MARC

Analysis
Research
Corporation

Mentat II

User's Guide

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

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

Document Title: **Mentat II User's Guide**
Part Number: UG-3009-2.3
Revision Date: June, 1996

PROPRIETARY NOTICE

RESTRICTED RIGHTS NOTICE

TRADEMARKS

Table of Contents

Chapter 8: Container

Chapter 9: Manhole

Chapter 10: Tire

Chapter 11: Transmission Tower

Chapter 12: Importing a Model

Chapter 13: Punch

Chapter 14: Bracket

Chapter 15: Cooling Fin

Preface

The purpose of this manual is to introduce the first time user to the Mentat II program. The User's Guide covers the basics of the program and helps the novice user in becoming comfortable with Mentat II 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 II.
Part II consists of a series of sample sessions that provide the user with hands-on experience with the functionality of the Mentat II 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 II is used as a tool to accomplish these steps.

Chapter 2 **The Mechanics of Mentat II**, describes the user interface aspects of the program

Chapter 3 **Background Information**, expands on the common features of Mentat II and describes some of the underlying philosophies of the program.

Chapter 4 **Getting Started**, introduces you to Mentat II with a simple example of how to create a finite element model.

Chapter 5 **A Simple Example**, introduces you to use Mentat II 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 II 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.

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 II screen are in UPPER CASE.
- Literal user input and program prompts are in `typewriter` 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 **< I >**.
- A filename implies a concatenation of pathname and filename. The pathname may be omitted if the filename is in your current directory.

Chapter 1: Introduction

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

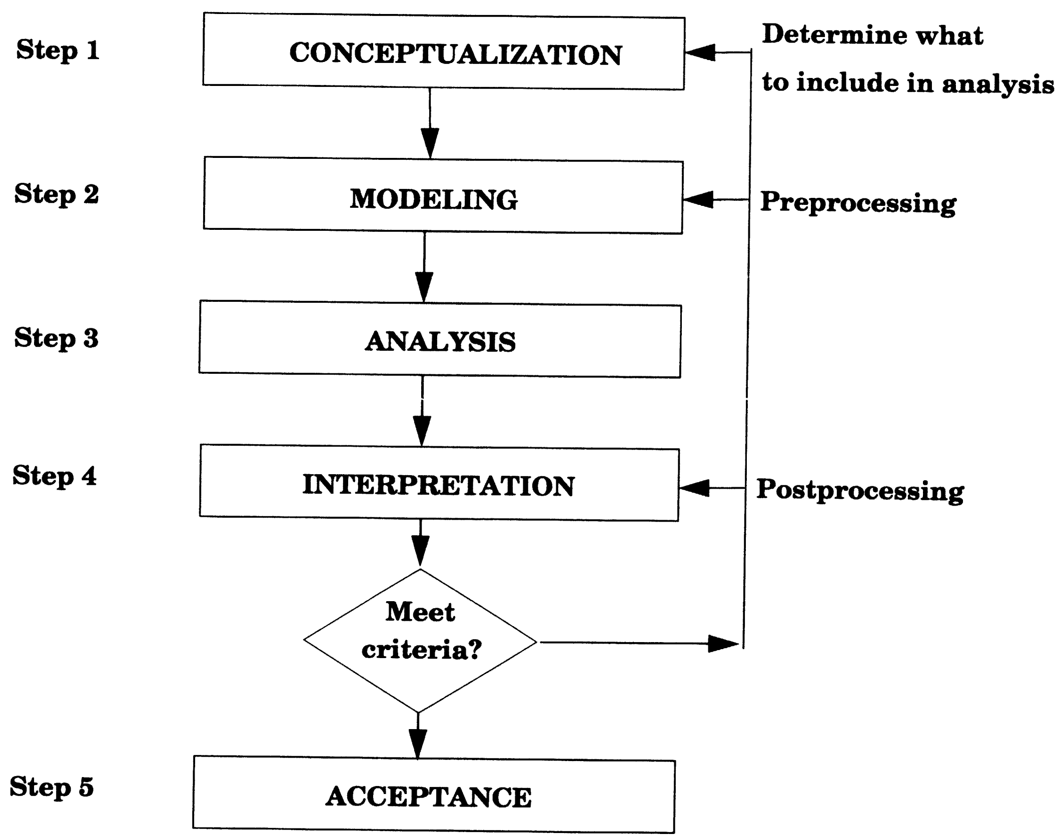


Figure 1.1 The Analysis Cycle

Brief Look at the Finite Element Analysis Process

In order to enhance your understanding of Mentat II, 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 II. 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 II 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.

Chapter 2: Mechanics of Mentat II

Chapter Overview

Before you get started with Mentat II you need to know how to communicate with the program. The goal of this chapter is to give you an overview of how Mentat II 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 II communicates with you
- How you communicate with Mentat II
- The menu system

2.1 Mentat II Window Layout

The starting point for all communication with Mentat II is the window shown in Figure 2.1 that appears at the start of the program.

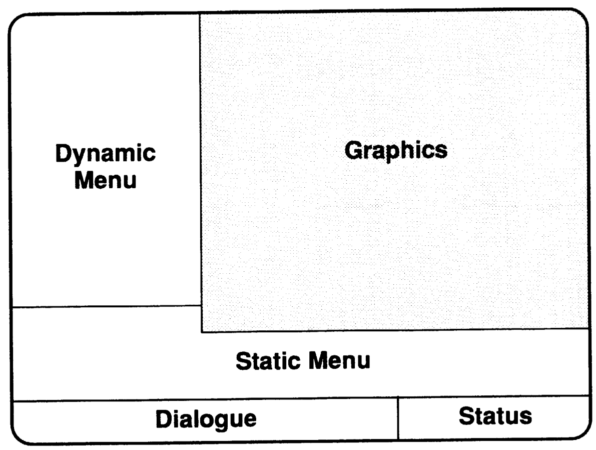


Figure 2.1 Basic Mentat II Window

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

- Graphics
- Menu
- Dialogue

The **graphics** area is used to display the current state of the database. When you start Mentat II, 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 sub-areas, 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.

2.2 How Mentat II Communicates with You

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

Mentat II'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 II warns you through a message posted in the dialogue area. Mentat II 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 <**CR**> instead of typing in the filename will tell Mentat II 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 II 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 <**CR**>.

2.3 How You Communicate with Mentat II

All interaction with Mentat II 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.

2.3.1 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 2.1 is a graphical representation of the mouse, mouse buttons, and corresponding cursor.

