTS
3 4
END LIST (#)
```

(Pick top 2 points)

The next step is to define a second patch for the cross section of the flaring tool. Due to the conical shape of tool, it is best to use the cylindrical coordinate system to enter a set of points for the cone angle of 15 degrees from the horizontal axis.

```
MAIN

MESH GENERATION

RECTANGULAR (to switch to CYLINDRICAL)

pts ADD

5 15 0

16 15 0

GRID (to switch off the grid)

FILL
```

		UTLS	FILES '	PLOT "	VIEW 1	DWN, MODEL	TK-	TY-	1Z=	RX-	RY-	RZ=	oun	OUT	otrates
RETURN	MAIN	UNDO	SAVE	DRAW	FILL	RESETVIEW	TK+	TY+	TZ+	RX+	RY+	RZ+	ZOOM	IN	USER
SELECT	END LIST (#)														1
EXIST. UNSEL	HINTS. SURFAC														
ALL: SELECT	VERELE OUTLINE													2	×
SURFACES	SOUDS													1	
PONTS	CURVES														
NODES	ELEMENTS														
RESET	1														
x		1													
Y															
x		1													
CORNELLAS	J.														
	0														
TRANSLATIONS	0														•
	0														
	0		1												2
ROTATIONS	0	1	à												a
	1														
State Prototions			y-5												
SCALE FACTORS	0														
	0														
PONT	0														
MUYE															20MMRC

Figure 7-3 First Patch (Cross Section of Cylindrical Tube)

Before duplicating the newly generated points it is important to realize that all operations are done in the local coordinate system. For now, simply change the coordinate system back to rectangular. This can be done by clicking on the CYLINDRICAL button twice.

Duplicate the just entered points and translate them 0.6 in the y-direction using the following button sequence to form the upper corners of the flaring tool.

```
MAIN
MESH GENERATION
CYLINDRICAL
SPHERICAL
DUPLICATE
TRANSLATIONS
0 0.6 0
POINTS
5 6
END LIST (#)
```

(to switch to SPHERICAL) (to switch to RECTANGULAR)

(Pick the two points generated above)

The four points for the second patch have now been defined. Use the srfs ADD command to enter the second patch.



(*Pick points in counter-clockwise direction*)



Figure 7-4 Second Patch (Cross Section of Flaring Tool)

Use the following button sequence to translate the second patch until it almost meets the cylindrical tube.



```
MAIN
MESH GENERATION
MOVE
TRANSLATIONS
0 1.25 0
SURFACES
2
END LIST (#)
```

(Pick the surface to move)

The two patches that outline the cylindrical tube and conical flaring tool respectively are shown in Figure 7-5.



Figure 7-5 Tube and Flaring Tool Patches Defined

The two patches are converted to elements. The number of subdivisions is set to 8×3 for the cylindrical tube and to 14×6 for the conical flaring tool. Use the following button sequence to convert the two patches.

```
MAIN
  MESH GENERATION
     CONVERT
        DIVISIONS
           8 3
        SURFACES TO ELEMENTS
           1
                                                   (Pick the surface to convert)
           END LIST (#)
        DIVISIONS
           14 6
        SURFACES TO ELEMENTS
                                                   (Pick the surface to convert)
           2
           END LIST (#)
        PLOT
           draw POINTS
                                                                      (off)
           label POINTS
                                                                      (off)
           draw SURFACES
                                                                      (off)
           REGENERATE
           RETURN
```

In this sample session, the **CONVERT** option is used instead of the **AUTOMESH** option. Although both options would create a finite element mesh, the **CONVERT** processor allows for better control of the element distribution.

Figure 7-6 shows the results of the conversion process.



Figure 7-6 Tube and Flaring Tool Patches Converted

Once you have converted the two patches to elements, you can assign a MARC element type to the elements. Although Mentat will assign a default type to elements, based on the dimensionality of the problem, it is advised to explicitly set the element type. The element type selected for this analysis is MARC element type 10, a four-noded axisymmetric quadrilateral element. Use the following button sequence to select this element type for all existing elements. (Pick from row FULL INTEGRATION and column QUAD(4)).

```
MAIN
JOBS
ELEMENT TYPES
mechanical elements AXISYMMETRIC SOLID
10 (FULL INTEGRATION / QUAD(4))
OK
all: EXIST.
```

SET ELEMENT TYPE	E																141.00	
MECHANICAL ELD	MENTS		ME	DIANICA	LANISYM	METRIC	SOLID ELEMENT T	VINES										
3-D TRUSS/BEAM												RIA.		QUND				
3-D MEMBRANE/											6	4	0	6	9			
3-0 SOLID	FULL INTEGRATION										126	10	28	52	94			
A05VAMETRIC SHELL				FULL & HERRMANN FORMULATION									179	82	33		<u> </u>	
				REDUCED INTEGRATION										116	55		-	
PLANE STRAIN			P2	REDUCED & HERRIMANN FORMULATION										119	58			
PLANE STRESS '			N	ILAR										141	48		-	
2-D TRUSS/BEAM			TV	NIST .										20	67			
MISCELLANEOUS			11	TWIST & HERRIMANN FORMULATION										10	65			
			TV	TWIST REBAR										145	142			
3-D TRUSS		BENDING											85	96				
		10	FOURER FULL INTEGRATION											62				
3-0 SHELL			FC	FOURER FULL & HERRMANN FORMULATION											63			
3-D SOLD			FC	FOURER REDUCED INTEGRATION											73	-	-	
AUSYMMETRIC S	RELL		FC	FOURIER REDUCED & HERRMANN FORMULATION											74		-	
AUSYMMETRIC S	oup							~	~									
PLANAK									n									
OTHER ELEMENTS								_	_									
MAGNETOSTATIC		- 1																
ELECTROMAGNE	IIC .																	
CLEAR	1															8		
D TYPES	D CLAS	58ES													1	î i		
ALL: SELECT	VENELE	OSTINIA														<u> </u>	ж.	
EXIST UNSEL	inves.	SURFAC																
SELECT	END US	06															1	
RETURN	MAIN		UNDO	SAVE	DRAW	FILL	RESET NEW	TK+	TY+	TZ+	RX+	RYA	RZ+	ZOOM	IN	USE	R	
			UTLS'	FILES '	PLOT 1	VIEW 7	F DWN, MODEL	TK-	TY-	TZ=	RX-	RY-	RZ-	908	OUT	DEE	NED	

Figure 7-7 Select MARC Element Type

The displacement degree of freedom in the x-direction for the nodes at the far left end of the cylindrical tube is fixed.

```
MAIN
BOUNDARY CONDITIONS
MECHANICAL
FIXED DISPLACEMENT
ON x displace
OK
nodes ADD
1 10 19 28
END LIST (#)
```

(*on*)

(Pick left row of nodes)



Figure 7-8 Fixed Nodes of Cylindrical Tube in X-Direction

The deformable tool will be loaded by a pressure load. If there is no contact between tube and tool, a rigid body mode is present. This rigid body mode can be removed by entering a week spring between tube and tool. Enter a spring by using the following button sequence on the next page:

MAIN LINKS SPRING/DASHPOT ZOOM BOX STIFFNESS 10.0e3 NODE 1 131 DOF 1 1 NODE 2 9 DOF 2 1

(create a zoom box by moving <↑> while keeping <ML> depressed)

SPRING	STASHPO	118															2014
MEW	REM			1													
NAME	links																
COPY	PREV	MEXT	EDIL														
UNK CL	ASS	spring		1													
STIFFIE	35	10000		1													
DAMPE	IG COEF	0		1			-							-			
USER	SUBROUT	NE USPRI	G						-	-							
REGIN							-							1			
NODE		131					-			-				+			
DOF		1							-	_				1	-	2	
an											_	_	1			_	
NODE		[n										_	-		100		
DOE		<u> </u>							1	5-			-	_		-	
		J.				_		1	\sim	2	_	_			~	10	
NTO 1 S	SPRINCES/D	ASHPOTS		1		-							-			_	_
NTO N	SPRINGS/C	DASHPOTS	+	-	2	-	_		2	_			-		2		
					-			1		÷.,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,					- 2	ć.,	
						6		-	~	2			1	_			
				-	20		_	-					1-				
					1				2	-							
				-				1									v
				[-		-								- 1	
	SUSCE	d vesee a	former	-	20												
	CONSEL	and a	SURFAC	-													
				1													
SELECT		COMPANY STOR															
SELECT		MAIN	/ (03	UNDO	SAVE	DRAW	FEL	RESETVIEW	TKe.	TV4	TZ+	FOX4	Rife.	RZ+	ZOOM	IN	USER

Figure 7-9 Specifying the Spring Between the Deformable Bodies


Step 2

Apply material properties to both the tube and flaring tool. The properties for the tube are different from those of the flaring tool. Use the following button sequence to assign the material to all elements of the tube.

MAIN FILL MATERIAL PROPERTIES ISOTROPIC YOUNG'S MODULUS 30.0e6 POISSON'S RATIO 0.3 PLASTICITY INITIAL YIELD STRESS 3.6e4 OK OK elements ADD END LIST (#)

ID MATERIALS

(use the Box Pick Method to select all tube elements)

(*on*)



Figure 7-10 Material Properties Applied to all Tube Elements

Apply the material properties for all elements of the flaring tool using the following button sequence on the next page:

MAIN MATERIAL PROPERTIES NEW ISOTROPIC YOUNG'S MODULUS 40.0e6 POISSON'S RATIO 0.3 PLASTICITY INITIAL YIELD STRESS 6.0e4 OK OK elements ADD

(Use the Polygon Pick Method to select all tool elements)





Figure 7-11 Material Properties Applied to all Elements



Step 3

Identify the two contact bodies by storing the elements of each deformable body in a set using the following button sequence:

MAIN CONTACT CONTACT BODIES NAME tube DEFORMABLE OK elements ADD END LIST (#) NEW NAME tool DEFORMABLE OK elements ADD END LIST (#)

(Use the Box Pick Method to select all tube elements)

(Use the Polygon Pick Method to select all tool elements)

The easiest way to identify the contact bodies is to request the program to draw the bodies in different colors.

MAIN CONTACT CONTACT BODIES ID CONTACT

(on)



Figure 7-12 Identifying the Tube and Flaring Tool by Color

Although maybe not apparent in Figure 7-12, the color of the tube is different from the flaring tool and is indicated in the key that appears in the upper left hand corner of the graphics area. Click on ID CONTACT once again to switch off the PLOT IDENTIFY mode

```
MAIN
CONTACT
CONTACT BODIES
ID CONTACT
```

(off)



Step 4

The following button sequence defines a table to specify the loading of the flaring tool. Figure 7-13 gives a graphical representation of the flaring tool being loaded.

MAIN **BOUNDARY CONDITIONS** MECHANICAL TABLES NAME loading TABLE TYPE time OK XMAX 87 **XSTEP** 87 **YMAX** 2400 ADD POINT 0 0 9 900 39 2400 87 0 SHOW MODEL



Figure 7-13 Loading of Flaring Tool

Apply the load to all edges at the far right end of the flaring tool.

```
MAIN
  BOUNDARY CONDITIONS
     MECHANICAL
       NEW
       EDGE LOAD
          pressure TABLE
            loading
            OK
          OK
       edges ADD
                                                          (Pick edges)
          38:1
                52:1
                       66:1
                             80:1 94:1 108:1
          END LIST (#)
```



Figure 7-14 Loading Applied to Far Right End of Flaring Tool

The following button sequence creates a loadcase with the default name *lcase1*.

```
MAIN

LOADCASE

mechanical analyses STATIC

LOADS

OK

TOTAL LOADCASE TIME

87

# STEPS

87

OK
```

(select all loads)



Step 5 & 6

Once you have defined the loadcase, activate the constant dilatation procedure for all elements to avoid numerical problems due to the incompressible plasticity and activate the large strain plasticity procedure based on the mean normal plasticity solution procedure. Select the result to be written on the post file, switch to axi-symmetric analysis, and submit the job.

<i>(on)</i>
STRAIN-MEAN NRM
(Switch to small-strain radial return procedure)
STRAIN-RADIAL R
(Switch to large strain mean normal procedure)

Step 7

The final phase of the analysis cycle (shown in Figure 1-1 of Chapter 1) is postprocessing. Postprocessing involves viewing and evaluating the results of an analysis.

In order to evaluate analysis results with Mentat, you must have a so-called post file which consists of analysis results from the finite element analysis program MARC.

A typical postprocessing session may consist of the following steps:

Reading the post file created by submitting the job

- Creating a history or path plot of the model
- Displaying a plot of the model at specific increments
- Viewing different levels of stress types on the model

The results of the flaring process analysis have been saved in a post file. Use the following button sequence to open the file:



Zoom in on the area of contact for better access of the node that represents the tip deflection of the tube. The resulting close-up of the contact area shown in Figure 7-15 should now appear in the graphics area.

MAIN

RESULTS

ZOOM BOX



Figure 7-15 Close up of Contact Area

The objective of the analysis, stated on page 7-3, requires a plot that demonstrates the tip displacement versus the load. Since the loading pattern is given in Figure 7-13, a displacement versus the increment plot can also be used. The tip displacement in y-direction on the inner diameter of the tube is collected and displayed using the HISTORY PLOT option.

```
MAIN
RESULTS
HISTORY PLOT
SET NODES
9
END LIST (#)
COLLECT DATA
0 100 1
```

The 0 is the first history increment, 100 the last history increment, and 1 is the increment step size. The program will read the increments indicated by the message Collected increment (number) in the dialogue area.

Once all the data for a plot has been collected, it can be displayed in a diagram where the increment number is the x-axis variable and the displacement in the y-direction is the y-axis variable. The FIT option allows you to view the history plot in the graphics area. Use the following button sequence to display the graph:

```
MAIN

RESULTS

HISTORY PLOT

NODES/VARIABLES

ADD CURVE

9

Increment

Displacement y

FIT
```

(from the NODES panel)

Recall the objective of our analysis: to expand the tube diameter by 10%. The Y-axis variable, displacement y, has to reach a value of 0.4 in the unloaded configuration to meet the objective. Click on the YMAX button and enter 0.5. Set the following plot settings to label the history graph.

```
MAIN
RESULTS
HISTORY PLOT
SHOW IDS
10
XSTEP
20
YSTEP
20
YMAX
0.5
```

The maximum value for y-displacement is obtained in increment 39. After this increment, the flaring tool is unloaded. The overshoot is necessary to obtain a 10% permanent diameter increase in the load-free state.



Figure 7-16 History Plot of Node 9 over 87 Increments

To better understand the process, it is helpful to look at an animation of the deformation of the tip of the tube. Return to the postprocessing results panel and click on the DEF & ORIG button to view both the original and the deformed structure. At this point, drawing the nodes and the internal edges of the mesh is no longer necessary.

Use the following button sequence on the next page to change the plot settings so that only the outline edges of the model are displayed.

MAIN PLOT MORE OUTLINE PREVIOUS draw element FACES draw NODES REGENERATE FILL

(off) (off)

Once the nodes and element faces of the interior mesh have been suppressed, leaving only an outline of the two structures, you get a much clearer picture of the extent of the deformation that has taken place.

For animation purposes, the data that is processed needs to be condensed. The data is automatically condensed and written to disk for each frame of animation. Once this process has been completed, the frames can be traversed when shown in playback mode. Use the following button sequence to condense the data and activate the playback.

```
MAIN

RESULTS

REWIND

NEXT INC

DEF&ORIG

MORE

animate INCREMENTS

100 1

animate ANIMATION

FILL

PLAY

SHOW MODEL
```

The 100 increments is a user defined upper limit of the number of frames that are to be created for the animation. The numeral 1 on the same line represents the interval at which to create a frame. In this case, each increment is defined to be a frame.

The von Mises stresses induced by the flaring process on the model can be viewed using the following button sequence:

```
MAIN

RESULTS

SKIP TO INC

39

SCALAR

DOWN

Equivalent von Mises Stress

OK

CONTOUR BANDS
```

The resulting model shown in Figure 7-17 clearly indicates that the von Mises Stress is concentrated in two areas: the tip of deflection, where the tube made contact with the tool, and in the area where the tube is deformed.



Figure 7-17 Plot of Original & Deformed Tube showing von Mises Stresses at Increment 39

Next, you can check the model for the plastic strain. Since you have already specified the increment and have contour bands selected, you only need to click on SCALAR and Total Equivalent Plastic Strain from the pop-up menu to check for permanent deformation. The resulting model, shown in Figure 7-18, indicates where plastic strain is found.



Figure 7-18 Plot of Original & Deformed Tube showing Plastic Strain at Increment 39

If you are interested in viewing the von Mises stresses over the course of 87 increments, make sure you have CONTOUR BANDS selected under the SCALAR PLOT panel, and von Mises as scalar quantity, prior to animating the model using the button sequence shown before.

Conclusion

As mentioned in the chapter overview, the goal of the analysis described in this sample session was threefold.

- **1.** To determine whether the final shape of the tube meets the objective of the analysis.
- 2. To determine whether residual stresses are present in the steel tube and flaring tool.
- **3.** If residual stresses are present, to determine what those residual stresses are.

The results of the analysis demonstrate that the goals of the analysis have been met.

- **1.** You have seen that the flaring tool expands the diameter of the tube by 10%.
- **2.** Residual stresses are present in the steel tube; however, there are not any noticeable stresses in the flaring tool.
- **3.** Figure 7-18 shows the equivalent plastic strain just before the tool is released.

Procedure File

```
| Version : Mentat 3.3
*alias action
*exec_p gen_image.proc
@top()
*action | figure 7.0 - dummy
*action | figure 7.1 - dummy
*action | figure 7.2 - dummy
 step 1
@push(meshgen)
@push(coord_system)
*grid u domain
0 8
*grid_u_spacing
1
*grid_v_domain
0 5
*grid v spacing
0.5
*set_grid on
*fill view
@pop(1)
@push(plot_screen)
*set_point_labels on
@pop(1)
 create a surface corresponding to the tube
   length=8.0, inner diam=4.0, outer diam=4.3
*add surfaces
point( 0.0000000000e+00, 4.000000000e+00,
0.00000000000e+00)
point( 8.0000000000e+00, 4.000000000e+00,
0.000000000000e+00)
point( 8.00000000000e+00, 4.5000000000e+00,
0.00000000000e+00)
point( 0.0000000000e+00, 4.5000000000e+00,
0.00000000000e+00)
```

```
@push(move)
*set_move_translations
0 -0.2 0
*move_points
 3 4
# | End of List
*action | figure 7.3
@pop(1)
@push(coord_system)
*system_cylindrical
@pop(1)
*add points
5 15 0
16 15 0
*set grid off
*fill_view
@push(coord_system)
*system_rectangular
@pop(1)
@push(duplicate)
*set_duplicate_translations
0.60
*duplicate_points
 5
 6
# | End of List
@pop(1)
*add_surfaces
 5
 6
 8
 7
*action | figure 7.4
@push(move)
*set_move_translations
0 1.25 0
*move_surfaces
 2
# | End of List
*action | figure 7.5
@pop(1)
```

mesh the tube 8x3 elements mesh the tool 14x6 elements @push(convert) *set_convert_divisions 8 3 *convert_surfaces 1 # | End of List *set_convert_divisions 14 6 *convert_surfaces 2 # | End of List @push(plot_screen) *set_points off *set_point_labels off *set_surfaces off *regenerate *action | figure 7.6 select the marc element type @pop(1) @pop(1) @pop(1) @push(jobs) @push(element_types) @popup(mech_eltype_axisym_solid_pm) *action | figure 7.7 *element_type 10 @popdown(mech_eltype_axisym_solid_pm) all_existing @pop(1) @pop(1) fix the far let side of the tube in axial direction @push(apply) @push(apply_mechanical)

```
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed_disp_pm)
*apply_dof x
@popdown(fixed_disp_pm)
*add apply nodes
 1 10 19 28
# | End of List
*action | figure 7.8
@pop(1)
@pop(1)
 create an artificial connection between the tube
 and the tool by means of a spring
@push(links)
@set($class,spring)
@set($notclass1,tie)
@set($notclass2,servo)
@push(springs dashpots)
*zoom box
*zoom_box(1,0.439909,0.375318,0.540816,0.496183)
*link_class spring *link_value stiffness
10e3
*link_class spring *spring_node 0
131
*link class spring *spring dof 0
1
*link class spring *spring node 1
9
*link_class spring *spring_dof 1
1
*action | figure 7.9
@pop(1)
@pop(1)
 step 2
 define and assign material properties to both tube and tool
*fill view
```

```
@push(mat_props)
*material_type mechanical:isotropic
@popup(mat_isotropic)
*material_value isotropic:youngs_modulus
30e6
*material value isotropic:poissons ratio
.3
*material type plasticity
@popup(iso_plastic_popmenu)
*material_value plasticity:yield_stress
3.6e4
@popdown(iso plastic popmenu)
@popdown(mat_isotropic)
*add material elements
 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24
# | End of List
*identify_materials *regen
*action | figure 7.10
*new material
*material type mechanical:isotropic
@popup(mat isotropic)
*material_value isotropic:youngs_modulus
40e6
*material_value isotropic:poissons_ratio
.3
*material_type plasticity
@popup(iso_plastic_popmenu)
*material_value plasticity:yield_stress
6.0e4
@popdown(iso_plastic_popmenu)
@popdown(mat_isotropic)
*add material elements
 25 26 27 28 29 30 31 32 33 34 35 36 37 38 39 40 41 42 43 44 45 46
47 48 49 50
 51 52 53 54 55 56 57 58 59 60 61 62 63 64 65 66 67 68 69 70 71 72
73 74 75 76
 77 78 79 80 81 82 83 84 85 86 87 88 89 90 91 92 93 94 95 96 97 98
99 100 101
 102 103 104 105 106 107 108
# | End of List
*action | figure 7.11
```

```
@pop(1)
 step 3
 assign the elements of tube and tool to contact bodies
@push(contact)
@push(contact bodies)
*contact_body_name
tube
*contact deformable
@popup(contact deform pm)
@popdown(contact_deform_pm)
*add contact body elements
 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24
# | End of List
*new_contact_body
*contact_body_name
tool
*contact deformable
@popup(contact deform pm)
@popdown(contact_deform_pm)
*add contact body elements
 25 26 27 28 29 30 31 32 33 34 35 36 37 38 39 40 41 42 43 44 45 46
47 48 49 50
 51 52 53 54 55 56 57 58 59 60 61 62 63 64 65 66 67 68 69 70 71 72
73 74 75 76
 77 78 79 80 81 82 83 84 85 86 87 88 89 90 91 92 93 94 95 96 97 98
99 100 101
 102 103 104 105 106 107 108
# | End of List
*identify_contact *regen
*action | figure 7.12
*identify_none *regen
@pop(1)
@pop(1)
 step 4
 define the time dependent loading of the flaring tool
 and apply this to the element edges at the far right
```

```
of the flaring tool
@push(apply)
@push(apply_mechanical)
@push(table)
*set table type
@popup(table_type_pm)
@list(table_type_list,0)
time
@popdown(table_type_pm)
*set table xmax
87
*set_table_xstep
87
*set_table_ymax
2400
*table add
0 0
9 900
39 2400
87 0
*action | figure 7.13
*show model
@pop(1)
*new_apply
*apply_type edge_load
@set($usersubfv,FORCEM)
@popup(apply_edge_load_popmenu)
*apply value p
*apply_table p0
@set($dof,p0)
@popup(apply_select_table_popmenu)
@list(select table list,0)
table1
@popdown(apply_select_table_popmenu)
@popdown(apply_edge_load_popmenu)
*add_apply_edges
 38:1 52:1 66:1 80:1 94:1 108:1
# | End of List
*action | figure 7.14
@pop(1)
```

```
@pop(1)
 define loadcase
@push(loadcase)
@push(lcase mechanical)
*loadcase_type static
@popup(lcase static)
*loadcase_value time
87
*loadcase value nsteps
87
@popdown(lcase_static)
@pop(1)
@pop(1)
 step 5 and 6
   define a job
@push(jobs)
*job class mechanical
@popup(job_mechanical)
@list(job rem lcase list,0)
@list(job_add_lcase_list,0)
*add job loadcases lcase1
@popup(job_an_opts_mech_popmenu)
*job option cdilatation:on
*job_option plasticity:s_strn_rr
*job option plasticity: 1 strn mn add
*job_option cdilatation:on
@popdown(job_an_opts_mech_popmenu)
@list(job add lcase list,0)
@list(job rem lcase list,0)
@popup(tensor_scalar_popmenu)
@list(post tensor list1,0)
@list(post_var_list1,0)
*add_post_tensor stress
*add_post_tensor pl_strain
*add_post_var von_mises
*add_post_var tepl_strain
@popdown(tensor scalar popmenu)
```

```
@list(job_add_lcase_list,0)
@list(job_rem_lcase_list,0)
*job_option dimen:axisym
@popdown(job_mechanical)
@push(file screen)
@set($file browser label,SAVE AS FILE)
@set($file_browser_command,*save_as_model)
@popup(save file browser popmenu)
@list(save_file_browser_filelist,0)
@list(save_file_browser_dirlist,0)
*save as model c07.mud yes
@popdown(save file browser popmenu)
@pop(1)
*update job
@popup(job_run_popmenu)
*submit_job 1
*monitor job
@popdown(job_run_popmenu)
@pop(1)
 step 7
 open the post file
@push(results)
*post_open_default
*zoom box
*zoom_box(1,0.456916,0.460560,0.578231,0.582697)
@push(post history plot)
*action | figure 7.15
*set history nodes
9
#
*history_collect
0 100 1
@push(history_nodes_vars)
@list(hist_node_list,0)
@list(hist var list,0)
*history_add
9
Increment
```

```
Displacement y
*history_fit
@pop(1)
*set_history_increment_id
10
*set_history_xstep
20
*set_history_ystep
20
*set_history_ymax
.5
*history_filled
*action | figure 7.16
@pop(1)
@push(post_history_plot)
*show_model
@pop(1)
@push(plot_screen)
@push(plot1_screen)
*edges outline
@pop(1)
*set_nodes off
*set faces off
*regenerate
*fill view
@pop(1)
*post_rewind
*set_deformed both
@push(results2)
*post_animate_increments
100
1
@push(animation_screen)
*fill_view
*animation play
*show model
@pop(1)
@pop(1)
*post_skip_to
39
*post_value
```

```
@popup(post_scalar_pm)
@list(post_scalar_list,0)
@list(post_scalar_list,1)
@list(post_scalar_list,22)
Equivalent Stress
@popdown(post_scalar_pm)
*post_contour_bands
*action | figure 7.17
*post_value
@popup(post_scalar_pm)
@list(post_scalar_list,22)
Equivalent Plastic Strain
@popdown(post_scalar_pm)
*action | figure 7.18
@pop(1)
```



Chapter Overview

This chapter demonstrates the modeling and analysis of the bottom of an aluminum container under internal pressure. The particular configuration of this container bottom leads to a snap-through problem. Accurate modeling of the geometry is essential since it dramatically influences the snap-through process.

The primary goal of this chapter is to show you three Mentat functionalities.

- Using nonlinear analysis to solve a snap-through analysis problem
- Using the TABLES option to specify input data that changes with time, plastic strain, etc.
- Animating the results of an analysis



Background Information

Description

The container, a soft drink can, is assumed to be a circle cylinder with a radius of 1.3 inches and a total height of 4.8 inches. The container* is made out of aluminum and has a wall thickness of 0.025 inches



Figure 8-1 Aluminum Container

^{*} Note: At times the container may be referred to as *can* in the text.



Idealization

The geometry of this problem is fairly simple due to two factors. The first is that the geometry and the loading of the container are axisymmetric and allow you to perform an axisymmetric analysis. The second factor is that the focus of the analysis is restricted to the phenomena that occur at the bottom of the container. In this analysis, the height of the container, h, is limited to a length where the edge effects are damped out. The theory behind this assumption is explained below.

If $h = 2.5 \sqrt{rt}$

where r = the radius of the container,

t = the wall thickness,

the solution decreases to about 4% of its value at the bottom edge. In this example, it means you can safely ignore the influence of the top edge since the critical height, h, is equal to 0.4519, calculated as follows:

 $h = 2.5\sqrt{1.307 \times 0.025} = 0.4519$

An awareness of this decay distance is very important in numerical calculations. If you wish to correctly capture the behavior of the solution in the edge region, the typical finite element size must be small in comparison to the decay distance.





Figure 8-2 Section of Container to be Analyzed

Requirements for a Successful Analysis

Nonlinear problems that involve buckling or snap-through are prime candidates for displacement controlled incremental strategies. Unfortunately, the problem at hand is a load controlled problem. In order to able to traverse the load versus displacement curve of a point on the bottom of the container you must use a loading pattern such that the load increment is scaled in size and applied in the correct direction. The arc-length method combined with a Newton-Raphson iterative scheme will guarantee you that the entire load displacement curve can be traversed. Needless to say, the solution of this problem consists of large displacements and finite strains.



Full Disclosure

- Analysis Type Nonlinear snap-through.
- Element Type MARC Type 89, axisymmetric shell.

Material Properties

Aluminum with workhardening. Isotropic with Young's Modulus = 11.0e6 p.s.i and Poisson's Ratio = 0.3.

The stress-strain data used to define the workhardening of the aluminum is listed in Table 8-1 and graphically represented in Figure 8-3.

Log Plastic Strain (x)	Cauchy Stress (y)	Total Engineering Strain
0.0	42000.0	0.0038
0.001748	44577.0	0.0057
0.003494	45157.0	0.0075
0.06766	63665.0	0.0755
0.09531	70950.0	0.1058
0.1570	81315.0	0.1763
0.2070	88560.0	0.2365
0.2623	95216.0	0.3066

Table 8-1	Stress	Strain	Data
	01000	ouani	Duiu



Figure 8-3 Graphical Representation of Cauchy Stress vs. Logarithmic Plastic Strain Data


Overview of Steps

- **Step 1** Input all arcs according to the measurements specified in Figure 8-2.
- **Step 2** Input straight lines to connect the arcs.
- **Step 3** Convert the geometric entities to finite elements.
- **Step 4** Use SWEEP to eliminate all duplicate nodes, then switch the element class to quadratic shell elements and attach the midside nodes to the curves.
- **Step 5** Add kinematic boundary conditions to enforce the symmetry and restrain rigid body motion.
- Step 6 Specify edge loads.
- Step 7 Rectify connectivity to ensure consistent normals.
- **Step 8** Add material properties.
- Step 9 Add geometric properties.
- Step 10 Define the loadcase.
- Step 11 Submit the job.
- Step 12 Postprocess the results by looking at the deformed shape and the load displacement curve of the node located on the symmetry axis.

Detailed Session Description

Step 1

A structure that is modeled with axisymmetric elements requires the global x-axis to point into the axial direction of that structure. As a result of this type of modeling, the container will be displayed in a horizontal position.

As in Chapter 7, this sample session demonstrates the use of the geometric meshing technique. The geometric entities used to create the mesh are two types of curves: arcs and lines. Once you have generated the geometric model, the arcs and lines are converted to finite elements. Refer to Chapter 3 for more information on mesh generation techniques.

Use the Center/Radius/Angle(begin)/Angle(end) arc type (CRAA) to create the first arcs of the geometry. Use the following button sequence to select and add the CRAA arc type. The values for the measurements of the arcs are given in Figure 8-2.

```
MAIN
  MESH GENERATION
     CURVE TYPE
        CENTER/RADIUS/ANGLE/ANGLE
           RETURN
        crvs ADD
           0 0 0
                                                                    (center)
           2.345
                                                                    (radius)
           0 22
                                                                (angle limits)
           2.026 1 0
                                                                    (center)
           0.063
                                                                    (radius)
           129 265
                                                                (angle limits)
           2.0 1.2 0
                                                                    (center)
           0.125
                                                                    (radius)
           296 334
                                                                (angle limits)
           2.3 1.127 0
                                                                    (center)
           0.18
                                                                    (radius)
           90 162
                                                                (angle limits)
```

Switch on the labeling of points.

MAIN FILL PLOT label POINTS REGENERATE

(*on*)





Figure 8-4 Using CRAA Type Arcs to Create First Four Curves

The next step is to add a new arc (number 5) so that it is tangent to arc 1, the lower arc, using the following button sequence on the next page:



Figure 8-5 Using TRA Type Arc to Create Arc 5



Finally, add the straight lines to complete the geometric description of the model. Set the curve type to LINE. Use the crvs ADD button on the mesh generation panel and click on the existing points that need to be connected. As we noted in the section on *Idealization*, page 8-3, it is necessary to extend the wall of the cylinder to at least 0.4519 inches from the edge to ensure that the edge effects are negligible.

```
MAIN
MESH GENERATION
CURVE TYPE
LINE
RETURN
crvs ADD
11 14
4 9
8 17
```

(Click on points to connect) (Click on points to connect) (Click on points to connect)

The origin chosen for this problem is 2.354 inches to the left of point 1 of arc 1. The total extent of the wall necessary is therefore 2.354 + 0.4519; that is, approximately 3 inches. Therefore, we will add a point at 3.0 1.307 0.0.

```
MAIN
MESH GENERATION
pts ADD
3.0 1.307 0.0
FILL
crvs ADD
12 18
```

(Click on points to connect)



Figure 8-6 Connect the Arcs by Lines

Use the **CONVERT** processor, located on the mesh generation panel, to convert the geometric entities (in this case, curves) to finite elements. You must specify the number of elements for each curve. A higher mesh density is required at sections of high curvature and large displacements than in regions where the values for stress and strain are expected to be less severe. For this reason, you must specify a larger number of convert divisions for those arcs of high curvature and large displacements as indicated in the button sequence given below. Figure 8-7 shows the result of converting the curves to finite elements.



MAIN PLOT label POINTS (off) label CURVES (on)REGENERATE RETURN **MESH GENERATION** CONVERT **DIVISIONS** 8 1 (Number of subdivisions) **CURVES TO ELEMENTS** 1 9 END LIST (#) DIVISIONS 6 1 **CURVES TO ELEMENTS** 4 2 END LIST (#) DIVISIONS 4 1 **CURVES TO ELEMENTS** 6 3 END LIST (#) DIVISIONS 3 1 **CURVES TO ELEMENTS** 7 5 8 END LIST (#)

Step 4

The previous operations may have left duplicate nodes; that is nodes with different identification numbers but occupying the same space. In finite element terms, these nodes are not connected which may introduce undesirable mechanisms in the structure.



Figure 8-7 Model after Converting Curves to LINE(2) Elements

Use the **SWEEP** processor introduced to you in the sample session of Chapter 4 to eliminate the duplicate nodes that occupy the same location. Since this involves a comparison of real numbers that cannot be done exactly in a computer, nodes are swept together if they are within a certain tolerance from each other. This tolerance can be changed from its default value. Be careful when adjusting the tolerance: too large a tolerance can collapse the entire structure into a single point.

MAIN MESH GENERATION SWEEP sweep NODES all: EXIST. In order to describe the curved geometry as precise as possible, the linear (LINE (2) elements) will be converted to elements with a quadratic interpolation function (LINE (3) elements).

```
MAIN
MESH GENERATION
CHANGE CLASS
LINE (3)
ELEMENTS
all: EXIST.
```

The mid-side nodes will be positioned on the straight line that connects the nodes of the original LINE(2) element. In order to take full advantage of the higher interpolation function the mid-side nodes will have to be positioned on the arcs. This is done using the **ATTACH** processor, more in particular the ATTACH NODES TO CURVE option.

The procedure requires that the user zooms in on each arc, activates the proper attach option, selects the curve and identifies the nodes to be attached. Non-attached nodes can be recognized by the square marker while attached nodes are drawn as circles.

MAIN								
MESH G	BENE	RATIC	ON					
ATTA	ACH							
at	tach r	nodes	CUR	VE				
	1							(Pick curve)
	54	55	56	57	58	59	60	61 (<i>Pick nodes</i>)
								(these are the mid-side nodes in the
								definition area of curve #1)
	FΝΓ) I IST	(#)					
	5		(")					(Pick curve)
	92	91	90					(Pick nodes)
	FNΓ) ÎST	· (#)					(Tren notes)
	2		(")					(Pick curve)
	70	71	72	73	74	75		(Pick nodes)
	FNL	, ist	· (#)	15	/ 1	15		(Tree notes)
	2		(#)					(Pick curve)
	5 0 0	02	01	0 5				(Pick rodes)
		ου τοιιοτ	· (#)	00				(1 ick hodes)
			(#)					(Dick course)
	4	0.0		80				(Pick curve)
		80	- /9	/8	11	/6		(Pick nodes)
	ENL	LISI	(#)					

The situation prior to and after the attach operation on curve 2 are shown in Figure 8-8 and Figure 8-9.



Figure 8-8 Situation prior to Attach to Curve 2



Figure 8-9 Situation after Attach to Curve 2



In an axisymmetric shell analysis, there are three types of applicable displacements or degrees of freedom (d.o.f.): axial, radial, and rotational. The axial d.o.f. is represented as a global x, and the radial as a global y.

For this particular model, the boundary conditions are simple. This is due to the symmetry conditions applied to the center line node through the suppression of radial displacement and in-plane rotation.

MAIN	
PLOT	
label CURVES	(off)
draw CURVES	(off)
draw POINTS	(off)
RETURN	
BOUNDARY CONDITIONS	
MECHANICAL	
NEW	
FIXED DISPLACEMENT	
ON y displace	<i>(on)</i>
ON z displace	(see note*) (on)
OK	
nodes ADD	
1	(Pick the lower left node)
END LIST (#)	
FILL	

^{*} Note: The buttons x displace, y displace, z displace, x rotation, y rotation, and z rotation refer to the 6 d.o.f. that generally exist for a node of a 3-D shell element. However, this problem uses an axisymmetric shell element with the following 3 d.o.f.: displacement in x-direction, displacement in y-direction, and rotation about the z-axis. In such cases, the button x displace refers to the displacements in x-direction, y displace refers to displacements in y-direction and z displace refers to the 3rd degree of freedom, the rotation about the z-axis.



Figure 8-10 shows you the model with the boundary conditions added.

Figure 8-10 Fixed Displacement at Axis of Rotation

Next, suppress two degrees of freedom of the extreme node at the circumference of the can: 1) suppress the movement in the axial direction, and 2) suppress the rotational degree of freedom.