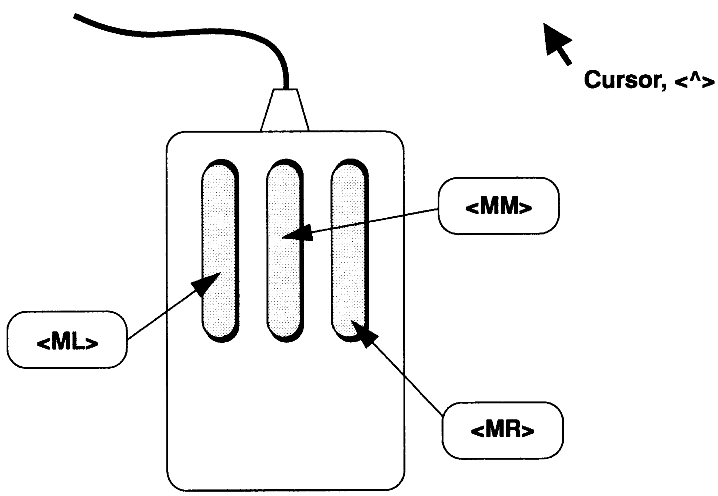


Figure 2.2 The Mouse, Mouse Buttons, and Corresponding Cursor

As mentioned under Documentation Conventions in the Preface, we have developed name conventions for each mouse button. The left button is represented by <**ML**>, the middle button by <**MM**>, and the right button by <**MR**>. The movements and location of the mouse are visually communicated to the user by the mouse cursor which is represented by a **<↑>** 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.

2.3.1.1 Using the Mouse to Select a Menu Item

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

On-line Help

An important companion for novice users in Mentat II 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 <↑> 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.

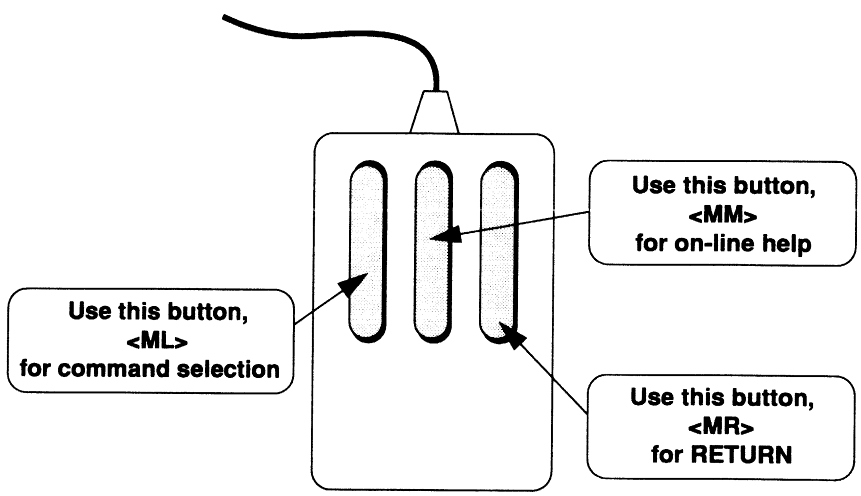


Figure 2.3 Using the Mouse in the Menu Area

2.3.1.2 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.

1 - The need to identify existing items displayed in the graphics area occurs frequently during a user's interaction with the Mentat II program. The mouse is used for this by moving the <↑> 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.

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

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 II 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.

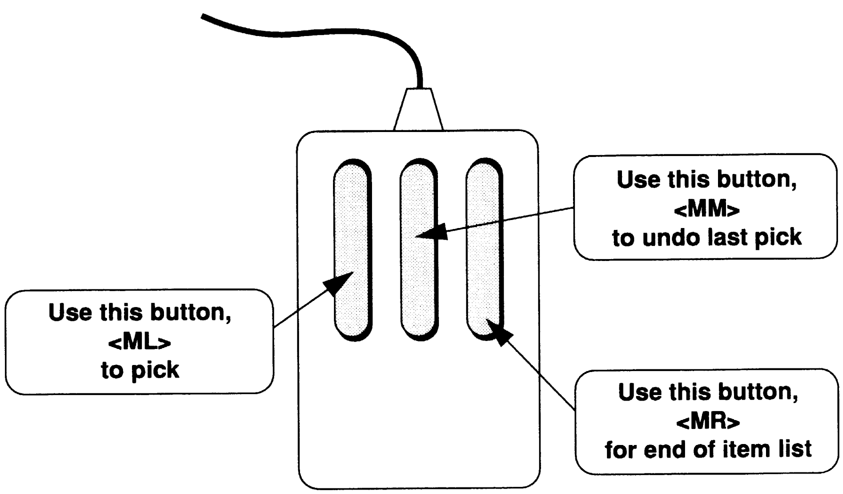


Figure 2.4 Using the Mouse in the Graphics Area

2.3.2 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 II 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 of Section 2.2 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 II 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 II 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 II 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 **<CR>** 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 II maintains a history of lines that are entered and offers limited recall and editing capabilities for the command line. The arrow keys ∧ and ∨ 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`.

2.4 Menu Structure

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

2.4.1 Menu System

The kernel of the Mentat II 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 II 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 2.5 shows you the organization of the main menu that appears when you start Mentat II 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!

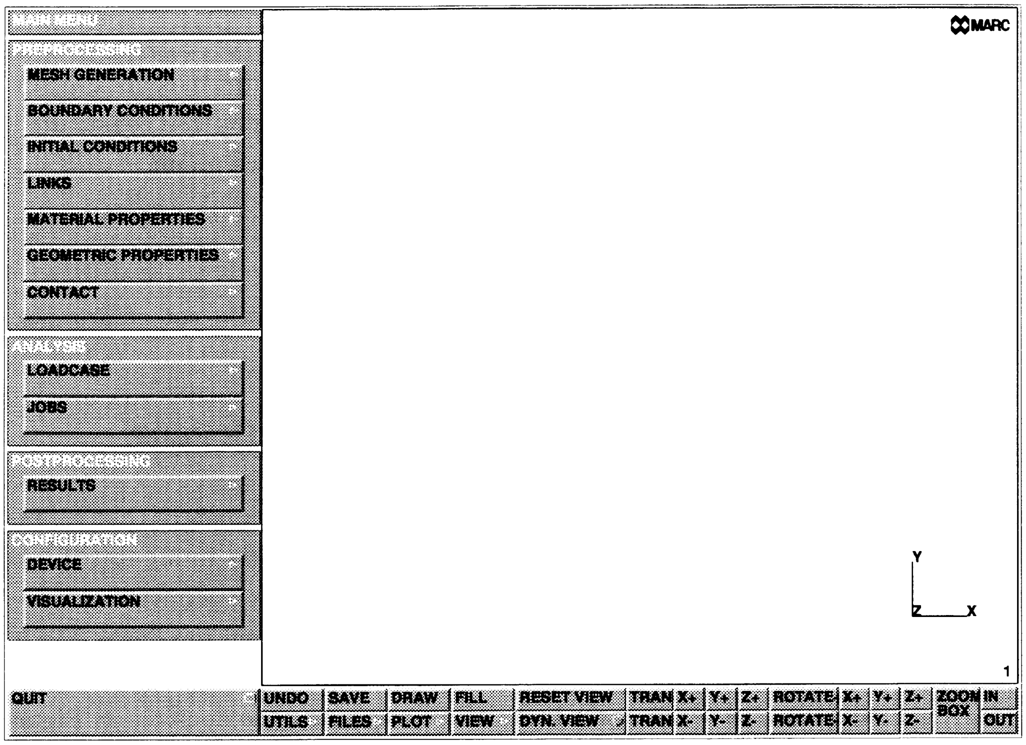


Figure 2.5 Organization of Main Menu

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 <↑> over the menu area and click the <**MR**>. The result of this sequence is equivalent to clicking on the RETURN menu item.

2.4.2 Menu Components

This section describes the anatomy or different components of the Mentat II 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 <**ML**>. Flat rectangles are not selectable; they convey the setting of parameters. The program does not respond to clicking the <**ML**> or <**MM**> on the flat rectangles.

Mentat II 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.

1. **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.

2. **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.

3. **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.

4. **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.

5. **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 ▤ 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 2.9.

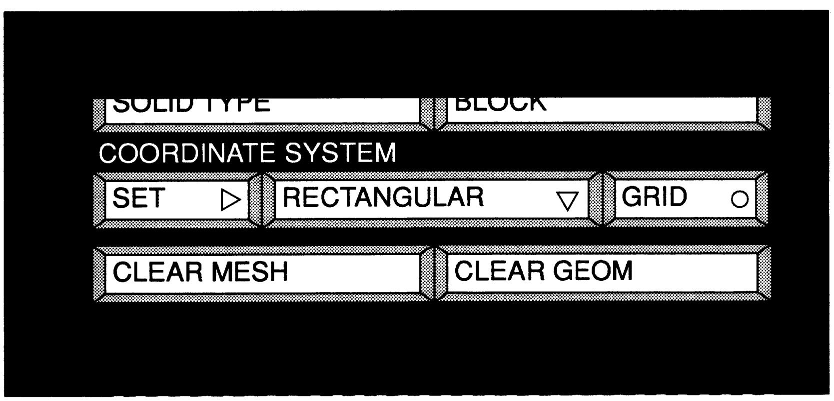


Figure 2.6 Coordinate System Panel

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.

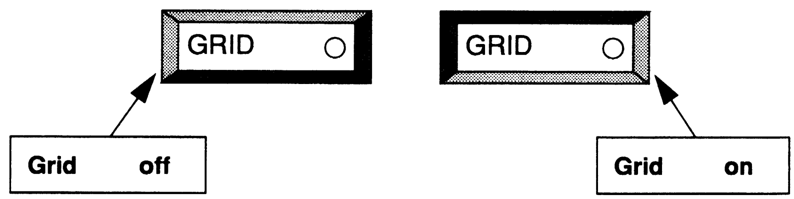


Figure 2.7 Released and Depressed States of a Toggle Button

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.

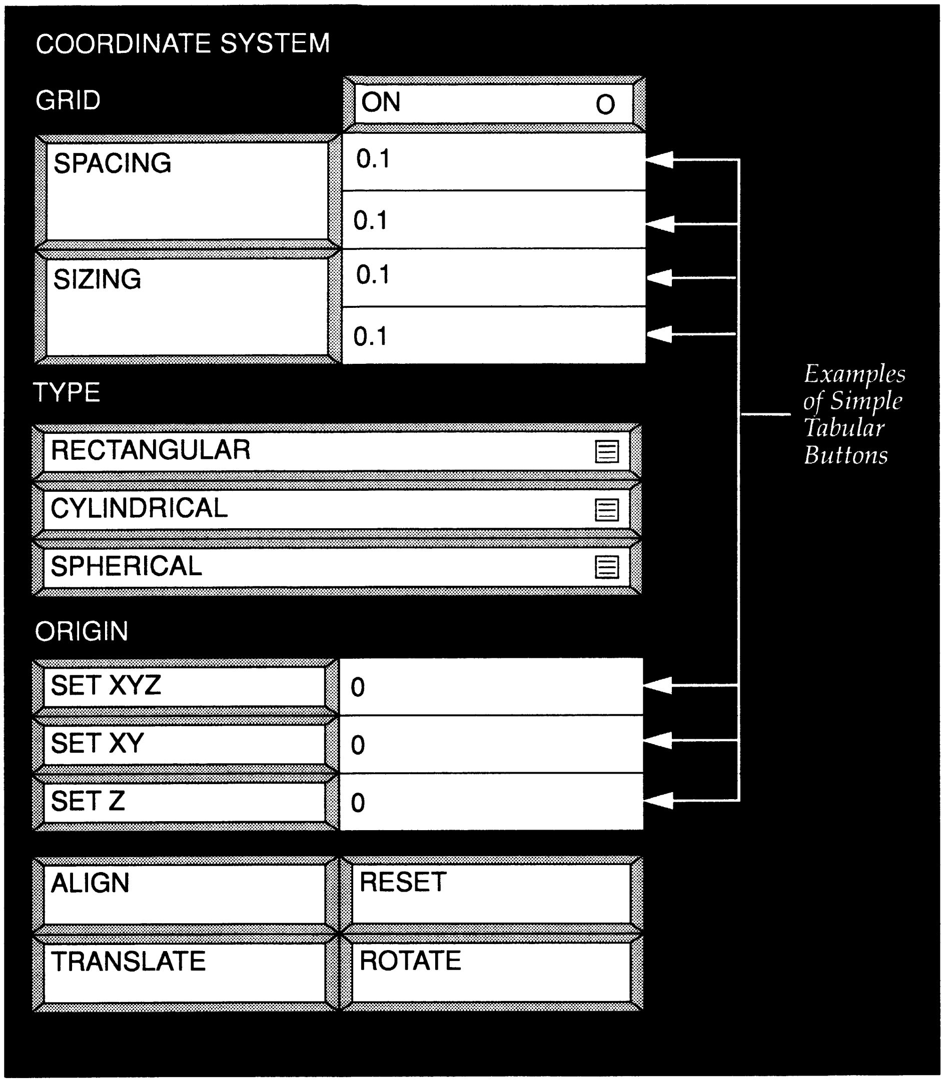


Figure 2.8 Example of Simple Tabular Buttons

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 II menu.