```
MAIN
BOUNDARY CONDITIONS
MECHANICAL
NEW
FIXED DISPLACEMENT
ON x displace (on)
ON z displace (on)
OK
nodes ADD
18 (Pick the top right node)
END LIST (#)
```



Figure 8-11 Boundary Conditions at the Circumference of Container



To specify the loading sequence, use the TABLES option to create the load table using the following button sequence:

```
MAIN
  BOUNDARY CONDITIONS
     MECHANICAL
       TABLES
          TABLE TYPE
             time
             OK
          XMIN
             0
          XMAX
             1
          YMIN
             0
          YMAX
             500
          NAME
             loading
          ADD POINT
             0 0
                500
             1
          label X-AXIS
             time
          label Y-AXIS
             pressure
```

You will refer to the table name, loading, when you apply the edge loads. The xmin, xmax, ymin and ymax values specify the table limits. The 0 to 1 range is for the x value and 0 to 500 is the range for the y value. The x-axis represents the time (which is to be regarded a dummy variable for this analysis) and the y-axis represents the pressure load. The loading pattern is specified as zero edge load at time 0 and an edge load of 500 at time 1. Since the *total loadcase time*, used for quasi static analysis (LOADCASE menu), will be set to 1.0, this table will result in a total load of 500 to be reached at the end of the loadcase. The table points can be entered either via the keyboard or by using the mouse to pick the (0, 0.0) point in the graph followed by the (1, 500.0) point.



Figure 8-12 Load History Table

As mentioned on page 8-4, it is the task of the analysis program to define a load incrementation that reaches the target load.

Now that the load type and the load path have both been defined, use the following button sequence on the next page to specify where to apply this load.

MAIN BOUNDARY CONDITIONS MECHANICAL NEW EDGE LOAD PRESSURE 1 pressure TABLE loading OK OK edges ADD all: EXIST. FILL

The actual load applied to the structure is 1 (the base value entered at the pressure prompt), multiplied by the values defined in the table.



Figure 8-13 Pressure Load Applied



Figure 8-13 clearly indicates the pressure load has not been applied in the correct direction for all elements. This is caused by the way the curves were created. The outward normal that determines the positive direction of the load is directly dependent on which point of the arc was defined first. Figure 8-14 depicts this dependency on an element that has a 1-2 connectivity.



Figure 8-14 The Outward Normal in Arc Definition

The connectivity of the elements can easily be corrected using the following button sequence.

```
MAIN
MESH GENERATION
CHECK
FLIP ELEMENTS
END LIST (#)
```

(Pick the elements to be flipped)

You may want to use the ZOOM option for closer view of the areas where the flipped elements are located to make it easier for you to pick the elements that have been loaded in the opposite direction. Pick the elements by moving the cursor over each element and clicking $\langle ML \rangle$. Don't forget to specify end of list by clicking $\langle MR \rangle$ in the graphics area when you have picked all the flipped elements. Click on FILL to rescale the model to fill the graphics area if you have used the ZOOM option. Figure 8-15 shows the model with corrected loading.



Figure 8-15 Correctly Directed Loads for all Elements



The specific values for the isotropic material specification can be entered using the following sequence of buttons:

```
MAIN
MATERIAL PROPERTIES
ISOTROPIC
YOUNG'S MODULUS
11.0e6
POISSON'S RATIO
0.3
OK
```

The stress-strain data of the material requires the use of the TABLES option similar to Step 6 when you added the edge loads. Remember that the table values are multiplied by the base value as was explained before in the section on specification of the pressure load. The table name is specified by clicking on the NAME button followed by typing in workhard as the name of the table. You will use this table name later. The values for the plastic strain and stress are listed in Table 8-1. Note that based on the solution procedure (i.e. large displacements, updated Lagrange procedure, finite strain plasticity), this data must be of the form listed below.

Procedure	Stress	Strain
Default	Engineering	Engineering
Large Displacements	2nd Piola-Kirchhoff	Green-Lagrange
Large Strain Plasticity	True (Cauchy)	Logarithmic

The following button sequence is used to define the stress-strain data table.

```
MAIN
  MATERIAL PROPERTIES
     TABLES
     NEW
       TABLE TYPE
          plastic_strain
          OK
       XMIN
          0
       XMAX
          0.5
       YMIN
          0
       YMAX
          1.0e5
       NAME
          workhard
       label X-AXIS
          log_strain
       label Y-AXIS
          true_stress
       ADD POINT
       SHOW MODEL
```

RETURN

(Refer to plastic strain stress data in Table 8-1 on page 8-5)



Figure 8-16 Workhardening Slope

Plasticity may occur due to the extreme loading. Click on the PLASTICITY button which shows the plasticity properties panel. Choose the defaults von Mises yield criteria and the isotropic hardening rule. The initial yield stress is set to 1, which means that the table values are multiplied by a factor of 1.

```
MAIN

MATERIAL PROPERTIES

ISOTROPIC

PLASTICITY

INITIAL YIELD STRESS

1

initial yield stress TABLE 1

workhard

OK

OK

OK

elements ADD

all: EXIST.
```



Specify the element thickness for all elements using the following button sequence:

```
MAIN
GEOMETRIC PROPERTIES
AXISYMMETRIC
SHELL
THICKNESS
0.025
OK
elements ADD
all: EXIST.
```

Step 10

Now that you have defined the individual loads and kinematic constraints, define a loadcase that combines these boundary conditions.

A pop-up menu appears over the graphics area containing a list of available boundary conditions and their status (selected or not). Combine these individual loads in a loadcase that can be referred to by a name. The default name for this loadcase is *lcase1*. Clearly you want all kinematic constraints and distributed loads (pressures) to be applied so all boundary conditions must be selected. Use the SELECT and DESELECT buttons to activate and deactivate the boundary conditions respectively. Confirm the correctness of this loadcase definition by clicking on the OK button.

```
MAIN
LOADCASE
mechanical analyses STATIC
LOADS
OK
```

The following individual components for an analysis have already been specified:

- **1.** You defined the topology and connectivity of the finite element model.
- 2. You assigned boundary conditions, material properties, and geometric properties.
- **3.** You combined the boundary conditions in a loadcase.

Since snap-through is likely to occur in this problem the user has to instruct MARC to solve a system of equations with a non-positive definite tangent stiffness matrix.

MAIN LOADCASE mechanical analyses STATIC SOLUTION CONTROL NON-POSITIVE DEFINITE OK

(*on*)

Furthermore, the default settings for convergence testing are not well suited for this particular problem. Although the default type of testing, relative testing on residual forces, is appropriate, the relative force tolerance needs to be reduced. The necessity of this action can be explained by looking at the boundary conditions: the constraint on the axial displacement degree of freedom is found at a large radius (node 18). Due to the internal pressure we will find a large reaction-force at this node. Allowing a certain percentage of this reaction-force to be present as residuals anywhere in the structure will result in undesired interference of those residuals with the automatic load stepping process.

```
MAIN
LOADCASE
mechanical analyses STATIC
CONVERGENCE TESTING
RELATIVE FORCE TOLERANCE
0.05
OK
```

Finally, the adaptive load stepping algorithm of MARC will be activated. This algorithm allows for the analysis of snap-through phenomena in which the load incrementation needs to be scaled depending on the amount of nonlinearity that is occurring. Various parameters control this procedure. In this case, we will allow for a maximum of 600 increments. In the first increment 0.05 (5%) of the total load will be applied. Also, the user needs to specify that the arc length will never exceed the value used in the first increment. The default MODIFIED RIKS procedure will be used.

```
MAIN

LOADCASE

mechanical analyses STATIC

mech. load (arc length) PARAMETERS

MAX # INCREMENTS

600

INITIAL FRACTION

0.05

MAX ARC LENGTH MULTIPLIER

1.0

OK
```

Step 11

It is time to prepare the loadcase for a job and to submit it for finite element analysis.

Prior to defining the job parameters, the appropriate MARC element type is set.

Next, the analysis class MECHANICAL is activated, resulting in a pop-up menu over the graphics area. Click the SELECT button and pick the *lcase1* button from the available loadcases list to select the only available loadcase for this job.

```
MAIN

JOBS

ELEMENT TYPES

mechanical elements AXISYMMETRIC SHELL

89 (LINE 3 / thick shell)

OK

all: EXIST.

RETURN

MECHANICAL

loadcases SELECT

Icase1
```

As we have indicated before, this analysis involves large displacements, finite strain plasticity, updated Lagrange procedure, and follower forces. The finite element program requires directives that indicate this. From the mechanical analysis pop-up menu, click on the ANALYSIS OPTIONS button and activate the following options:

```
MAIN

JOBS

MECHANICAL

ANALYSIS OPTIONS

SMALL STRAIN-MEAN NRM

SMALL STRAIN-RADIAL R

(Switch on large strain radial return procedure)

NO FOLLOWER FORCE

(to switch to FOLLOWER FORCE)

OK
```

The results of the analysis will appear in a results file. Specify the results variables you are interested in by clicking on the JOB RESULTS button from the mechanical analysis pop-up menu. Select Equivalent Plastic Strain and Equivalent von Mises Stress variables. Finally, set the numbers of layers used for integration through the shell thickness to 5.

```
MAIN
JOBS
MECHANICAL
JOB RESULTS
scalars
von_mises
OUT
tepl_strain
OUT
OK
JOB PARAMETERS
# SHELL/BEAM LAYERS
5
```

This analysis may involve a large number of increments. For this reason, you may want to write the results every 10 increments using the FREQUENCY button. For this sample session, however, write every increment which is the default value of the FREQUENCY option and confirm the settings by clicking on the OK button.

The following button sequence submits the job. The job can be monitored using the MONITOR option which, in case of automatically running the procedure file, prevents Mentat from proceeding to Step 12 before the analysis has run to completion.

```
MAIN
JOBS
CHECK
SAVE
RUN
SUBMIT 1
MONITOR
```

This analysis takes a few minutes, depending on the power of the host to which you are submitting the job.



The results of the analysis step are stored in a disk file. To access the results, it is necessary to open this file and (selectively) extract data from it. Use the following button sequence to open the file.

MAIN RESULTS OPEN DEFAULT

Click on the FILL button located in the static menu area to scale the model to fill the graphics area. The following button sequence removes the node labeling and gives you a clearer picture of the model shown in Figure 8-15.

MAIN FILL PLOT draw NODES REGENERATE RETURN

(off)

We are interested in the deformed shape as a function of the increasing/decreasing load. The best way to obtain a good overview of the deformations is to animate the deformed shape. Since it is impractical to incorporate the animated picture in this guide, a detailed description follows on how to obtain and play back the animation frames. Only 3 out of a sequence of 25 frames are shown in this guide.

Before you collect the animation frames, click on the DEF & ORIG button so that the deformed and original shapes of the model are shown simultaneously.



Figure 8-17 Resulting Post File of the Container

MAIN RESULTS NEXT INC DEF & ORIG MORE animate INCREMENTS 25 10

(Number of frames to save) (Increment step size)

The program responds by scanning the results file and extracting the appropriate data.

When replaying the sequence of animation files, you may have to scale the deformed and original models to fit in the graphics area. Use the FILL button located in the static menu to scale the models while still having the last increment of the animation sequence displayed. Use the following button sequence to play the animation sequence.

MAIN FILL RESULTS MORE ANIMATION FILL PLAY SHOW MODEL

The following two figures capture 3 of the 25 animation frames.

```
MAIN
RESULTS
SKIP TO INC
80
SKIP TO INC
160
SKIP TO INC
240
```

Figure 8-18 and Figure 8-19 show you that the bottom ridge first un-rolls and is followed by a snap-through of the arch. Ultimately, the shape of the bottom of the can will become spherical as is shown in Figure 8-20.



Figure 8-18 Original and Deformed Model at Increment 80

The user can use the BEAM CONTOURS or BEAM VALUES plot option to display elemental quantities. Alternatively, a path plot where the position of the nodes is plotted on the x-axis and the equivalent plastic strain is plotted on the y-axis, can be created. The nodes are logically connected though the connectivity of the elements. The plot shown in Figure 8-21 displays the equivalent plastic strain for increment 240 of this analysis.

Use the following button sequence on page 8-41 to plot the total equivalent plastic strain.



Figure 8-19 Original and Deformed Model at Increment 160



Figure 8-20 Original and Deformed Model at Increment 240

Container

MAIN PLOT draw NODES (on)REGENERATE RETURN RESULTS PATH PLOT NODE PATH (select nodes from lower left to upper right, approximately each 10th node will do) END LIST (#) VARIABLES ADD CURVE Arc Length **Total Equivalent Plastic Strain Layer 3**



Figure 8-21 Path Plot of Equivalent Plastic Strain



It is possible to monitor these diagrams and obtain an overview of the location and degree of plastic strain as a function of the loading.

Finally, a diagram is given that indicates how the bottom collapses as a function of the total load that was (adaptively) put onto the structure. The total load is represented by the reaction force in x-direction of node 18 which is at the end of the line segment. The displacements of the bottom are represented by the axial displacements of node 1 which is at radius 0.0.

The user can use the following button sequence to generate this figure:

```
MAIN
  RESULTS
    HISTORY PLOT
       SHOW MODEL
       SET NODES
         18
         1
         END LIST (#)
       COLLECT DATA
          0 1000 1
       NODES/VARIABLES
         ADD 2-NODE CRV
            1
            Displacement x
            18
            DOWN
            Dist Load 1
         FIT
         RETURN
       SHOW IDS
         10
       YMAX
          5000.0
```

(upper right) (lower left)


Figure 8-22 Axial Displacement of Bottom versus Applied Pressure Diagram

Conclusion

The loading path versus displacement has been successfully traced for the snap-through analysis of the bottom of an aluminum container.



Procedure File

```
| Version : Mentat 3.3
*alias action
*exec_p gen_image.proc
*action | figure 8.0 - dummy
*action | figure 8.1 - dummy
*action | figure 8.2 - dummy
*action | figure 8.3 - dummy
@top()
 step 1
@push(meshgen)
@push(curve_type)
*set_curve_type arc_craa
@pop(1)
*add_curves
0 0 0
2.345
0 22
2.026 1 0
0.063
129 265
2.0 1.2 0
0.125
296 334
2.3 1.127 0
0.18
90 162
*fill view
@push(plot_screen)
*set point labels on
*regenerate
*action | figure 8.4
@pop(1)
@push(curve_type)
*set_curve_type arc_tra
@pop(1)
*add_curves
```

```
3
0.05
50
*action | figure 8.5
 step 2
@push(curve_type)
*set_curve_type line
@pop(1)
*add curves
11 14
 4 9
 8 17
*add points
3.0 1.307 0
*fill_view
*add curves
12 18
# | End of List
*action | figure 8.6
 step 3
 generate elements based on the geometric entities
@push(plot_screen)
*set_point_labels off
*set curve labels on
*regenerate
@pop(1)
@push(convert)
*set_convert_divisions
8 1
*convert curves
19
# | End of List
*set_convert_divisions
6 1
*convert_curves
 4 2
```

```
# | End of List
*set_convert_divisions
4 1
*convert_curves
 63
# | End of List
*set_convert_divisions
3 1
*convert_curves
 7 5 8
# | End of List
@pop(1)
*action | figure 8.7
@push(sweep)
*sweep nodes
all_existing
@pop(1)
@push(change_class)
*set_change_class line3
*change elements
all_existing
@pop(1)
@push(attach)
*attach nodes curve
 1
 54 55 56 57 58 59 60 61
# | End of List
*zoom_box
*zoom box(1,0.236961,0.286260,0.328798,0.430025)
*attach_nodes_curve
*attach_nodes_curve
 5
 92 91 90
# | End of List
*fill view
*zoom box
*zoom_box(1,0.133787,0.161578,0.225624,0.344784)
*action | figure 8.8
*attach_nodes_curve
 2
 70 71 72 73 74 75
```



```
# | End of List
*action | figure 8.9
*fill view
*zoom box
*zoom_box(1,0.226757,0.006361,0.382086,0.199746)
*attach_nodes_curve
 3
82 83 84 85
# | End of List
 4
81 80 79 78 77 76
# | End of List
*fill_view
@pop(1)
@pop(1)
 step 5
@push(plot_screen)
*set curve labels off
*set curves off
*set_points off
*regenerate
@pop(1)
@push(apply)
@push(apply_mechanical)
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed disp pm)
*apply_dof y
*apply_dof z
@popdown(fixed_disp_pm)
*add_apply_points
*add_apply_nodes
 1
# | End of List
*fill_view
*action | figure 8.10
*new_apply
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
```

```
@popup(fixed_disp_pm)
*apply_dof x
*apply_dof z
@popdown(fixed_disp_pm)
*add_apply_nodes
 18
# | End of List
*action | figure 8.11
@push(table)
*set_table_type
@popup(table_type_pm)
@list(table type list,0)
time
@popdown(table_type_pm)
*set table xmin
0
*set table xmax
1
*set_table_ymin
0
*set table ymax
500
*table name
loading
*table add
0 0
1 500
*set_table_xname
time
*set_table_yname
pressure
*action | figure 8.12
*fill view
@pop(1)
*new apply
*apply_type edge_load
@set($usersubfv,FORCEM)
@popup(apply_edge_load_popmenu)
*apply_table p0
@set($dof,p0)
@popup(apply_select_table_popmenu)
```

```
@list(select_table_list,0)
loading
@popdown(apply_select_table_popmenu)
@popdown(apply_edge_load_popmenu)
*add_apply_edges
all_existing
*action | figure 8.13
@pop(1)
@pop(1)
@push(meshgen)
@push(check_elements)
*flip elements
 90
 88
 89
 66
 67
 64
 65
 63
 62
# | End of List
 72
 73
 70
 71
 69
 68
# | End of List
*fill_view
*action | figure 8.14 - dummy
*action | figure 8.15
@pop(1)
@pop(1)
  step 8
  material properties
@push(mat_props)
*material_type mechanical:isotropic
```

```
@popup(mat_isotropic)
*material_value isotropic:youngs_modulus
11e6
*material_value isotropic:poissons_ratio
.3
@popdown(mat_isotropic)
@push(table)
*new table
*set_table_type
@popup(table_type_pm)
@list(table_type_list,0)
plastic strain
@popdown(table_type_pm)
*set table xmin
0
*set_table_xmax
0.5
*set_table_ymin
0
*set table ymax
100000
*table_name
workhard
*set_table_xname
log_strain
*set_table_yname
true stress
*table add
0 42000
1.748e-3 44577
3.494e-3 45157
6.766e-2 63665
9.531e-2 70950
.1570 81315
.2070 88560
.2623 95216
*action | figure 8.16
*show model
@pop(1)
*material_type mechanical:isotropic
@popup(mat_isotropic)
```

```
*material_type plasticity
@popup(iso_plastic_popmenu)
*material_value plasticity:yield_stress
1
*material_table plasticity:yield_stress0
@set($dof,plasticity:yield_stress0)
@popup(material_select_table_popmenu)
@list(select table list,0)
workhard
@popdown(material_select_table_popmenu)
@popdown(iso_plastic_popmenu)
@popdown(mat isotropic)
*add_material_elements
all existing
@pop(1)
  step 9
  specify the thickness of the shell elements
@push(geom props)
@push(geometry_mech_axisym)
*geometry type mech axisym shell
@popup(geom_mech_axi_shell_pm)
*geometry_value thick
0.0250
@popdown(geom_mech_axi_shell_pm)
*add_geometry_elements
all existing
@pop(1)
@pop(1)
  step 10
@push(loadcase)
@push(lcase mechanical)
*loadcase_type static
@popup(lcase static)
@popup(select_loads_popmenu)
@list(lcase_apply_list,0)
@popdown(select loads popmenu)
```

```
@popup(mech_solution_control)
*loadcase_option nonpos:on
@popdown(mech_solution_control)
@popup(mech_convergence_popmenu)
*loadcase option converge:force only
*loadcase option converge:residuals
*loadcase_value force
0.05
@popdown(mech_convergence_popmenu)
*loadcase_option stepping:arclength
*loadcase option stepping:arclength
@popup(arclength popmenu)
*loadcase_value maxinc
600
*loadcase value initfraction
0.05
*loadcase value maxmultiplier
1
@popdown(arclength_popmenu)
@popdown(lcase static)
@pop(1)
@pop(1)
 step 11
 define a mechanical analysis and submit the job
@push(jobs)
@push(element types)
@popup(eltype_axisym_pm)
*element_type 89
@popdown(eltype_axisym_pm)
all existing
@pop(1)
*job class mechanical
@popup(job_mechanical)
@list(job_rem_lcase_list,0)
@list(job add lcase list,0)
*add_job_loadcases lcase1
@popup(job_an_opts_mech_popmenu)
*job option plasticity:s strn rr
```

```
*job_option plasticity:l_strn_mn_add
*job_option follow:on
@popdown(job_an_opts_mech_popmenu)
@list(job_add_lcase_list,0)
@list(job rem lcase list,0)
@popup(tensor_scalar_popmenu)
@list(post_tensor_list1,0)
@list(post var list1,0)
*add_post_var von_mises
*add_post_var tepl_strain
*post_var_outer_layers tepl_strain
*post var outer layers von mises
@popdown(tensor_scalar_popmenu)
@list(job add lcase list,0)
@list(job rem lcase list,0)
@popup(job_param_mech_popmenu)
*job param layers
5
@popdown(job_param_mech_popmenu)
@list(job rem lcase list,0)
@popdown(job_mechanical)
@push(file_screen)
@set($file browser label,SAVE AS FILE)
@set($file_browser_command,*save_as_model)
@popup(save_file_browser_popmenu)
@list(save_file_browser_filelist,0)
@list(save file browser dirlist,0)
*save_as_model c08.mud yes
@popdown(save file browser popmenu)
@pop(1)
*update job
@popup(job_run_popmenu)
*submit job 1
*monitor_job
@popdown(job run popmenu)
@top()
@push(results)
*post open default
@push(plot_screen)
*set_nodes off
*regenerate
```

```
*action | figure 8.17
  animation sequence
@pop(1)
@push(results2)
*post_animate_increments
40
@pop(1)
*set_deformed both
*post_next
@push(results2)
*post_animate_increments
40
10
*fill_view
@push(animation_screen)
*fill view
*animation_play
*fill view
*animation_play
*show_model
@pop(1)
@pop(1)
*post_skip_to
80
*action | figure 8.18
*post_skip_to
160
*action | figure 8.19
*post_skip_to
240
*action | figure 8.20
@push(plot_screen)
*set nodes on
*regenerate
@pop(1)
@push(post_pathplot)
*set_pathplot_path
 1
 18
```

```
# | End of List
@push(pathplot_vars)
@list(path_var_list,0)
*pathplot_add
Arc Length
Total Equivalent Plastic Strain Layer 3
*pathplot_fit
*action | figure 8.21
@pop(1)
*show_model
@pop(1)
@push(post_history_plot)
*set_history_nodes
 18 1
# | End of List
*history_collect
0 1000 1
@push(history_nodes_vars)
@list(hist_node_list,0)
@list(hist var list,0)
*history_add
1
Displacement x
Dist Load 1
*history_fit
@pop(1)
*set_history_increment_id
10
*action | figure 8.22
```





Chapter Overview

This chapter demonstrates the analysis of a region where one cylinder penetrates another cylinder of a larger radius and where the structure is loaded by an internal pressure. The radius thickness ratio of the structure warrants the use of shell theory instead of a full three-dimensional analysis using hexahedral elements.

The objective of this chapter is to highlight the following three Mentat capabilities.

- Generation of a cylinder-cylinder intersection.
- Application of face loads.
- Display of results in a contour plot.



Background Information

Description

In this session you analyze a cylindrical pressure vessel penetrated by an off-centered manhole. The diameter of the vessel is 168 inches and the plate thickness is 0.54 inches. The manhole has a diameter of 48 inches and a plate thickness of 1.0 inches. The dimensions of the structure are shown in Figure 9-1.



Figure 9-1 Vessel Dimensions





The manhole is positioned 45 inches from the center line as indicated by Figure 9-2.

Figure 9-2 Side View of Vessel



Idealization

Only a portion of the vessel needs to be modeled due to symmetry and to the way arched structures respond to localized loads. The thickness/radius ratio is small enough that it allows you to use the shell approximation instead of a full three-dimensional analysis. As the focus of this analysis is on the response in the vicinity of the penetration, the mesh can be limited to the portion of the structure shown in Figure 9-3.

It can be theoretically proven that for a material with a Poisson's ratio of 0.3, when measured at a distance $2.5\sqrt{rt}$ removed from the edge, the influence of the penetration is reduced to 4% of the value at the edge. Here *r* is the radius and *t* the thickness. For this particular case, the decay distance is 16.84 inches.

Therefore, the boundary conditions can be applied at the shell edge without affecting the stresses at the vessel-manhole intersection. Due to symmetry it is sufficient to analyze half the vessel section shown in Figure 9-3.





Figure 9-3 Section to be Analyzed



Requirements for a Successful Analysis

The analysis is considered successful if the localized stresses are known at the intersection of the two cylinders. The decay distance of 16.84 inches is assumed to be valid and therefore, the stresses at the edge of the analyzed structure should be less than 4% of the peak value.

Full Disclosure

Analysis Type Linear static.

Element Type MARC element type 75, four-noded shell element.

Material Properties Steel Young's Modulus = 30e6 psi Poisson's Ratio = 0.3



Overview of Steps

Step 1	Create two cylindrical	surfaces: one	for the vessel	l and one for the	manhole.
--------	------------------------	---------------	----------------	-------------------	----------

- **Step 2** Convert the surface of the vessel into a finite element mesh.
- **Step 3** Remove the elements in the vicinity of the manhole, creating a hole in the surface of the vessel.
- **Step 4** Attach the nodes of the circumference of the hole to the intersection of the vessel and the manhole surfaces.
- **Step 5** Redistribute the nodes on the perimeter of the hole.
- **Step 6** Add line elements to the circumference of the circular hole.
- **Step 7** Drag the line into shell elements thus creating the manhole.
- **Step 8** Attach the rim of the manhole to a flat surface.
- **Step 9** Subdivide the top row of elements of the manhole to improve the element aspect ratio of these elements.
- Step 10 Sweep the entire mesh to remove duplicate nodes.
- Step 11 Apply boundary conditions.
- Step 12 Apply material properties.
- Step 13 Apply geometrical properties.
- Step 14 Submit the analysis.
- Step 15 Postprocessing: contour the equivalent von Mises stress on the structure.

(on)

Detailed Session Description

As mentioned in earlier sample sessions, the first step is to establish a coordinate system. It seems natural to orient the z-axis of the global coordinate system in the direction of the axial axis of the vessel.

Step 1

Choose an origin that lies in the plane of the end cap of the vessel. This way the x- y axes of the global coordinate system span a plane that coincides with the plane of the end cap. See page 9-4 on *Idealization* which mentions the need to model only a quarter section in circumferential direction. It is in this quarter section of the hull of the vessel that the manhole is modeled.

Make use of the ruled surface to create the quarter section of the vessel. The two curves necessary for ruled surfaces are arcs of equal radius extending 90 degrees at a z-coordinate of 0 and 116 respectively. Click on the following button sequence to use the Center/Radius/ begin Angle/end Angle arc type (CRAA) and to enter the data for the two curves.

```
MAIN
  MESH GENERATION
     CURVE TYPE
        CENTER/RADIUS/ANGLE/ANGLE
        RETURN
     crvs ADD
        0 0
              0
                                                                (Center point)
        84
                                                                     (Radius)
        0
                                                             (Beginning angle)
        90
                                                                (Ending angle)
        0 0 116
                                                                (Center point)
        84
                                                                     (Radius)
        0
                                                             (Beginning angle)
        90
                                                                (Ending angle)
     PLOT
        label CURVES
        RETURN
     FILL
```

To make the two arcs visible you need to deviate from the default viewpoint of $0\ 0\ 1$. There are two ways to do this: you can change the view (and the viewpoint) by clicking the appropriate view number on the view menu, or you can rotate the picture by an increment of 45 degrees about the y-axis.

Use the latter method and set the rotate increment in the VIEW menu using the following button sequence:

```
MAIN
VIEW
VIEW SETTINGS
model increments ROTATE
45
RETURN
rotate Y+
FILL
```

(in the static menu next to ROTATE+)

Now that both curves can be distinguished, add the surface by first specifying the surface type:

```
MAIN

MESH GENERATION

SURFACE TYPE

RULED

RETURN

srfs ADD

1 (Pick first curve)

2 (Pick second curve)
```

To pick the two previously defined curves, use the $\langle ML \rangle$ with the $\langle \uparrow \rangle$ in the vicinity of the curve. The program displays the surface. Similar to the button sequence outlined above, set the surface type to CYLINDER to add the surface of the manhole. The surface of the manhole is only used here to determine the intersecting curve; it is not used as a primitive entity to be converted to elements.

```
MAIN
MESH GENERATION
SURFACE TYPE
CYLINDER
RETURN
srfs ADD
45 40 58
45 120 58
24 24
```

(1st point on the axis of the cylinder) (2nd point on the axis of the cylinder) (Radii at 1st and 2nd point)

The basic geometry of the model is now complete. Rotate the picture about the y-axis over -45 degrees. Switch off the drawing of points and display four views of the model. Fill the graphic area for all views after activating them.

MAIN	
rotate Y-	(in the static menu next to ROTATE -)
PLOT	
draw POINTS	(off)
RETURN	
VIEW	
SHOW ALL VIEWS	
ACTIVATE ALL	
FILL	

Step 2

Use the **CONVERT** processor to convert the surface of the vessel to finite elements. Click on the following button sequence to create a mesh of 20x20 elements on the quarter cylinder surface.

```
MAIN
MESH GENERATION
CONVERT
DIVISIONS
20 20
SURFACES TO ELEMENTS
1
END LIST (#)
```

(Pick the ruled surface)



Figure 9-4 Four Views of Completed Model Geometry

To get a better overview of the model, change the view option to 2 and deactivate the face identification option to unclutter the picture. Display curves and surfaces using a high accuracy. Figure 9-5 illustrates where the manhole cylinder penetrates the surface of the vessel.

```
PLOT
draw element FACES
MORE
DIVISIONS
HIGH
REGENERATE
VIEW
show 2
RETURN
```

(off)

(Below SHOW ALL VIEWS)



Figure 9-5 Elements Generated on Vessel Surface

Step 3

Remove a group of 6x6 elements from the vessel surface that occupy the hole caused by the penetrating manhole. Next, all unused nodes must be removed.

MAIN MESH GENERATION elems REM

> END LIST (#) SWEEP remove unused NODES

(Box pick the elements)



Figure 9-6 Vessel with Elements Removed

Step 4

The surface of the vessel now has a square hole. The nodes on the perimeter of the square hole must now be attached to the intersection of the vessel and manhole surfaces which is done using the following button sequence:

```
MAIN

MESH GENERATION

ATTACH

attach nodes INTERSECT

2 (Pick the manhole surface)

1 (Pick the vessel surface)

(Box pick the nodes on the perimeter of the hole)

END LIST (#)
```

Relax the nodes while keeping the outline of the mesh fixed, using the button sequence given below. The resulting mesh is shown in Figure 9-7.





Figure 9-7 Peripheral Nodes Attached to Cylinder

Step 5

Figure 9-7 clearly indicates that the mesh pattern around the hole is not optimal. The primary cause of this is the irregular node distribution around the hole. In order to redistribute the nodes, you must STRETCH the nodes in groups so that the nodes are evenly distributed as indicated in Figure 9-8.



Figure 9-8 Evenly Distributed Nodes

The following button sequence is used to stretch the nodes:

```
MAIN
MESH GENERATION
STRETCH
NODES
226
161
END LIST (#)
```

(Pick the first node of the stretch node path) (Pick the last node of the stretch node path)

Repeat this operation for the other nodes on the perimeter of the hole as indicated in Figure 9-8.

It is obvious from the picture, the stretch operation no longer preserves the requirement that the perimeter of the hole is on the intersection of the two main surfaces.

To re-attach the nodes, use a directed attach method which will guarantee that the nodes will be moved to the intersection along a specified direction. The following button sequence demonstrates the steps required to apply the directed attach method.

```
MAIN

MESH GENERATION

ATTACH

DIRECTED

DIRECTION

-1 0 1

attach nodes INTERSECT

2

1

247 268 289 288

END LIST (#)
```

(Pick the surface) (Pick the surface) (Pick the nodes)

Repeat this process for all four sides that have been stretched using a different direction for each side. The result of the first attach operation is shown in Figure 9-9.



Figure 9-9 Using the Directed Attach Method to Re-Attach the Nodes

As noted before, it is sufficient to create only half of the model shown in Figure 9-10 due to symmetry. The reason for generating the entire model is that the nodes on the boundary of a mesh remain at their location during relax operation and only interior nodes are moved. Had we generated only half the model, the nodes on the line of symmetry (in the XY plane) would have required a manual redistribution.

```
MAIN
MESH GENERATION
elems REM
END LIST (#)
SWEEP
remove unused NODES
```

(Box pick all elements below the symmetry line)



Figure 9-10 Removing Unused Nodes

Step 6

There are several ways to create the manhole. The user is to follow the same steps used for the vessel. The cylindrical surface is first converted to elements. The bottom edge of the manhole elements is then attached to the intersecting line of the vessel and manhole surface. Instead you will use a different approach that involves the use of line elements. The edge of the existing hole in the vessel is *plated* with line elements that serve as the generating elements in an expand operation. Use the following button sequence to create line elements to the exposed side of the quadrilateral elements:

```
MAIN
MESH GENERATION
CONVERT
EDGES TO ELEMENTS
```

END LIST (#)

(Pick the edges at the perimeter of the hole)

Use the following button sequence to select and store the line elements generated by this operation into a set name for later reference.

```
MAIN
MESH GENERATION
SELECT
select by: ELEM CLASS
LINE(2)
OK
elems STORE
sticks
all: SELECT.
CLEAR SELECT
```

The **EXPAND** processor drags line elements into shell elements and shell elements into volume elements, effectively increasing the dimensionality of the element type by one. Use the EXPAND operation to drag the line elements equidistantly over 10 inches for 3 layers. The rim of the manhole created in this manner has the same shape as the intersection line of the two cylinders.

Step 7

Use the following button sequence to perform the expand operation.

```
MAIN
VIEW
SHOW ALL VIEWS
RETURN
MESH GENERATION
EXPAND
TRANSLATIONS
0 10 0
REPETITIONS
3
ELEMENTS
sticks
```

Use the **SWEEP** processor and click on **NODES** from the **SWEEP** panel to eliminate the duplicate nodes created by the expand operation. Click on the all: **EXIST**. button of the static menu to indicate that you want to rid the entire mesh of duplicate nodes. You can verify the elimination of the nodes by only drawing the outline of the mesh.

MAIN MESH GENERATION SWEEP sweep NODES all: EXIST.



Figure 9-11 Manhole with Nearly Correct Coordinates



Step 8

Attach the doubly curved rim of the manhole to a patch. To create the patch, select QUAD as the current surface type. Use the coordinates given below to add the patch in a local coordinate system. Create the local coordinate system by rotating 90 degrees about the global x-axis and translating it 112 inches in the global y-direction.

MAIN MESH GENERATION SURFACE TYPE QUAD RETURN coordinate system SET **U DOMAIN** -100 100 **U SPACING** 10 **V DOMAIN** -100 100 **V SPACING** 10 grid ON ROTATE 90 0 0 TRANSLATE 0 112 0 RETURN PLOT draw POINTS RETURN pts ADD -10 0 0 100 0 0 100 100 0 -10 100 0 FILL srfs ADD 29 30 31 32 GRID FILL

(on)

(*on*)

(pick the four points generated above) (off)



Figure 9-12 Creating the Patch

The nodes on the cut-off boundary of the manhole need to be attached to the intersection of the two surfaces. Use the following button sequence to attach the nodes:

```
MAIN
MESH GENERATION
ATTACH
attach nodes INTERSECT
2
3
```

(Pick the cylinder) (Pick the quad) (Use the Polygon Pick Method to pick the nodes from view 1)

Figure 9-13 shows the results of the attach operation.


Figure 9-13 Attached Nodes to Patch

Step 9

Subdivide the top row of elements in the second direction of connectivity to improve the aspect ratio. Once again, it is most convenient to use the Polygon Pick Method to select the elements.

```
MAIN
MESH GENERATION
SUBDIVIDE
DIVISIONS
1 2 1
ELEMENTS
END LIST (#)
```

(Pick the top row of elements)



Figure 9-14 Improved Aspect Ratio for Top Row Elements

Step 10

Sweep the mesh to eliminate duplicate nodes after each operation that generates elements.