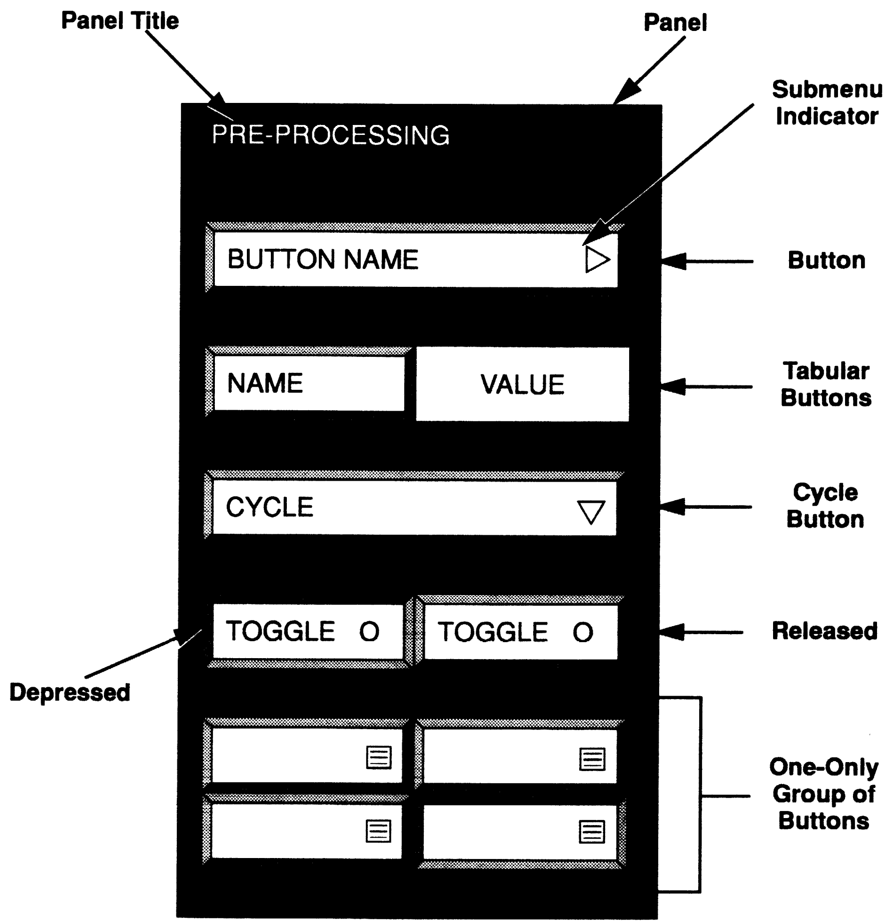


Figure 2.9 Summary of Mentat II Menu Buttons

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 <**MR**> since the <↑> 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 <↑> at one of the corners of the box. Depress the <**ML**> and move the <↑> 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 <**ML**>. Every element that is *completely* inside the box is included in the list specification.

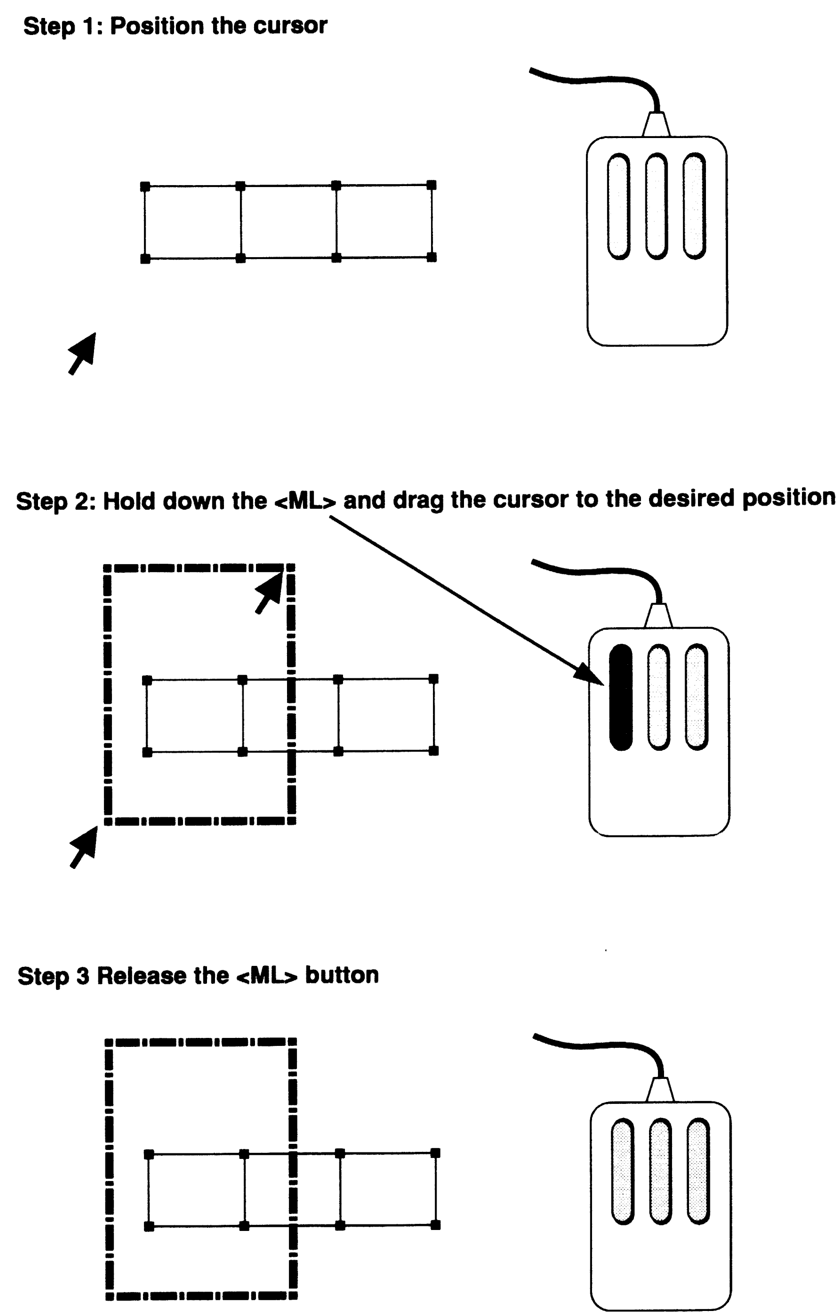


Figure 2.11 Selecting an Element Using the Box Pick Method

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 **<↑>** 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 **<↑>** 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 position near the beginning of 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.

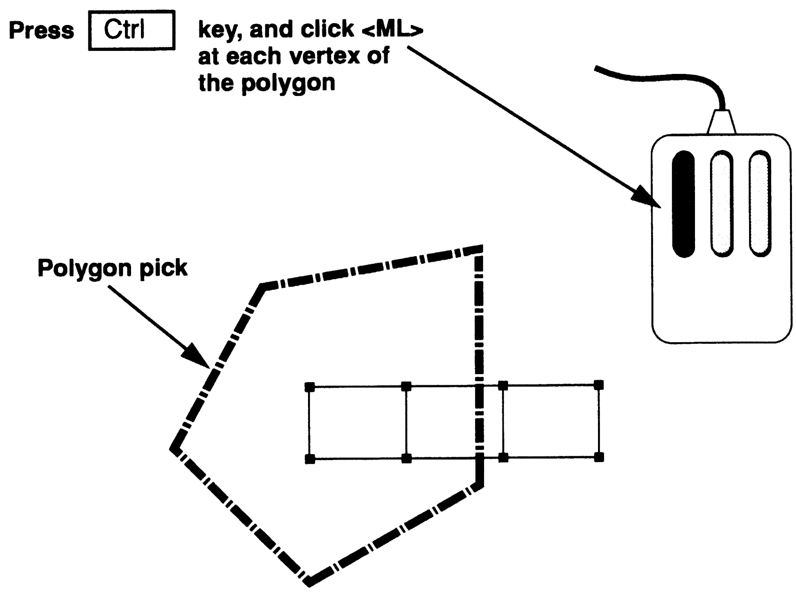


Figure 2.12 Selecting an Element Using the Polygon Pick Method

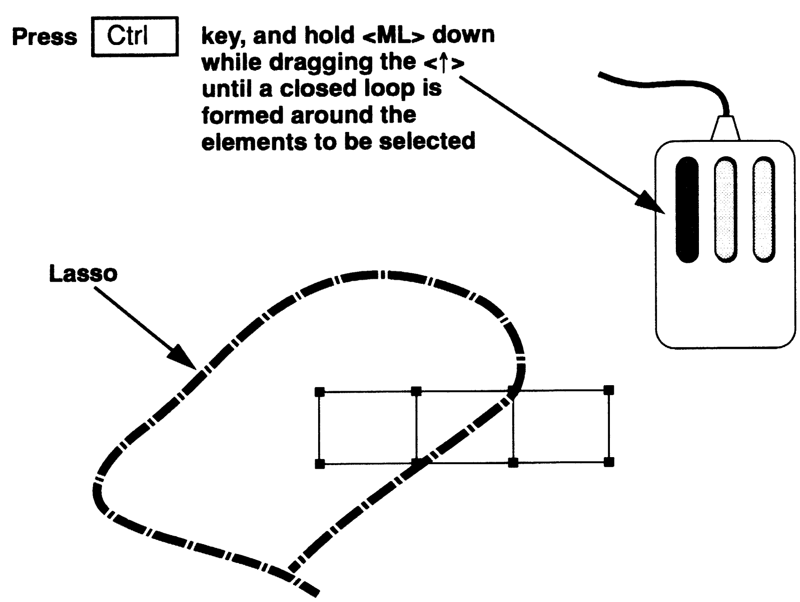


Figure 2.13 Selecting an Element Using the Lasso Pick Method

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.

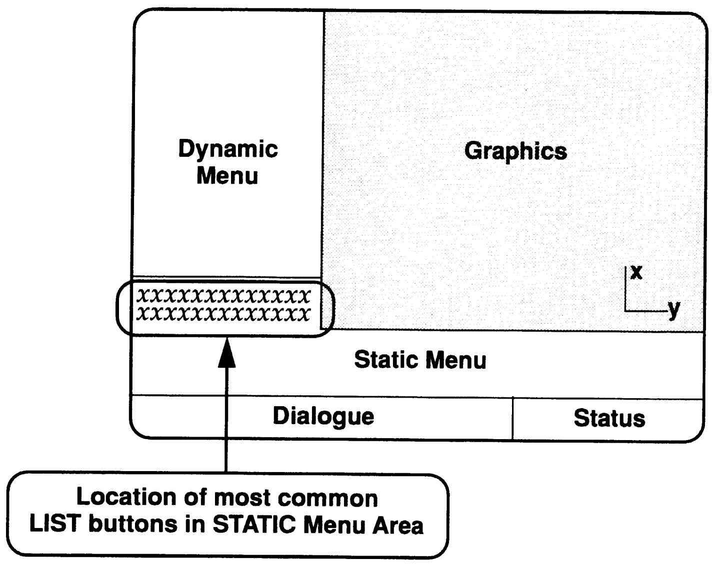


Figure 2.14 Location of LIST Buttons

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)

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.

The usage of **SELECT** will be demonstrated throughout the sessions in Part II of this manual.

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.

Table 2.1 Mouse Button Functions in Graphics Area

	<ML>	<MM>	<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

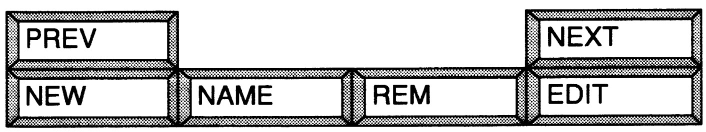
Table 2.2 Mouse Button Functions in Menu Area

	<ML>	<MM>	<MR>
	command selection	on-line help	return

2.6 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.



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.

REM

removes the current application id and the associated data.

EDIT

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

PREV

selects the previous id and makes it the current application.

NEXT

selects the next id and makes it the current application.

Chapter 3: Background Information

Chapter Overview

The purpose of this chapter is to give you an overview of the most common Mentat II features. These features recur throughout the sample sessions in Part II 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 II 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

3.1 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 II 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 II is rectangular Cartesian. A cylindrical or spherical coordinate system may be more suitable for a particular model.