```
MAIN
MESH GENERATION
SWEEP
NODES
all: EXIST.
```

You have now completed the topological part of the mesh.

For subsequent tasks, it is no longer required to use the geometric entities points, curves, and surfaces and therefore, the plotting of these items will be switched off.

PLOT	
draw POINTS	(off)
draw CURVES	(off)
draw SURFACES	(off)
REGENERATE	
FILL	

Step 11

There are two types of symmetry conditions across the edge that cut the vessel and manhole in half:

- 1. Zero displacement in z direction,
- **2.** Zero local rotations along the edge.

The first boundary condition (1) is expressed in global coordinates. To apply the second boundary condition, it is necessary to apply a transformation to the nodes of the vessel such that the boundary conditions can be expressed as a function of the global d.o.f.'s. Use the following button sequence to create the transformations.

```
MAIN
BOUNDARY CONDITIONS
MECHANICAL
VIEW
show 2
RETURN
TRANSFORMS
CYLINDRICAL
0 0 0
0 100
END LIST (#)
```

(center line) (Pick the nodes along the curved and straight edges of the vessel; not those of the manhole)

Mentat User's Guide 9-25

The boundary conditions (1) and (2) mentioned above are then applied through the following button sequence:



VIEW

show 4

(on) (on)

(Pick the nodes along the symmetry plane of the vessel and manhole)



Figure 9-15 Boundary Conditions Applied

The other curved edge of the vessel has an edge load applied in the direction of the center line.



Figure 9-16 Edge Loads Applied in Direction of Center Line

The vessel is under internal pressure which is applied through the following button sequence:

```
MAIN
BOUNDARY CONDITIONS
MECHANICAL
NEW
FACE LOAD
PRESSURE
-100.0
OK
faces ADD
all: EXIST.
```

Note that the definition of a positive pressure is one that is directed towards the face of the element.



Figure 9-17 Internal Pressure Applied



Figure 9-17 shows that the pressure on the manhole is applied as an external pressure. Two methods can be used to correct the direction in which the load is applied: either the sign of the applied pressure load for the manhole elements is changed or the direction of the connectivity of the elements of the manhole is changed. The latter method, used in the container sample session described in Chapter 8, is also used in this session and invoked with the following button sequence:

MAIN MESH GENERATION CHECK VIEW show 2 RETURN FLIP ELEMENTS END LIST (#)

VIEW show 4 RETURN (Use the Polygon Pick Method to select the elements of the manhole)



Figure 9-18 Corrected Load Direction

The following two types of symmetry conditions exist along the straight edges of the vessel: displacement in tangential direction is zero, rotation in axial direction is zero.

Due to the previously defined transformations, these boundary conditions can be applied using the following button sequence:

```
MAIN
BOUNDARY CONDITIONS
MECHANICAL
NEW
FIXED DISPLACEMENT
ON y displace
ON z rotation
OK
nodes ADD
```

(on) (on)

(Pick the nodes on the straight edges of the vessel)

```
END LIST (#)
```



Figure 9-19 Boundary Conditions Applied to Vessel Edges

Finally, an edge load is applied to the top perimeter of the manhole.

MAIN BOUNDARY CONDITIONS MECHANICAL NEW VIEW show 1 RETURN EDGE LOAD PRESSURE -1200 OK edges ADD END LIST (#) VIEW show 4

RETURN

(Pick the edges at the top rim of the manhole)



Figure 9-20 Edge Load Applied to Top Rim of Manhole



Step 12

The material properties for both the vessel and the manhole are specified on page 9-6. Use the following button sequence to apply steel properties to the two structures.

```
MAIN
MATERIAL PROPERTIES
ISOTROPIC
YOUNG'S MODULUS
30.0e6
POISSON'S RATIO
0.3
OK
elements ADD
all: EXIST.
```

Step 13

The manhole is manufactured out of a steel plate with a thickness of 1 inch. The thickness of the vessel is 0.54 inches. Click on the GEOMETRIC PROPERTIES button of the main menu and go to the "mechanical elements 3-D" submenu. Enter the SHELL pop-up menu, click on the THICKNESS button and type in 0.54. To confirm the correctness, click on the OK button. Assign the thickness to the elements of the vessel only. Repeat this process for the manhole using the following button sequence:

```
MAIN
GEOMETRIC PROPERTIES
mechanical elements 3-D
SHELL
THICKNESS
1.0
OK
VIEW
show 2
RETURN
elements ADD
END LIST (#)
```

(Pick the elements of the manhole)

Confirm the correctness of the thickness application using the ID GEOMETRIES button. The resulting figure is not shown here.

You have now completed the modeling process, Step 2 of the Analysis Cycle. Continue with the preparatory steps for the finite element analysis.

This analysis is a linear static in which case you do not need to create a loadcase. The INITIAL LOADS option in the JOBS menu is used to specify the loading pattern.

Step 14

Use the following button sequence to define the MARC element type, to verify that the appropriate initial loads have been activated, to specify the desired result variables, and to submit the job.

MAIN JOBS **ELEMENT TYPES 3-D MEMBRANE/SHELL** 75 OK all: EXIST. RETURN **MECHANICAL** JOB RESULTS tensors stress OUT scalars von_mises OUT OK **INITIAL LOADS** OK JOB PARAMETERS **#SHELL/BEAM LAYERS** 3 OK OK CHECK SAVE RUN **SUBMIT 1** MONITOR

Step 15

The screen is updated periodically to report the progress of the job. If the job has been successfully completed, the exit message on the panel will be 3004.

To display the results of the analysis for interpretation, use the following button sequence:

(off)

MAIN RESULTS OPEN DEFAULT NEXT INC PLOT draw NODES RETURN SCALAR Equivalent von Mises Stress Layer 1 OK CONTOUR BANDS DEF & ORIG FILL

Figure 9-21 shows the resulting model contoured with von Mises stresses.



Figure 9-21 Model with von Mises Stress Contours



Conclusion

Due to reproduction constraints, Figure 9-21 does not give you a clear representation of the actual resulting stress distribution that appears in the graphics area on your screen. The results indicate that, due to the penetration of the manhole into the vessel, the localized stress concentrations occur near the intersection.



Procedure File

```
| Version : Mentat 3.3
*alias action
*exec_p gen_image.proc
@top()
*action | image 0 - dummy
*action | figure 9.1 - dummy
*action | figure 9.2 - dummy
*action | figure 9.3 - dummy
@push(meshgen)
@push(curve_type)
*set_curve_type arc_craa
@pop(1)
*add curves
0 0 0
84
0
90
0 0 116
84
0
90
@push(plot_screen)
*set_curve_labels on
*regenerate
*fill view
@push(viewing_screen)
@push(view_settings_screen)
*rot model increment
45
@pop(1)
*rot_model_cspace_y_for
*fill view
 create a ruled surface between curves
@pop(1)
@pop(1)
@push(surface type)
*set_surface_type ruled
```

```
@pop(1)
*add_surfaces
 1
 2
 create cylindrical surface
@push(surface_type)
*set_surface_type cylinder
@pop(1)
*add surfaces
45 40 58
45 120 58
24 24
*rot_model_cspace_y_rev
@push(plot_screen)
*set_points off
@pop(1)
@push(viewing_screen)
*show view 4
*fill view
*show_all_views
*fill view
*action | figure 9.4
@pop(1)
  step 2
  convert surface 1 to a 20x20 mesh
@pop(1)
@push(meshgen)
@push(convert)
*set_convert_divisions
20 20
*convert_surfaces
 1
# | End of List
@pop(1)
@push(plot_screen)
*set faces off
```

```
@push(plot1_screen)
@push(divisions_screen)
*set_plot_divisions_high
*regenerate
@push(viewing_screen)
*show view 2
@pop(1)
*action | figure 9.5
@pop(1)
@pop(1)
@pop(1)
 step 3
 remove elements to create a hole
*remove elements
 151 152 153 154 155 171 172 173 174 175 191 192 193 194 195 211 212
213 214 215
 231 232 233 234 235 251 252 253 254 255
 150 170 190 210 230 250
# | End of List
*action | figure 9.6
 step 4
 remove unused nodes, left after deleting the elements and attach
  nodes on the perimeter of the hole to intersection of surface 2
and 1
@push(sweep)
*remove_unused_nodes
@pop(1)
@push(attach)
*attach nodes intersect
2
1
 158 159 160 161 162 163 184 205 226 247 268 284 285 286 287 288 289
 157 178 199 220 241 262 283
# | End of List
@pop(1)
```

```
@push(relax)
*relax_nodes
all_existing
*action | figure 9.7
 stretch the four diagonal sides of the hole
@pop(1)
@push(stretch)
*stretch_nodes
 226
 161
# | End of List
 160
 220
# | End of List
*stretch_nodes
 220
 286
# | End of List
 287
 226
# | End of List
*action | figure 9.8
@pop(1)
 perform a directed attach in each quadrant
@push(attach)
*set_attach_direction
-1 0 1
*attach_nodes_intersect
2
1
247 268 289 288
# | End of List
*action | figure 9.9
*set_attach_direction
1 0 1
*attach_nodes_intersect
```

2

```
1
 285
 284
 283
 262
 241
# | End of List
*set_attach_direction
-1 0 -1
*attach_nodes_intersect
2
1
 205
 184
 163
 162
# | End of List
*set_attach_direction
1 0 -1
*attach nodes intersect
2 1
 159
 158
 157
 178
 199
# | End of List
@pop(1)
 cut away half of the model
*remove elements
 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24 25
26 27 28 29
 30 31 32 33 34 35 36 37 38 39 40 41 42 43 44 45 46 47 48 49 50 51
52 53 54 55
 56 57 58 59 60 61 62 63 64 65 66 67 68 69 70 71 72 73 74 75 76 77
78 79 80 81
 82 83 84 85 86 87 88 89 90 91 92 93 94 95 96 97 98 99 100 101 102
103 104 105
```

```
106 107 108 109 110 111 112 113 114 115 116 117 118 119 120 121 122
123 124 125
 126 127 128 129 130 131 132 133 134 135 136 137 138 139 140 141 142
143 144 145
 146 147 148 149 156 157 158 159 160 161 162 163 164 165 166 167 168
169 176 177
178 179 180 181 182 183 184 185 186 187 188 189 196 197 198 199 200
# | End of List
*action | figure 9.10
@push(sweep)
*remove_unused_nodes
@pop(1)
 step 6
  generate line elements on the edges on the perimeter of the hole
@push(convert)
*edge_lines
 216:3
 236:3
 256:3
 275:0
 274:0
 273:0
 272:0
 271:0
 270:0
 249:1
 229:1
 209:1
# | End of List
@pop(1)
  select the line elements and store them as set sticks
@push(select_screen)
@push(select by screen)
*select_elements_class
@popup(sel_elem_class_pm)
line2
```

```
@popdown(sel_elem_class_pm)
@pop(1)
*store_elements
sticks
all selected
*select clear
@push(viewing_screen)
*show all views
@pop(1)
  expand the line elements to form the boot of the manhole
@pop(1)
@push(move)
@pop(1)
@push(expand)
*set_expand_translations
0 10 0
*set_expand_repetitions
3
*expand elements
sticks
@pop(1)
@push(sweep)
*sweep_nodes
all_existing
*action | figure 9.11
@pop(1)
  generate an additional surface at the future top location
   of the manhole
@push(surface_type)
*set_surface_type quad
@pop(1)
*set_grid on
@push(coord_system)
*grid u domain
-10 10
*grid_u_spacing
10
```

S 9 Manhole

```
*grid_u_domain
-100 100
*grid_v_domain
-100
100
*grid_v_spacing
10
*system_rotate
90 0 0
*system_translate
0 112 0
@push(file screen)
@pop(1)
@push(plot_screen)
*set_points on
@pop(1)
@pop(1)
*add_points
-10 0 0
100 0 0
100 100 0
-10 100 0
*fill view
*add_surfaces
29 30 31 32
*set_grid off
*fill view
*action | figure 9.12
 attach the nodes at the top
@push(attach)
*attach nodes intersect
 2
 3
 448 455 462 469 476 483 490 497 504 511 518 525 532
# | End of List
*action | figure 9.13
@pop(1)
  step 9
```

```
@push(subdivide)
*sub_divisions
1 2 1
*subdivide elements
 415 418 421 424 427 430 433 436 439 442 445 448
# | End of List
*action | figure 9.14
@pop(1)
 sweep the nodes
@push(sweep)
*sweep_nodes
all_existing
@push(plot_screen)
*set_points off
*set curves off
*set_surfaces off
*regenerate
@push(viewing_screen)
*show_view 2
*fill view
@pop(1)
@pop(1)
@pop(1)
@pop(1)
  step 11
@push(apply)
@push(apply_mechanical)
@push(viewing_screen)
*show_view 2
@pop(1)
@push(transformations)
*transform_cylindrical
0 0 0
0 0 100
 252 273 294 315 336 357 378 399 420 441
 227 228 229 230 231
```

```
211 212 213 214 215 216 217 218 219
 232 253 274 295 316 337 358 379 400 421
# | End of List
@pop(1)
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed_disp_pm)
*apply dof z
*apply_dof ry
@popdown(fixed_disp_pm)
*add apply nodes
 211 212 213 214 215 216 217 218 219 220 226 227 228 229 230 231 444
446 448 451
 453 455 534 543
# | End of List
@push(viewing_screen)
*show view 4
@pop(1)
*fill view
*action | figure 9.15
*new apply
*apply_type edge_load
@set($usersubfv,FORCEM)
@popup(apply_edge_load_popmenu)
*apply_value p
-4200
@popdown(apply_edge_load_popmenu)
*add_apply_edges
 381:2 382:2 383:2 384:2 385:2 386:2 387:2 388:2 389:2 390:2 391:2
392:2 393:2
 394:2 395:2 396:2 397:2 398:2 399:2 400:2
# | End of List
*action | figure 9.16
*new_apply
*apply type face load
@set($usersubfv,FORCEM)
@popup(apply_face_load_popmenu)
*apply value p
-100
```

```
@popdown(apply_face_load_popmenu)
```

```
*add_apply_faces
all_existing
*action | figure 9.17
@pop(1)
@pop(1)
@push(meshgen)
@push(viewing_screen)
*show view 2
@pop(1)
@push(check_elements)
*flip elements
 413 414 416 417 419 420 422 423 425 426 428 429 431 432 434 435 437
438 440 441
 443 444 446 447 449 450 451 452 453 454 455 456 457 458 459 460 461
462 463 464
 465 466 467 468 469 470 471 472
# | End of List
@pop(1)
@push(viewing_screen)
*show view 4
*action | figure 9.18
@pop(1)
@pop(1)
@push(apply)
*new_apply
@push(apply_mechanical)
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed disp pm)
*apply_dof y
*apply_dof rz
@popdown(fixed_disp_pm)
*add apply nodes
 211 232 253 274 295 316 337 358 379 400 421
 231 252 273 294 315 336 357 378 399 420 441
# | End of List
*action | figure 9.19
@pop(1)
*new_apply
@push(apply_mechanical)
*apply type edge load
```

```
@set($usersubfv,FORCEM)
@popup(apply_edge_load_popmenu)
*apply_value p
-1200
@popdown(apply_edge_load_popmenu)
@push(viewing_screen)
*show_view 1
@pop(1)
*add_apply_edges
 450:0 452:0 454:0 456:0 458:0 460:0 462:0 464:0 466:0 468:0 470:0
472:0
# | End of List
@push(viewing_screen)
*show view 4
@pop(1)
*action | figure 9.20
@pop(1)
@pop(1)
 step 12
@push(mat_props)
*material type mechanical:isotropic
@popup(mat_isotropic)
*material_value isotropic:youngs_modulus
30e6
.3
@popdown(mat_isotropic)
*add material elements
all_existing
@pop(1)
step 13
@push(geom props)
*geometry_name
vessel
@push(geometry_mech_three)
*geometry_type mech_three_shell
@popup(geom_shell_pm)
*geometry_value thick
```

```
443 444 446 447 449 450 451 452 453 454 455 456 457 458 459 460 461
462 463 464
 465 466 467 468 469 470 471 472
# | End of List
@push(viewing_screen)
*show view 4
@pop(1)
*identify geometries *regen
@push(plot_screen)
*elements solid
*regenerate
*elements wireframe
*regenerate
@pop(1)
*identify_none *regen
@pop(1)
@pop(1)
  step 14
@push(jobs)
@push(element_types)
@popup(eltype membrane shell pm)
*element_type 75
@popdown(eltype_membrane_shell_pm)
all existing
@pop(1)
*job_class mechanical
@popup(job mechanical)
@list(job_rem_lcase_list,0)
@list(job_add_lcase_list,0)
@popup(tensor scalar popmenu)
@list(post tensor list1,0)
@list(post_var_list1,0)
*add post tensor stress
*post_tensor_outer_layers stress
*add_post_var von_mises
*post var outer layers von mises
@popdown(tensor_scalar_popmenu)
@list(job_add_lcase_list,0)
@list(job_rem_lcase_list,0)
```

```
@popup(job_param_mech_popmenu)
*job_param layers
3
@popdown(job_param_mech_popmenu)
@list(job_rem_lcase_list,0)
@popup(init_loads_popmenu)
@list(job_apply_list,0)
@list(job_icond_list,0)
@popdown(init_loads_popmenu)
@list(job_rem_lcase_list,0)
@popdown(job mechanical)
@push(file screen)
@set($file_browser_label,SAVE AS FILE)
@set($file browser command, *save as model)
@popup(save file browser popmenu)
@list(save_file_browser_filelist,0)
@list(save file browser dirlist,0)
*save_as_model c09.mud yes
@popdown(save_file_browser_popmenu)
@pop(1)
*update job
@popup(job_run_popmenu)
*submit job 1
*monitor job
@popdown(job_run_popmenu)
@pop(1)
@push(results)
*post_open_default
*post next
@push(plot_screen)
*set nodes off
@pop(1)
*post value
Equivalent Von Mises Stress Layer 1
*post contour bands
*set deformed both
*action | figure 9.21
@top()
```



Chapter Overview

This chapter describes the analysis of the cross section of an automobile tire. The model is loaded by an internal pressure and the contact between the tire and the rim is to be analyzed.

The method used in this chapter to obtain a solution is typical for tackling an engineering problem. This chapter demonstrates that it is useful to approach a problem by using simple models first before going on to large complicated structures. This approach not only gives you a better understanding of your problem, but it also enables you to better analyze the results.



Background Information

Description

An automobile tire is a complex composite structure, consisting of (nonlinear) materials, that comes into contact with the road.

Figure 10-1 identifies the different materials and parts of an automobile tire by part name.



Figure 10-1 Cross-Section of Automobile Tire



Idealization

The material properties of the tread, side wall, chafer, and apex are isotropic. The carcass is characterized by an orthotropic material property. The steel belts and beads behave as isotropic materials in the circumferential direction of the tire. In this analysis, both the carcass on the steel belts and beads have been given the same properties as the rubber and thus no special elements are required in modeling these parts. The tire comes into contact at the chafer with the wheel rim. The wheel rim is modeled as an infinitely stiff body.



Figure 10-2 Overall Dimensions of the Tire



Level of Analysis Detail

This section describes the different stages of idealization that are observed in this analysis. As was noted in the chapter overview, the best approach for analyzing a complicated structure is to start with simple models. This approach allows you to gain knowledge and confidence in problem-solving as you progress through the analysis process. This approach also helps you to predict behaviors and to identify potential problems.

In this sample session, you will only analyze the inflation process using a crude and easy to generate mesh. The analysis presents some of the main components of detailed analysis.



Analysis

The purpose of this initial analysis is to describe the inflation process by means of an idealization of the real structure.

Idealization

For purposes of this simplified analysis, assume all materials to be identical and ignore the treads in the tire. An axisymmetric model is used and because of symmetry, you only need to analyze half of the cross-section.

Requirements for a Successful Analysis

The analysis is considered successful if the closing behavior at the rim/chafer interface of this simplified model can be shown.

Full Disclosure

Type of analysis Contact

Materials

The rubber material for this structure is characterized by three Mooney constants for which the following values are chosen:

 $C_{10} = 965 kPa$ $C_{20} = 193 kPa$ $C_{30} = 193 kPa$

Elements

MARC Element Type 82, four-noded axisymmetric Herrmann formulation.



Overview of Steps

- **Step 1** Create the boundary using Bezier curves.
- Step 2 Use automatic meshing (OVERLAY) to create a mesh.
- **Step 3** Create the rim as a rigid die and identify the contact bodies.
- **Step 4** Add boundary conditions.
- **Step 5** Apply internal pressure.
- **Step 6** Submit the job.
- Step 7 Postprocess the results.


Detailed Session Description

The description of the tire boundary geometry is well suited for the use of Bezier curves. The defining polygon of a Bezier curve can easily be changed which results in a *smooth* change in the entire curve. To demonstrate the versatility of this curve type, we will generate the boundary of the tire using Bezier curves exclusively.

Step 1

Before entering the Bezier curves, however, first establish an input grid using the following button sequence.

```
MAIN
MESH GENERATION
SET
U DOMAIN
-17 0
U SPACING
1
V DOMAIN
0 17
V SPACING
1
grid ON
FILL
```

(*on*)

Observe that the dimension of the grid size is specified in centimeters. The material constants specified in *Idealization* on page 10-3 require a conversion from kPa into N/cm², in order to be consistent with the units used here. For a good resolution of the Bezier curve drawing, set the plotting of curves with high accuracy. Note that when drawing, the number of subdivisions is merely a drawing resolution. The information on every point on the curve is preserved. Use the following button sequence to change the resolution and to set the curve type to Bezier.



MAIN PLOT MORE DIVISIONS HIGH RETURN RETURN MESH GENERATION CURVE TYPE BEZIER

The curves are added by clicking on the ADD button of the crvs panel and entering the points for the defining polygon vertices of each curve. The beginning and end points of the Bezier curve are determined by the first and last point specified. The tangent at either end is defined by the neighboring points.

```
MAIN

MESH GENERATION

crvs ADD

point(0,14,0)

point(-11,13,0)

point(-11,6,0)

point(-5,4,0)

point(-5,0,0)

END LIST (#)
```

(Pick the appropriate grid points)



MESH GE	NERAI	ON												-					V=17
NODES	ADD	REM	EDIT	SHOW	1														W
ELEH45	ADD	REM	EDIT	SHOW															
PTS	ADD	REM	EDIT	SHOW															
CHVS	ADD	REM	EDIT	SHOW															
SNES	ADD	REM	EDIT	SHOW										-					+
SOUDS	ADD	REM		SHOW												~			
BETWEEP	NODE	8ET	WEEN	POINT										/	_				
ELEMENT	CLASS	- " OU	AD (4)	7			,						- 2	/-					
CURVE T	YPE	1 BEZ	ER.	Ť									Ζ.						
SURFACE	ETYPE	* OU	AD	+								1							
SOUDTY	PE	· 81.0	жк	+								- /-							
COORDE	ATE 51	STEM										1.							
SET *	RECTA	INGULA	a 11+1	GRID								1							
CITATIN	194	for	ALC OF				,					1.							
ATTACH		* A10	DOM STOR									١.							
CHANSE	C1455	· CH	REK.									1							
CONVER	r	- DU	PLICATE																
EXPAND		INT	EBSECT										Ν.						
MOVE		* REI	AN.	+									1						
RENUMB	ER	" REV	ADEVE	+										· ·	*				
SOLDS		1 819	ETCH	7										Λ.					
SUBDIVE	Æ	1 544	EEP*											1				1.1	÷ •
SYMMET	AV.	+													1.			- 1	
ALC:	SELEC	3. VB	815 (s	NUTLINE											1				×
EX451	UNSER	. 111	19. 1	SURFAC									-	-	ï	-			
SELECT		1 84	DUST 0	1									-	-	-1				, JHB ;
RETURN		MA	N	1	UNDO	SAVE	DRAW	FILL	RESE	TYTEW	TK+	TY+	TZ+	RX+	RYA	RZ+	ZOOM	IN	USER
					UTLS'	FILES'	PLOT	VIEW 1	DWN	MODEL	TK-	TY-	TZ-	RX-	RY-	82-	808	OUT	DEFINED

Figure 10-3 Interior Tire Wall

Increase the resolution of the grid to 0.5 units:

```
MAIN
MESH GENERATION
SET
U SPACING
0.5
V SPACING
0.5
```

Create the exterior side of the wall of the tire by adding the following curve, the result of which is shown in Figure 10-4.



	Μ	AIN																													
		MF	SH	GEN			0	N																							
			crv		רי		<u> </u>	•													0	Dic	l- +1	ha	a		or	i.	ato	ari	Indint
					,	~			,												(1		<i>n 11</i>	ie	щ	'P'	θp	'' u	ле	gru	i poini
				poın	t(-	9,	16	, 0)																						
				poin	t(-	9,	14	:,0)																						
				poin	t(-	10	.5	,1	3,	0))																				
				noin	+(-	10	5	, R	5	; (<u>،</u>																				
						т (д	. J	,0	• •	,,,	, ,																				
				END	LIS	I (#	•)																								
METH C	FREEAT	10M																											_	V=17	
NODER.		Lara	Inne	Invest							2	1	1		1	2						11		1					E	(MAR)	8
C CMS	400	REN	EDIT	SHOW		• •											- 1														
010943	400	REAL	ELAT	SHOW													• }													•	
-15	100	PR34	ELAT	SHOW			1								1	1	:7			1				1						:	
EBUS	400	DEM .	EDIT	SHOW							2						7.												_	-+	
Source S	400	DCM	- LUI	SHOW												۰,	į							5	مرز		-				
OCUME	1 1000	00	J	DOM/T							1			*	٠	1						2	~	1						•	
o cuer	n moot	- 00	AD AD	- CARIN -												Į.					2	Ξ.								:	
CLEMEN	TYPE	1 80	100104	-											-)	Į.,				/											
SHEEA	C TABE	1 00	AD.												1	1			1												
SOUD T	VPE	1 81/	0.00								1				1	1		-12	<u> </u>					1						:	
			0.041	_							÷.				ł	1		-7													
COORD	INATE S	VSTEM													÷			1													
261	RECT	ANGULZ	64 ·] •	GRED		•								-	*	1		1								-				•	
CLEAR	4.51	0.1	lan ge	OM (2				1	1		1.1													
ATTACH		* AU	TOMES	н 1														-\.												•	
CHANGE	CLASS	CH	ECK.	P														· /												•	
CONVER	RT	1 DU	PLICAT	E ''			1				1			1	1	1			1	1				1			1			:	
EXPAND)	' INT	ERSEC	r .															1	-											
MOVE		* RE	. XAL	1		• •														1										•	
RENUM	BER	' RE	VOLVE	P		1	1				1				1	1				1	11	1.1		1				1		:	
SOUDS		517	LETCH	P																	N.			1							
SUBDIV	DE .	500	CEP*	1																	, .)							- ř			
SYMMET	TRY	+					1				1				1	1				1		1.		1			1	· F			
	36153	CT. VB	313	oumas:		11					1				1	1						1		1				; ¥	-	÷×-	
	UNIT	L IN	19.	SUBJEAC																		. [
SELECT		* 8M	D LIST (15		• •					-		• •					• •				- 1		-				• •	-	,11-0	1
RETURN	1	MA	an i	1	UNDO	544	Æ	DRAW	N	FILL		RE	581	ME	W	Í	TKa	T¥4	17	Z+	RX+	FIY	- 18	Z+	20	OM	IN	1	UB	ER.	
					UTLS'	FILE	81	PLOT	1	VIEW	1	10	WH.	MO	OBL		TX-	TY-	T	E=	RX-	RY	R	2-	BC BC	N.	0	UT	DE	- NED	

Figure 10-4 Part of Exterior Tire Wall Added

Switch on the labeling of points in order to facilitate creating the curves as specified in the button sequences below.

MAIN PLOT label POINTS REGENERATE

(*on*)



Add the following curve to create the lower part of the exterior wall of the tire. Even though severe changes in curvature occur in this part, the overall curve remains smooth. The results are shown in Figure 10-5.

crvs AD	D	
9		(Pick point)
poir	nt(-10.5,4,0)	(Pick grid point)
poir	nt(-9.5.3.5,0)	(Pick grid point)
poir	nt(-8.5,1.5,0)	(Pick grid point)
poir	nt(-8,0,0)	(Pick grid point)
poir	nt(-7.5,0.5,0)	(Pick grid point)
5		(Pick point
END	LIST (#)	



Figure 10-5 Exterior Tire Wall Completed



Figure 10-5 clearly indicates that the shape of the portion of the tire that comes into contact with the rim of the wheel is incorrect. This can be remedied by relocating some of the support points of the Bezier curve.

Points 13, 10, and 11 are relocated using the pts EDIT button on the mesh generation panel, the results of which are shown in Figure 10-6.

```
MAIN

MESH GENERATION

pts EDIT

13

-9.5 -0.5 0

10

-11.0 -0.5 0

11

-7 5 0

END LIST (#)
```





Figure 10-6 Correction of Tire Geometry



To add the tread, use a Bezier curve made up of point 6 and three new points. The exact location of these points are shown in Figure 10-7. The following button sequence specifies where these points are located in the local u-v-w coordinate system. Refrain from entering the points by typing in their coordinates; instead, always use the mouse to pick the points as it is a much easier method.

MAIN
MESH GENERATION
crvs ADD
6
point(-7.5,16,
point(-1.5,17,
point(0,17,0)
END LIST (#)

0)

0)

(Pick point) (Pick grid point) (Pick grid point) (Pick grid point)

MESH GENERA	лон																	,									+1	6,	2017
NODES ADD	REM	EDIT	[SHOW	1																-	-			-					Comments:
REMS ADD	REM	EDIT	SHOW		•				-	-						1.5	p.,	-	-10.					-					
		CONT.	Entrant.													- 1													
PTS ADD	P/CP4	EDIT	BHUW												1	1													
CHVS ADD	REM	EDIT	SHOW												÷	7	2												
SRES ADD	REM	EDIT	SHOW												0	Ľ									-	-	÷	17	
SOLIDS ADD	REM		SHOW											23	Û.	Ľ.,							2	~					
BETWEEN NOD	€ 8	TWEEN	POINT												1						1	ς.							
BLEMENT CLAP	s' 0	DID (M)	Ŧ												ŀ					/.									
CURVE TYPE	1 81	200													Į.				1	· · ·									
CONFLUE TO D	10													- 1				۰.	/ ·										
STRO-WCE LAND	0	UMD												-1				1											
SOUD TYPE	· 81	JOCK												1	1		1	(* -											
COORDINATE	SVSTEM													-1	1		: 1												
SET REC	ANGUL	AR []*	GRID											þ	C.		1												
CLEAR MESH	(a	LAR GE	DM		1	2	1		1	1	1	1			ĩ		1	1	1.1	1.1	1			1	1.1	1	1	1	
ATTACH	* AI	TOMES	н ^г						-					÷			\							-					
CHANGE CLAS	s ' Ci	4ECK												1				ŀ											
CONVERT	1 DI	JPLICAT	E "											٦1	1			1											
EXPAND	7 IN	TERSECT	r '											- 1					1. 5	÷.									
MOVE	1 8	AA.F	-												ι.				λ.										
RENUMBER	- R	VOLVE													λ			1)	1		A							
SOUDS	1 81	RETCH			•				-						•]	5				λ.									•
SUBDIVIDE	1 59	WEEP*							2						0	2	ς.,	1		0.5	Ċ.							ź.	
SYMMETRY	+			1													· \.				1							ſ.	
ALL SEL	CT. 198	and a	normal d														- ų	2,			-1							1	1 x
CARL COM	CI 184		51195.00		• •									-		-)	Ċ	14		- 1	i							
RELECT		0.1011					1.1		1						1			2				5			1.1			1	
SCCC-1		COLUMN 1		_		-										-					-	-		-					
METURN	M	NEN	1	UNDO	54	NE -	DI	WW		LL.	N	1981	MI	W		TK:	T	/+ ·	12+	RX4	m	(h)	RZ+		1006 3006	'	IN OUT	-16	REFINED

Figure 10-7 Tire Tread



Finally, to complete the boundary, add a straight line between points 1 and 17 which form the symmetry boundary. A Bezier curve is used here simply to demonstrate how it degenerates into a straight line when only two points are specified. The completed boundary is shown in Figure 10-8.

				-											_		_	-	_					and the second se			- CO. 10	the second se
ETURN	M	AN .	1	UNDO	54	ME	DP	LAW	11	u.	1	151	TYPE	W	T	Ke	T¥+	TZ4	10	\$4	RY-	RZ+	2	004	4 1	IN .	1	ISEN .
ELECT	" (B	D LIST (8		• •				-	• •			• •		• •		• •				-15							, <i>u</i> -e ;
15/1 (UH1	IEL IN	15.	SURFAC														1	.,14			1							
.: SEL	SCT. VS	1111	OUTUNE		11								11				17				1.	11	1				Ķ.	×
METRY	+																100			•)								
DIVIDE	59	611P	P.													2				٩.							ř.	
JD8	51	RETCH	P												. `													
UMBER	' RE	VOLVE	1		11								11		Λ.		1.1		1			1.					1	
VE	R	DAX	P		1		1					1	1.1	1	1		1.1	1	17		j.	1	1	1		1	1	
AND	- IN	ENSEC	· ·		•					•				-}			• •	1.	л.									•
WERI	00	PLICAT	t '											4				\· ·										
MARCENE	0 0	es.k											. "J	1			. \											
NUT OF ST	A	CONTRACT OF			11								1.1	[1.					1.1					1	
100.000	1 11	DOARD												1			1.											
ALL METERS	1 m	LAB CE	DIM .														1.											
AEC	TANGUL	AR .	GRID		• •				-					-19			F +											•
RDINATE	SYSTEM																1.											
DINE	84	unin.			11								11	I.			1										1	
D Delle		ner ner	-											1			17											
CASE THE	6 1 M	MD.												1			1 1	1									1	
VE THEFT	1 11	788	T											-1				1	6									
ENECIA	38 0	ND IN	T											. [ί.,				1	ς.								
WEEN NO	DE	IWEEN	POINT		11			1					11		71		1.1				2		1				1	
DS ADD	REM	-	SHOW				1						1.5	à.	:/:		1.1					12	÷	_			1	
ADC	REM	EDIT	SHOW		•				-						. 1	4									-	÷	1-	
s ADS	NEM C	EDIT	SHOW													/•												· · ·
ADC	REM	EDIT	SHOW													J.												
45 ADG	REM	EDIT	SHOW		11											T							1				1	
ES ADD	REM	EDIT	SHOW	1	•											je.		-is	-	-	-							
I OCHER	ADION							• •		•			• •		• •		• •					• •		•	1	_+1	6 <u>.</u>	24MMac

Figure 10-8 Completed Boundary



Step 2

Use the automatic overlay meshing option to create the mesh. This automatic mesh generator requires a closed boundary. The only input needed from the user is the number of subdivisions in the x- and y-direction respectively, and the identification of the closed boundary.



Figure 10-9 Mesh Generated by OVERLAY



It should be clear from the visual inspection that the mesh is rather coarse in the lower area where the tire comes into contact with the wheel rim. A local refinement is necessary and can be accomplished by using the **SUBDIVIDE** and **REFINE** processors.



Figure 10-10 Step 1 of Mesh Refinement



The **REFINE** option can be used to effectively create a transition between *layers* or *rows* of elements. It requires two sets of information:

The node about which the refinement is made;

The elements that will participate in the refinement.

Note that only those elements that have the refined node as part of the connectivity are eligible.



(Pick refine node) (Pick element)

Complete this action by subdividing two more elements according to Figure 10-12.

MAIN MESH GENERATION SUBDIVIDE 48 47 END LIST (#)

(Pick elements)

Remember that some processors such as **SUBDIVIDE**, **EXPAND**, **SYMMETRY**, and **DUPLICATE** may create duplicate nodes. Although the nodes are in the same position, they are not connected. The node that is picked as the refine node may be part of one element's connectivity but not of the neighboring element. The **REFINE** processor can, in such an instance, produce unexpected and undesired results. To prevent this, it is usually prudent to activate the **SWEEP** processor *before* a refine operation is performed. Compression of all nodes located within a specified distance is accomplished by activating the **NODES** button in the SWEEP menu followed by a list of nodes that you want to sweep. Generally, you will use the all: EXIST. list button to sweep all existing nodes. Finally, renumber all items in the database in order to obtain a sequential node and element numbering.



MAIN MESH GENERATION SWEEP sweep NODES all: EXIST. remove unused NODES RETURN RENUMBER ALL



Figure 10-11 Local Refinement Around a Node





Figure 10-12 Completion of Local Mesh Refinement

Step 3

The rim of the wheel is considered to be a rigid body and is constructed using a Bezier curve. Use the following button sequence to add the rim.

```
MAIN
  MESH GENERATION
     FILL
     ZOOM BOX
                                                        (Zoom in on the lower area)
     GRID
                                                                            (on)
     crvs ADD
        5
                                                                     (Pick point)
                                                                  (Pick grid point)
        point(-12,0,0)
        point(-5.5,2.5,0)
                                                                  (Pick grid point)
        point(-11,3.5,0)
                                                                  (Pick grid point)
        point(-11,1.5,0)
                                                                  (Pick grid point)
        END LIST (#)
```





Figure 10-13 Wheel Rim Added

The mesh has been conveniently generated so that the origins coincide with the center line and the bottom of the tire. Use the following button sequence to move the entire mesh and rim over a distance that is equivalent to the radius of the wheel:

```
MAIN
MESH GENERATION
GRID
MOVE
TRANSLATIONS
0 13 0
ELEMENTS
all: EXIST.
CURVES
all: EXIST.
RETURN
FILL
```

(off)



Not all elements of the tire will come into contact with the rim. You can drastically minimize the analysis time by identifying the elements that make up the deformable body that is expected to come into contact with the rim.

MAIN CONTACT CONTACT BODIES NEW DEFORMABLE OK elements ADD

(Pick the elements that may come into contact with the rigid body)

END LIST (#)

To identify the rim (curve) as a rigid contact body, use the following button sequence:

Note: It is important to use *NEW* in the following button sequence. If NEW is not used, you will overwrite the contact body just entered.

MAIN	
CONTACT	
CONTACT BODIES	
NEW	
RIGID	
OK	
CURVES ADD	
б	(Pick curve)
END LIST (#)	
PLOT	
label POINTS	(off)
draw POINTS	(off)
elements SOLID	
RETURN	
ID CONTACT	(<i>on</i>)





Figure 10-14 Identification of Contact Bodies

The contact bodies are identified on the graphics screen by clicking on the ID CONTACT button of the contact bodies panel. The curve that represents the rigid body is enhanced by cross-hatching the side where the body is located. If the display indicates that the body is located on the incorrect side, use the FLIP CURVES option to flip the curve. Refer to Chapter 8 for a detailed description on using the FLIP CURVES option.

Now switch off the identification of contact bodies.