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

3.1.1 Mesh Entities

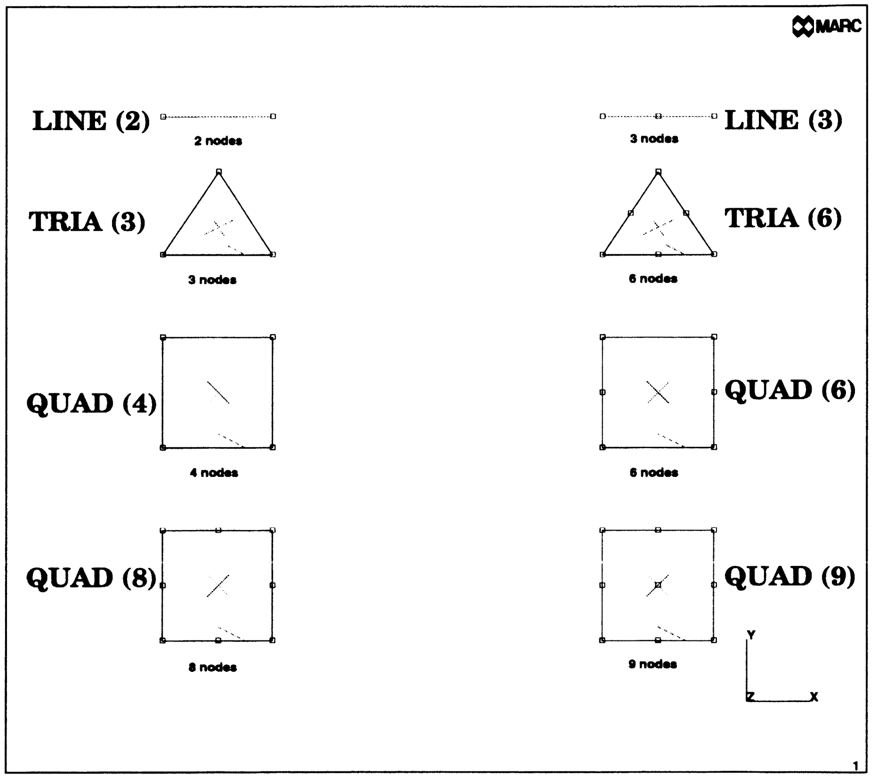


Figure 3.1a Element Classes

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

- **Nodes**

Nodes are characterized by three coordinates and symbolized by a □ on the screen. When nodes are attached to a geometric entity, the symbol ○ 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 II employs a wide variety of element classes which are identified in Figure 3.1a and

Figure 3.1 b. 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.

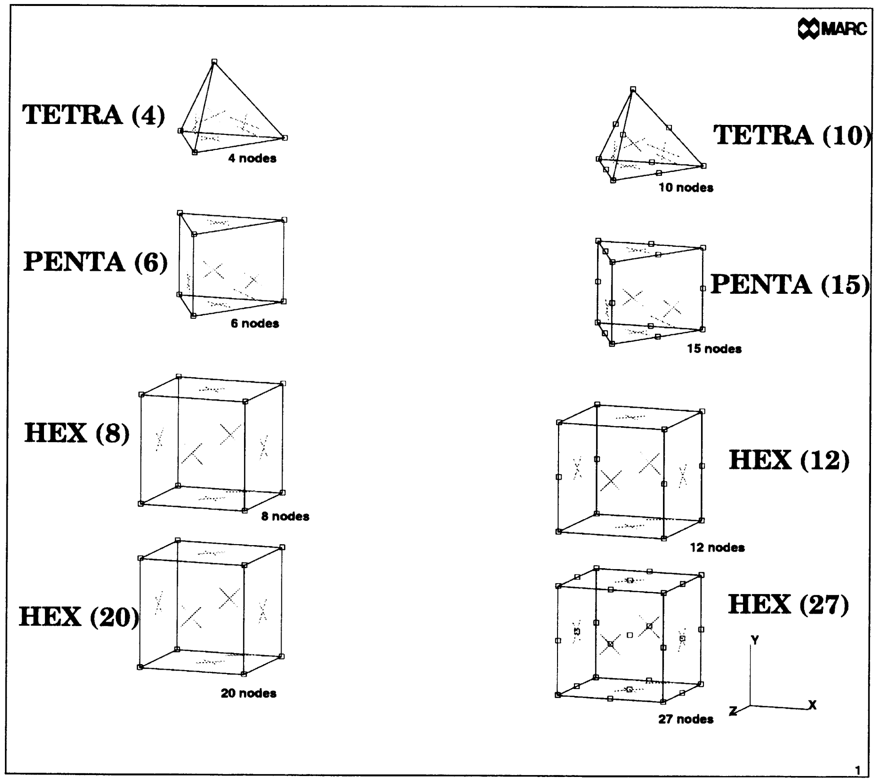


Figure 3.1 b Element Classes

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.

3.1.2 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 II 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 II 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 II 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.

3.1.3 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 II 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.

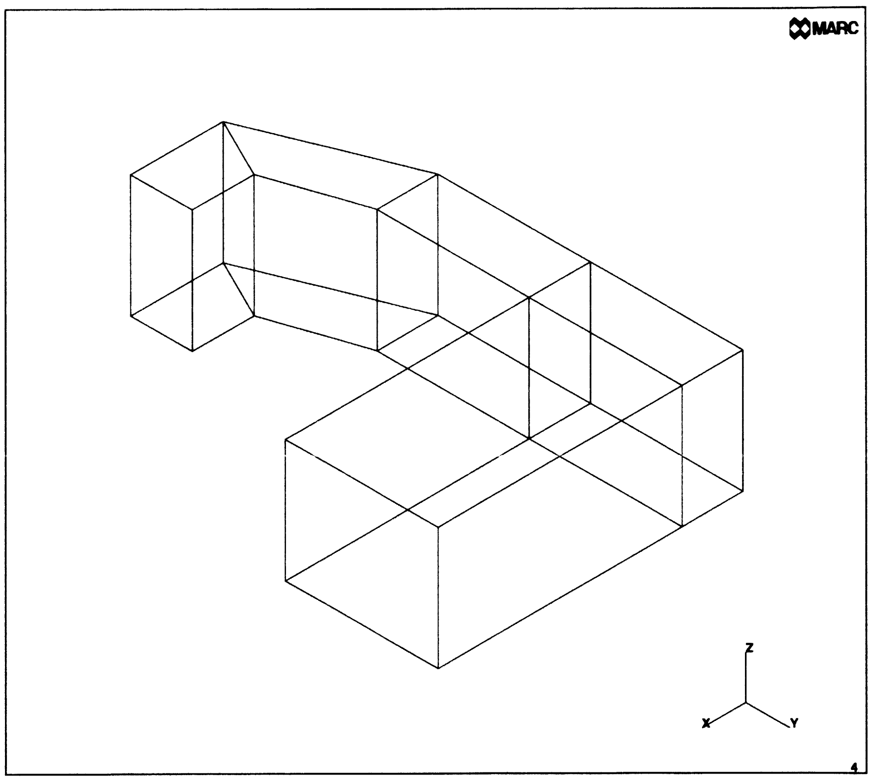


Figure 3.2 Example of a Coarse Mesh

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.

The direct mesh generation process in Mentat II 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.

3.1.4 The Geometric Meshing Technique

The basic building blocks for this technique are geometric entities rather than mesh entities. The geometric entities available in Mentat II 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.