```
MAIN
CONTACT
CONTACT BODIES
ID CONTACT
PLOT
elements WIREFRAME
FILL
```

(off)



Step 4

Symmetry conditions are applied to the nodes along the symmetry line using the following button sequence:

MAIN BOUNDARY CONDITIONS MECHANICAL FIXED DISPLACEMENT ON x displace OK nodes ADD 131 139 143 END LIST (#)

(*on*)

(Pick the three nodes at the right)

The symmetry boundary conditions are displayed in Figure 10-15.



Figure 10-15 Symmetry Boundary Conditions Applied



Step 5

The tire is loaded by an internal pressure. Use the following button sequence to specify the loading history through a table.

```
MAIN
  BOUNDARY CONDITIONS
    MECHANICAL
       TABLES
         NAME
            loading
         TABLE TYPE
            time
            OK
         XMAX
            300
         YMAX
            220
         ADD POINT
            0 0
            300 214
         SHOW MODEL
```

It is important to specify the table type because a table will only be applied if the appropriate type is assigned. For boundary conditions, only table type *time* is valid.

Apply this load to the interior of the tire using the following button sequence:

```
MAIN
BOUNDARY CONDITIONS
NEW
EDGE LOAD
pressure TABLE
loading
OK
edges ADD
END LIST (#)
```

(Pick the edges on the inside)





The results of the applied internal pressure are depicted in Figure 10-16.

Figure 10-16 Internal Pressure Applied

The material for this mesh is assumed to be uniform over the entire mesh. Specify the material properties using the following button sequence:

```
MAIN
MATERIAL PROPERTIES
MORE
MOONEY
C10
96.5
C20
-19.3
C30
19.3
OK
elems ADD
all EXIST.
```



Step 6

Use the following button sequence to prepare a loadcase.

```
MAIN

LOADCASE

mechanical analyses STATIC

LOADS

OK

TOTAL LOADCASE TIME

300

# STEPS

300

OK
```

This loadcase is to be used in the job that ultimately is submitted for analysis. Use the following button sequence to specify the job.





Use the following button sequence to submit the job.

```
MAIN
JOBS
SAVE
RUN
SUBMIT 1
MONITOR
```

The analysis stops with an exit number 2004, indicating that a rigid body motion is present. (The tire separates from the rim).

Step 7

The purpose of the preliminary analysis is to gain experience in completing a relatively simple analysis. The following results will be displayed:

- **1.** Animation of the deformation. Only the first and last frame are shown here.
- 2. Contouring of the von Mises stress on the tire cross section.

Use the following button sequence to open the results file.

```
MAIN
RESULTS
OPEN DEFAULT
NEXT INC
```

To focus on the geometry, it is necessary to turn the node labeling and face identification off as is shown in Figure 10-17.

Use the button sequence given below to turn the node labeling and face identification off.

MAIN	
FILL	
PLOT	
draw NODES	(off)
draw FACES	(off)
REGENERATE	





Figure 10-17 Mesh without Node Labeling and Face Identification

Click on DEF & ORIG to request the original and deformed structure to be shown. The animation buttons are in the second part of the postprocessing results menu and can be reached by clicking on the MORE button. To create the animation frames, use the following button sequence:



MAIN RESULTS MORE animate INCREMENTS 100 1

The numeral 100 is entered here as a response to the number of increments that need to be processed.

From the analysis, we know that the results stretch out over approximately 13 increments. Hence, 100 is a safe upper limit. The program will now prepare the frames for animation. Figure 10-18 and Figure 10-19 show the second and last of the animation frames.



Figure 10-18 Second Animation Frame





Figure 10-19 Last Animation Frame

To animate the sequence of frames use the following button sequence:

MAIN RESULTS FILL MORE ANIMATION FILL PLAY

The equivalent Cauchy stress can be displayed by using the following button sequence:

MAIN RESULTS SCALAR Equivalent Cauchy Stress CONTOUR BANDS





Figure 10-20 shows the results of the model with equivalent stress contour bands.

Figure 10-20 Mesh with Equivalent Cauchy Stress Contours



Conclusion

This example demonstrates that it is relatively easy to complete a contact analysis using a simple to generate geometry and an incompressible material. The tire looses contact with the rim. This is caused by the fact that the steel belt is not present in this analysis. Although the results shown in this analysis have little engineering value, the analysis is valuable in reassuring that the available tools in MARC will enable you to solve a more complex problem.



Procedure File

```
| Version : Mentat 3.3
*alias action
*exec_p gen_image.proc
*action | figure 10.0 - dummy
*action | figure 10.1 - dummy
*action | figure 10.2 - dummy
@top()
 step 1
@push(meshgen)
@push(coord_system)
*grid u domain
-17 0
*grid_u_spacing
1
*grid_v_domain
0 17
*grid v spacing
1
*set_grid on
*fill view
@pop(1)
@push(plot_screen)
@push(plot1_screen)
@push(divisions screen)
*set_plot_divisions_high
*regenerate
@pop(1)
@pop(1)
@pop(1)
@push(curve type)
*set_curve_type bezier
@pop(1)
*add curves
point(0,14,0)
point(-11,13,0)
point(-11,6,0)
point(-5, 4, 0)
```



```
point(-5, 0, 0)
#
*action | figure 10.3 -
@push(coord_system)
*grid_u_spacing
0.5
*grid_v_spacing
0.5
@pop(1)
*add_curves
point(-9,16,0)
point(-9, 14, 0)
point(-10.5,13,0)
point(-10.5,8.5,0)
#
*action | figure 10.4 -
@push(plot_screen)
*set_point_labels on
*regenerate
@pop(1)
*add curves
 9
point(-10.5,4,0)
point(-9.5,3.5,0)
point(-8.5, 1.5, 0)
point(-8,0,0)
point(-7.5,0.5,0)
5
#
*action | figure 10.5
*edit_points
 13
-9.5 -0.5 0
 10
-11.0 -0.5 0
 11
-7 5 0
#
*action | figure 10.6
*add_curves
6
```



```
point(-7.5,16,0)
point(-1.5, 17, 0)
point(0,17,0)
#
*action | figure 10.7
*add curves
 17
 1
# | End of List
*action | figure 10.8
 step 2: automatic meshing
*set grid off
@push(automesh)
@push(automesh_planar)
*set_overlay_divisions
20 20
*overlay_mesh
all existing
*action | figure 10.9
@pop(1)
@pop(1)
@push(subdivide)
*sub divisions
2 1 1
*subdivide elements
52 3 5 7 61
# | End of List
*action | figure 10.10
*zoom box
*zoom_box(1,0.399093,0.703562,0.717687,0.986005)
*refine node
 1
 1
# | End of List
*action | figure 10.11
*subdivide elements
 48 47
# | End of List
*action | figure 10.12
```



```
@pop(1)
@push(sweep)
*sweep_nodes
all_existing
*remove_unused_nodes
@pop(1)
@push(renumber)
*renumber all
@pop(1)
@pop(1)
 step 3: create the rim as a rigid body and identify the contact
bodies
@push(meshgen)
*set_grid on
*add curves
 5
point(-12,0,0)
point(-5.5, 2.5, 0)
point(-11,3.5,0)
point(-11,1.5,0)
#
*action | figure 10.13
*set_grid off
@push(move)
*set_move_translations
0 13 0
*move elements
all_existing
*move_curves
all existing
*fill view
@pop(1)
@pop(1)
@push(contact)
@push(contact_bodies)
*new_contact_body
*contact_deformable
@popup(contact_deform_pm)
@popdown(contact_deform_pm)
```



```
*add_contact_body_elements
 1 2 3 4 5 6 7 44 45 46 47 48 49 50 51 52 53 54 55 56 57 58 59 104
105 106
 107 108 109 110 111 112 113 114 115 116 117 118 119 103 43
# | End of List
*new contact body
*contact_rigid
@popup(contact rigid pm)
@popdown(contact_rigid_pm)
*add_contact_body_curves
 6
# | End of List
@push(plot_screen)
*set point labels off
*set points off
*elements_solid
@pop(1)
*identify_contact *regen
*action | figure 10.14
@pop(1)
@pop(1)
 step 4: add boundary conditions
@push(contact)
@push(contact_bodies)
*identify none *regen
@pop(1)
@pop(1)
@push(apply)
@push(plot_screen)
*elements wireframe
*regenerate
@pop(1)
@push(apply mechanical)
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed disp pm)
*apply_dof x
@popdown(fixed_disp_pm)
*add apply nodes
```



```
131 139 143
# | End of List
*action | figure 10.15
  internal pressure
*new_apply
@push(table)
*table name
loading
*set_table_type
@popup(table type pm)
@list(table_type_list,0)
time
@popdown(table_type_pm)
*set_table_xmax
300
*set_table_ymax
220
*table add
0 0
300 214
*show model
@pop(1)
*apply_type edge_load
@set($usersubfv,FORCEM)
@popup(apply_edge_load_popmenu)
*apply_table p0
@set($dof,p0)
@popup(apply_select_table_popmenu)
@list(select_table_list,0)
loading
@popdown(apply_select_table_popmenu)
@popdown(apply_edge_load_popmenu)
*add apply edges
 43:2 44:2 45:2 47:2 48:2 49:2 53:2 56:2 58:2 59:2 60:2 62:2 64:2
66:2 68:2 70:2
 73:2 74:2 77:2 78:2 80:2 81:2 82:2 84:2 85:2 86:2 87:2 88:2 96:2
111:2 112:2
# | End of List
*action | figure 10.16
```



```
@pop(1)
@pop(1)
@push(mat_props)
@push(mat_props2)
*material_type mechanical:mooney
@popup(mat mooney)
*material_value mooney:c10
96.5
-19.3
*material_value mooney:c01
0
*material value mooney:c20
-19.3
*material value mooney:c30
19.3
@popdown(mat_mooney)
*add material elements
all existing
@pop(1)
@pop(1)
@push(loadcase)
@push(lcase_mechanical)
*loadcase type static
@popup(lcase_static)
*loadcase value time
300
*loadcase value nsteps
300
@popdown(lcase static)
@pop(1)
@pop(1)
@push(jobs)
*job class mechanical
@popup(job_mechanical)
@list(job rem lcase list,0)
@list(job_add_lcase_list,0)
*add_job_loadcases lcase1
@popup(job an opts mech popmenu)
*job_option large:on
*job_option follow:on
@popdown(job_an_opts_mech_popmenu)
```



```
@list(job_add_lcase_list,0)
@list(job_rem_lcase_list,0)
@popup(tensor_scalar_popmenu)
@list(post_tensor_list1,0)
@list(post var list1,0)
*add post tensor cauchy
@popdown(tensor_scalar_popmenu)
@list(job add lcase list,0)
@list(job_rem_lcase_list,0)
*job_option dimen:axisym
@popdown(job mechanical)
@push(element types)
@popup(mech_eltype_axisym_solid_pm)
*element type 82
@popdown(mech_eltype_axisym_solid_pm)
all_existing
@push(file screen)
@set($file_browser_label,SAVE AS FILE)
@set($file_browser_command, *save_as_model)
@popup(save file browser popmenu)
@list(save file browser filelist,0)
@list(save_file_browser_dirlist,0)
*save as model c10.mud yes
@popdown(save_file_browser_popmenu)
@pop(1)
@pop(1)
*update job
@popup(job_run_popmenu)
*submit job 1
*monitor job
@popdown(job_run_popmenu)
@pop(1)
@push(results)
*post_open_default
*post next
@push(plot screen)
*set_nodes off
*set faces off
*regenerate
@pop(1)
*action | figure 10.17
```



```
*set_deformed both
@push(results2)
*post_animate_increments
100
1
@pop(1)
*action | figure 10.18
*action | figure 10.19
*post_value
@popup(post_scalar_pm)
Equivalent Cauchy Stress
@popdown(post_scalar_pm)
*post_contour_bands
*action | figure 10.20
@pop(1)
```




Transmission Tower

Chapter Overview

This chapter describes a sample session that illustrates the functionality of the Mentat program through the modeling and analysis of a tower structure. The goal of this chapter is to give you hands-on experience with the following Mentat capabilities.

- To show you how to create a mesh of linear beam elements using the following mesh generation features:
 - user-defined local coordinate systems
 - node and element creation,
 - element subdivision and duplication.
- To demonstrate a static and a modal analysis of a model.
- To view and examine the results of an analysis.

Background Information

Tower Description

The tower is 68 feet high, 18 feet square at the base, 4 feet square at the top, and has 6 cablearms, each 6 feet wide. The tower is made of steel angles (L3x3x1/4 and L2x2x1/4) and is loaded by member self-weight, wind, and cable loads. The feet at the base of the tower are fixed. A sketch of the tower is depicted in Figure 11-1.



Figure 11-1 Transmission Tower



Idealization

By virtue of the element type used in this model, MARC Element Type 52, the sketch in Figure 11-1 and the finite element model itself are identical. This may not be true in all cases. Members of the tower are idealized as beam elements with 6 degrees of freedom at each node (ux, uy, uz, rxx, ryy, rzz).

The wind loads are applied as distributed loads along the main vertical members of the tower. Cable loads are applied as point loads at the end of the cable-arms. Self-weight is applied as distributed loads on all members.

Requirements for a Successful Analysis

The analysis is considered successful if the displacements of the structure, as a result of its external loading, can be determined. The second part of the analysis (modal analysis) is successful if the eigenmodes of the structure can be predicted.

Full Disclosure

Analysis Types

Linear static Modal

Material

Steel, Young's modulus = 4.176e9 psf, Poisson's ratio = 0.3. Mass density = 15.217 slugs/ft³.

Elements

MARC Type 52, three-dimensional, two-noded beam element.

Element Properties

Those obtained from the AISC steel manual. L3x3x1/4Weight = 4.9 lbs/ft. Area = 0.01 ft². Ixx = Iyy = 6.0e-5 ft⁴. L2x2x1/4 Weight = 3.19 lbs/ft.Area = 0.00651 ft^2 . Ixx = Iyy = $2.0e-5 \text{ ft}^4$.

Overview of Steps

- **Step 1** Create the first face of the main tower structure by adding nodes and elements and using user-defined coordinate systems, subdivision, and symmetry.
- **Step 2** Duplicate the first face to create the remaining faces of the main tower structure. It is crucial to sweep nodes and elements after using symmetry and duplicate.
- Step 3 Create one cable arm by adding nodes and elements and using subdivide.
- **Step 4** Use symmetry on the first arm to create the second arm. Then use duplicate on the first two to create the remaining cable arms.
- **Step 5** Add boundary conditions. Specify fixed displacements at nodes to constrain the nodes at the tower base. Apply a gravity load to the elements. Group the elements in sets so that you can easily refer to them later. Add distributed wind loads. Apply cable loads as point loads on the nodes at the ends of the cable arms.
- **Step 6** Define the material and apply it to all elements. Define the geometric properties and apply them to the appropriate elements.
- Step 7 Job submission of the static analysis.
- **Step 8** Static analysis results processing.
- Step 9 Job submission of the modal analysis.
- Step 10 Modal analysis results processing.

Detailed Session Description

When modeling a structure, it is very important to define a coordinate system that can be referred to as you create different parts of the structure. The **global coordinate system**, called the *x-y-z system*, is the coordinate system attached to the earth and can be used for this purpose. The global coordinate system may not always be the optimal choice in Mentat because the design of the program restricts the orientation of the model, particularly when graphical input is desired.

A **local coordinate system** is a set of three independent directions and an origin, defined with respect to the global coordinate system. For easy reference we refer to the local coordinate system as the *u-v-w system*. The nature of this system may be cartesian, cylindrical or spherical according to the commonly accepted definitions. Initially the local coordinate system coincides with the global x-y-z system. We emphasize once again that the position and orientation of the local coordinate system is defined in terms of the global coordinate system. Everything else, except viewpoint, is defined in terms of the local coordinate system.

A Note on Grid Space

If you are using the mouse to input entities such as nodes, you need to relate the twodimensional space of your screen to three-dimensional reality. Choose the u-v plane of the local coordinate system as a plane that is sensitive to mouse picks. You can orient the input grid anywhere in space simply by translating and rotating the local coordinate system. By virtue of the fact that the local u-v-w coordinate system initially lines up with the global coordinate system, the input grid also initially lies in the global x-y plane.



Figure 11-2 Local Coordinate System (u-v-w)

A Note on Viewpoint

The ability to orient the local coordinate system anywhere in space does not necessarily mean that it is optimal for graphical input. The best resolution of the grid is obtained by viewing the grid plane with the eye positioned along the normal to that plane. Use VIEWPOINT in the MANIPULATE CAMERA (ABSOLUTE) menu - which can be entered via the VIEW menu - to define the appropriate eye position measured with respect to the global coordinate system. Once again, we emphasize that viewpoint and the position of the local coordinate system are the only two exceptions to the rule that everything in Mentat is measured in local coordinates. Keep the following three points in mind with respect to viewpoint.

- Changing the viewpoint is not the same as rotating the object that you are viewing. Although the end result may appear to be the same, there is a fundamental difference. Changing the viewpoint does not change the position of the model, while a transformation of the object permanently changes the position of that object.
- Changing the viewpoint is *not* related to changing the local coordinate system. These are two independent actions.
- Changing the position and sense of the local coordinate system only affects the position of entities that have yet to be defined. It does not influence or change the position of entities, such as nodes or points, that have already been defined.

Step 1

As described in previous sample sessions, the first step in building a finite element mesh is to establish an input grid. Activate the grid and set the grid spacing to 1 unit and the grid size to 10 units.

The best approach to use for creating the transmission tower is to align the center line of the structure with the global z-axis. Rotate the local coordinate system about the global x-axis over 90 degrees, and translate it over 9 units in the global y-direction. Set and activate view 2.

MAIN **MESH GENERATION** SET **U DOMAIN** -10 10 **U SPACING** 1 **V DOMAIN** -10 10 **V SPACING** 1 qrid ON (on)ROTATE 90 0 0 TRANSLATE 0 9 0 VIEW activate 2 (on)show 2 FILL PLOT label NODES (on)label ELEMENTS (on)

Prior to adding elements, the user has to change the default element class for newly generated elements from QUAD(4) to LINE (2). Use the ADD button from the ELEMS panel to add the first three elements to form a triangle that will constitute the base of the tower:

```
MAIN

MESH GENERATION

ELEMENT CLASS

LINE (2)

RETURN

elems ADD

node(-9,0,0)

node(-9,10,0)

1

2

3
```

(pick grid point) (pick grid point) (pick grid point) (pick node) (pick node) (pick node)

MESHIGE	NERATI	ON				- 2			. 1.			- 10	ο, .		-	-			Manage.
NODES	ADD	REM	103	SHOW	l							1.							COLUMN T
ILENS.	ADD	REM	THEE	SHOW							/								
PTS	ADD	REM	TICLE	SHOW		· I				÷.,	1 -								
CRVS	ADD	REM	TICLE	SHOW		- 1				-/									
SNES	ADD	REM	1103	SHOW						1.									
501/05	ADD	REM	1	SHOW															
DETWEEP	INDDE	00	WEEN P	THIO		• f			·/·			•				-			
ELEMENT	CLASS	⁺ UN	E (2)	1		· +		. /								-			
CURVE T	VPE	* UN	6	1				/.								-			
SURFAC	ETYPE	" OU	WD					/											
SOLD TY	PE	61	DCK.				· /	,											
COORDI	UNIE SY	STEM					1.												
887	RECIA	NGULA	vi 1=0	IND		. <i>y</i>										-			. ,a-10
CLEAR M	ESH	[cu	AR GEO	364															
ATTACH		* AU	TOMESH	4 *															
CHANGE	CLASS	CH	ECK	1															
CONVER	T	* DU	PLICATE									•							
EXPAND		* INT	ERSECT	r +												-			
MOVE		7 RE	LARC													-			
RENUMB	89.	7 BE	VOLVE																
30U08		1 811	TELCH																
SUBDIVID	Æ	7 8W	EEP														1.1		£
SUPPLY	RY	P																	
(ALL)	BELEC	T. VIS	13188	NUTUNE															× _×
E2857	UNISEL	. [INV	15. 1	URFAC															
SELECT		* E74	O USE M	0						-				• •	-	-		·	· · 2
		-																	
RETURN		MA	н	1	UNDO	SAVE	DRAW	FILL	RESE	WEW	TX	TY	12+	RDD	RYN	RZ+	20064	IN	USER

Figure 11-3 First Three Elements of Tower Base

Having obtained Figure 11-3, continue to subdivide the vertical side and the hypotenuse, and add the two cross members. Notice how you have started to create to the left of the local v-axis. Although this does not look at all like the *base* yet, you will continue to add to the left side of the tower face and use symmetry to complete the first face. Figure 11-4 shows the results of this operation.

```
MAIN

MESH GENERATION

SUBDIVIDE

ELEMENTS

1 2

END LIST (#)

RETURN

elems ADD

5

8

8

6
```

(pick node) (pick node) (pick node) (pick node at upper left position)



Figure 11-4 Left Base of Tower

The next step is to establish a new local coordinate system with a grid spacing of 2 and a grid size of 20 to create the *head* of the tower that is parallel to the local coordinate system of the base but shifted 50 units in z-direction. Figure 11-5 shows the new coordinate system relative to the part of the *base* that you have already defined. Use the SHOW ALL VIEWS option to view the model from the four default angles. View 2 shows that the new local coordinate system is shifted 7 units in the negative y-direction.

MAIN **MESH GENERATION** SET **U DOMAIN** -20 20 **U SPACING** 2 **V DOMAIN** -20 20 **V SPACING** 2 TRANSLATE 0 -7 50 VIEW activate 1 activate 2 activate 4 PERSPECTIVE ACTIVATE ALL SHOW ALL VIEWS FILL show 2 RETURN

(off) (off) (on)



Figure 11-5 Four Views of New Local Coordinate System

Use the technique of generating one master element and subdividing it into 6 equally sized elements to generate the left side of the *head*. Keep the bias factor equal to zero but change the number of subdivisions to 6 in the first direction of the element.

Since line elements are one-dimensional elements, it is not necessary to define the number of elements in the second and third direction. In general, the direction in which the subdivisions are made is dependent on the connectivity; that is, the order in which nodes are entered to create an element.

MESH GENERATION elems ADD node (-2,0,0) node (-2,18,0) SUBDIVIDE DIVISIONS 6 2 2 2 ELEMENTS 10 (the newly generated elem END LIST (#)	MA	AIN																	
elems ADD (Pick from the product of the p		MESH GEN	JFRA		N														
In code (-2,0,0) node (-2,18,0) SUBDIVIDE DIVISIONS 6 2 2 ELEMENTS 10 END LIST (#) SUBDIVIDE DIVISIONS 6 2 2 ELEMENTS 10 END LIST (#)			חח														(Picl	from t	ha arid
ALC: VOULE SUBJECT VOULE SAVE DEAW FEL RESETVENT TO TY TZ FIG FIG RC 2004 HI USER .				0 0													1 100	.]1011 11	ie griu
NOCE (-2,18,0) SUBDIVISIONS 6 2 2 ELEMENTS 10 END LIST (#) SUBDIVISIONS 6 2 2 ELEMENTS 10 END LIST (#) SUBDIVISIONS 7 7 8055 ACCORS 0 0 0 0 0 0 0 0 0 0 0 0 0		node	2 (- 2	,0,0))														
SUBDIVISIONS 6 2 2 2 ELEMENTS 10 END LIST (#) HISTORIE CHARGES 0 0 ELEMENTS CHARGES 0 0 ELEMENTS CHARGES 0 0 ELEMENTS CHARGES 0 0 0 ELEMENTS CHARGES 0 0 0 0 0 0 0 0 0 0 0 0 0		node	e (- 2	,18,	, 0)														
DIVISIONS 6 2 2 ELEMENTS 10 END LIST (#) SUBSOVEE CAVEROUSE 0 0 0 ELEMENTS CAVEROUSE 0 0 0 ELEMENTS CAVEROUSE 0 0 0 ELEMENTS CAVEROUSE 0 0 0 ELEMENTS CAVEROUSE 0 0 0 ELEMENTS CAVEROUSE 0 0 0 ELEMENTS CAVEROUSE 0 0 0 ELEMENTS CAVEROUSE 0 0 0 ELEMENTS CAVEROUSE 0 0 0 ELEMENTS CAVEROUSE 0 0 0 CALENT CA		SUBDIV	IDE																
6 2 2 ELEMENTS 10 END LIST (#)		DIVIS	SION	S															
ELEMENTS 10 END LIST (#)		6	2	2															
10 END LIST (#) SUBDOWDE 0 SWERONS 0 2 0 0 0 <		ELEN	MEN	rs															
ENDLIST (#)		10	0											(1	the 1	newlv	gene	erated e	lement
		E	ND L	IST	(#)									(-			0		
SUBJECT 0 </th <th></th> <th></th> <th></th> <th></th> <th>()</th> <th></th>					()														
Divisions 0	SUBDIVIDE											۰						Maanoo	
2 2 BNS FACTORS 0 0 0 0 0 BLLEMENTS CURVES RESET 0 REFINE 0 FILL YSHELE SELECT YSHELE	DIVISIONS	6				:	1.1		1.1		16			11	1.1	:			
Image: Participant set in the set of the se		2	1								5.								
BIAS FACTORS 0 0 0 BLEMENTS CURVES RESET 0 REFINE 0 ALL SELECT VORELE DUTTING SELECT VORELE VINDO SAVE DELENTIN MAN		2					1.1	1.1	1.1		10.			1	1.1	:			
0 0 0 0 0 0 REFRE 0 ALL SELECT VORSE F 0000000 SELECT VORSE F SELECT VORSE F VENDO SWE DRW FEL RETURN MAN	EWS FACTORS	0									13. I								
0 1/15 BLEMENNES CURVES RESET 1/6 REFINE 1/6 ALL: SELECT VOID LIST OF 1/6 SELECT CONT USI OF REFUNN MAN UNDD SAVE DRAW RETURN MAN		0							1.1		12				1				
BLE MENTS CURVES RESET		0									13								
RESET Image: Select Selec	ELEMENTS	CURVES					1.1	1.1	1.1	• •	118. ·		1.1	11		µ-20			
ALL: SELECT: YORELF OKTURE	RESET		1																
ALL: SELECT YORELE OKITURE SELECT ' DID LIST (P) RETURN MAN ' UNDO SAVE DRAW FILL RESETVIEW TO: TY: TZ: Ros Res Res ZOOM IN USER D	arrar.					:	1.1		1.1					11	1.1	:			
ALL: SELECT VORELE OKTURE DESI UNSEL MANY SLUFFIC SELECT ' DID LIST (M) RETURN MANY ' UNDO SAVE DRAW FILL RESETVEW TIG TY TZ K R66 RT6 RZ ZOOM IN USER T	NAT INC.																		
ALL: SELECT VORELE OUTURE DESI UNSEL NOTES SELECT ' DID LIST (M) RETURN MAN ' UNDO SAVE DRAW FILL RESETVIEW TO: TY: TZ: Ros Res Res ZOOM IN USER - 2 DIVISION - 2 2						:	1				1.1			1	1				
ALL: SELECT YORELS DOTTING SELECT TOD LIST (M) RETURN MAN ' UNDO SAVE DRAW FILL RESETVIEW TIG TY+ TZ+ RG+ RT+ RZ+ ZOOM IN USER T																			
ALL: SELECT VESELS DOTTING DESI UNSEL INFOR SLOF AC SELECT DIS DIS AC RETURN MAN ' UNDO SAVE DRAW FILL RESETVIEW TIG TYA TZA RGA RYA RZA ZODA IN USER '							1.1	1.1	1.1				1.1	1	1.1				
ALL: SFLECT VESELE DUTUES DESI UNSEL INTO SUBFAC SELECT DUSEL INTO SUBFAC RETURN MAIN UNDO SAVE DRAW FILL RESETVIEW TIG TY+ TZ+ RK+ RY+ RZ+ ZOOM IN USER 7																			
ALL: SFLECT VESELE DOTTING DEST UNSEL INTO SUBTAC SELECT ' DID LIST OF RETURN MAIN ' UNDO SAVE DRAW FILL RESETVIEW TIG TY+ TZ+ RK+ RY+ RZ+ ZOOM IN USER '																			
ALL: STLSCT VORELE DUTINE DEST UNSEL INTOL SUBTAC SELECT ' DID LIST OF RETURN MANY ' UNDO SAVE DRAW FILL RESETVIEW TIG TY+ TZ+ RK+ RY+ RZ+ ZOOM IN USER '																			
ALL: STLSCT VENELE OUTDING DEST UNSEL INVESTIGATION SELECT ' DAD LIST OF RETURN MAAN ' UNDO SAVE DRAW FILL RESETVIEW TIG TY+ TZ+ RK+ RY+ RZ+ ZOOM IN USER '																			
ALL: STESCT VENELE COUTING DEST UNSEL OWTH SLOFAC SELECT ' DAD LET (M) RETURN MANY ' UNDO SAVE DRAW FILL RESETVIEW TIG TY+ TZ+ RK+ RY+ RZ+ ZOOM IN USER '																			
ALL: STLISCT VESELE OUTLINE COST UNSEL DIVINE SELECT CONDUCTOR DRAW FILL RESETVIEW TIG TY+ TZ+ RG+ RZ+ ZOOM IN USER CONTENTS																			
ALL: STLECT VERLE ORTHON SELECT ' DAD LIST OF RETURN MAAN ' UNDO SAVE DRAW FILL RESETVIEW T64 TV4 TZ4 R64 R74 R24 ZOOM IN USER '									R.	3	1						- 6		
NUC YEARCH (1998) OPENING N X COST UNDO SAVE DRAW FILL RESET VIEW T64 TV4 TZ4 R64 RV5 R2004 IN USER 2 RETURN MAAN UNDO SAVE DRAW FILL RESET VIEW T64 TV4 TZ4 R64 RV6 REF DRAW UNDO SAVE DRAW FILL RESET VIEW T64 TV4 TZ4 R64 RV6 RV6 RV6 RV6 DRAW VIEW T64 TV4 TZ4 R64 RV6 RV6 <t< td=""><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>12</td><td>52</td><td>pr -</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></t<>									12	52	pr -								
SELECT PRO LOT (P) 2 RETURN MAIN UNDO SAVE DRAW FILL RESETVIEW T64 TY4 TZ4 R64 RY6 RZ4 ZOOM IN USER OF MULTICED 2	Dest Cuenci	AL WORKLA COUTLINE							r.	2							4		
RETURN MAIN UNDO SAVE DRAW FILL RESETVIEW TO THE REAL ROAD TO THE REAL ROAD IN USER	SELECT	" END LIST ON							1										
DIT OF THE OF TH	RETURN	MAIN	UNDO	SAVE	DRAW	FILL	B	ESETA	NEW.	TK	TY4	17.4	FX4	Re-	RZ+	ZOOM	IN	USER	
UTLS' FILES' FLOT' VIEW ' DYNLMODEL TX- TY- TZ- RX- RY- RZ- OW OUT CEPHED			UTLS'	FILES '	PLOT	VIEW	r F	DVN. N	RODEL	TK-	TY-	12-	RX-	RY-	82-	80%	OUT	DEFINED	

Figure 11-6 Creating the Left Side of the Tower Head

The next step is to generate the sloping mid-section. Set the grid spacing to 1 and the grid size to 50 to define yet another local coordinate system to connect the two coordinate systems that were already defined.

Use ALIGN to line up the local u and v axes so that the plane spanned by these two axes contains the origins of both previous coordinate systems. This way you can use the grid to define the mid-section elements.

```
MAIN
  MESH GENERATION
    SET
       ALIGN
         0 9 10
         -9 9 10
         -2
             2
               50
       U DOMAIN
         -50 50
       U SPACING
         1
       V DOMAIN
         -50 50
       V SPACING
         1
       VIEW
         show 3
         RETURN
```

(upper right of lower section) (upper left of lower section) (bottom node of top section)



Figure 11-7 Local u and v Axes Aligned

The subdivisions along the mid-section are not equally spaced. This is where the **bias factor** can be used to successfully generate a weighted subdivision. The following example illustrates the theory behind the bias factor.

Suppose you want to subdivide the line element in Figure 11-8 so that the length of the elements are biased towards the right. The desired number of subdivisions is assumed to be 4 and a local coordinate t is introduced, which ranges from -1 at the first node to +1 at the second node. An unbiased subdivision would produce 4 elements with their respective nodes located at t = -1, t = -1/2, t = 0, t = +1/2 and t = +1. A biased subdivision relocates these nodes using the formula

$$t^* = t + b(1 - t^2)$$

where *b* is the bias factor and t^* is the biased local coordinate.

Using a bias factor b = 0.5 will result in nodes at $t^* = -1$, $t^* = -1/8$, $t^* = +1/2$, $t^* = +7/8$ and $t^* = +1$. The unconditionally valid range of the bias factor is $-0.5 \le b \le +0.5$.



Figure 11-8 Subdivision of Line Element (Biased to the Right)

The mid-section consists of a biased and an unbiased part (LO-mid-section and HI-midsection). Continue to generate the tower creating two elements and subdividing them, once using a bias factor of 0.2 and once using a bias factor of 0.0. MAIN **MESH GENERATION** VIEW show 2 RETURN nodes ADD 0 13 0 elems ADD 6 12 **SUBDIVIDE DIVISIONS** 2 1 1 BIAS -0.2 0 0 ELEMENTS 17 END LIST (#) DIVISIONS 5 1 1 BIAS 0.2 0 0 ELEMENTS 19 END LIST (#) DIVISIONS 4 1 1 BIAS 0 0 0 **ELEMENTS** 18 END LIST (#)

(pick grid point)

(pick upper left node of lower section) (pick bottom node of top section)



Figure 11-9 Subdivided Elements in the Mid-Section

As a next step, the lower-mid-section is completed and cross members are added in this part.

```
MAIN

MESH GENERATION

elems ADD

33

19

19

29

SUBDIVIDE

DIVISIONS

3 1 1

ELEMENTS

30

END LIST (#)
```

(pick middle node of mid-section)
 (pick auxiliary node at u= 0)
 (pick auxiliary node at u= 0)
 (pick upper left node of base)

MAIN **MESH GENERATION** elems ADD 34 32 32 35 35 31 31 36 36 30 PLOT label ELEMENTS REGENERATE RETURN

(pick auxiliary node at u= 0) (almost horizontal to the left) (work the diagonals)

(off)



Figure 11-10 Completed Mid-Section



Step 2

The stage is now set for a symmetry operation. Please note that this operation is always carried out in the local coordinate system. The point with coordinates (0,0,0) and the normal vector (1,0,0), both in the U,V,W coordinate system, define the symmetry plane. These are the default settings for Mentat.



Figure 11-11 Cross Section of Tower Member

The object of the symmetry operation is to establish the key points in space so that you can complete one face of the tower. Use the existing points and click on the nodes in succession to add the cross members. Note that the program assumes you want to add new elements until you instruct it otherwise. Note also that it is not necessary to activate the 'elems ADD' button every time you enter a new element.

MAIN MESH GENERATION elems ADD

> PLOT label NODES REGENERATE RETURN

(first generate the horizontal cross member between nodes 33 and 64, then work the diagonals in the high-mid-section)

(off)



Figure 11-12 Tower Face Completed

The next step is to duplicate the geometry of one tower face three times to complete the structure. There are some remarks to be made prior to executing this step:

- Perform a sweep operation on all nodes in order to remove the duplicate nodes from the model.
- The current local coordinate system is not ideal to use for specification of the point about which to duplicate nor for specification of the rotation vector. Reset the coordinate system so that it lines up with the global coordinate system: make sure the u-direction is parallel to the x-direction, the v-direction parallel to the y- direction, and the w-direction parallel to the z-direction.

Now duplicate the face you just created. The rotation duplication vector should be 90 degrees in the z-direction and the point of duplication 0, 0, 0. Figure 11-13 shows the results of the duplication operation from 4 different viewpoints.