- **Overlay**

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 II to specify the boundary. (This also implies that a combination of curve types is permitted.)

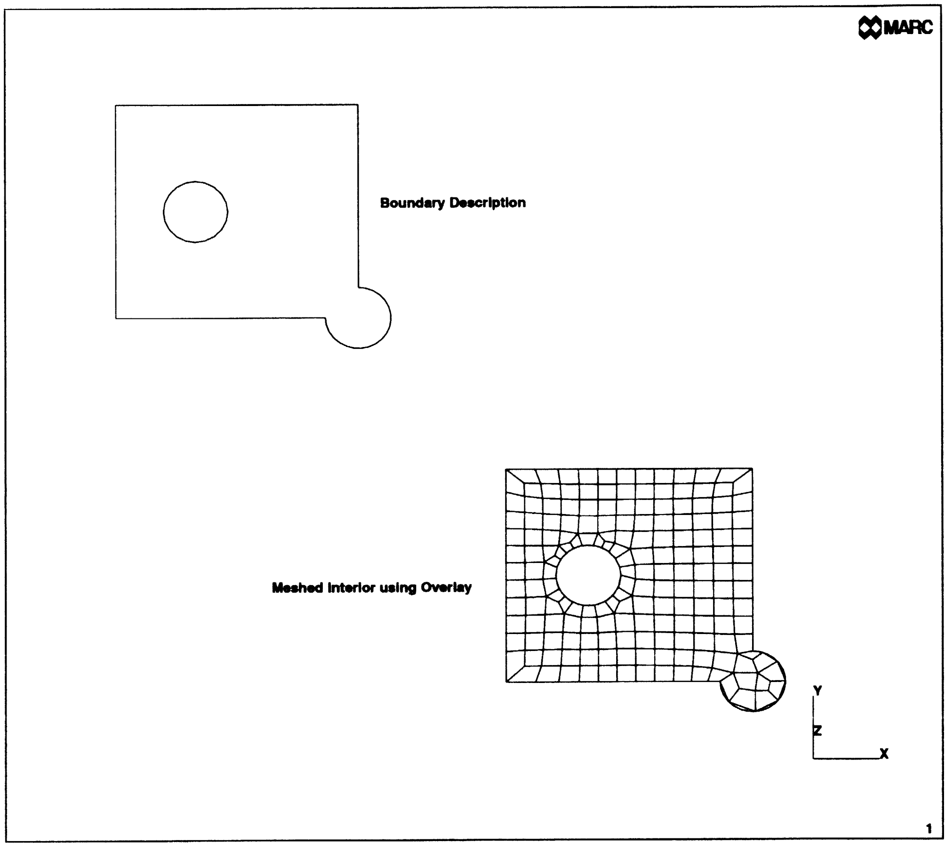


Figure 3.3 Hole in a Plate

There are restrictions that apply to the domain that is to be meshed:

- The boundary contour must be closed. If gaps exist in the boundary, the mesh will bleed into the ambient space.
- The minimal distance between two boundary contours must be greater than 2h, where h is the typical size of the element.
- Only flat regions and trimmed surfaces can be meshed.
- The resulting mesh contains elements of the quadrilateral family only.

NOTE

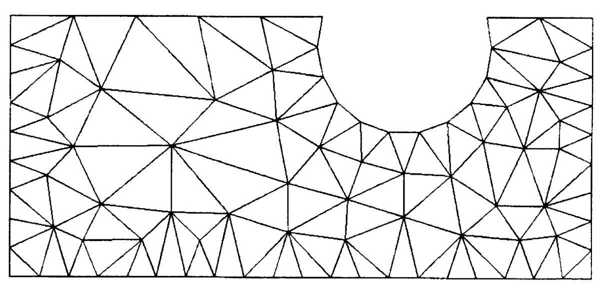
Be aware that this overlay method generally creates more elements than expected as the typical element size is uniform over the entire domain to be meshed.

An alternative technique is often to ignore the gaps at first, and to create them later by using the appropriate editing tools. If element densities in certain regions of the model are an issue, you may want to split the region into parts leaving some room to manually create a transition mesh.

- **Automesh**

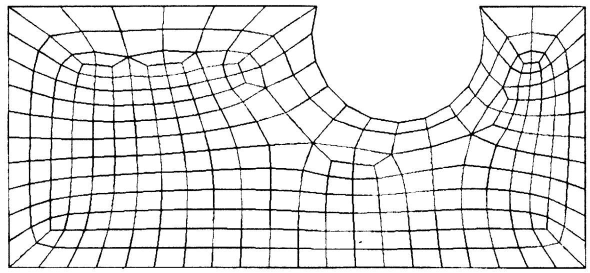
Mentat II contains as optional products automatic mesh generators which generate finite element meshes on solids, on trimmed surfaces and within curves in a plane. Three automatic mesh generators are currently available:

The **TRI mesh generator** creates triangular elements on the faces of a solid, on a trimmed surface or within curves in a plane.



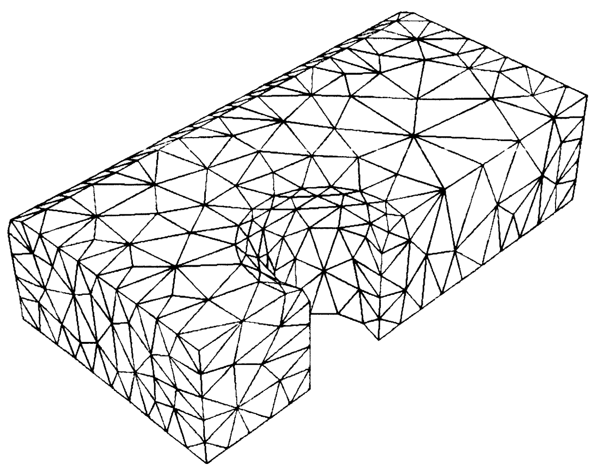
Triangular Mesh

The **QUAD mesh generator** creates quadrilateral elements on the faces of a solid, on a trimmed surface or within curves in a plane.



Quadrilateral Mesh

The **TETRA mesh generator** creates tetrahedral elements in a solid volume or within a volume bounded by triangular elements.



Tetrahedral Mesh

Either the number of divisions or the average element length can be specified on the bounding curves or on the edges of the solid. Mentat II will try match the element mesh as closely as possible to these specifications. Note that due to current limitations in the ACIS solid modeler it is necessary to SPLIT the faces of periodical surfaces (such as cylinders and spheres) before meshing.

3.1.5 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:

$AR \leq 3$ for linear elements
$AR \leq 10$ 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.