```
MAIN
  MESH GENERATION
     SWEEP
       sweep NODES
          all: EXIST.
       RETURN
     SET
       RESET
       grid ON
       RETURN
     DUPLICATE
       ROTATIONS
          0 0 90
       REPETITIONS
          3
       ELEMENTS
          all: EXIST.
       VIEW
          SHOW ALL VIEWS
          FILL
          RETURN
```

(to switch off grid)



Figure 11-13 Four Views of Tower Face Duplication

The various operations you have just executed may have left duplicate nodes, that is nodes with different identification numbers but occupying the same space. In finite element terms, these nodes are not connected which may introduce undesirable mechanisms in the structure.

Use the **SWEEP** processor to eliminate the duplicate nodes that occupy the same location into one node with a single identification number. Since this involves a comparison of real coordinates that cannot be done exactly in a computer, nodes are swept together if they are within a certain tolerance from each other. This tolerance can be changed from its default value. Be careful when adjusting the tolerance as too large a tolerance can collapse the entire structure into a single point. MAIN MESH GENERATION SWEEP sweep NODES all: EXIST. sweep ELEMENTS all: EXIST.

Now that you have generated the gross anatomy of the tower structure, it is useful to identify parts of the structure by name. The concept of **set naming** is a very powerful tool; it can be used in any place where a list is required and allows you to manipulate a group of entities rather than individual entities.



Figure 11-14 Boxes Used for Element Set Selection

11 Transmission Tower

You have already defined the base, mid-section, and head of the tower. Use the Box Pick Method to fence off the different portions of the mesh. The STORE ELEMENTS command in the SELECT menu prompts for a set name first, followed by a list of elements. Position the cursor at one corner of the portion of elements to be fenced off. Depress the left mouse button. Drag the cursor to the opposite corner of the box and release. You have just selected every item that is inside the box, indicated by a change in color. The extent of the box are $+\infty$ and $-\infty$ in the direction perpendicular to the screen.

MAIN MESH GENERATION VIEW show 2 RETURN SELECT elements STORE base

END LIST (#) MAIN DEVICE picking PARTIAL SELECT elements STORE mid_sect_lo

> END LIST (#) RETURN picking COMPLETE SELECT elements STORE mid_sect_hi

> > END LIST (#) elements STORE head

> > > END LIST (#)

(Box Pick the elements in the base of the tower, see Figure 11-14)

(Box Pick the elements, realizing that all elements partially within the box will also be included)

(Box Pick the elements)

(Box Pick the elements)



Figure 11-15 Element Sets Created

The current structure still contains a number of mechanisms that need to be eliminated by adding members. For example, the legs of the base contain mechanisms that are eliminated by adding cross members. In order to accomplish this, limit the *visible* elements to the base. Proceed to the SELECT submenu and click SELECT SET, activate the desired set and click MAKE VISIBLE. Only the elements contained in the set 'base' and their nodes remain visible. The base only occupies a small portion of the screen. Select view 4 for display and FILL the screen.

The current view point is perhaps not optimal. Nodes may overlap, making it impossible to pick nodes at some locations. Use *dynamic viewing* to change the position of the mesh so that you can add the additional cross-members.

Dynamic viewing can be switched on by clicking DYN. VIEW in the static menu. Next, position the cursor in the middle of the graphics area and hold down <**MM**>. Move the mouse to the left or right and see how the mesh rotates about the screen axis.

As an alternative, in the button sequence the model is rotated around its z-axis using one of the buttons in the MANIPULATE MODEL menu.

Use the basic element ADD command to add the cross members. Repeat this for parts in the lower mid-section.

Note: Elements can only be added when dynamic viewing is off. If you try to add an element with dynamic viewing on, the result will be a null operation.

MESH GEI	NERAD	DN															
NODES	ADD	REM	EDIT	SHOW	1												
BLEMS	ADD	REM	EDIT	SHOW													
PTS	ADD	REM	EDIT	SHOW													
CHVS	ADD	REM	EDIT	SHOW	1												
SRES	ADD	REM	EDIT	SHOW	1					~	*	-					
SOUDS	ADD	REM		SHOW					_	_	41		-				
BETWEEN	NODE	861	WEEN P	POINT					N.	- /		\		- 74			
BLEMENT	CLASS	" UN	(2)	Ť				/		1.	1	$\langle \cdot \rangle$			_		
CURVE TO	PE .	' [UN		T			~	-	/			24	5		\		<u> </u>
SURFACE	TYPE	1 00	Ø	+			K			\geq	15	\leq			1		X
SOUDTY	PE	· 81.0	CK.	Ŧ			1	< h			fe					/	11
COORDIN	ATE 51	STEM					17	×								r.	(
SET 1	RECTA	NGULA	a i fin	GRID			Jan (/	X					1	~	\mathbb{N}^{\times}	\$
		1 mil						812					1	1		.X-	1
CLEAR M	1.5/1		AR GEG				-17	×**.		_		1		/		K/	1
ALLWER .	C1 + E E	- A0	Central CFL				16		\	_	_ar			/		1	4
COMPARENT	1	- DV	1.01.070							/	$ \frown $	-	_/				_
EXPLAND		1 100	BRECH						1.42				_64 -				
MOVE		1 00	614 614	-					7	-		1	7				
RENIME	FR	1 80	DEVE	-						1	-Ver	1					
SOUDS		1 819	ETCH	-						/		/					
SUBDIVID	Æ	544	2.7*	P.							17	(
SYMMETR	ŧv.	+			1						VL.						- 1
ALL:	SELEC	1. VB	RE 1	NUTLINE							8-0						
Dest.	UNSEL	. 1011	0. 1	SURFAC													*
		T FM	LISTO	0													
SELECT							2		I menerated	Land	Laur	I ma	[max	1	1	1	
RETURN		MA	N	1	UNDO	SAVE	DRAW	PILL .	HESET NEW	TR ₀ +	TY+	TZ+	P(X+	RY-	RZ+	200M	EM .

Figure 11-16 Elements to be Added in the Base of the Tower

MAIN **MESH GENERATION** PLOT label NODES (on)RETURN SELECT SELECT SET base OK MAKE VISIBLE RETURN VIEW show 4 FILL activate 1 (off)activate 2 (off) activate 3 (off)MANIPULATE MODEL rotate in model space Z-RETURN RETURN elems ADD (add elements as indicated in Figure 11-16) SELECT elements STORE base all: VISIBLE SELECT SET mid_sect_lo OK MAKE VISIBLE RETURN FILL elems ADD (add elements as indicated in Figure 11-17) SELECT elements STORE mid_sect_lo all: VISIBLE



Figure 11-17 Elements to be Added in the Lower Mid-Section

Step 3

The next step is to generate the arms of the tower from which the high voltage cables are suspended. Make the *head* of the tower visible so that you can easily pick nodes with the mouse. The position of the node at the extreme end of the arm is known and added through the node ADD command. Use the mouse pick to create the elements between the node at the extreme end and the different portions of the head.



Figure 11-18 Elements to be Added

MAIN **MESH GENERATION** SELECT SELECT SET head OK MAKE VISIBLE RETURN FILL **ZOOM BOX** VIEW MANIPULATE MODEL rotate in model space Z-RETURN RETURN nodes ADD -8 0 63

elems ADD

(zoom in on top section)

(click 7 times)

(Add 4 elements according to Figure 11-18)



Figure 11-19 Elements to be Added



(Pick the 4 elements just created)

(Add 9 elements according to Figure 11-19)



Figure 11-20 Box Pick of Transmission Tower Arm

Store the elements of the arm in an element set called arm for later reference.

MAIN DEVICE picking PARTIAL SELECT elements STORE arm

> END LIST (#) RETURN picking COMPLETE

(Box Pick the elements in the arm)



Step 4

Use symmetry and duplicate to reproduce the arm on either side of the tower head.

MAIN MESH GENERATION DUPLICATE RESET ROTATIONS 0 0 180 ELEMENTS arm FILL RESET TRANSLATIONS 0 0 -6 REPETITIONS 2 **ELEMENTS** arm RETURN SWEEP sweep NODES all: EXIST. SELECT **ELEMENTS** all: EXIST. MAKE VISIBLE FILL RETURN PLOT label NODES MORE **IDENTIFY** SETS REGENERATE NONE

(*off*)

You have now completed Steps 1 through 4 outlined in *Overview of Steps* on page 11-4. The goal of these four steps was to show you the importance of defining a local coordinate system and using it to add, duplicate, and symmetry parts of the tower structure.



Figure 11-21 Completed Transmission Tower Model

This section outlined the advantage of using the symmetry of a structure in twodimensional before creating a three-dimensional structure using the duplication process.

You have been using the direct meshing technique described in Chapter 3 to model the transmission tower. At this point, you have only completed the geometrical part of the finite element mesh. The next step is to specify the *correct* boundary conditions and loads.
There are basically two types of boundary conditions for this model: kinematic and load.

- **1.** Kinematic: The base of the structure is attached to the ground.
- 2. Load: Gravity
 Point loads are applied to the cable arms to simulate the weight of the cables.
 Wind loads are applied as distributed loads perpendicular to the tower.

Chapter 3 discussed how the application of boundary conditions is equal to the process of finding the answer to the question: "Apply *what*, *where*, and *when*"

First concentrate on what.

The **BOUNDARY CONDITIONS** button in the main menu reveals a submenu that allows you to specify mechanical boundary conditions.

Kinematic Boundary Conditions

Click on FIXED DISPLACEMENT. A pop-up appears over the graphics area. Constrain the first three degrees of freedom by clicking the ON button.

Note that while the pop-up is activated your view of the graphics area is obstructed and all other buttons of the regular menu are inactive. You must confirm the values entered in the pop-up, by clicking on the OK button, before you can access the regular menu again.

Now that you have answered *what*, you can concentrate on *where*.

If you limit your view to the base, you can easily pick on the nodes that attach the structure to the ground. This operation completes the *where* and ties it to the *what* portion of the equation. Since the problem is time independent, the equation is complete because there is no need to answer *when*. The application of *what* is confirmed by the display of arrows in the direction it was applied.

MAIN	
BOUNDARY CONDITIONS	
MECHANICAL	
NAME	
bolts	
SELECT	
SELECT SET	
base	
OK	
MAKE VISIBLE	
FILL	
RETURN	
FIXED DISPLACEMENT	
ON x displace	(<i>on</i>)
ON y displace	(<i>on</i>)
ON z displace	(<i>on</i>)
OK	
nodes ADD	
	(pick 4 nodes at base)
END LIST (#)	



Figure 11-22 Boundary Conditions

For future reference, it is useful to group elements and store them in a set that can be referenced later by name. We already mentioned that not all members are of the same geometry. Use the SELECT option to group all elements that are L3x3 angles. Store this group in a set called L3x3. All other members are L2x2 angles. Generate a list of these elements by inverting the previous list and storing them in a set called L2x2.



Figure 11-23 Elements to be Contained in Set L3x3

MAIN MESH GENERATION SELECT elements STORE L3x3 END LIST (#) SELECT SET mid_sect_lo OK MAKE VISIBLE FILL elements STORE L3x3

END LIST (#)

(pick elements from base)

(pick elements from mid_sect_lo)

MAIN **MESH GENERATION** SELECT SET mid_sect_hi OK MAKE VISIBLE FILL elements STORE L3x3 END LIST (#) SELECT SET head OK MAKE VISIBLE FILL elements STORE L3x3 END LIST (#) SELECT SET arm OK MAKE VISIBLE FILL elements STORE L3x3 END LIST (#) **ELEMENTS** all: EXIST. MAKE VISIBLE FILL **ELEMENTS** all: EXIST. select mode AND L3x3 elements STORE L2x2all: SELECT.

(pick elements from mid_sect_hi)

(pick elements from head)

(pick elements from arm)

MAIN MESH GENERATION SELECT CLEAR SELECT RESET PLOT MORE IDENTIFY SETS REGENERATE NONE REGENERATE



Figure 11-24 Identify Sets

Load (Gravity)

Similar to the FIXED DISPLACEMENT button, when you activate the GRAVITY LOAD button, a pop-up appears over the graphics area where you can enter appropriate values of the gravity load. The program expects the magnitude of the gravity acceleration in the negative z direction here. This answers the *what* portion of the equation. Use the all: EXIST. button to answer the *where* part of the equation.

```
MAIN
BOUNDARY CONDITIONS
MECHANICAL
RESET VIEW
FILL
NEW
NAME
gravity
GRAVITY LOAD
Z ACCEL.
-32.2
OK
elements ADD
all: EXIST.
```

Wind Load

Assume that the transmission tower is subjected to a wind load with a stronger load applied to the upper part of the tower and a weaker load to the lower part of the tower. Simulate the wind loads by applying a distributed load in y-direction. Assume that only one face of the tower is loaded by this wind load.

Prior to applying the loads element sets of all elements in the lower and upper frontal face of the tower will be generated. The upper portion is stored in an element set called hi_front while the lower portion is stored in a set called lo_front.



Figure 11-25 Gravity Load for the Structure



Figure 11-26 Box Pick from base for lo_front



Figure 11-27 Polygon Pick from mid_sect_lo for lo_front



Figure 11-28 Polygon Pick from mid_sect_hi for hi_front



Figure 11-29 Box Pick from head for hi_front



Figure 11-30 Box Pick from arm for hi_front

MAIN **BOUNDARY CONDITIONS MECHANICAL** SELECT VIEW show 1 RETURN SELECT SET base OK MAKE VISIBLE elements STORE lo_front (select according to Figure 11-26) END LIST (#) SELECT SET mid_sect_lo OK MAKE VISIBLE elements STORE lo_front (select according to Figure 11-27) END LIST (#) SELECT SET mid_sect_hi OK MAKE VISIBLE elements STORE hi_front (select according to Figure 11-28) END LIST (#)

Complete the set hi-front by processing the sets 'head' and 'arm' in an identical way.

Now actually apply the loads:





Figure 11-31 Strong Wind Load

In an identical way, a Y FORCE of -80 can be applied to all elements contained in lo_front.



Figure 11-32 Weak Wind Load

Point Loads

The cables are suspended from the arms of the tower and are simulated as point loads hanging from each tip of the arm. A load of -500 in this direction is applied to each of the six arm extremities. The boundary conditions menu allows you to enter these point loads through the POINT LOADS option. The already familiar pop-up appears over the graphics area. Enter the values in the appropriate fields.

Use the mouse to pick the nodes that are to receive a load. Enter the end of list character (#) after you have picked the 6 nodes.

MAIN BOUNDARY CONDITIONS MECHANICAL NEW NAME cable_load POINT LOAD Z FORCE -500 OK nodes ADD

END LIST (#)

(pick the 6 nodes on the tip of the arms)



Figure 11-33 Cable Loads



Geometric Properties

The finite element analysis program requires you to specify the properties such as the material type and area of the members. Use the **GEOMETRIC PROPERTIES** processor to enter the area. So far, you have already separated the L3x3 angles from the L2x2 angles and can refer to them by set name. The L3x3 angles have an area of 0.01 while the L2x2 angles have an area of 0.00651. The moments of inertia about the x and y axis are equal and must be entered here. The data is listed under *Element Properties* on page 11-3 of this chapter. However, beam elements require some additional geometric data defining the direction of the local x and y axis. This data can also be entered through the **GEOMETRIC PROPERTIES** processor.

The local x and y axis of all elements not exactly pointing in z-direction can be defined using a reference vector (0, 0, 1). For all elements pointing in z-direction we will use a reference vector (1, 0, 0), although the reality may be more complex. This requires the definition of a set of elements called "upright" that contains all elements pointing in z-direction. As demonstrated previously, the user can make the sets base, mid-sect-lo, mid-sect-hi, head and arms visible one after the other, select the upright elements and add those to the stored list. There is a total of 44 such elements to be found. (8 in base, 24 in head and 12 in arm).

MAIN	
MESH GENERATION	
PLOT	
draw NODES	(off)
RETURN	
SELECT	
SELECT SET	
base	
MAKE VISIBLE	
FILL	
elements STORE	
upright	
	(pick the 8 elements pointing in z-direction)
END LIST (#)	
SELECT SET	
head	

(*etc.*, *etc.*)

After having selected all 44 elements, make all elements visible again.

MAIN **GEOMETRIC PROPERTIES** 3-D NEW NAME L3x3_z **ELASTIC BEAM** AREA 0.01 6.0e-05 6.0e-05 0 0 1 OK SELECT SELECT SET L3x3 OK select mode AND SELECT SET upright OK RETURN elements ADD all: SELECT. **ID GEOMETRIES**

(for Ixx) (for Iyy) (for direction) (for direction) (for direction)

(to switch to EXCEPT)

(*on*)

MAIN **GEOMETRIC PROPERTIES** 3-D NEW NAME $L3x3_x$ **ELASTIC BEAM** AREA 0.01 6.0e-05 6.0e-05 1 0 0 OK SELECT **CLEAR SELECT** RESET SELECT SET upright OK select mode AND SELECT SET L2x2OK RETURN elements ADD all: SELECT.

(for Ixx) (for Iyy) (for direction) (for direction) (for direction)



Figure 11-34 Geometric Properties Assignment for L3x3 Angles

MAIN **GEOMETRIC PROPERTIES** 3-D NEW NAME $L2x2_z$ **ELASTIC BEAM** AREA 0.00651 2.0e-05 2.0e-05 0 0 1 OK SELECT **CLEAR SELECT** RESET SELECT SET L2x2OK select mode AND SELECT SET upright OK RETURN elements ADD all: SELECT.

(for Ixx) (for Iyy) (for direction) (for direction) (for direction)

MAIN **GEOMETRIC PROPERTIES** 3-D NEW NAME $L2x2_x$ **ELASTIC BEAM** AREA 0.00651 2.0e-05 2.0e-05 1 0 0 OK SELECT **CLEAR SELECT** RESET SELECT SET upright OK select mode AND SELECT SET L3x3 OK RETURN elements ADD all: SELECT. SELECT **CLEAR SELECT** RESET

(for Ixx) (for Iyy) (for direction) (for direction) (for direction)



Figure 11-35 Geometric Properties Assignment for L2x2 Angles

Material Properties

The last step is to assign material properties. The members of this tower are made out of steel. For this analysis, you need to specify the Young's Modulus and Poisson's Ratio and mass density, all located in the MATERIAL PROPERTIES menu. Assign this material to all existing elements.

MAIN MATERIAL PROPERTIES NEW NAME steel ISOTROPIC YOUNG'S MODULUS 4.176e9 0.3 15.217 OK elements ADD all: EXIST. ID MATERIALS ID MATERIALS

(for Poisson's Ratio) (for mass density)

> (on) (off)



Figure 11-36 Material Properties Assignment



Job Submission of the Static Analysis

Finally, you submit the job. This is easily done in the JOBS menu. SUBMIT submits the job in the background. The status of the job can be checked or monitored continuously. Once you have successfully submitted the job, you must carefully analyze the results.

MAIN JOBS **NEW** NAME static MECHANICAL **INITIAL LOADS** (default: all loads selected) OK OK ELEMENT TYPES mechanical elements 3-D TRUSS/BEAM 52 (LINE(2), THIN ELASTIC BEAM) OK all: EXIST. RETURN CHECK SAVE RUN SUBMIT 1 MONITOR

Step 8

Static Analysis Results Processing

The static analysis considers the wind load, gravitational load, and point loads. The structure will undergo a bending out of the x-z plane as a result of the wind load. Figure 11-37 shows the results. We switched on the **automatic deformation scaling** option resulting in an exaggerated display of the displacements.

MAIN RESULTS OPEN DEFAULT VIEW show 3 RETURN DEF & ORIG NEXT INC deformed shape SETTINGS AUTOMATIC FILL



Figure 11-37 Deformation of Tower under Combined Load



Job Submission of the Modal Analysis

To run the modal analysis, it is necessary to restore the model file. After a new, dynamic, loadcase has been defined, a new job is created and submitted.

MAIN RESULTS CLOSE deformed shape OFF RETURN FILES RESTORE RETURN LOADCASE NEW NAME dynamic DYNAMIC MODAL **# MODES** 15 OK RETURN JOBS NEW NAME dynamic **MECHANICAL** loadcases SELECT dynamic OK CHECK SAVE RUN SUBMIT 1 MONITOR



Modal Analysis Results Processing

Open the results file by clicking the RESULTS button from the main menu, followed by the OPEN DEFAULT button. The modal shapes are stored in subincrements and can be accessed through the NEXT INC button. As is demonstrated in previous chapters, it is useful to animate the different modal shapes. Figure 11-38 and Figure 11-39 display examples of mode shapes found during this analysis.

The postprocessing can be carried out as follows:

MAIN RESULTS **OPEN DEFAULT** VIEW show 4 RETURN **DEF & ORIG NEXT INC NEXT INC** deformed shape SETTINGS **AUTOMATIC** FILL RETURN DEF ONLY MORE animate MODE 15 ANIMATION FILL REPEAT **PLAY** STOP SHOW MODEL RETURN PREVIOUS SKIP TO INC 0:15 **DEF & ORIG** FILL



Figure 11-38 Eigenmode of Tower, f = 5.366 Hz



Figure 11-39 Eigenmode of Tower, f = 20.64 Hz

Conclusion

This structure is an example where automatic mesh generators cannot be utilized to create a finite element model. It is demonstrated in this chapter that by using the 'conventional' tools available in Mentat, a fairly complicated mesh can be generated without any difficulty.

The displacements, as a result of a load in negative y direction, shown in Figure 11-37 are as expected. The results of the modal analysis can be fully appreciated by animation of the different modes.

Procedure File

```
| Version : Mentat 3.3
*alias action
*exec_p gen_image.proc
*action | figure 11.0 - dummy
*action | figure 11.1 - dummy
*action | figure 11.2 - dummy
@top()
 step 1
@push(meshgen)
@push(coord_system)
*grid u domain
-10 10
*grid_u_spacing
1
*grid_v_domain
-10 10
*grid v spacing
1
*set_grid on
*system rotate
90 0 0
*system_translate
0 9 0
@push(viewing screen)
*show_view 2
*fill view
@push(plot_screen)
*set_node_labels on
*set_element_labels on
@pop(1)
@pop(1)
@pop(1)
@push(element_class)
*set_element_class line2
@pop(1)
*add elements
node( -9.0000000000e+00, 0.000000000e+00,
```

```
0.00000000000e+00)
node( -9.000000000000e+00,
                             1.00000000000e+01,
0.00000000000e+00)
 node( 0.00000000000e+00, 1.0000000000e+01,
0.00000000000e+00)
 1
 2
 3
*action | figure 11.3
@push(subdivide)
*subdivide elements
 1
 2
# | End of List
@pop(1)
*add_elements
 5
 8
 8
 6
*action | figure 11.4
@push(coord_system)
*grid_u_domain
-20 20
*grid_v_domain
-20 20
*grid_u_spacing
2
*grid_v_spacing
2
*system_translate
0 -7 50
@push(viewing_screen)
*show_view 4
*fill view
*view_perspective
*show_all_views
*fill view
@pop(1)
*action | figure 11.5
@push(viewing_screen)
```

```
*show_view 2
@pop(1)
@pop(1)
*add_elements
node( -2.0000000000e+00, 0.0000000000e+00,
0.000000000000e+00)
node( -2.00000000000e+00, 1.8000000000e+01, 2.220446049250e-
16)
@push(subdivide)
*sub_divisions
6 2 2
*subdivide elements
 10
# | End of List
*action | figure 11.6
@pop(1)
@push(coord_system)
*system_align
0 9 10
-9 9 10
-2 2 50
*grid_u_spacing
1
*grid_v_spacing
1
*grid_u_domain
-50 50
*grid_v_domain
-50 50
@push(viewing_screen)
*show view 3
@pop(1)
*action | figure 11.7
*action | figure 11.8 - dummy
@push(viewing screen)
*show view 2
@pop(1)
@pop(1)
*add nodes
0 13 0
*add_elements
```
11 Transmission Tower

б 12 @push(subdivide) *sub_bias_factors -.2 0 0 *sub divisions 2 1 1 *subdivide_elements 17 # *sub_divisions 511 *sub_bias_factors .2 0 0 *subdivide_elements 19 # | End of List *sub divisions 4 1 1 *sub_bias_factors 0 0 0 *subdivide_elements 18 # | End of List *action | figure 11.9 @pop(1) *add_elements 33 19 19 29 @push(subdivide) *sub divisions 3 1 1 *subdivide elements 30 # @pop(1) *add_elements 34 32

```
32
35
35
31
31
36
36
30
@push(plot_screen)
*set_element_labels off
*regenerate
@pop(1)
*action | figure 11.10
  step 2
@push(symmetry)
*symmetry_elements
all_existing
*action | figure 11.11
@pop(1)
*add_elements
33
64
64
24
24
56
56
26
26
58
58
28
28
47
47
14
14
49
49
```

```
@pop(1)
@push(duplicate)
*set_duplicate_rotations
0 0 90
*set_duplicate_repetitions
3
*duplicate_elements
all existing
@push(viewing_screen)
*show_all_views
*fill view
@pop(1)
*action | figure 11.13
*action | figure 11.14 - dummy
@pop(1)
@push(sweep)
*sweep_nodes
all_existing
*sweep_elements
all existing
@pop(1)
@push(viewing_screen)
*show view 2
@pop(1)
@push(select_screen)
 create a set of elements contained in the base of the structure
*show view 2
*store elements
base
 3 4 5 6 7 8 9 39 40 41 42 43 44 45 94 95 96 97 98 100 101 103 104
105 106 107
 108 109 110 111 112 113 114 187 188 189 196 197 198 199 200 201 202
203 204 205
 206 207
# | End of List
create a set of elements contained in the lower mid-section of the
structure
```

```
*pick_inside_partial
*store_elements
mid sect lo
 25 26 27 28 29 31 32 33 34 35 36 37 38 57 58 59 60 61 62 63 64 65
66 67 68 69
148 149 151 152 154 155 157 158 160 161 162 163 164 165 166 167 168
169 170 171
172 173 174 175 176 177 178 179 180 181 182 183 184 185 186 253 254
255 256 257
 258 259 260 261 262 263 264 265 266 267 268 269 270 271 272 273 274
275 276 277
 278 279
# | End of List
create a set of elements contained in the higher mid-section of the
structure
*pick_inside_complete
*store_elements
mid sect hi
20 21 22 23 24 52 53 54 55 56 70 71 72 73 74 75 89 90 91 92 93 133
134 136 137
139 140 142 143 145 146 280 281 282 283 284 285 286 287 288 289 290
291 292 293
 294 295 296 297 337 338 339 340 341 342 343 344 345 346 347 348 349
350 351
# | End of List
 create a set of elements contained in the head of the structure
*store_elements
head
 11 12 13 14 15 16 46 47 48 49 50 51 76 77 78 79 80 81 82 83 84 85
86 87 88 115
116 118 119 121 122 124 125 127 128 130 131 298 299 300 301 302 303
304 305 306
 307 308 309 310 311 312 313 314 315 316 317 318 319 320 321 322 323
324 325 326
 327 328 329 330 331 332 333 334 335 336
# | End of List
*identify_sets *regen
```

```
*action | figure 11.15
*identify_none *regen
@push(plot_screen)
*set_node_labels on
@pop(1)
*select sets
@popup(select_set_popmenu)
@list(select_set_list,0)
base
@popdown(select_set_popmenu)
*visible selected
@push(viewing screen)
*show_view 4
*fill view
@pop(1)
@pop(1)
*action | figure 11.16
@push(viewing_screen)
*activate_view 1
*activate view 2
*activate view 3
*deactivate_view 3
*deactivate view 2
*deactivate_view 1
*rot_model_cspace_z_rev
@pop(1)
*add elements
146 8
80
147
81
148
82
44
74
3
3
76
76
75
75
```

74 3 75 74 76 @push(select_screen) *store_elements base all_visible *select_sets @popup(select_set_popmenu) @list(select_set_list,0) mid_sect_lo @popdown(select_set_popmenu) *visible selected *fill_view @pop(1) *action | figure 11.17 *add_elements 134 198 197 36 67 136 135 199 194 35 131 195 132 196 133 66 104 19 19 106 106 105

```
105
104
104
106
19
105
@push(select_screen)
*store elements
mid_sect_lo
all_visible
*select sets
@popup(select_set_popmenu)
@list(select_set_list,0)
head
@popdown(select_set_popmenu)
*visible_selected
*fill view
@push(viewing_screen)
@push(man_model_screen)
*rot model mspace z rev
*rot_model_mspace_z_rev
*rot_model_mspace_z_rev
*rot_model_mspace_z_rev
*rot_model_mspace_z_rev
*rot_model_mspace_z_rev
*rot_model_mspace_z_rev
@pop(1)
@pop(1)
@pop(1)
*add nodes
-8 0 63
*action | figure 11.18
*add elements
200
101
200
17
200
86
200
11
```

```
@push(subdivide)
*sub_divisions
2 1 1
*subdivide_elements
378
379
377
376
#
@pop(1)
*action | figure 11.19
*add elements
202 208
208 211
211 205
205 202
202 209
209 211
211 206
206 202
202 211
@push(viewing_screen)
*show view 2
*fill view
@pop(1)
@top()
@push(device)
*pick_inside_partial
@pop(1)
@push(meshgen)
@push(select_screen)
*action | figure 11.20
*store elements
arm
 380 381 382 383 384 385 386 387 388 389 390 391 392 393 394 395 396
# | End of List
@pop(1)
@pop(1)
@push(device)
*pick_inside_complete
@pop(1)
```

```
@push(meshgen)
  step 4
@push(duplicate)
*duplicate_reset
*set_duplicate_rotations
0 0 180
*set_duplicate_repetitions
1
*duplicate_elements
arm
#
*fill view
*duplicate reset
*set_duplicate_translations
0 0 -6
*set_duplicate_repetitions
2
*duplicate elements
arm
@pop(1)
@push(sweep)
*sweep_nodes
all_existing
@pop(1)
@push(select_screen)
*select_elements
all existing
*visible_selected
*fill view
@push(plot_screen)
*set node labels off
@push(plot1_screen)
@push(identify screen)
*identify_sets
*regenerate
*action | figure 11.21
*identify_none
@pop(1)
@pop(1)
```

```
@pop(1)
@pop(1)
@pop(1)
  step 5
@push(apply)
@push(apply_mechanical)
*apply_name
bolts
@push(select_screen)
*select sets
@popup(select_set_popmenu)
@list(select set list,0)
base
@popdown(select_set_popmenu)
*invisible selected
*undo
*select_sets
@popup(select_set_popmenu)
@list(select_set_list,0)
base
@popdown(select_set_popmenu)
*visible selected
*fill view
@pop(1)
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed disp pm)
*apply_dof x
*apply_dof y
*apply dof z
@popdown(fixed disp pm)
*add_apply_nodes
 1 40 68 69
# | End of List
@push(viewing_screen)
*show view 4
*reset_view
*fill_view
@pop(1)
```

```
*action | figure 11.22
 Select all elements that will be heavy L3x3 profiles
 first select from set base then mid_sect_lo, mid_sect_hi,
 head and arms.
@push(select_screen)
*store_elements
L3x3
 201
 198
 105
 108
 43
 42
 6
 7
 199
 196
 103
 106
 200
 197
 104
 107
 40
 41
 4
 5
 97
 100
 98
 101
 189
 95
 188
 94
 187
 3
 39
 96
```

*visible_selected
*store_elements
L3x3
404
403
399
401
397
398
402
400
448
456
460
452
461
453
457
449
451
459
455
463
450
458
462
454
419
415
427
423
421
417
429
425
420
418 414
414 426
4∠0 400
422
424

416 428 383 381 387 385 384 386 380 382 82 316 317 318 # | End of List *select_sets L3x3 *visible selected *fill_view *action | figure 11.23 *select clear *select_elements all_existing *visible_selected *fill_view *select_elements all_existing *select_mode_except L3x3 All other elements will be L2x2 profiles *store_elements L2x2all selected *select_clear *select_reset *identify_sets *regen *action | figure 11.24 *identify_none

*regen @pop(1) *new_apply *apply_name gravity *apply_type gravity_load @popup(apply_gravity_load_popmenu) *apply value z -32.2@popdown(apply_gravity_load_popmenu) *add_apply_elements all existing *action | figure 11.25 @push(viewing screen) *show view 1 *fill_view @pop(1)@push(select_screen) *select_sets @popup(select set popmenu) @list(select_set_list,0) base @popdown(select_set_popmenu) *visible selected *action | figure 11.26 *store elements lo front 3 4 5 6 7 8 9 39 40 41 42 43 44 45 # *select sets @popup(select_set_popmenu) @list(select_set_list,0) mid sect lo @popdown(select_set_popmenu) *visible selected *action | figure 11.27 *store_elements lo front 25 26 27 28 29 31 32 33 34 35 36 37 38 57 58 59 60 61 62 63 64 65 66 67 68 69 #

```
*select_sets
@popup(select_set_popmenu)
@list(select_set_list,0)
mid_sect_hi
@popdown(select_set_popmenu)
*visible selected
*action | figure 11.28
*store elements
hi front
20 21 22 23 24 52 53 54 55 56 70 71 72 73 74 75 89 90 91 92 93
#
*select sets
@popup(select_set_popmenu)
@list(select set list,0)
head
@popdown(select_set_popmenu)
*visible selected
*action | figure 11.29
*store_elements
hi front
 11 12 13 14 15 16 46 47 48 49 50 51 76 77 78 79 80 81 82 83 84 85
86 87 88
# | End of List
*select_sets
@popup(select_set_popmenu)
@list(select_set_list,0)
arm
@popdown(select_set_popmenu)
*visible selected
*action | figure 11.30
*store elements
hi front
 382 383 386 387 390 394 397 398 401 402 405 409 418 419 420 421 426
427 428 429 434 435 442 443 448 449 450 451 456 457 458 459 464
 465 472 473
# | End of List
*select_clear
*select elements
all_existing
*visible_selected
@push(viewing_screen)
```

```
*show view 4
@pop(1)
@pop(1)
*new_apply
*apply name
hi wind
*apply_type global_load
@set($usersubfv,FORCEM)
@popup(apply_global_load_popmenu)
*apply_value y
-120
@popdown(apply_global_load_popmenu)
*add_apply_elements
hi front
*action | figure 11.31
*new_apply
*apply name
lo wind
*apply_type global_load
@set($usersubfv,FORCEM)
@popup(apply_global_load_popmenu)
*apply_value y
-80
@popdown(apply_global_load_popmenu)
*add_apply_elements
lo front
*action | figure 11.32
*new_apply
*apply name
cable load
*apply_type point_load
@set($usersubfv,FORCDT)
@popup(apply_point_load_popmenu)
*apply_value z
-500
@popdown(apply_point_load_popmenu)
*add_apply_nodes
 213
 249
 250
 226
```

```
225
 200
# | End of List
*action | figure 11.33
@pop(1)
@pop(1)
@push(plot_screen)
*set nodes off
@pop(1)
step 6
@push(geom_props)
@push(geometry_mech_three)
@push(select_screen)
*select sets
@popup(select_set_popmenu)
@list(select_set_list,0)
base
@popdown(select_set_popmenu)
*visible selected
*fill view
*store_elements
upright
97 100
98 101
40 41
4 5
#
*select sets
@popup(select_set_popmenu)
@list(select_set_list,0)
head
@popdown(select_set_popmenu)
*visible_selected
*fill view
*store_elements
upright
116 119 122 125 128 131
 46 47 48 49 50 51
 11 12 13 14 15 16
 115 118 121 124 127 130
```

```
# | End of List
*select_sets
@popup(select_set_popmenu)
@list(select_set_list,0)
arm
@popdown(select_set_popmenu)
*visible_selected
*store elements
upright
407
405
468
464
469
465
431
430
388
390
434
435
#
*select clear
*select_reset
*select_elements
all_existing
*visible_selected
*fill_view
@pop(1)
*new_geometry
*geometry_name
L3x3 z
*geometry_type mech_three_beam_ela
@popup(geom_el_beam_pm)
*geometry value area
0.01
6.0e-5
6.0e-5
0
0
1
```

```
@popdown(geom_el_beam_pm)
@push(select_screen)
*select sets
@popup(select_set_popmenu)
@list(select_set_list,0)
L3x3
@popdown(select_set_popmenu)
@push(select method screen)
@pop(1)
@push(select_mode_screen)
*select_mode_except
@pop(1)
*select_sets
@popup(select set popmenu)
@list(select set list,0)
@list(select_set_list,1)
upright
@popdown(select_set_popmenu)
@pop(1)
*add_geometry_elements
all selected
*identify_geometries *regen
*new geometry
*geometry_name
L3x3_x
*geometry_type mech_three_beam_ela
@popup(geom_el_beam_pm)
*geometry_value area
.01
6.0e-5
6.0e-5
1
0
0
@popdown(geom_el_beam_pm)
@push(select_screen)
*select clear
*select_reset
*select_sets
@popup(select_set_popmenu)
```

```
@list(select_set_list,1)
upright
@popdown(select_set_popmenu)
@push(select_mode_screen)
*select mode except
@pop(1)
*select_sets
@popup(select set popmenu)
@list(select_set_list,1)
L2x2
@popdown(select_set_popmenu)
@pop(1)
*add_geometry_elements
all selected
*action | figure 11.34
*new_geometry
*geometry_name
L2x2 z
*geometry_type mech_three_beam_ela
@popup(geom el beam pm)
*geometry value area
0.00651
2e-5
2e-5
0
0
1
@popdown(geom_el_beam_pm)
@push(select screen)
*select clear
*select reset
*select sets
@popup(select set popmenu)
@list(select_set_list,1)
L2x2
@popdown(select_set_popmenu)
*select_mode_except
*select sets
@popup(select_set_popmenu)
@list(select_set_list,1)
upright
```

11 Transmission Tower

```
@popdown(select_set_popmenu)
@pop(1)
*add_geometry_elements
all_selected
*new_geometry
*geometry_name
L2x2_x
*geometry_type mech_three_beam_ela
@popup(geom_el_beam_pm)
*geometry_value area
0.00651
2.0e-5
2.0e-5
1
0
0
@popdown(geom_el_beam_pm)
@push(select_screen)
*select_clear
*select reset
*select sets
@popup(select_set_popmenu)
@list(select_set_list,1)
upright
@popdown(select_set_popmenu)
*select_mode_except
*select sets
@popup(select_set_popmenu)
@list(select set list,1)
L3x3
@popdown(select_set_popmenu)
@pop(1)
*add_geometry_elements
all_selected
@push(select screen)
*select clear
*select_reset
@pop(1)
*action | figure 11.35
@pop(1)
@pop(1)
```

```
@push(mat_props)
*material name
steel
*material_type mechanical:isotropic
@popup(mat_isotropic)
*material value isotropic:youngs modulus
4.176e9
0.3
*material_value isotropic:mass_density
15.217
@popdown(mat_isotropic)
*add material elements
all_existing
*identify materials *regen
*action | figure 11.36
*identify_none *regen
@pop(1)
 step 7
@push(jobs)
*job_class mechanical
@popup(job mechanical)
@list(job_rem_lcase_list,0)
@list(job_add_lcase_list,0)
@popdown(job_mechanical)
*job name
static
*job class mechanical
@popup(job_mechanical)
@list(job_rem_lcase_list,0)
@list(job add lcase list,0)
@popdown(job mechanical)
@push(element_types)
@popup(eltype truss beam 3d pm)
*element type 52
@popdown(eltype_truss_beam_3d_pm)
all existing
@pop(1)
@push(file_screen)
@set($file_browser_label,SAVE AS FILE)
```

```
Procedure File
```

```
@set($file_browser_command,*save_as_model)
@popup(save_file_browser_popmenu)
@list(save_file_browser_filelist,0)
@list(save_file_browser_dirlist,0)
*save as model c11.mud yes
@popdown(save_file_browser_popmenu)
@pop(1)
*update job
@popup(job_run_popmenu)
*submit_job 1
*monitor job
@popdown(job_run_popmenu)
@top()
@push(results)
*post_open_default
@push(viewing_screen)
*show view 3
*fill view
@pop(1)
*set deformed both
*post next
@push(deformed_settings)
*set automag on
*action | figure 11.37
(1)qoq@
*fill view
 step 9
*post_close
@pop(1)
@push(loadcase)
*new loadcase
*loadcase_name
dynamic
@push(lcase_mechanical)
*loadcase_type dynamic_modal
@popup(lcase dynamic modal)
*loadcase_value nmodes
15
@popdown(lcase_dynamic_modal)
```

```
@pop(1)
@pop(1)
@push(jobs)
*new_job
*job name
dynamic
*job_class mechanical
@popup(job mechanical)
@list(job_rem_lcase_list,0)
@list(job_add_lcase_list,0)
*add_job_loadcases dynamic
@popdown(job mechanical)
*update_job
@popup(job_run_popmenu)
*submit job 1
*monitor_job
@popdown(job_run_popmenu)
@pop(1)
  step 10
@push(results)
*post_open_default
@push(viewing_screen)
*show view 4
@pop(1)
*set deformed both
*post_next
*post next
@push(deformed_settings)
*set_automag on
@pop(1)
@push(results2)
*post_animate_mode
15
@push(animation_screen)
*animation_play
@push(viewing_screen)
*fill view
*show_view 4
*fill view
```

```
@pop(1)
*animation_play
*action | figure 11.38
*show_model
@pop(1)
@pop(1)
*post_skip_to
0:15
*action | figure 11.39
```



Chapter Overview

This chapter describes the process of importing a geometric or finite element model from a supported CAD or FEM program. The process is illustrated through a sample session that involves importing and meshing a geometric model specified in IGES format.

Background Information

Description

The structure you are importing is a seal made out of rubber that will undergo large deformations caused by coming into contact with other parts. The structure is modeled using a boundary representation of straight lines and curves.

After reading the IGES file, you will select a portion of the model and transform it into a finite element mesh. This process is described in the steps listed below.

The IGES file will be found in the Mentat installation directory, in the subdirectory *examples/userguide* and is named *c12.igs*.

Detailed Session Description

Step 1

Assume you are already in Mentat and in the directory where the file you wish to import is located.

Use the following button sequence to read the IGES file. Click on the FILL button located in the static menu area to scale the model to fill the graphics area. The scaled model that appears in the graphics area is shown in Figure 12-1.

```
MAIN
FILES
IMPORT
IGES
cl2.igs
OK
FILL
```

(Select file)

Step 2

Prior to manipulating the model in any way, you are advised to eliminate all duplicate points and curves using the **SWEEP** processor. Use the following button sequence to sweep the model of all duplicate entities.

```
MAIN
MESH GENERATION
SWEEP
sweep POINTS
all: EXIST.
sweep CURVES
all: EXIST.
```

Mentat responds by sweeping all duplicate points and curves, respectively.



Figure 12-1 Scaled Model of Imported IGES File

To improve the quality of the display, change the default plot settings to a higher accuracy.

```
MAIN
MESH GENERATION
PLOT
MORE
DIVISIONS
HIGH
RETURN
RETURN
RETURN
```

Step 3

Assume you only need to mesh the lower part of the model shown in Figure 12-1. It is useful to store the upper and lower parts of the model in two separate sets as it makes it much easier to reference when working with only part of the model. An option in Mentat that aids you in focusing on the part of the model you want to mesh is the VISIBLE option which is used to hide extraneous information.

You are going to use the automatic overlay meshing feature which requires a closed boundary description. The lower part of the geometry therefore needs an additional line segment. Create this line in the vicinity of the lower neck of the model using the following button sequence.

MAIN MESH GENERATION	
	(Zoom in on the base of the neck)
crvs ADD	(Use the default curve type)
235	(Pick point)
188	(Pick point)

Use the following button sequence to create the two sets: one for the upper part of the model, the other for the lower part. Due to the awkward shape of the model, it is best to use the polygon pick method (CTRL key + <ML>) described in *List Specification* of Chapter 2 to select the members for each set.

MAIN	
PLOT	
draw POINTS	(off)
REGEN	
RETURN	
FILL	
SELECT	
crvs STORE	
upperpart	(The curve set name)
	(Use the Polygon Pick Method to
	select the curves)
END LIST (#)	



Figure 12-2 Curved Added at Base of Neck

Repeat this operation for the lower part of the model and save the set as lowerpart. A suggestion for the contour of the polygon pick is depicted in Figure 12-3 and Figure 12-4.

To verify that you have created two sets, click on the sets SELECT SET button. A pop-up menu appears over the graphics area listing the currently defined sets. Both *lowerpart* and *upperpart* should be listed. Click on OK to return to the SELECT menu.



Figure 12-3 Polygon Pick Contour for Upper Part

Step 4

To focus on the lower part of the structure, use the following button sequence to hide the upper part of the model.

```
MAIN
MESH GENERATION
SELECT
SELECT SET
lowerpart
OK
MAKE VISIBLE
FILL
```



Figure 12-4 Polygon Pick Contour for Lower Part

The upper part of the model is hidden and the lower part scaled to fill the graphics area as is shown in Figure 12-5.

A closer look at Figure 12-5 reveals there is an extra curve in the geometry that interferes with the boundary description of the part. This curve must be removed before the automatic meshing feature is invoked. The curve is located in the inner part of the seal on the right hand side of the model. Use the following button sequence to remove this curve.

```
MAIN
MESH GENERATION
crvs REM
180
END LIST (#)
```

(Pick curve)


Figure 12-5 Lower Part of Model Scaled to Fill the Graphics Area

The model is ready to be meshed using the 2D PLANAR MESHING from the AUTOMESH processor from the MESH GENERATION menu. First, the overlay mesher will be used. Due to the intricate shape of the model, it is necessary to use a density of 70 elements in both the X and Y direction. A setting of less than 70 will cause holes to appear in the mesh. Make sure you specify all: VISIBLE curves for the Enter overlay curve list: prompt. Use the following button sequence to mesh the model. Keep in mind that it will take the program some time to generate the model due to the number of divisions specified.

```
MAIN
MESH GENERATION
AUTO MESH
2D PLANAR MESHING
quadrilaterals (overlay) DIVISIONS
70 70
quadrilaterals (overlay) QUAD MESH!
all: VISIBLE
```



Figure 12-6 Lower Part of Model with Curve Removed

It is helpful to turn off some of the plot entities to produce a cleaner view of the mesh.

MAIN	
PLOT	
draw NODES	(off)
draw ELEMENT FACES	(off)
draw CURVES	(off)
REGENERATE	
RETURN	
SAVE	

Figure 12-7 shows the resulting mesh that should appear in the graphics area.



Figure 12-7 Mesh Generated with OVERLAY Mesher

In order to demonstrate the use of the other 2D planar meshers, the element mesh will be removed and the display of the curves and points will be activated.

```
MAIN
MESH GENERATION
AUTO MESH
2D PLANAR MESHING
CLEAR MESH
PLOT
draw POINTS
draw CURVES
REGENERATE
RETURN
```

(*on*) (*on*) 12 Importing a Model

Since we have already assured that a closed loop exist for the curves, we do not have to enter the REPAIR GEOMETRY menu in AUTOMESH. Instead we go to CURVE DIVISIONS directly. Meshers other than the overlay mesher require a curve division. We first determine the distance between the two parallel curves. Based on this distance, a proper curve division will be set. Note that the Advancing Front QUAD mesher requires an even division on the loops.

MAIN MESH GENERATION AUTO MESH CURVE DIVISIONS UTILS MORE DISTANCE RETURN AVG LENGTH 0.3 restriction FORCE EVEN DIV apply restriction LOOPS APPLY CURVE DIVISIONS ALL VISIBLE

(Click two points on the parallel curves)



Figure 12-8 Determine Distance between Parallel Curves



Figure 12-9 Curve Divisions

The mesh based using the Advancing Front QUAD mesher is now obtained with:

```
MAIN

MESH GENERATION

AUTO MESH

2D PLANAR MESH

quadrilaterals (adv frnt) QUAD MESH!

ALL VISIBLE

PLOT

draw CURVES

draw POINTS

REGENERATE

RETURN
```

(off) (off)



Figure 12-10 Mesh Generated with Advancing Front QUAD Mesher

Clear the mesh and repeat the meshing with the Delaunay triangular mesher.

```
MAIN

MESH GENERATION

AUTO MESH

2D PLANAR MESHING

CLEAR MESH

PLOT

draw CURVES

REGENERATE

RETURN

triangles (delaunay) TRI MESH!

ALL VISIBLE

PLOT

draw CURVES

REGENERATE

REGENERATE

REGENERATE

RETURN
```

(On)

(Off)



Figure 12-11 Mesh Generated with a Delaunay Triangular Mesher

Clear the mesh and repeat the meshing with the Advancing Front triangular mesher.

```
MAIN

MESH GENERATION

AUTO MESH

2D PLANAR MESHING

CLEAR MESH

PLOT

draw CURVES

REGENERATE

RETURN

triangles (adv frnt) TRI MESH!

ALL VISIBLE

PLOT

draw CURVES

REGENERATE
```

(*on*)

(*off*)



Figure 12-12 Mesh Generated with a Advancing Front Triangular Mesher

Procedure File

```
| Version : Mentat 3.3
*alias action
*exec_p gen_image.proc
@top()
*action | figure 12.0 - dummy
 read iges file
@push(file_screen)
@push(import_screen)
@set($iges browser label,IMPORT IGES)
@set($iges_browser_command,*import iges)
@popup(read iges browser popmenu)
@list(read_iges_browser_filelist,0)
@list(read_iges_browser_dirlist,0)
*import iges c12.igs
@popdown(read_iges_browser_popmenu)
*fill view
@pop(1)
*action | figure 12.1
@pop(1)
 remove duplicate points and curves
@push(meshqen)
@push(sweep)
*sweep_points
all existing
*sweep_curves
all_existing
@pop(1)
@push(select screen)
@push(plot_screen)
@push(plot1_screen)
@push(divisions_screen)
*set_plot_divisions_high
*regenerate
@pop(1)
@pop(1)
```

```
*zoom box
*zoom_box(1,0.339587,0.651685,0.515947,0.768539)
@pop(1)
@pop(1)
*add curves
 235
 188
*action | figure 12.2
@push(plot_screen)
*set_points off
*regenerate
*fill view
@push(select_screen)
*store curves
upperpart
 8 16 17 18 102 103 104 105 106 107 111 112 139 140 141 142 209 213
 2 92 93
 1 3 4 5 6 7 9 10 11 12 13 14 15 91 94 95 96 97 98 99 100 101 108 109
110 113
 114 115 116 117 118 119 120 121 124 125 126 127 128 129 130 131 132
133 134 135
 136 137 138
 122 123
*action | figure 12.3
# | End of List
*fill view
*store curves
lowerpart
 19 20 29 30 41 52 78 85 87 88 153 156 158 174 175 203 211
 210
 21 22 23 24 25 26 27 28 31 32 33 34 35 36 37 38 39 40 42 43 44 45
46 47 48 49
 50 51 53 54 55 56 57 58 59 60 61 62 63 64 65 66 67 68 69 70 71 72
73 74 75 76
 77 79 80 81 82 83 84 86 89 90 143 144 145 146 147 148 149 150 151
152 154 155
 157 159 160 161 162 163 164 165 166 167 168 169 170 171 172 173 177
178 179 180
 181 182 183 184 185 186 187 188 189 190 191 192 193 194 195 196 197
198 199 200
 201 202 204 205 206 207 208 209 212 213
```

```
176
# | End of List
*action | figure 12.4
*select_sets
@popup(select_set_popmenu)
@list(select_set_list,0)
lowerpart
@popdown(select_set_popmenu)
*visible_selected
*fill_view
*action | figure 12.5
*select curves
@pop(1)
@pop(1)
*remove curves
 180
# | End of List
*action | figure 12.6
  automatic overlay mesher
@push(automesh)
@push(repair_geometry)
*check curves
all_visible
*action | figure 12.7
@pop(1)
@push(automesh_planar)
*set overlay divisions
70 70
*overlay_mesh
all visible
@push(plot screen)
*set_nodes off
*set faces off
*set_curves off
*regenerate
@pop(1)
*action | figure 12.8
@push(file_screen)
@set($file_browser_label,SAVE AS FILE)
```

```
@set($file_browser_command,*save_as_model)
@popup(save_file_browser_popmenu)
@list(save_file_browser_filelist,0)
@list(save_file_browser_dirlist,0)
*save as model c12.mud yes
@popdown(save_file_browser_popmenu)
@pop(1)
@pop(1)
@pop(1)
 other planar meshers
*clear_mesh
@push(plot screen)
*set curves on
*set_points on
*regenerate
@pop(1)
 auto mesh menus
@push(automesh)
 @push(repair_geometry)
 *clean_2d_loops
 all_visible
@pop(1)
 set curve divisions
@push(curve_divisions)
@push(utility_screen)
@push(utility1_screen)
*distance
 -1.618117982359e+001 -1.010336052905e+001 0.00000000000e+000
 -1.619734125936e+001 -9.803796165359e+000 0.00000000000e+000
*action | figure 12.9
@pop(1)
@pop(1)
*set_curve_div_type_fix_avgl *set_curve_div_avgl
.3
*set_curve_div_rest_evn
```

```
@popup(curve_div_apply_pm)
*set_curve_div_applyrest_lps
*action | figure 12.10
@popdown(curve_div_apply_pm)
*apply_curve_divisions
all visible
@pop(1)
@push(automesh planar)
*af_planar_quadmesh
all_visible
@push(plot_screen)
*set curves off
*set_points off
*regenerate
@pop(1)
*action | figure 12.11
*clear mesh
@push(plot_screen)
*set_curves on
*regenerate
@pop(1)
 delaunay triangular mesh
*dt_planar_trimesh
all visible
@push(plot_screen)
*set_curves off
*regenerate
@pop(1)
*action | figure 12.12
*clear mesh
@push(plot screen)
*set_curves on
*regenerate
@pop(1)
 advancing front triangular mesh
*af_planar_trimesh
all_visible
```



```
@push(plot_screen)
*set_curves off
*regenerate
*set_curves on
*regenerate
@pop(1)
*action | figure 12.13
```



Chapter Overview

The sample session described in this chapter analyzes the process of punching. A tool with a rigid dimple is pushed into a circular plate. The object of this process is to produce a circular plate with spherical indentation. The goal of the static analysis described in this chapter is to determine the residual stresses and plastic strains in the workpiece after the operation.



Background Information

Description

This problem demonstrates the preparation of a contact analysis involving multiple rigid bodies (the tool) and a deformable body (the workpiece). The top of the tool is a sphere blended in with a flat rigid plate. The workpiece is supported such that radial displacements are constrained at the outer diameter while axial displacements are constrained at the node positioned at the corner of the outer diameter and the backing plate. The bottom part of the tool is a flat backing plate with a hole at the same location as the dimple of the top part of the tool. The plate of the tool supports the entire workpiece, except for the region of the hole.

Idealization

Because of the axisymmetric nature of the geometry and the loading, this process can be idealized to an axisymmetric model. The edge of the workpiece is clamped which prevents rigid body motion of the workpiece. The backing plate that backs the workpiece is modeled as a rigid body and remains in place during the analysis. The punch is modeled as a rigid body and moves during the analysis towards the static backing plate, while indenting the workpiece.

The tool is stopped when the flat surfaces of both parts of the tool are in full contact with the workpiece. This occurs when the total displacement of the punch is 0.1488 inches, which is reached in 0.4 seconds. Hence, the velocity of the top part of the tool (i.e. punch) is 0.372 inch per second. The friction between tool and workpiece is assumed to be negligible and is therefore not taken into consideration in this analysis.





Figure 13-1 Punch, Workpiece, and Backing Plate

Requirements for a Successful Analysis

The analysis is considered successful when the punch becomes flush with the workpiece and is released afterwards to determine the residual stresses.

Full Disclosure

The workpiece is constructed out of steel with a Young's Modulus of 30.0e6 psi and a Poisson's Ratio of 0.3. It has a yield stress of 39,000 psi. The material exhibits work-hardening. The workpiece has a radius of 0.7874 inches and a thickness of 0.117 inches.





Figure 13-2 Dimensions of Punch, Workpiece, and Backing Plate

The punch is a sphere of radius 0.24 with a fillet of radius 0.109 that brings it tangent to a horizontal piece. It will move over a total distance of 0.1488 inches in a period of 0.4 seconds. The backing plate has a cylindrical hole of radius 0.25 inches into which the workpiece is forced. Both punch and backing plate are considered to be rigid during the analysis.





Figure 13-3 The Workhardening Curve for the Workpiece Material



Overview of Steps

Step 1	Create a model of a rectangular patch and convert it to finite elements.

- **Step 2** Create the curves required for the punch & backing plate.
- **Step 3** Apply the required fixed displacements to the rim of the workpiece. Apply the material data.
- **Step 4** Identify the contact bodies and create the table that defines the motion of the rigid die, representing the punch.
- **Step 5** Define the incremental steps and convergence testing parameters.
- **Step 6** Activate the large strain parameters and submit the job.
- **Step 7** Postprocess the results by displaying the deformed structure and the residual stresses and strains.



Detailed Session Description

Step 1

The approach used in this session to generate the model is the geometric meshing technique.

The first step is to create the workpiece. The recommended method is to create a point and expand it to a line curve, followed by expanding this curve to a quad surface. Use the following button sequence to create the first point.

```
MAIN
MESH GENERATION
pts ADD
0.24 0 0
```

Next, expand the point using a translation of 0.117 inches in the x-direction and then expand the resulting curve using a translation of 0.7874 in the y-direction. Use the following button sequence to create the quad surface.

```
MAIN
MESH GENERATION
EXPAND
TRANSLATIONS
0.117 0 0
POINTS
all: EXIST.
TRANSLATIONS
0 0.7874 0
CURVES
all: EXIST.
FILL
```

The next step is to convert the geometric entities to finite elements. This is done using the **CONVERT** processor. Five divisions will be used through the thickness and 20 along the radius. Use the following button sequence to mesh the surface.



MAIN MESH GENERATION CONVERT DIVISIONS 5 20 SURFACES TO ELEMENTS all: EXIST. PLOT draw SURFACES REGENERATE RETURN

(*on*)



Figure 13-4 Result of the Convert Command

An important portion of the analysis requires that a sharp corner will be developed at the lip of the cylinder. To do this, the mesh must be refined in that area. The user will need to zoom in on that area. The nodes near the radius of 0.25 will be moved to exactly that location. The y-coordinates of these nodes can be determined by the SHOW command on the NODES panel. The move operation is done using the following button sequence:

```
MAIN
  MESH GENERATION
     nodes SHOW
        42
                                                (Pick a node on the 8th row of nodes
                                                        from the axis of symmetry)
     MOVE
        FORMULAS
           х
           0.25
           \mathbf{Z}
        NODES
                                                    (Box pick the 8th row of nodes)
           37 38 39 40 41 42
           END LIST (#)
        RETURN
```

The next step is to subdivide the sixth row of elements. Use the following button sequence to subdivide the elements.

```
MAIN

MESH GENERATION

SUBDIVIDE

DIVISIONS

1 2 1

ELEMENTS

26 27 28 29 30

END LIST (#)
```

(Box pick the 7th row of elements)

After subdividing, it is usually necessary to remove all the duplicate nodes. It is also advisable to renumber the elements because there is a gap in the numbering from the subdivide operation. This can be done with the following button sequence.

```
MAIN
MESH GENERATION
SWEEP
sweep NODES
all: EXIST.
RETURN
RENUMBER
ALL
```



Step 2

The next step is to create the dies. The dies will be represented by geometric entities. These entities are a combination of curves.

For the punch, the first step is to put a point at the center of the sphere. Then use that point to create an arc. It is easier to have a rigid body almost touching the deformable body. That is why the center point will be created by duplicating the top center point of the workpiece at a distance equal to the sphere radius. The following button sequence will create the center point and arc.

```
MAIN
  MESH GENERATION
     DUPLICATE
        TRANSLATIONS
           -0.24 0 0
        POINTS
                                                      (Pick the lower left point)
           1
           END LIST (#)
        RETURN
     CURVE TYPE
        CENTER/RADIUS/ANGLE/ANGLE
        RETURN
     crvs ADD
        0 0 0
                                             (Pick the point just created (Center))
        0.24
                                                                   (Radius)
        0 55
                                                 (Beginning angle, ending angle)
```

The next curve must be tangent to the one just created. Therefore, the curve type must be changed to arc type tangent/radius/angle before creating the curve. The radius of the arc is 0.109 inches and the angle will be a negative 55 degrees. The negative sign makes the arc go clockwise. The following button sequence will create the arc.





Figure 13-5 The Spherical Part for the Punch

The next step is to finish the rigid body. There is one line required to finish the punch. This is a horizontal line tangent to the second arc. The following button sequence will create the line.





Figure 13-6 The Geometry of the Punch

The next step is to create the backing plate. First, a point will be added at the bottom of the workpiece at a y location of 0.25. It will then be expanded in the x and y direction creating the two lines required for the rigid body. The following button sequence will generate these curves.





Figure 13-7 Punch, Workpiece, and Backing Plate



Step 3

The first step is to create a table with the stress versus plastic strain table. The following button sequence will create the table.

```
MAIN
  MATERIAL PROPERTIES
    TABLES
       TABLE TYPE
         plastic_strain
         OK
       SHOW TABLE
       ADD POINT
         0 39000
         0.7e-3 58500
         1.6e-3 63765
         2.55e-3 67265
         3.3e-3 68250
         10e-3 72150
       FIT
       NAME
         work-hard
    MORE
       label X-AXIS
         plastic strain
       label Y-AXIS
         yield stress
       SHOW MODEL
```

The next step is to input the material properties and assign them to the elements. The table must be assigned to the yield stress value to include the workhardening. The following button sequence will assign the material properties.







Figure 13-8 Workhardening Curve



The next step is to clamp the end of the workpiece. The model is axisymmetric and therefore, has only 2 degrees of freedom at each node. The first set of boundary conditions will clamp the node on the top right in both axial and radial direction. The second set of boundary conditions will constrain the radial motion of both the nodes on the axis of symmetry and the nodes on the outer radius of the workpiece.

MAIN	
BOUNDARY CONDITIONS	
MECHANICAL	
FIXED DISPLACEMENT	
ON x displace	<i>(on)</i>
ON y displace	(on)
OK	
nodes ADD	
126	(Pick the node at the top right point)
END LIST (#)	
NEW	
FIXED DISPLACEMENT	
ON y displace	(on)
OK	
nodes ADD	
	(Pick the bottom edge nodes)
	(Pick the top edge nodes)

END LIST (#)

Step 4

This step assigns the elements and curves to the correct contact bodies. Rigid bodies must always follow all deformable bodies. The following button sequence will assign all the elements to deformable body 1.

MAIN CONTACT CONTACT BODIES DEFORMABLE NAME workpiece elements ADD all: EXIST.



The next step is to assign the curves to rigid bodies. By default, analytical curves will be used for rigid bodies composed of curved entities. Therefore, no manual interference is required to specify the number of subdivisions used to discretize the curves.

The following button sequence will create the 2 rigid bodies.

```
MAIN
  CONTACT
     CONTACT BODIES
        NEW
        NAME
           punch
        crvs ADD
           2 3 4
                                                        (Pick curves of punch)
           END LIST (#)
        NEW
        NAME
           back
        crvs ADD
           5 6
                                                   (Pick curves of backing plate)
           END LIST (#)
```

At this point, it is advisable to check the correctness of the definition direction of the curves used in the rigid bodies.



MAIN CONTACT CONTACT BODIES PLOT elements SOLID REGENERATE RETURN ID CONTACT

(on)



Figure 13-9 Incorrect Definition Direction of Curves in Back-Plate

The ID CONTACT button will show the rigid bodies and their direction. If either of the curves is defined such that the rigid body is on the same side as the deformable body, the curve can be flipped by using the FLIP CURVES button.





Figure 13-10 Corrected Definition

The punch will move during the analysis. To define the motion, a table of time versus velocity must be defined. The axial distance of the straight section of the punch and the workpiece is 0.1488 inches. This value can be determined with the DISTANCE command on the second page of the UTILITIES menu (use MORE).



As stated before, this gap will be closed in 0.4 seconds. As soon as the horizontal part of the punch touches the workpiece, the motion will be reversed and the release option will be switched on. In order to accomplish separation within this single increment, the punch will be withdrawn at high velocity.

The following button sequence will define the table.

```
MAIN
  CONTACT
    CONTACT BODIES
       TABLES
         NEW
         NAME
           punch_motion
         TABLE TYPE
           time
            OK
         ADD POINT
            0 0.1488/0.4
            0.4 0.1488/0.4
            0.4 -10*0.1488/0.4
            0.5 -10*0.1488/0.4
         FIT
         SHOW MODEL
```

(0.1488/0.4 = velocity)

(-10*velocity)


Figure 13-11 Velocity as a Function of Time

The following button sequence will assign the table to the punch motion (please notice that the current and therefore active body is body 3, the backing plate!).

```
MAIN

CONTACT

CONTACT BODIES

PREV

CONTACT BODY PROPERTIES

velocity X

1

velocity x TABLE

punch_motion

OK

OK
```

(to activate body 2)



The loadcases describe the first and second part of the loading history and the loads used during those parts. The following button sequence will create the loadcases.

```
MAIN
  LOADCASE
    NAME
       indent
    mechanical analyses STATIC
       LOADS
         OK
       TOTAL LOADCASE TIME
         0.4
       # STEPS
         100
       SOLUTION CONTROL
         MAX # RECYCLES
            20
         OK
       OK
    NEW
    NAME
       release
    mechanical analyses STATIC
         LOADS
         OK
       TOTAL LOADCASE TIME
         0.1
       # STEPS
         1
       SOLUTION CONTROL
         MAX # RECYCLES
            20
         OK
       CONTACT
         CONTACT RELEASES
            SELECT
              punch
            OK
```



MECHA	RCAL LOA	ADCASES										At Lance						
MEW	REM			ME	DIANICA	L STATIC	PARAM	TERS										4 0000
ANNE	licase1			10	NDS				-							2		
COPY	PREV	NEXT	EDIL .	80	LUTION	CONTRO	L		~									
LOADCA	SE TYPE			C	NVERG	NCE TES	TING		~									
+ STAT	IC I			C	NTACT				~									
° BUO	are.		P-	9	P CIWI	GES			~									
° CREE	P		P	M	IMERICA	L PREFER	ENCES				_							
° DYN	MIC MOD	WL.	P	DE	SIGN CO	INSTRAM	as .											
C DVIN	MIC TRA	NSIENT	P.	TOTAL LOADCASE TIME									-		-8			
* SPEC	TRUMRE	SPOMSE	P	81	EPPING	PROCED	JRE											
* RECIT	PLASTIC	;	-	•	FDÆD	8	100		-		-							
° 800	APPROA	ACH	1	A	MITTER	LOADING												
				2	VECH. LO	DAD (ARC	PARA	METER	15		<u> </u>			_				
				* THERMAL LOAD * MECK LOAD (EQUILIERIUM)						METER	15		~					
										METER	15		n					
				2	VECH.LC	PWRM	METER	IS										
				84	561	1				ок			1		-			
IIILE													-		-			
DEACTN	MTED ELE	IMENTS																
ADD	REAL								-	+						- 6		_
ALC:	SELECT	venus	ounue															
	UNSEL	innes.	SURFAC	-	3.72			_		_			_					e e
SELECT		END LS	FT (04)		0 K641													
RETURN		MAIN		UNDO	SAVE	DRAW	FILL	RESET	NEW	Die T	X (s. 1) X+ TY+	TZ+	RX+	RYs	RZ+	200M	IN .	USER
					FILES '	PLOT '	VIEW 1	DVN. N	JODEL	TX-	TY-	12-	RX-	RY-	RZ-	80%	OUT	DEFINED

Figure 13-12 Specify Release of Contact Body



The final preprocessing step is to create the job and submit it to run in the background. The job menu defines the special analysis options, the results saved, and other global parameters. This is also where the loadcases can be selected in the desired order. The following button sequence will create the job and submit it.



Figure 13-13 Select Solution Procedure



MAIN JOBS **MECHANICAL** available loadcases indent release ANALYSIS OPTIONS CONSTANT DILATATION SMALL STRAIN-MEAN NRM SMALL STRAIN-RADIAL R JOB RESULTS tensors stress scalars von_mises tepl_strain OK **AXISYMMETRIC** OK CHECK SAVE RUN SUBMIT 1 MONITOR

(Switch on large strain, mean normal based on additive decomposition)

(This makes sure that the default element type 10 is used)

The MONITOR option will continually update the log and return the program control to the user when the analysis is complete. If the user wishes, there is a monitor capability in the RESULTS menu which will allow the user to watch the deformations and stresses during the analysis.



Figure 13-14 Select Quantities to be Written on Post Tape



The analysis requires the final deformed shape and the stresses at that time. The following button sequence will present the results.

```
MAIN
  RESULTS
     OPEN DEFAULT
     FILL
     PLOT
       draw NODES
       MORE
          edges OUTLINE
          PREVIOUS
       RETURN
     DEF & ORIG
     SCALAR
       Equivalent von Mises Stress
       OK
     CONTOUR BANDS
     MONITOR
```

(*off*)

The deformed shape shows that the 90 degree lip is well developed. It also shows the final stresses after the punch operation. The last increment (101) shows the residual stresses after the punch has been withdrawn.

		1	_	_	_		Add and
OIL RESULTS							
POST FILE	ASCI " NARVE	OUTPUT PE	E	• N	OPRIN	T	
K7 STYLE	FREQUENCY 1			5	UMMAR	LA.	
TENSORS	CLEAR						
stress	Stress	LAYERS	ALL	our	DEF	DEFAULT	
cauchy	Caaxity Stress	LAVERS	ALL	OUT	DEF		
ha_stress_r	Real Harmonic Stress	LAVERS	ALL	OUT	DEF		
ha_stress_i	Imaginary Harmonio Stress	LAYERS	ALL	OUT	DEF		
stress_p	Stress in Preferred Sys	LAYERS	ALL	OUT	DEF		
ratness_ue	Reber Stress in Undeformed Conf	LAVERS	ALL	OUT	DEF		
ntnin_dc	Rebar Stress in Deformed Card	LAVERS	ALL	OUT	DEF		
strain	Total Strain	LAVERS	ALL.	OUT	DEF		
el_strain	Elastic Strain	LAYERS	ALL	OUT	DEF		
SCALARS	CLEAR			1	1	for succession in the second s	
• von_inisins	Equivalent Von Mises Stress	LAYERS	ALL	our	DEF	Derword	/
mean_normal	Mean Normal Stress	LAVERS	ML.	our	DEF		
ecausity.	Equivalent Cauchy Stress	LAYERS	WL	our	DEF		
eha_stress_r	Equivalent Real Hannovsie Stress	LAYERS	ALL	001	DEF		
ehe_stress_i	Equivalent Imag Harmonic Stress	LAVERS	ALL	001	DEF		
te_energy	Total Strain Energy Density	LAVERS	ALL	our	DEF		
eel_strain	Equivalent Elastic Strain	LAVERS	ALL .	our	DEF		
ee_energy	Elastic Strain Energy Density	LAYERS	ALL.	OUT	DEF		
tepl_strain	Total Equivalent Plastic Strain	LAYERS	ALL	our	DEF	DEFAULT	
VECTORS	USER DEFINED NODAL VECTORS (USER SUB	UPOSTV)	F	_		_	
			-				
		1.28.					

Figure 13-15 Deformations and Stresses at Maximum Indentation



Figure 13-16 Deformations and Stresses after Springback



Procedure File

```
| Version : Mentat 3.3
*alias action
*exec_p gen_image.proc
*action | figure 13.0 - dummy
*action | figure 13.1 - dummy
*action | figure 13.2 - dummy
*action | figure 13.3 - dummy
@top()
@push(meshgen)
| step 1
*add_points
0.24 0 0
@push(expand)
*set_expand_translations
0.117 0 0
*expand points
all_existing
   expand the curve in y-direction creating a surface
*set expand translations
0 0.7874 0
*expand_curves
all_existing
*fill view
@pop(1)
  create a 5x20 mesh on the surface
@push(convert)
*set convert divisions
5 20
*convert surfaces
all existing
@push(plot_screen)
*set surfaces off
*regenerate
@pop(1)
```



```
*action | figure 13.4
@pop(1)
  find the row closest to R=0.25
  and shift those nodes to the exact position
@push(coord_system)
*grid u domain
*grid_v_domain
0 0.25
*grid_v_spacing
0.25
*set_grid on
@pop(1)
@push(move)
*set_move_formulas
x
0.25
\mathbf{z}
*move nodes
 37 38 39 40 41 42
# | End of List
@pop(1)
@push(subdivide)
*sub divisions
1 2 1
*subdivide_elements
 26 27 28 29 30
# | End of List
@pop(1)
*set_grid off
@push(sweep)
*sweep_nodes
all_existing
@pop(1)
@push(renumber)
*renumber_all
@pop(1)
  step 2
```



```
@push(duplicate)
*set_duplicate_translations
-0.24 0 0
*duplicate_points
1
#
@pop(1)
@push(curve_type)
*set_curve_type arc_craa
@pop(1)
*add curves
0 0 0
0.24
0 55
*action | figure 13.5
   add a curve that bends back to vertical again
@push(curve_type)
*set_curve_type arc_tra
@pop(1)
*add_curves
11
0.109
-55
  finish with straight line segment
@push(expand)
*set_expand_translations
0 0.6 0
*expand_points
 14
# | End of List
*fill view
*action | figure 13.6
  make the backing plate, starting with the corner point
@pop(1)
*add points
```



```
0.357 0.25 0
@push(expand)
*expand_points
17
# | End of List
*set_expand_translations
0.4 0 0
*expand points
17
# | End of List
*fill view
*action | figure 13.7
@pop(1)
@pop(1)
  step 3 : apply material data and boundary conditions
  define work hardening curve
@push(mat_props)
@push(table)
*set_table_type
@popup(table_type_pm)
@list(table_type_list,0)
plastic_strain
@popdown(table_type_pm)
*show_table
*table add
 0.
        39000
  .7e-3 58500
 1.6e-3 63765
 2.55e-3 67265
 3.3e-3 68250
 10.e-3 72150
*table fit
*table name
work-hard
*set_table_xname
plastic strain
*set_table_yname
yield stress
*action | figure 13.8
```



```
*show_model
```

```
apply Young's modulus, Poisson ratio and
    work hardening to all elements
@pop(1)
*material type mechanical:isotropic
@popup(mat_isotropic)
*material_value isotropic:youngs_modulus
30e6
.3
*material type plasticity
@popup(iso_plastic_popmenu)
*material_value plasticity:yield_stress
1
*material_table plasticity:yield_stress0
@set($dof,plasticity:yield_stress0)
@popup(material select table popmenu)
@list(select table list,0)
work-hard
@popdown(material select table popmenu)
@popdown(iso_plastic_popmenu)
@popdown(mat_isotropic)
*add material elements
all existing
@pop(1)
  fix the upper right corner node
@push(apply)
@push(apply mechanical)
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed_disp_pm)
*apply_dof x
*apply dof y
@popdown(fixed_disp_pm)
*add_apply_nodes
126
```



```
#
*new_apply
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed_disp_pm)
*apply_dof y
@popdown(fixed_disp_pm)
*add apply nodes
1 2 3 4 5 6
 121 122 123 124 125 126
# | End of List
@pop(1)
@pop(1)
 step 4
   identify the contact bodies and define their motion
@push(contact)
@push(contact bodies)
*contact_body_name
workpiece
*contact deformable
@popup(contact_deform_pm)
@popdown(contact_deform_pm)
*add_contact_body_elements
all_existing
*new_contact_body
*contact body name
punch
*add_contact_body_curves
2
3
4
*undo
*new_contact_body
*add_contact_body_curves
 1
 2
 3
# | End of List
```



```
*contact_body_name
punch
*new_contact_body
*contact_body_name
back
*add_contact_body_curves
 5
 4
# | End of List
@push(plot_screen)
*elements solid
@pop(1)
*identify_contact *regen
*action | figure 13.9
*flip curves
 4
# | End of List
*action | figure 13.10
*identify_none *regen
@push(table)
*new table
*table_name
punch motion
*set_table_type
@popup(table_type_pm)
@list(table_type_list,0)
time
@popdown(table_type_pm)
*table add
0.0 0.1488/0.4
0.4
       0.1488/0.4
0.4 - 10 \times 0.1488 / 0.4
0.5 -10*0.1488/0.4
*table fit
*action | figure 13.11
@pop(1)
*previous_contact_body
*contact rigid
@popup(contact_rigid_pm)
*contact_option control:velocity
@popup(contact_velocity_pm)
```



```
*contact_value vx
1
*cbody_table vx0
@set($dof,vx0)
@popup(contact_select_table_popmenu)
@list(select table list,0)
punch_motion
@popdown(contact select table popmenu)
@popdown(contact_velocity_pm)
@popdown(contact_rigid_pm)
@pop(1)
@pop(1)
 step 5
 define two loadcases, one for the identation and one for the release
@push(loadcase)
@push(lcase_heat)
@pop(1)
@push(lcase mechanical)
*loadcase_type static
@popup(lcase static)
*loadcase_value nsteps
*loadcase value time
.4
*loadcase value nsteps
100
*action | figure 13.12
@popup(mech_solution_control)
*loadcase value maxrec
20
@popdown(mech solution control)
@popdown(lcase_static)
*loadcase name
indent
*new loadcase
*loadcase name
release
*loadcase_type static
@popup(lcase_static)
```



```
*loadcase_value time
.1
*loadcase_value nsteps
1
@popup(mech_solution_control)
*loadcase value maxrec
20
@popdown(mech solution control)
@popup(contact_lcase_popmenu)
@list(lcase_cbody_list,0)
@list(lcase exseq list,0)
@list(lcase carea list,0)
*add_loadcase_cbodies punch
*action | figure 13.13
@popdown(contact_lcase_popmenu)
@popdown(lcase_static)
@pop(1)
@pop(1)
 step 6
 activate the non-linear job options and submit a job
@push(jobs)
*job class mechanical
@popup(job_mechanical)
@list(job add lcase list,0)
@list(job_rem_lcase_list,0)
*add job loadcases indent
*add job loadcases release
@popup(job_an_opts_mech_popmenu)
*job option cdilatation:on
*job option plasticity:s strn rr
*job_option plasticity:l_strn_mn_add
*action | figure 13.14
@popdown(job_an_opts_mech_popmenu)
@list(job_add_lcase_list,0)
@list(job_rem_lcase_list,0)
@popup(tensor_scalar_popmenu)
@list(post_var_list1,0)
@list(post_tensor_list1,0)
```



```
*add_post_tensor stress
*add_post_var von_mises
*add_post_var tepl_strain
*action | figure 13.15
@popdown(tensor scalar popmenu)
@list(job add lcase list,0)
@list(job_rem_lcase_list,0)
*job option dimen:axisym
@popdown(job_mechanical)
@push(file_screen)
@set($file browser label,SAVE AS FILE)
@set($file browser command, *save as model)
@popup(save_file_browser_popmenu)
@list(save file browser filelist,0)
@list(save file browser dirlist,0)
*save_as_model c13.mud yes
@popdown(save file browser popmenu)
@pop(1)
@pop(1)
@push(jobs)
*update job
@popup(job_run_popmenu)
*submit job 1
*monitor job
@popdown(job_run_popmenu)
@pop(1)
 step 7
 evaluate the results
@push(results)
*post open default
*post_next
*fill view
@push(plot screen)
*set_nodes off
@pop(1)
@push(deformed_settings)
*edges_outline *regen
@pop(1)
```



```
*set_deformed both
*post_value
@popup(post_scalar_pm)
@list(post_scalar_list,0)
Equivalent Von Mises Stress
@popdown(post_scalar_pm)
*post_contour_bands
*post_contour_bands
*post_monitor
*post_skip_to
100
*action | figure 13.16
*post_next
*action | figure 13.17
```



Chapter Overview

The sample session described in this chapter demonstrates a simple linear static and dynamic analysis on a steel bracket. The bracket restrains a vertical pipe. The bracket also supports some mechanical equipment. First, the bracket will be subjected to a static load. A dynamic analysis will predict the normal frequencies and mode shapes of vibration to determine if there is any interaction with the bracket and surrounding excitation frequencies. Finally, the bracket will be subjected to a time dependent pressure and the dynamic response will be determined.



Background Information

Description

This problem demonstrates the preparation of a model using two different meshing techniques, multiple geometric properties, three loadcase types, and corresponding boundary conditions and loads. It will also demonstrate the application of boundary conditions to geometric entities and the merging of different meshes.

The bracket is 15x30x10 with a hole to support a pipe. The bracket must support standard operating loads. It must not have a frequency that can be excited by the mechanical equipment which it supports. It must not fail for a given time dependent pressure loading.

Idealization

The bracket will be represented by flat plate elements which have 4 nodes and the thickness is considered a property of the element. The vertical support plates will require quadrilateral elements to be degenerated to triangular elements in the portion where the arc is tangent to the horizontal plate.

The bracket is welded to a column and therefore will be considered fully fixed on that edge. The weight of the mechanical equipment will be applied to the cantilevered section of the horizontal plate as a distributed load of 1 psi.

Requirements for a Successful Analysis

The analysis will be considered successful if none of the stresses are above 36000 psi during standard operating loads and there are no modes in the range of the mechanical equipment. The bracket cannot cause the pipe to break during a dynamic loading event.





Figure 14-1 Bracket

Full Disclosure

The steel bracket is modeled by four-noded plate elements with a Young's Modulus of 30e6 psi and a Poisson's Ratio of 0.3. It is assumed that the material will not exceed the yield point of 36000 psi. The horizontal plate is 15 inches by 30 inches with a hole of 3.5 inch radius centered in the half of the plate where the vertical support plates are attached. The 2 vertical support plates are 10 inches by 15 inches with a filleted edge. The horizontal plate is 0.25 inches thick and the vertical plates are 0.5 inches thick.





Figure 14-2 Dimensions and Loads for the Bracket





Overview of Steps

Step 1	Create the boundary of a flat area representing the half of the plate with the hole in it. Use the overlay mesh generator to create finite elements.
Step 2	Create the cantilevered section of the plate. Convert it to finite elements. Merge the two parts.
Step 3	Fold the vertical sections and modify the elements in the triangular region.
Step 4	Apply boundary conditions.
Step 5	Assign material and geometric properties.
Step 6	Create the loadcases and submit the jobs.
Step 7	Postprocess the results.



Detailed Session Description of the Linear Static Case

Step 1

The approach used in this session to generate the model is to use the geometric meshing technique to create 2 different areas and mesh them. The first area will be meshed using the overlay mesh generator, and the second will be meshed using the **CONVERT** processor. The entire model will be created as a flat piece and, subsequently, the 2 support pieces will be folded.

As in the sample session described in Chapter 4, the first step in building the mesh is to establish an input grid. Click on the MESH GENERATION button of the main menu. Next click on the SET button to access the coordinate system menu where the grid settings are located. Use the following button sequence to set the grid spacing to 5 inches and the grid size of 30 inches.

```
MAIN
MESH GENERATION
SET
U DOMAIN
-30 30
U SPACING
5
V DOMAIN
-30 30
V SPACING
5
grid ON
RETURN
```

(on)

The next step will be to create the 3 vertical lines of the model. The following button sequence will create the lines.



MAIN MESH GENERATION FILL crvs ADD point(0,-10,0) point(0,25,0) point(15,0,0) point(15,15,0) point(30,0,0) point(30,15,0)

(Pick points from grid)

MESH CE	NERAI	ON					-				-	y-38				•		-	20MARC
NODES	ADD	REM	EDIT	SHOW	1														
BLEMS	ADD	REM	EDIT	SHOW								t.							
PTS .	ADD	REM	EDIT	SHOW															
CHVS	ADD	REM	EDIT	SHOW								1							
SRES	ADD	REM	EDIT	SHOW	1														
SOUDS	ADD	REM		SHOW															
BETWEET	N NODE	BEI	WEEN	POINT			-				-	T.				Ť			T
BLEMEN	T CLASS	00	AD (4)	7															
CURVE 1	MPE	" UN	E	T			-					t.							÷ .
SURFAC	E TYPE	1 00	AD	*															
SOUDTY	rPE	61.0	жк	1			-				-	ł				÷			ł
COORD	NATE 51	STEM																	
SET *	RECTA	NGULA	a 1	GRID			-				-	ł				Ļ		-	30-ىر
CLEAR	6.51	0.1	AR GE	M	i i														
ATTACH		* AU	rote:si	۲ I								t.							
CHANNEE	CLASS	CH	ECK																
CONVER	IT	100	PLICATI	E "			-				-	Ļ							
EXPAND		" INT	ERSECT	r '															
MOVE		* REI	.AX	+															
RENUM	ER	" RE	VOLVE																
SOUDS		518	LETCH																
SUBDIVI	DE	594	EE1*									-						- 7	-
SYMMET	RY	*																	
ALL:	SELEC	 VS 	815 3	NULLINE														- ¥	X
EX851	UNSER	. HY	19. 1	SURFAC															
SELECT		* EM	D LIST (1					•	•	-	•	•			•	•	-	· 1
RETURN		MA	EN .	1	UNDO	SAVE	DRAW	PILL	RESET	MEW	TK+	T¥+	TZ+	RX+	RY+	RZ+	ZOOM	IN	USER
					UTLS'	FILES '	PLOT 1	MEM .	DVN	MODEL	TK-	TY-	12-	RX-	RY-	R2-	BUN	OUT	DEFINED

Figure 14-3 Grid and Straight Line Segments

The next geometric entity to be added will be the 2 fillets. The curve type must be changed to arc and then the 2 curves added. To insure that the arc end points are the end points of the line the CENTER/POINT/POINT arc type will be used. The following button sequence will add the 2 arcs.



	M	AIN																		
		ME	SH	GEN	IERATI	NC														
			CUF	RVE	TYPE															
			(TER/PC	Ν														
	F	RETI	JRN																	
	crvs		5																	
			,	22 5	27	5	(Center point)													
			-	1 5 0	0		(Dick lower and point of the second line)													
			-	15 0	0		(Tick lower ena point of the second line)													
		() -1	0 0		(Pick lower end point of the first line)														
			2	22.5	42.	5	0			(Center point)										
			() 2	5 0					(Pick upper end point of the first line)										
			-	15	15 0					(Pick upper end point of the second line)										
										(2 ten apper ena pour of the second the)										
MESH CE	NERAD	ON			•	-		•	-	y-38	·	-	•		-	20MARC				
NODES	ADD	REM	EDIT	SHOW																
BLEMS	ADD	REM	EDIT	SHOW						N										
P13	ADD	REM	EDIT	SHOW																
CHVS	ADD	REM	EDIT	SHOW						ł										
50005	400	DOM:	100	SHOW							•									
RETWEET	NODE	1 RET	J WEEN F	POINT						+			1			t				
REMEN	CLASS	1 011	AD OD																	

					UTLS'	FILES'	PLOT '	MEM.	DWN	MODEL	TK-	TY-	TZ-	RX-	RY-	82-	0.05	OUT	GETTRED
RETURN		MA	MAIN		UNDO	SAVE	DRAW	PILL	RESE	MEW	TK+	T¥+	TZ+	R(X)	RYA	RZ+	ZOOM	IN	USER
SELECT		* 8M	LIST (6			-								-				· 1
		. 100	19. 1	SURFAC															
ALL:	SELEC	3. ¥89	815 3	OUTUBE														- ×	;X
SYMMET	HY	+																T	
SUBDIVI	DE	1 244	101															1	
SOUDS		518	ETCH																
RENUM	ER	" REV	ADEVE																
MOVE		* REL	AK.	1															
EXPAND		1 INT	ERSECT	r r															
CONVER	T	DU	DUPLICATE									\mathbf{V}							
CHANGE	CHANSE CLASS CHECK									12									
AFTACH * AUTOMESH *										1									
CLEARN	6.51	[a.	AR GE	DM									+	/	_				
SET "RECTANGULAR "GRID						-					ł				_ل_		-	30-ىر	
COORD	NATE 51	STEM																	
SOUD TY	/PE	· 81.0	юк	7			-					+			-	+			-
SURFAC	E TYPE	* OU	ND D	+															
CURVE T	MPE	AN	C.01P	T												1			-
ELEMEN	CLASS	' OU	AD (4)	τ															
BETWEET	NODE	BET	WEEN I	POINT			-			-		ł				7			t
SOUDS	ADD	REM	1.Dit	SHOW										~					
CHVS	ADD	REM	EDIT	SHOW								÷							
PTS	ADD	REM	EDIT	SHOW															
ELEMS	ADD	REM	EDIT	SHOW								Ν.							

Figure 14-4 Line Segments and Fillets



The next step will be the center hole. The coordinate system will be moved such that it has an origin that is the center of the hole. The hole will be added using the grid.

```
MAIN
  MESH GENERATION
    SET
       set origin XYZ
         7.5 7.5
                    0
       U DOMAIN
            -10 10
       U SPACING
         0.5 0.5
       V DOMAIN
           -10 10
       V SPACING
         0.5 0.5
       RETURN
    ZOOM BOX
    CURVE TYPE
       CENTER/POINT
       RETURN
    crvs ADD
       0 0 0
       3.5 0 0
    FILL
```

(Zoom in on the center of the grid)

(Pick the center point) (Pick a point on the circle)





Figure 14-5 Generation of the Circular Hole

The next step is to create finite elements from the geometric entities. This will be done using the overlay mesh generator. The following button sequence will generate the mesh.

```
MAIN
  MESH GENERATION
     GRID
                                                                      (off)
     AUTOMESH
        2D PLANAR MESHING
           quadrilaterals (overlay) DIVISIONS
           15
               20
           quadrilaterals (overlay) QUAD MESH!
              2 4 5
                                                    (Use the Box Pick Method)
           1
                         6
           END LIST (#)
           RETURN
        RETURN
```



Figure 14-6 The Closed Contour for the OVERLAY Command



Figure 14-7 The Automeshed Part



The next step is to mesh the cantilevered portion. This section will be modeled as a 4 point quadrilateral surface and then converted to a 6x6 finite element mesh. The following button sequence will create the mesh.

```
MAIN
  MESH GENERATION
     srfs ADD
                                         (Pick the points in counter-clockwise order)
        3
        5
        6
        4
     CONVERT
        DIVISIONS
           6 6
        SURFACES TO ELEMENTS
                                                                 (Pick surface)
           1
           END LIST (#)
           RETURN
```

The overlay mesh generator may have created some unused nodes which must be removed. Furthermore, the nodes on the interface of the two meshes will not be coincident. Therefore, to merge them, the sweep tolerance should be large, approximately 0.5. and only the nodes along the interface selected. The following button sequence will merge the nodes. The sweep tolerance should be changed back to the default when the merge operation is finished.



Figure 14-8 Elements on the Square Surface

(Box Pick the nodes on the interface)







Figure 14-9 Correctly Connected Mesh



The final mesh general operation is to fold the two sides down. First, the nodes must be along the line of the fold. This will be done by creating a line along each edge and attaching the nodes to these lines. Then, the 2 corner elements will be divided into 2 triangular elements. These elements must still have the class of QUAD(4). This can be achieved by generating the triangular elements as degenerated quad elements, double clicking one node in the connectivity list.

The lines will be created by using the grid with a spacing of 5 and a size of 15. The origin of the grid must be set to the global origin. The following button sequence will create the lines and attach the nodes to them.


MAIN **MESH GENERATION** SET RESET **U SPACING** 5 **V SPACING** 5 **U DOMAIN** 0 15 **V DOMAIN** 0 15 grid ON RETURN **CURVE TYPE** LINE RETURN crvs ADD point(0, 15, 0) 12 point(0, 0, 0) 7 GRID ATTACH attach nodes CURVE 7 END LIST (#) 8 END LIST (#)

(on)

(Pick grid point) (Pick upper left point of the surface) (Pick grid point) (Pick lower left point of the surface) (off)

> (Pick lower line) (Pick nodes near lower line)

> (Pick upper line) (Pick nodes near upper line)





Figure 14-10 Nodes at the Top Half Attached to the Line

The 2 corner elements at the transition of the fillet to the square plate must be removed and 2 triangular elements will replace them. To create the triangular elements, the last node should be selected twice. The triangular elements have to be defined such that they allow for folding over the line segment. The following button sequence will create the first of 4 triangular elements.



	MAIN
	MESH GENERATION
	elems REM
(Pick elements at the triangular corners)	82 53
	END LIST (#)
(Pick nodes)	elems ADD
	113
	58
(First click on this node)	109
(Second click on this node)	109

Add three more triangular elements in the same way.



Figure 14-11 Corner Elements Replaced by Triangular Elements



The final step in the process of folding is to actually move the elements. This is done using the **MOVE** processor and rotating the elements. The lower edge will rotate 90 degrees and the upper edge will rotate -90 degrees. The following button sequence will fold the 2 edges.

MAIN	
MESH GENERATION	
MOVE	
ROTATIONS	
90 0 0 POINT	
15 0 0	(Pick point at end of bottom line of horizontal plate)
ELEMENTS	
	(Box Pick the lower elements to be folded)
END LIST (#)	
ROTATIONS	
-90 0 0	
POINT	
15 15 0	(Pick point at end of top line of horizontal plate)
ELEMENTS	
	(Box Pick the upper elements to be folded
END LIST (#)	



The following button sequence will show all four views and turn off the points and curves. It makes the viewing easier.



(off) (off)



Figure 14-12 The Complete FE Model



The next step is to apply the boundary conditions. First, the back edge of the bracket will be fixed in the 3 translational degrees of freedom. The following button sequence will fix the edge.

MAIN	
BOUNDARY CONDITIONS	
MECHANICAL	
FIXED DISPLACEMENT	
ON x displace	<i>(on)</i>
ON y displace	<i>(on)</i>
ON z displace	<i>(on)</i>
OK	
nodes ADD	
	(Box Pick the left edge of the plate,
	preferably in view 1 or 2)

END LIST (#)

The next step is to apply the face loads to the cantilevered portion of the plate. The loads will be 1 psi downward to represent the mechanical equipment. The following button sequence will apply the distributed loads.

```
NEW
FACE LOAD
PRESSURE
1
OK
surfaces ADD
1
END LIST (#)
```

(Pick the surface)





Figure 14-13 Loads Applied to All Elements Attached to the Surface

In reviewing the loads, it should be noted that the elements of which nodes have been moved do not have loads applied to them. This problem can be fixed by re-attaching these elements to the surface. The following button sequence will attach the elements and the program will then automatically apply the loads.



MAIN MESH GENERATION ATTACH attach elements SURFACE

(Pick surface) (Box Pick the Elements which are detached from the surface)

END LIST (#) RETURN



Figure 14-14 Added Surface Loads due to Renewed Attach



The next step is to assign the material properties. The material is steel and the mass density must be included because a dynamic analysis will be done. The following button sequence will assign the material properties.

```
MAIN
MATERIAL PROPERTIES
ISOTROPIC
YOUNG'S MODULUS
30e6
0.3
MASS DENSITY
0.283/386.4
OK
elements ADD
all: EXIST
```

(Poisson's ratio)

(Mass density)

The next step is to assign the thickness of the plates. Because the plates have different thicknesses, two geometric properties will be required. To make the picking of the elements easier, the user should select the partial picking capability in the device menu. The default is full which requires that the item of the requested type be fully contained in the graphical pick. The partial mode will select any item of the requested type that is partially in the graphical pick. The following button sequence will change the picking type and then assign the geometric properties to the elements.



```
MAIN
  DEVICE
     picking PARTIAL
     RETURN
  GEOMETRIC PROPERTIES
     3-D
        SHELL
           THICKNESS
              0.25
           OK
        elements ADD
                                             (Pick the Horizontal Plate Elements)
           END LIST (#)
        NEW
        SHELL
           THICKNESS
              0.5
           OK
        elements ADD
                                                (Pick the Vertical Plate Elements)
           END LIST (#)
```

To verify the geometric property assignment, the following button sequence changes some plot defaults and switches on the identification of geometries.

MAIN	
GEOMETRIC PROPERTIES	
3-D	
ID GEOMETRIES	<i>(on)</i>
PLOT	
elements SOLID	
REGENERATE	
RETURN	
ID GEOMETRIES	(off)





Figure 14-15 Graphical Confirmation of Applied Geometries

The next step is to create the static loadcase. It is a linear static with all loads selected and, therefore, no special options need to be selected. The following button sequence will create the loadcase.

MAIN LOADCASE mechanical analyses STATIC OK





The following button sequence will create the job and execute it.

Step 7

The final step in any analysis is to postprocess the results. This is done by opening the post file and reviewing the results. The deformations will be drawn using automatic scaling. The following button sequence will do this. However, there may be other results that the user wishes to review.



MAIN RESULTS OPEN DEFAULT DEF & ORIG NEXT INC NEXT INC SCALAR Equivalent von Mises Stress Layer 2 OK CONTOUR BANDS deformed shape SETTINGS deformation scaling AUTOMATIC

Note: The results for increment 0 and increment 1 are identical. This is caused by the fact that the pressure loading is still selected as an initial load. The MARC writer generates for increment 1 (i.e.the complete loadcase) the incremental load between the load vector and the initial load, which is a zero load increment. As a result, the deformation in increment 0 and increment 1 is identical.



Figure 14-16 Deformed Structure and Contours of von Mises Stress



Conclusion

The bracket will sustain the required static loads.



Dynamic Modal Shape Analysis

Overview of Steps

- **Step 1** Restore the database from the static analysis.
- Step 2 Create a modal dynamic loadcase and submit it.
- Step 3 Postprocess the results.



Detailed Session Description of the Modal Shape Analysis

Step 1

For the dynamic analysis, the same geometry will be used. The first step is to restore the database. Closing the post file will automatically restore the database. In addition, the commands are shown for restoring the database and resetting the program. (These commands are not necessary here). Finally, the plotting of points, curves, and surfaces will be switched off.

MAIN	
RESULTS	
CLOSE	
FILES	
RESTORE	
RESET PROGRAM	
RETURN	
VIEW	
show 4	
PLOT	
draw POINTS	(off)
draw CURVES	(off)
draw SURFACES	(off)
RETURN	
FILL	



The next step is to create a modal dynamic loadcase. The default is that 10 modes are determined which is enough for this structure. (Note that the determination of higher order modes in general required higher mesh densities). The following button sequence will create the loadcase.

MAIN LOADCASE NEW DYNAMIC MODAL OK

The next step is to create and execute the modal analysis job. The following button sequence will create and submit the job.

MAIN JOBS NEW MECHANICAL available Icase2 ANALYSIS OPTIONS OK OK RUN SAVE SUBMIT 1 MONITOR

(Verify that the LANCZOS method is used)



The next step is to postprocess the results. For modal analyses, not only the values of the eigenfrequencies but also the shape of the deflections or modal shapes are of interest. For the deformed shape, the automatically scaled deformations should be viewed. For ease of understanding, it is best to show all four views. The following button sequence will do the postprocessing.

MAIN RESULTS **OPEN DEFAULT** DEF & ORIG PLOT draw NODES MORE edges OUTLINE RETURN RETURN SCALAR Displacement z OK CONTOUR BANDS deformed shape SETTINGS deformation scaling AUTOMATIC RETURN **NEXT INC**

(off)

(Repeat until all modes have been viewed)

Finally, generate an animation sequence of one modal shape:

MAIN RESULTS MORE animate MODE 9 ANIMATION VIEW show view 4 FILL RETURN PLAY SHOW MODEL

(Number of frames for animation)





Figure 14-17 The Second Eigenmode

Observe that the calculated eigenfrequencies and corresponding eigenvectors are stored as so-called subincrements on the post file. The eigenfrequency value (in cycles/time unit) corresponding to a specific mode is printed on the top left of the screen.



increment	mode	value (cycles/time)
0:1	1	27.5
0:2	2	85.3
0:3	3	150.7
0:4	4	240.8
0:5	5	298.6
0:6	6	304.8
0:7	7	413.0
0:8	8	456.9
0:9	9	505.9
0:10	10	569.3

Table 14-1	Eigenfrequencies	Bracket

Observe that the eigen period of mode 1 will be 0.036 seconds.



Dynamic Transient Analysis

Overview of Steps

- **Step 1** Create an pressure time table for the loading. Then apply it as a face load in a loadcase.
- **Step 2** Create and submit a transient analysis.
- **Step 3** Postprocess the results.



Detailed Session Description of Dynamic Transient Analysis

Step 1

Restore the database to continue with the analysis preparation. The next step is to include a time dependent pressure history using the table option in the boundary condition menu. Here, the table which is included in the file c14.tbl on the Mentat installation tape is used. To load the table, the user will need the full path name to the Mentat subdirectory *examples/userguide* or will have to copy this file into his own directory. The following button sequences will restore the model and input the table.

MAIN RESULTS CLOSE FILES RESTORE **RESET PROGRAM** RETURN PLOT draw CURVES draw POINTS draw SURFACES FILL MAIN **BOUNDARY CONDITIONS MECHANICAL** TABLE LOAD c14.tbl FIT SHOW MODEL RETURN

(off) (off) (off)

The pressure history as function of the time is shown in Figure 14-18.





Figure 14-18 Transient Pressure Loading as Function of Time

Observe that the maximum pressure magnitude is 1 which is equal to the pressure as defined in the static loadcase. The maximum pressure is reached within 0.01 seconds and the pressure is equal to zero at 0.04 seconds and kept to zero until 0.09 seconds. Looking at the results of the eigenfrequencies in Table 14-1, it is to be expected that the lower modes will be dominating during the transient.

The next step is to assign this table to the pressure to the same elements as in the pressure load used in loadcase 1.



MAIN BOUNDARY CONDITIONS MECHANICAL NEW FACE LOAD pressure TABLE table1 OK OK faces ADD

END LIST (#)

(Box Pick the cantilevered elements)



Figure 14-19 Transient Pressure Loading



The next step is to create the loadcase for the transient analysis. The following button sequence will create the new loadcase.

```
MAIN

LOADCASE

NEW

DYNAMIC TRANSIENT

LOADS

apply2

OK

TOTAL LOADCASE TIME

0.09

fixed PARAMETERS

# STEPS

90

OK

OK

OK
```

(Deselect static load)



The next step is to create the job and submit it. The following button sequence will create and submit the job.

MAIN JOBS NEW **MECHANICAL** available lcase3 (Select loadcase 3) ANALYSIS OPTIONS (Verify that dynamic transient operator Newmark is used) OK **INITIAL LOADS** boundary conditions apply2 (Remove static load) OK JOB RESULTS scalars von mises OUT (Select outer and midplanes) OK JOB PARAMETERS **# SHELL/BEAM LAYERS** 3 OK OK SAVE RUN **SUBMIT 1** MONITOR



The final step in any analysis is to postprocess the results. This is done by opening the post file and reviewing the results. The following button sequence will do this. However, there may be other results that the user wishes to review.

```
MAIN

RESULTS

OPEN DEFAULT

NEXT INC

DEF & ORIG

deformed shape SETTINGS

deformation scaling FACTOR

5.

deformation scaling MANUAL

OK

SCALAR

Equivalent von Mises Stress Layer 2

OK

CONTOUR BANDS

MONITOR
```

This will walk through all 90 increments.



It is also possible to monitor a path plot. The following button sequence will show this.

```
MAIN
  RESULTS
    REWIND
    PATH PLOT
       NODE PATH
         171
         109
         121
         END LIST (#)
       SHOW IDS
         5
       VARIABLES
         ADD CURVE
            Arc Length
            Equivalent von Mises Stress Layer 1
            Arc Length
            Equivalent von Mises Stress Layer 2
            Arc Length
            Equivalent von Mises Stress Layer 3
         RETURN
       FIT
       YMAX
         36000
       MONITOR
       SHOW MODEL
```

The stresses for the nodes in the node path do not exceed the yield stress of 36000 psi.



As a last step, history plots are made. First, a plot of the stresses versus time is shown in Figure 14-20.

```
MAIN
  RESULTS
    REWIND
    HISTORY PLOT
     SET NODES
       166
       109
       113
       END LIST (#)
     COLLECT DATA
       0 90 1
       SHOW IDS
       0
    NODES/VARIABLES
       ADD VARIABLE
         Time
         Equivalent von Mises Stress Layer 3
       FIT
```





Figure 14-20 History Plot of von Mises Stress

Finally, the tip displacement in z-direction will be shown and compared with the value of due to the static load with the same magnitude (loadcase 1), which was found to be 0.28

```
MAIN

RESULTS

REWIND

HISTORY PLOT

SET NODES

171

END LIST (#)

COLLECT DATA

0 90 1

NODES/VARIABLES

ADD VARIABLE

Time

Displacement z

FIT
```





Figure 14-21 History Plot of Tip Z-displacement





Conclusion

Due to the dynamic loading, a larger value of the tip displacement is found for the same value of the maximum pressure. The bracket is shown to withstand the dynamic loads. The stresses never exceed yield.





Pressure Table

```
Pressure as function of time file c14.tbl:
# Title
table1
# X-axis Label
Х
# Y-axis Label
Υ
# Type
                  1
# Steps in X and Y
                 10
                                     20
# X-min, X-max, Y-min, Ymax
  0.00000000000e+00 9.0000000000e+00 -1.0000000000e+00
1.000000000000e+00
                 04
  0.0000000000e+00 0.0000000000e+00
                                                         1
  1.0000000000e+02 1.0000000000e+00
                                                         2
  4.0000000000e+02 0.0000000000e+00
                                                         3
  9.0000000000e+02 0.0000000000e+00
                                                         4
```



Procedure File

```
| Version : Mentat 3.3
*alias action
*exec_p gen_image.proc
*action | figure 14.0 - dummy
*action | figure 14.1 - dummy
*action | figure 14.2 - dummy
@top()
@push(meshgen)
@push(coord_system)
*grid_u_domain
-30 30
*grid_u_spacing
5
*grid_v_domain
-30 30
*grid_v_spacing
5
*set grid on
*fill view
@pop(1)
*add_curves
 point( 0.00000000000e+000,-1.0000000000e+001,
0.00000000000e+000)
point( 0.0000000000e+000, 2.5000000000e+001,
0.000000000000e+000)
 point( 1.5000000000e+001, 0.0000000000e+000,
0.00000000000e+000)
 point( 1.5000000000e+001, 1.5000000000e+001,
0.000000000000e+000)
 point( 3.00000000000e+001, 0.0000000000e+000,
0.000000000000e+000)
point( 3.0000000000e+001, 1.5000000000e+001,
0.000000000000e+000)
*action | figure 14.3
@push(curve_type)
*set_curve_type arc_cpp
@pop(1)
*add curves
22.5 - 27.5 0
```



```
15 0 0
0 -10 0
22.5 42.5 0
0 25 0
15 15 0
*action | figure 14.4
@push(coord_system)
*origin
7.5 7.5 0.0
*grid_u_domain
-10 10
*grid u spacing
0.5
*grid_v_domain
-10 10
*grid_v_spacing
0.5
@pop(1)
@push(curve_type)
*set_curve_type circle_cp
@pop(1)
*add_curves
0 0 0
3.5 0 0
*fill view
*action | figure 14.5
*set grid off
@popdown(append_procedure_browser_popmenu)
@push(automesh)
@push(automesh_planar)
*set_overlay_divisions
15 20
*action | figure 14.6
*overlay_mesh
1 2 4 5 6
#
*action | figure 14.7
@pop(1)
@pop(1)
  step 2
```



```
*add_surfaces
 3
 5
 6
 4
@push(convert)
*set_convert_divisions
66
*convert_surfaces
 1
# | End of List
*action | figure 14.8
@pop(1)
@push(sweep)
*remove_unused_nodes
*set_sweep_tolerance
0.5
*sweep_nodes
 83 89 93 97 101 107 109 123 130 137 144 151 158 165
# | End of List
*action | figure 14.9
*set_sweep_tolerance
0.001
@pop(1)
  step 3
@push(coord_system)
*system_reset
*grid_u_spacing
5
*grid_v_spacing
5
*grid u domain
0 15
*grid_v_domain
0 15
*set_grid on
@pop(1)
@push(curve_type)
```


```
*set_curve_type line
@pop(1)
*add_curves
point(0,15,0)
12
point(0,0,0)
7
*set grid off
@push(attach)
*attach_nodes_curve
 7
53 54 55 56 57 58 110
# | End of List
*attach_nodes_curve
 8
 11 12 13 14 15 16 78
# | End of List
*action | figure 14.10
@pop(1)
*remove elements
82 53
# | End of List
@pop(1)
@push(meshgen)
#
*add_elements
113
58
109
109
58
52
109
109
22
16
83
83
16
81
83
```



```
83
*action | figure 14.11
@push(move)
*set_move_translations
*set move rotations
90 0 0
*set_move_point
15 0 0
*move elements
1 2 3 4 5 6 41 42 43 44 45 46 47 48 49 51 52 132
#
*set move rotations
-90 0 0
*set move point
15 15 0
*move_elements
 35 36 37 38 39 40 81 83 84 85 86 87 88 89 90 91 92 129
# | End of List
@push(viewing_screen)
*show all views
*fill view
@push(plot_screen)
*set_points off
*set_curves off
*regenerate
*fill view
*action | figure 14.12
@pop(1)
@pop(1)
*move_curves
@pop(1)
@push(sweep)
*remove unused nodes
*fill_view
@pop(1)
@top()
  step 4 Apply boundary conditions and loads
@push(apply)
@push(apply_mechanical)
```



```
*apply_type fixed_displacement
@set($usersubfv,FORCDT)
@popup(fixed_disp_pm)
*apply_dof x
*apply_dof y
*apply dof z
@popdown(fixed_disp_pm)
*add apply nodes
70 71 72 75 78 82 84 90 94 98 102 108 110 114 117 120 121
#
*new_apply
*apply type face load
@set($usersubfv,FORCEM)
@popup(apply_face_load_popmenu)
*apply value p
1
@popdown(apply_face_load_popmenu)
*add_apply_surfaces
1
# | End of List
*action | figure 14.13
@pop(1)
@pop(1)
@push(meshgen)
@push(attach)
*attach_elements_surface
1
93 99 105 111 117 123
# | End of List
*action | figure 14.14
@pop(1)
#
@pop(1)
 step 5
@push(mat_props)
*material_type mechanical:isotropic
@popup(mat_isotropic)
*material_value isotropic:youngs_modulus
30e6
```



```
.3
*material_value isotropic:mass_density
.283/386.4
@popdown(mat_isotropic)
*add material elements
all existing
@pop(1)
@push(device)
*pick_inside_partial
@pop(1)
@push(geom_props)
@push(geometry mech three)
*geometry_type mech_three_shell
@popup(geom shell pm)
*geometry value thick
0.25
@popdown(geom_shell_pm)
*add_geometry_elements
 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24 25 26 27 28 29
30 31 32 33
 34 50 54 55 56 57 58 59 60 61 62 63 64 65 66 67 68 69 70 71 72 73
74 75 76 77
 78 79 80 93 94 95 96 97 98 99 100 101 102 103 104 105 106 107 108
109 110 111
 112 113 114 115 116 117 118 119 120 121 122 123 124 125 126 127 128
130 131
# | End of List
*new_geometry
*geometry type mech three shell
@popup(geom_shell_pm)
*geometry_value thick
0.5
@popdown(geom shell pm)
*add_geometry_elements
 1 2 3 4 5 6 35 36 37 38 39 40 41 42 43 44 45 46 47 48 49 51 52 81
83 84 85 86
 87 88 89 90 91 92 129 132
# | End of List
*identify_geometries *regen
@push(plot_screen)
*elements_solid
```



```
*regenerate
*action | figure 14.15
  step 6
@push(loadcase)
@push(lcase_mechanical)
*loadcase_type static
@popup(lcase static)
@popdown(lcase_static)
@pop(1)
@pop(1)
@push(jobs)
*job class mechanical
@popup(job mechanical)
@list(job_rem_lcase_list,0)
@list(job add lcase list,0)
*add job loadcases lcase1
@popup(tensor_scalar_popmenu)
@list(post tensor list1,0)
@list(post var list1,0)
*add_post_var von_mises
*post var outer layers von mises
*add_post_tensor stress
*post_tensor_outer_layers stress
@popdown(tensor_scalar_popmenu)
@list(job_add_lcase_list,0)
@list(job_rem_lcase_list,0)
@popup(job param mech popmenu)
*job_param layers
3
@popdown(job param mech popmenu)
@list(job rem lcase list,0)
@popdown(job_mechanical)
@push(file screen)
@set($file browser label,SAVE AS FILE)
@set($file_browser_command,*save_as_model)
@popup(save file browser popmenu)
@list(save_file_browser_filelist,0)
@list(save_file_browser_dirlist,0)
@popup(interface_confirm)
```



```
@popdown(save_file_browser_popmenu)
*save_as_model c14.mud yes
@popdown(interface_confirm)
@pop(1)
*update_job
@popup(job_run_popmenu)
*submit_job 1
*monitor job
@popdown(job_run_popmenu)
@top()
@push(results)
*post open default
*set_deformed both
*post next
*post next
*post_value
@popup(post_scalar_pm)
Equivalent Von Mises Stress Layer 2
@popdown(post_scalar_pm)
*post contour bands
@push(deformed_settings)
*set_automag on
*action | figure 14.16
@pop(1)
@pop(1)
 mode shape analysis
   step 1
@push(results)
*post close
@push(file screen)
*restore_model
*reset
*alias action
*exec_p gen_image.proc
@pop(1)
@push(viewing_screen)
*show_view 4
@push(plot screen)
```



```
*set_points off
*set_curves off
*set surfaces off
*regenerate
*fill view
@pop(1)
@pop(1)
@pop(1)
   step 2
@push(loadcase)
*new_loadcase
@push(lcase mechanical)
*loadcase type dynamic modal
@popup(lcase_dynamic_modal)
@popdown(lcase_dynamic_modal)
@pop(1)
@pop(1)
@push(jobs)
*new job
*job_class mechanical
@popup(job mechanical)
@list(job_rem_lcase_list,0)
@list(job_add_lcase_list,0)
*add_job_loadcases lcase2
@popup(job_an_opts_mech_popmenu)
@popdown(job_an_opts_mech_popmenu)
@list(job add lcase list,0)
@list(job_rem_lcase_list,0)
*save model
@popdown(job_mechanical)
*update job
@popup(job_run_popmenu)
*submit job 1
*monitor job
@popdown(job_run_popmenu)
@pop(1)
  step 3
```



```
@push(results)
*post_open_default
*set_deformed both
@push(plot_screen)
*set nodes off
@push(plot1_screen)
*edges_outline
@pop(1)
@pop(1)
*post_value
@popup(post_scalar_pm)
Displacement z
@popdown(post_scalar_pm)
*post_contour_bands
@push(deformed_settings)
*set_automag on
@pop(1)
*post next
*post_next
*post next
*action | figure 14.17
@push(results2)
*post_animate_mode
9
@push(animation_screen)
*animation_play
@push(viewing_screen)
*show_view 4
*fill view
@pop(1)
*animation_play
*show model
@pop(1)
@pop(1)
   dynamic transient analysis
   step 1
*post_close
@push(file_screen)
*restore_model
```



```
*reset
*alias action
*exec_p gen_image.proc
@push(plot_screen)
*set curves off
*set surfaces off
*set_points off
*regenerate
*fill view
@pop(1)
@pop(1)
@pop(1)
@push(apply)
@push(apply mechanical)
@push(table)
@set($table_browser_label,READ TABLE)
@set($table_browser_command,*table_read)
@popup(read_table_browser_popmenu)
*table_read c14.tbl
@popdown(read table browser popmenu)
*table_fit
*action | figure 14.18
*show model
@pop(1)
*new_apply
*apply_type face_load
@set($usersubfv,FORCEM)
@popup(apply_face_load_popmenu)
*apply table p0
@set($dof,p0)
@popup(apply_select_table_popmenu)
@list(select table list,0)
table1
@popdown(apply_select_table_popmenu)
@popdown(apply face load popmenu)
*add apply faces
 93:0 94:0 95:0 96:0 97:0 98:0 99:0 100:0 101:0 102:0 103:0 104:0
105:0 106:0
 107:0 108:0 109:0 110:0 111:0 112:0 113:0 114:0 115:0 116:0 117:0
118:0 119:0
 120:0 121:0 122:0 123:0 124:0 125:0 126:0 127:0 128:0
```



```
# | End of List
*action | figure 14.19
@pop(1)
@pop(1)
  step 2
@push(loadcase)
*new loadcase
@push(lcase_mechanical)
*loadcase type dynamic transient
@popup(lcase dynamic transient)
@popup(select_loads_popmenu)
@list(lcase_apply_list,0)
*remove loadcase loads apply2
@popdown(select_loads_popmenu)
*loadcase value time
.09
*loadcase_value nsteps
90
@popdown(lcase dynamic transient)
@pop(1)
@pop(1)
  step 3
@push(jobs)
*new_job
*job class mechanical
@popup(job_mechanical)
@list(job_rem_lcase_list,0)
@list(job add lcase list,0)
@popdown(job mechanical)
*job_class mechanical
@popup(job mechanical)
@list(job_rem_lcase_list,0)
@list(job_add_lcase_list,0)
*add job loadcases lcase3
@popup(job_an_opts_mech_popmenu)
@popdown(job_an_opts_mech_popmenu)
@list(job_add_lcase_list,0)
```



```
@list(job_rem_lcase_list,0)
@popup(init_loads_popmenu)
@list(job_apply_list,0)
@list(job_icond_list,0)
*remove job applys apply2
@popdown(init_loads_popmenu)
@list(job_rem_lcase_list,0)
@popup(tensor scalar popmenu)
@list(post_tensor_list1,0)
@list(post_var_list1,0)
*add_post_var von_mises
*post var outer layers von mises
@popdown(tensor_scalar_popmenu)
@list(job add lcase list,0)
@list(job rem lcase list,0)
@popup(job_param_mech_popmenu)
*job param layers
3
@popdown(job_param_mech_popmenu)
@list(job rem lcase list,0)
@popdown(job_mechanical)
*save_model
*update job
@popup(job_run_popmenu)
*submit job 1
*monitor job
@popdown(job run popmenu)
@pop(1)
@push(results)
*post_open_default
*post next
*set deformed both
@push(deformed settings)
*set_defmag
5.
@pop(1)
*post_value
@popup(post scalar pm)
Equivalent Von Mises Stress Layer 2
@popdown(post_scalar_pm)
*post_contour_bands
```



```
*post_monitor
*post_rewind
@push(post_pathplot)
*set_pathplot_path
 171
 109
 121
# | End of List
*set_pathplot_node_id
5
@push(pathplot_vars)
@list(path_var_list,0)
*pathplot_add
Arc Length
Equivalent Von Mises Stress Layer 1
*pathplot_add
Arc Length
Equivalent Von Mises Stress Layer 2
*pathplot_add
Arc Length
Equivalent Von Mises Stress Layer 3
@pop(1)
*pathplot fit
*set_pathplot_ymax
36000
*post_monitor
*show model
*post_rewind
@pop(1)
@push(post_history_plot)
*set_history_nodes
 166
 109
 113
# | End of List
*history_collect
0 90 1
*set_history_increment_id
0
@push(history_nodes_vars)
@list(hist_node_list,0)
```



```
@list(hist_var_list,0)
*history_add_var
Time
Equivalent Von Mises Stress Layer 3
*history_fit
*action | figure 14.20
@pop(1)
*show model
*set_history_nodes
 171
# | End of List
*history_collect
0 1000 1
@push(history_nodes_vars)
@list(hist_node_list,0)
@list(hist_var_list,11)
*history_add
171
Time
Displacement z
*history_fit
*action | figure 14.21
```





Chapter Overview

An effective means of augmenting the cooling effectiveness of a given thermal cooling design, is to increase the area exposed to the cooling fluid by means of adding fins. In the fin design, the effectiveness is judged by comparing the temperatures of the structure for conditions with and without fins. This sample problem determines two sets of temperatures reflecting the structure with and without a fin.

Background Information

Description

This problem demonstrates the preparation of a heat transfer model including convection boundary conditions.

Idealization

The model is a 0.15" X 0.05" rectangle with a 0.05" square fin centered vertically on the right side. The vertical sides have convection boundary conditions and the top and bottom are adiabatic.

Requirements for a Successful Analysis

The analysis is considered completed if a steady state analysis is performed for a structure with fin and a structure without fin.





Figure 15-1 Cross-Section of Cooling Fin



Full Disclosure

The model is 0.15" high and 0.05" wide. The fin is centered vertically and is 0.05" square. The convection on the left side is:

 $q = h(T - T_{\infty})$

with $h = 750 Btu/HR Ft^{2} {}^{o}F = 750 \times \frac{1}{3600(12)^{2}} Btu/in^{2} {}^{o}F$

$$T_{\infty} = 2500 \ ^{o}F$$

 $T_{m} = 1000 {}^{o}F$

The right side has a convection:

 $q = h(T - T_{\infty})$ with $h = 500 Btu/HR Ft^{2} {}^{o}F = 500 \times \frac{1}{3600(12)^{2}} Btu/in^{2} {}^{o}F$

The material has a coefficient of thermal conduction $\lambda = 1.157 \ 10^{-4} Btu \ in^2 \ ^oF$ For a steady-state analysis, it is not required to enter the mass density and the heat capacity.



Overview of Steps

- **Step 1** Create 2 surfaces and convert to finite elements.
- **Step 2** Add convection boundary conditions.
- **Step 3** Add material data.
- **Step 4** Create a steady-state loadcase.
- **Step 5** Create a thermal job and submit.
- **Step 6** Postprocess results.
- **Step 7** Delete fin elements.
- **Step 8** Modify convection boundary conditions.
- Step 9 Create new job and submit.
- Step 10 Postprocess results.



Detailed Session Description

Step 1

The first step will create 2 surfaces and convert them to finite elements. The following button sequence will create the surfaces and convert them.

```
MAIN
  MESH GENERATION
     SET
        U SPACING
          0.05
        V SPACING
          0.05
        U DOMAIN
          0 0.2
        V DOMAIN
          0 0.2
        grid ON
                                                                     (on)
        RETURN
     FILL
     ZOOM BOX
                                              (Box pick right upper half of grid)
     srfs ADD
                                                           (Pick grid points)
       point(0,0,0)
        point(0.05,0,0)
        point(0.05,0.15,0)
        point(0,0.15,0)
        point(0.05,0.05,0)
        point(0.1,0.05,0)
        point(0.1,0.1,0)
        point(0.05,0.1,0)
     GRID
                                                                     (off)
     CONVERT
        DIVISIONS
           3 6
        SURFACES TO ELEMENTS
                                                       (Pick the first surface)
           1
          END LIST (#)
        DIVISIONS
           3 2
        SURFACES TO ELEMENTS
                                                     (Pick the second surface)
           2
          END LIST (#)
```



The next button sequence will merge the duplicate nodes on the interface of the two surfaces.



(off) (off)



Figure 15-2 The Mesh Generated using the Convert Option



The next step is to add the convection boundary conditions. The following button sequence creates the boundary conditions.

```
MAIN
  BOUNDARY CONDITIONS
     THERMAL
        NAME
          hotside
        EDGE FILM
          COEFFICIENT (TOP)
              750/(3600*144)
          TEMP@INF (TOP)
             2500
          OK
        edges ADD
                                                      (Box Pick the left edge)
          END LIST (#)
        NEW
        NAME
          coolant
        EDGE FILM
          COEFFICIENT (TOP)
              500/(3600*144)
          TEMP@INF (TOP)
             1000
          OK
        edges ADD
                                                     (Box Pick the right edge;
                                                   several Boxes are required)
          END LIST (#)
        RETURN
     ID BOUNDARY CONDS
                                                                    (on)
     ID BOUNDARY CONDS
                                                                    (off)
```

Note that for the adiabatic conditions at top and bottom edges, no boundary conditions have to be applied.



Figure 15-3 The Film Conditions on the Hot and Coolant Side

The next step is to add the material data. The material is isotropic. The following button sequence assigns the material properties.

```
MAIN
MATERIAL PROPERTIES
HEAT TRANSFER
isotropic CONDUCTIVITY
1.157e-4
OK
elements ADD
all: EXIST.
```



The next step is to create a steady-state loadcase. The following button sequence will do this.

MAIN LOADCASE heat transfer analyses STEADY STATE LOADS OK OK

Step 5

The next step is to create a thermal job and submit the job for analysis. The following buttons sequence will do this.

```
MAIN
  JOBS
     HEAT TRANSFER
       loadcases SELECT
          Icase1
       analysis dimension PLANAR
       OK
     ELEMENT TYPES
       PLANAR
          39
          OK
       all: EXIST.
       RETURN
       SAVE
     RUN
       SUBMIT 1
       MONITOR
```



The final step for the first analysis is to postprocess results. The following button sequence will review the results.

MAIN RESULTS OPEN DEFAULT CONTOUR BANDS NEXT INC NEXT INC



Figure 15-4 Contours of Temperature for Structure with Fin



First, restore the database with the geometry. Then delete the fin elements. The following button sequence will modify the model.

MAIN RESULTS CLOSE scalar plot OFF RETURN FILES RESTORE RESET PROGRAM RETURN MESH GENERATION elems REM

END LIST (#) SWEEP remove unused NODES (Box Pick the fin elements)



Figure 15-5 Mesh without Fin



The next step is to modify convection boundary conditions on the edge where the fin was previously. The following button sequence will modify the convection.



(on) (off)

(to edit the second boundary condition)

(Box Pick the right edges)

(*on*) (*off*)



Figure 15-6 New Thermal Boundary Conditions



The next step is to create a new job and submit it. This will prevent overwriting of the previous post file. The following button sequence will do this.

MAIN JOBS NEW HEAT TRANSFER loadcases SELECT lcase1 analysis dimension PLANAR OK CHECK SAVE RUN SUBMIT 1 MONITOR

Step 10

The final step is to postprocess results of the second analysis. The following button sequence will review the results.

MAIN RESULTS OPEN DEFAULT CONTOUR BANDS NEXT INC NEXT INC



Figure 15-7 Results for Structure without Fin



Procedure File

```
| Version : Mentat 3.3
*alias action
*exec_p gen_image.proc
*action | figure 15.0 - dummy
*action | figure 15.1 - dummy
@top()
 step 1
@push(meshgen)
@push(coord system)
*grid_u_domain
0 0.2
*grid u spacing
0.05
*grid_v_domain
0 0.2
*grid_v_spacing
0.05
*set grid on
*fill view
@pop(1)
*add surfaces
point( 0.00000000000e+000, 0.0000000000e+000,
0.00000000000e+000)
 point( 5.0000000000e-002, 0.0000000000e+000,
0.00000000000e+000)
 point( 5.00000000000e-002, 1.5000000000e-001,
0.00000000000e+000)
point( 0.0000000000e+000, 1.5000000000e-001,
0.000000000000e+000)
*add surfaces
point( 5.0000000000e-002, 5.0000000000e-002,
0.000000000000e+000)
 point( 1.00000000000e-001, 5.0000000000e-002,
0.00000000000e+000)
 point( 1.00000000000e-001, 1.0000000000e-001,
0.000000000000e+000)
point( 5.0000000000e-002, 1.0000000000e-001,
```



```
0.00000000000e+000)
*set_grid off
@push(convert)
*set_convert_divisions
36
*convert surfaces
 1
# | End of List
*set_convert_divisions
3 2
*convert_surfaces
 2
# | End of List
@pop(1)
@push(sweep)
*sweep_nodes
all existing
@push(plot_screen)
*set_points off
*set surfaces off
*regenerate
*fill_view
@pop(1)
*action | figure 15.2
@pop(1)
 step 2
@push(apply)
*apply_name
hotside
@push(apply_thermal)
*apply_type edge_film
@set($usersubfv,FILM)
@popup(therm edge film pm)
*apply_value h
750/(3600*144)
*apply_value tinf
2500
@popdown(therm_edge_film_pm)
*add apply edges
```



```
1:3 4:3 7:3 10:3 13:3 16:3
# | End of List
*new_apply
*apply_name
coolant
*apply_type edge_film
@set($usersubfv,FILM)
@popup(therm_edge_film_pm)
*apply_value h
500/(3600*144)
*apply_value tinf
1000
@popdown(therm_edge_film_pm)
*add_apply_edges
 3:1
 6:1
 19:0
 20:0
 21:0
 21:1
 24:1
 24:2
 23:2
 22:2
15:1
 18:1
#
@pop(1)
*identify applys *regen
*action | figure 15.3
*identify_none *regen
@pop(1)
 step 3
@push(mat_props)
*material_type heat
@popup(mat_heat)
*material_value heat:conductivity
1.157e-4
@popdown(mat_heat)
```



```
*add_material_elements
all_existing
@pop(1)
  step 4
@push(loadcase)
@push(lcase_heat)
*loadcase_type steady_state
@popup(lcase_heat_steady_state)
@popdown(lcase_heat_steady_state)
@pop(1)
@pop(1)
@push(jobs)
*job_class heat
@popup(job_heat)
@list(job_add_lcase_list,0)
@list(job_rem_lcase_list,0)
*add job loadcases lcase1
*job_option dimen:two
@popdown(job_heat)
@push(element types)
@popup(eltype_planar_pm)
*element_type 39
@popdown(eltype_planar_pm)
all existing
@push(file_screen)
@set($file browser label,SAVE AS FILE)
@set($file_browser_command,*save_as_model)
@popup(save_file_browser_popmenu)
@list(save file browser filelist,0)
@list(save file browser dirlist,0)
*save_as_model c15.mud yes
@popdown(save file browser popmenu)
@pop(1)
@pop(1)
*update job
@popup(job_run_popmenu)
*submit_job 1
*monitor_job
```



```
@popdown(job_run_popmenu)
@pop(1)
@push(results)
*post_open_default
*post_contour_bands
*post_next
*post_next
*action | figure 15.4
  recapture the database
*post close
@push(file_screen)
*restore model
*reset
@pop(1)
@pop(1)
*alias action
*exec_p gen_image.proc
@push(meshgen)
  delete the fin elements
*edit_surfaces
*remove elements
 19 20 21 22 23 24
# | End of List
@push(sweep)
*remove unused nodes
*action | figure 15.5
@pop(1)
@pop(1)
@push(apply)
*next_apply
@push(apply thermal)
*add_apply_edges
 12:1
 9:1
# | End of List
*identify_applys *regen
*action | figure 15.6
```



```
*identify_none *regen
@pop(1)
@pop(1)
@push(jobs)
*new_job
*job_class heat
@popup(job_heat)
@list(job add lcase list,0)
@list(job_rem_lcase_list,0)
*add_job_loadcases lcase1
*job_option dimen:two
@popdown(job heat)
*save_model
*update job
@popup(job_run_popmenu)
*submit_job 1
*monitor_job
@popdown(job_run_popmenu)
@pop(1)
@push(results)
*post_open_default
*post_contour_bands
*post_next
*post_next
*action | figure 15.7
```


Index

Numerics

2D PLANAR MESHING 12-11

A

adaptive load stepping algorithm 8-32 ADD 3-7, 4-21, 8-11, 10-8, 11-8, 11-27, 11-30 adding an element 4-21 adiabatic thermal conditions 15-8 advancing front QUAD mesher 12-12, 12-14 ALIGN 11-14 analysis cycle 1-1 ANALYSIS OPTIONS 8-33 animation frames 10-29 APPEND 4-4 ATTACH NODES TO CURVE 8-15 automatic deformation scaling 11-63 automatic load stepping 8-31 automatic overlay meshing 10-15, 12-5 AUTOMESH 7-10 automesh 3-9

В

backing plate create 13-12 current active body 13-21 curves of 13-17 description 13-2 dimensions of 13-4 sample of 13-3, 13-13 basic solid types block 3-6 cylinder 3-6 prism 3-6 sphere 3-6 torus 3-6 BEAM CONTOURS 8-38 BEAM VALUES 8-38 Bezier curve 10-7, 10-8, 10-12, 10-13, 10-14, 10-19 bias factor 11-15 BIAS FACTORS 5-7 BLEND 6-2 boolean operations BLEND 3-6 CHAMFER 3-6 INTERSECT 3-6 SUBTRACT 3-6 UNITE 3-6 boundary conditions convection 15-2, 15-14 edge load 5-10 fixed displacement 5-9 kinematic 11-37 load 11-37 mechanical 5-9 menu 11-39, 11-53 thermal 15-8, 15-14 box pick method 11-25 button sequence diagram 4-5 buttons cycle 2-13 GRID 2-15 OK 2-14

1

one-only 2-14 RESET 2-14 submenu 2-13 tabular 2-14 toggle 2-14

С

CAD program 12-1 CANCEL 4-7 Cauchy stress 10-30 CENTER/POINT/POINT 14-7 CENTER/RADIUS/ANGLE/ANGLE (CRAA) 8-8, 9-8, 13-10 CHECK correct definition of mesh 3-7 FLIP ELEMENTS 5-8 UPSIDE DOWN 5-8 CHECK menu 5-8 CLEAR GEOM 3-7 CLEAR MESH 3-4 conceptualization 1-2 conductivity 15-9 conical flaring tool 7-9, 7-10 CONTACT tasks CONTACT AREAS 3-21 CONTACT BODIES 3-21 CONTACT TABLES 3-21 CONTOUR BANDS 5-18, 7-32 contour plots 3-25 contour plotting 5-18, 15-11 convection 15-2, 15-4, 15-14 CONVERGENCE TESTING 8-31 CONVERT 3-9 convert bias factors 5-7 surface to elements 5-7, 15-6 coordinate system 4-10, 5-4

coordinate system menu 7-5, 14-6 COORDINATE SYSTEM panel 4-10, 4-34 CREATE 4-4 CTRL-C 4-7 CURVE DIVISIONS 12-12 curve type arc/center/point/point 5-5 polyline 5-5 CYLINDER 9-9 cylinder 6-2 CYLINDRICAL 2-16, 7-7 cylindrical coordinate system 7-6 cylindrical tube 7-9, 7-10

D

data mode 2-8DEF & ORIG 5-17, 7-28, 8-35, 10-28 defaults isotropic hardening rule 8-29 von Mises yield criteria 8-29 deformable body 7-18 deformable contact body 3-21 deformed structure plotting 5-17 deformed structure scaling 5-17 degrees of freedom (d.o.f.) axial 8-19 radial 8-19 rotational 8-19 Delaunay triangular mesher 12-15 DESELECT 8-30 dialogue area 2-3directed attach method 9-16 DISTANCE 13-19 DIVISIONS 4-36, 7-10 DIVISIONS submenu 3-6 domain setting 5-4 DUPLICATE 7-7



duplicate nodes 15-7 solids 6-12 duplicate nodes 8-13 DYN. VIEW 11-27 dynamic viewing 11-27

Ε

EDGE LOAD 9-32 EDIT - identifier 2-28 element class QUAD4 4-21 type 15-10 type definition 5-15 ELEMENT TYPES 7-11 ELEMENTS 2-21, 4-19, 4-37 elements linear 3-13quadratic 3-13 ELEMS 4-21. 11-8 END LIST 2-21 END LIST (#) 4-38, 7-6 equivalent Cauchy stress 10-30 plastic strain 8-38 stress contour bands 10-31 EXECUTE 4-4 EXIST. 4-38, 4-42, 9-20, 10-17, 11-44 EXISTING 2-25 EXIT 4-7

F

face identification 10-27 FACE LOAD 9-28 FEM program 12-1 FILL 4-15, 4-35, 7-6, 8-26, 8-35, 8-37, 11-26, 12-3 film conditions 15-8 FIT 7-26 FIXED DISPLACEMENT 7-12, 11-37, 11-44 flaring tool 7-1, 7-2, 7-3, 7-4, 7-15 FLIP CURVES 10-22, 13-18 FLIP ELEMENTS 5-8 four main tasks analysis 2-11 configuration 2-11 postprocessing 2-11 FREQUENCY 8-34 FULL INTEGRATION 7-11

G

geometric entities curves 3-5 points 3-5 solids 3-6 surfaces 3-5 geometric meshing technique 13-7 geometric propertie - thickness 5-13 global coordinate system 11-5, 11-6, 11-22 graphics area 2-2 GRAVITY LOAD 11-44 GRID 4-34, 7-6 grid display 4-13 grid ON 4-13 grid spacing horizontal 5-4 vertical 5-4

Η

heat capacity 15-4 heat transfer 15-2 heat transfer job 15-10 Herrmann formulation 10-5 HISTORY PLOT 7-25

I

ID CONTACT 7-19, 10-22, 13-18 **ID GEOMETRIES** 9-34 **ID MATERIALS** 7-15 identifiers EDIT 2-28 NAME 2-28 NEW 2-28 NEXT 2-29 PREV 2-29 REM 2-28 identify contact bodies 7-18 IGES file 12-2 IGES format 12-1 INITIAL LOADS 3-18, 5-13, 9-34 INITIAL YIELD STRESS 7-15 initial yield stress 7-4 input grid 7-5 **INVISIBLE 2-26** ISOLATE 3-25 iso-surfaces plot 3-25 ISOTROPIC 7-15, 7-17

J

job definition initial loads 5-13 select results 5-14 JOB RESULTS 5-13, 8-33 JOBS 5-13, 7-23, 11-63 JOBS menu 5-13, 9-34, 11-63

Κ

keyboard input literal data 2-8 numerical 2-8

L

LABEL 4-18 LANCZOS method 14-33 LINE 8-11 LINE (2) 8-15, 11-8 LINE (3) 8-15 LINKS 7-14 literal data 2-8, 2-9 data mode 2-8string data 2-9 LOAD 4-4 LOADCASE menu 8-22 option 5-13, 7-22, 10-26 loading - flaring tool 7-20 loads cable 11-2, 11-3 gravitational 11-63 point 11-3, 11-37, 11-53, 11-63 self-weight 11-2, 11-3 wind 11-3, 11-37, 11-44, 11-63 local coordinate system 7-7, 11-5, 11-6, 11-7, 11-10, 11-12, 11-13, 11-20, 11-22, 11-35



Μ

MAIN 2-12 main menu 2-12 MAKE VISIBLE 11-26 manhole diameter 9-2 plate thickness 9-2 MANIPULATE CAMERA (ABSOLUTE) menu 11-6 MANIPULATE MODEL menu 11-27 MARC Element Type 52 11-3 mass density 15-4 MATERIAL PROPERTIES 7-15 material properties assignment 11-62 isotropic 5-12, 7-15, 11-61 tube elements 7-16 Young's modulus 5-12 MECHANICAL 5-13, 7-12, 8-32 Mentat window 2-2 menu area 2-3 MENU EXECUTE 4-4 MENU RECORD 4-4 mesh entities elements 3-3 nodes 3-3 MESH GENERATION 4-9, 5-4, 7-5, 12-9, 14-6 mesh generation panel 8-12 mesh generator QUAD MESH! 3-10 TETRA MESH! 3-12 **TRI MESH!** 3-10 mid-section completed 11-19 elements 11-14 HI-mid-section 11-16 LO-mid-section 11-16

sloping 11-13 subdivided elements 11-18 modal analysis 11-65, 11-66 modal dynamic loadcase 14-33 modal shapes 11-66 modeling 1-2 MODIFIED RIKS procedure 8-32 MONITOR 3-23, 5-15, 8-34, 13-25 MOORE 7-29, 13-19 mouse - 3 button functions 2-5 MOVE 7-6, 7-9

Ν

NAME identifier 2-28 tables 8-27 NEW contact bodies 10-21 identifier 2-28 NEXT identifier 2-29 results 3-24 NEXT INC 5-17, 11-66 node labeling 10-27 NODES 4-18, 4-41, 9-14, 9-20, 10-17 NON-POSITIVE DEFINITE 8-31 number of subdivisions 7-10 numerical 2-8

0

on-line help 2-6 OPEN 3-24 OPEN DEFAULT 11-66



OpenGL graphics 4-2 opening post files 5-17 operations SUBTRACT 6-7 UNITE 6-6 OVERLAY 14-11 overlay mesh generator 14-5, 14-6, 14-10 overlay mesher 12-9, 12-12

Ρ

panel COORDINATE SYSTEM 4-10, 4-34 DRAW 4-18 ELEMS 4-21, 11-8 LABEL 4-18 SWEEP 4-41 path plot show model 5-18 show path plot 5-18 variable definition 5-18 path plotting option 5-18 select node path 5-18 PLASTICITY 7-15, 8-29 PLOT 4-17 PLOT IDENTIFY 7-19 PLOT menu 3-6 plots deformed shape 3-25 history 3-26 label elements 4-17, 4-19 label nodes 4-17 label solid edges 6-8 path 3-26 scalar 3-25 tensor 3-25 vector 3-25

POINT LOADS 11-53 POINTS 7-6, 7-7 Poisson's ratio 5-11, 7-4, 11-61, 13-3, 14-3 polygon pick method 9-23, 12-5 post file contents 5-17 postprocessing - open post file 15-11, 15-15 preprocessing 2-11 PREPROCESSING panel 4-9 PREV - identifier 2-29 prism 6-2 procedure file playback 4-4 predefined stops 4-4 recording 4-4 stepwise playback 4-4 PROCEDURES 4-4 processors ATTACH 3-9, 8-15 AUTOMESH 12-9 BOUNDARY CONDITIONS 3-16, 11-37 CHANGE CLASS 3-4 COMPOSITE 3-20 CONTACT 3-21, 7-18 CONVERT 3-7, 5-7, 7-10, 8-12, 9-10, 13-7, 14-6 DUPLICATE 3-2, 3-7, 6-3, 6-21, 10-17 EXPAND 3-3, 3-7, 9-19, 10-17 GEOMETRIC PROPERTIES 3-20, 9-33, 11-55 INITIAL CONDITIONS 3-18 JOB RESULTS 3-23 JOBS 3-17, 3-23 LINKS 3-18 LOADCASE 3-17, 3-23 MATERIAL PROPERTIES 3-19, 7-15, 11-61 MOVE 3-7, 14-20 **ORIENTATION 3-20**



```
OVERLAY 3-10, 10-15
   PATH PLOT 2-11
   PLOT 2-26
   REFINE 10-16, 10-17
   RELAX 3-7
   RESULTS 3-24
   REVOLVE 3-6
   SELECT 2-25
   SOLIDS 3-6, 3-7
   STRETCH 3-7
   SUBDIVIDE 2-11, 2-21, 3-7, 10-16, 10-17
   SWEEP 3-7, 4-39, 6-17, 6-21, 8-7, 8-14,
    9-20, 10-17, 11-23, 12-3
   SYMMETRY 3-2, 3-7, 6-3, 6-19, 6-21,
    10-17
pts EDIT 10-12
punch
   dimensions of 13-4
   process of 13-2
   sample of 13-3
```

Q

QUAD 7-5, 9-21 QUAD(4) 7-11, 11-8, 14-16 QUIT 4-7

R

RECTANGULAR 2-16, 7-6 REFINE 3-15, 10-17 REGENERATE 7-10 RELATIVE FORCE TOLERANCE 8-31 REM - identifier 2-28 remove elements 15-12 RENUMBER 3-7 RESET PROGRAM 4-6 restore database 15-12 RESULTS 11-66, 13-25 results interpretation 1-2 RETURN 2-12, 4-16, 4-20, 4-39 rigid contact body 3-21, 10-21 RUN 5-15

S

SAVE 4-7, 4-43 SAVE & EXIT 4-7 save a model 4-43 SCALAR 7-31 SCALAR PLOT panel 7-32 SELECT 8-30, 8-32, 11-40, 12-6 SELECT menu 11-25 SELECT SET 11-26, 12-6 SELECT submenu 11-26 SELECTED 2-25, 5-9 SET 2-16, 4-10, 5-4, 7-5, 14-6 set naming 11-24 SHELL 9-33 SHOW 13-8 SHOW ALL VIEWS 11-10 SHOW MODEL 5-18 SKIP 3-24 solids blend 6-7 blending 6-2 block 6-10boolean operations 6-2cylinder 6-4 prism 6-4, 6-6 subtract 6-6, 6-13 unite 6-5 SPHERICAL 2-16, 7-7 SPRING/DASHPOT 7-14 srfs ADD 7-8



START/CONT 4-4 starting Mentat 4-2 static loadcase 14-27 steady-state heat transfer 15-4, 15-10 loadcase 15-10 STIFFNESS 7-14 stopping Mentat emergency stop 4-7 normal stop 4-7 STORE 2-25 STORE ELEMENTS 11-25 stress-strain data 8-5 definitions 8-27 STRETCH 9-14 string data 2-9 SUBDIVIDE 4-6, 4-35 subdivide divisions 4-36 elements 4-35, 4-37 SUBMIT 3-23, 5-15, 11-63 submitting a job 5-15, 15-10, 15-15 surface - add 15-6 surface type (ruled) 5-6SURFACES 7-9 SURFACES TO ELEMENTS 7-10 **SWEEP** menu 10-17 panel 4-41, 9-20 sweep nodes 4-39, 15-12

Т

 TABLE
 3-17, 3-22

 TABLE TYPE
 10-24

 TABLES
 8-1, 8-22, 8-27

TANGENT/RADIUS/ANGLE (TRA) 8-10 thermal boundary conditions 15-14 conduction 15-4 job 15-10, 15-15 material properties 15-9 THICKNESS 9-33 total equivalent plastic strain 7-31 TRANSLATIONS 7-6

U

U DOMAIN 4-11 U SPACING 4-12 UNSELECTED 2-25 UPDATE 3-23 UPSIDE DOWN 5-8 UTILITIES menu 13-19 UTILS 4-5

V

V DOMAIN 4-12 V SPACING 4-12 variables equivalent plastic strain 8-33 equivalent von Mises stress 8-33 VIEW 6-4 view activate 6-4 deactivate 6-4 perspective 6-4 show 6-4 VIEW menu 9-9, 11-6 VIEWPOINT 11-6 VISIBLE 2-26, 12-5, 12-9 von Mises stresses 7-30, 7-32

Index

W

wind load distributed 11-3, 11-37, 11-44 static analysis 11-63 strong 11-52 weak 11-53 workhardening curve 13-5, 13-15 workpiece dimensions of 13-4 sample of 13-3

Y

Y FORCE 11-52 YMAX 7-26 Young's modulus 5-11, 11-61, 13-3, 14-3

Ζ

Z FORCE 11-54 ZOOM 8-26 ZOOM BOX 7-14





Document Title:Mentat User's Guide, Version 3.3Part Number:UG-3009-03.3Revision Date:March, 1999

UNITED STATES

CALIFORNIA - Palo Alto MARC Corporate Headquarters 260 Sheridan Avenue, Suite 309 Palo Alto, CA 94306, USA Tel: 1 650 329 6800 Fax: 1 650 323 5892 Email: support@marc.com

CALIFORNIA - San Diego MARC Analysis Research Corp. 4330 LaJolla Village Drive, Suite 320 San Diego, CA 92122 Tel: 1 619 658 9588 Fax: 1 619 758 8710

CONNECTICUT - E. Berlin MARC Analysis Research Corp. 1224 Mill Street E. Berlin, CT 06023 Tel: 1 860 828 2055 Fax: 1 860 828 2031

MICHIGAN - Ann Arbor MARC Analysis Research Corp. 24 Frank Lloyd Wright Drive Ann Arbor, MI 48106-0523 Tel: 1 734 998 0540 Fax: 1 734 998 0542 GERMANY - Munich MARC Software Deutschland GmbH Ismaninger Str. 9 85609 Aschheim (bie Munich) Germany Tel: 49 (89) 904 50 33 Fax: 49 (89) 903 06 76 Email: support@marc.de

GERMANY - Hannover MARC Software Deutschland GmbH Alte Dohrener Str. 66 D 30173 Hannover, Germany Tel: 49 (511) 980 5182 Fax: 49 (511) 980 5187

ITALY

Espri-MARC s.r.l. Viale Brigata Bisagno 2/10 16129 Genova, Italy Tel: 39 (010) 585 949 Fax: 39 (010) 585 949 Email: espri.marc@interbusiness.it

THE NETHERLANDS MARC Analysis Research Corporation Dublinstraat 32 2713 HS Zoetermeer The Netherlands Tel: 31 (79) 3510 411 Fax: 31 (79) 3517 560 Email: support@marc.nl

EUROPE

CZECH REPUBLIC MARC Overseas, Inc. Podolska 50 147 00 Praha 4 Czech Republic Tel: 420 (2) 6121 4123 420 (2) 6121 4111 x252 Fax: 420 (2) 6121 4123 Email: support@marc.cz

FRANCE

MARC France S.A. Bercy Expo SR 6101A 40 Avenue des Terroirs de France 75611 Paris Cedex 12, France Tel: 33 (1) 4474 1550 Fax: 33 (1) 4474 1554 Email: support@marc.fr UNITED KINGDOM MARC UK Ltd. 11 Linford Forum, Rockingham Drive Linford Wood Milton Keynes, MK14 6LY Tel: 44 (1908) 606 070 Fax: 44 (1908) 606 633 Email: support@marc.co.uk

PACIFIC RIM

CHINA MARC Overseas, Inc. Bright Chang An Building, Suite 321 7# Jian Guo Men Nei Street Beijing, China 100 005 Tel: 86 (10) 6510 2056 /57 /58 Fax: 86 (10) 6510 2053 Email: marcbj@public.bta.net.cn

JAPAN - Nagoya Nippon MARC Co., Ltd. Nagoya Nishiki Daiichi Seimei Bldg. 2F, 6-5 Nishiki 1-chome Naka-ku, Nagoya, Aichi 460-0003 Tel: 81 (52) 202 5661 Fax: 81 (52) 202 5665

JAPAN - Osaka Nippon MARC Co., Ltd. 4F 2nd Kimi Building 2-11 Toyotsucho Suita-shi Osaka 564-0051 Tel: 81 (6) 385 1101 Fax: 81 (6) 385 4343

JAPAN - Tokyo (Asian Operations Headquarters) Nippon MARC Co., Ltd. Shinjuku Daiichi Seimei Building P.O. Box 5056 2-7-1 Nishi-Shinjuku, Shinjuku-ku Tokyo 163-0704 Tel: 81 (3) 3345 0181 Fax: 81 (3) 3345 1529 Email: support@marc.co.jp

KOREA Dong Kyung Bldg. 7FL 824-19 Yuksam-Dong Kangnam-Ku, Seoul, Korea Tel: 82 (2) 561 7543 Fax: 82 (2) 561 7767 Email: marck@unitel.co.kr