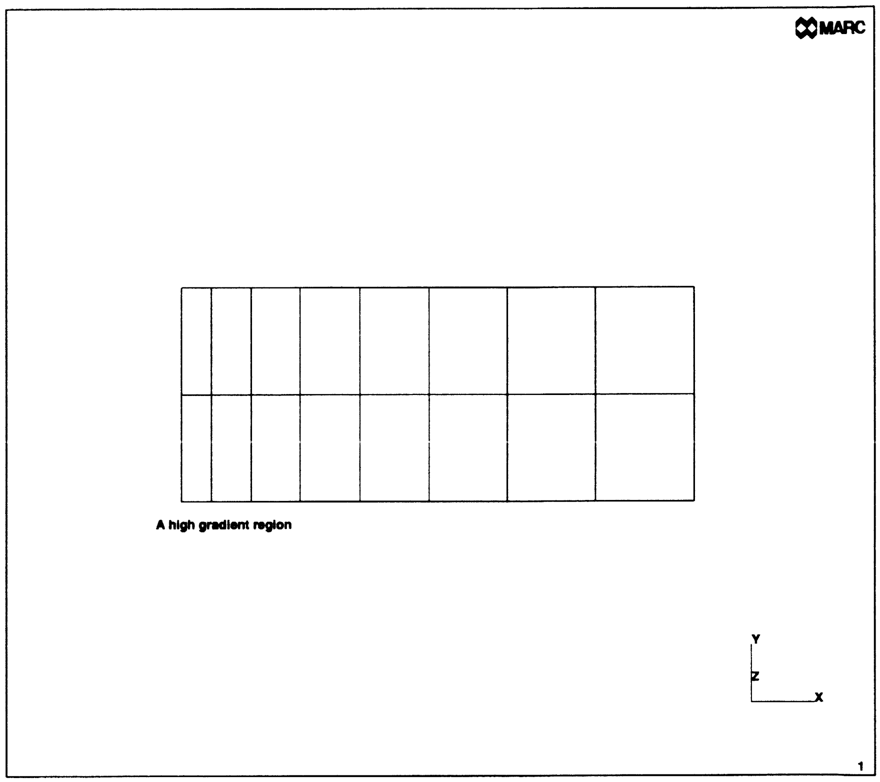


Figure 3.4 A Biased Mesh

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

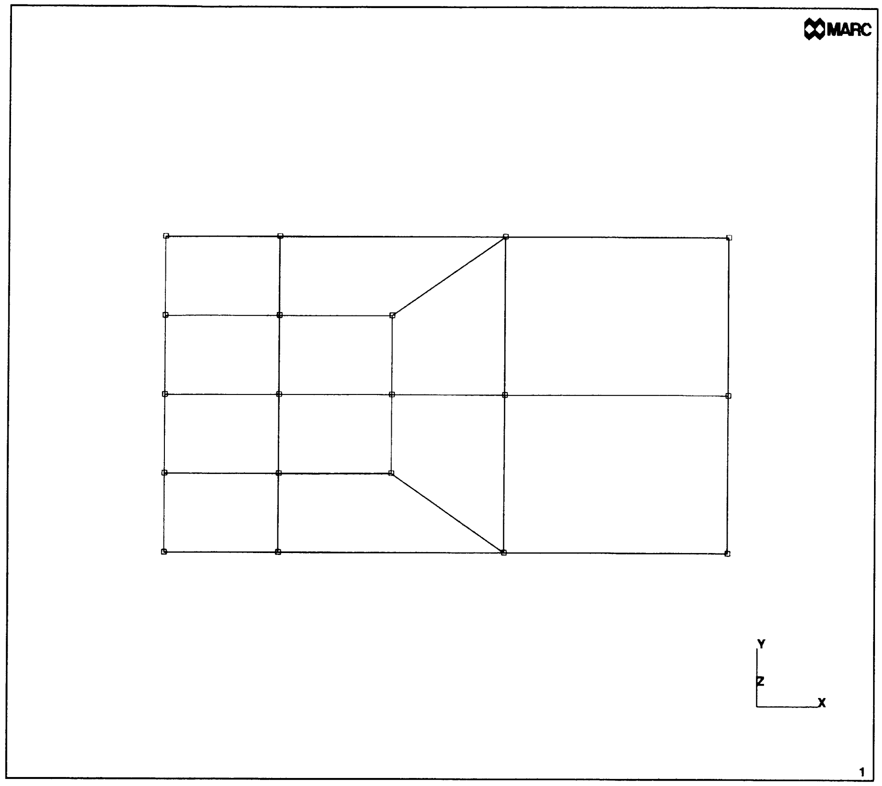


Figure 3.5 A Transition Mesh

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 II allows you to create a transition region.

NOTE

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

3.2 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 II 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.)

3.3 Material and Geometric Properties

Virtually all of the required material data for an analysis with MARC may be entered through Mentat II. 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.

Section 2.5 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, axi-symmetric, 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.

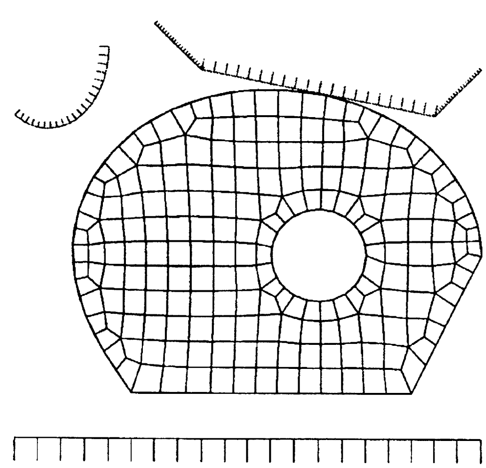
For some element types, special options may be flagged in order to get more accurate results. For instance the classical 4 node plane strain element is known to give a too stiff behavior if the element is subjected to bending. By selecting the assumed strain formulation the element type is modified into 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.

3.4 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 II 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 2-D applications and surfaces for 3-D 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.

3.5 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 eigenfrequency analysis based on pre-stressed structure
- Loadcase 3: Perform transient analysis using superposition of eigenmodes. 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.

3.6 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 II.

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

The deformations found in a mechanical analysis can be shown in what is known to Mentat II 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 II 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.

Chapter 4: Getting Started

Chapter Overview

This chapter describes the routine interactions with Mentat II listed below.

- ☐ Starting Mentat II,
- ☐ Using the PROCEDURE option,
- ☐ Stopping a Mentat II 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 II to venture on to the sample sessions in Part II of this manual.

4.1 Starting the Mentat II Program

Before you start the Mentat II program...

1. You will need an account so you can use Mentat II on your system.

2. If you don't know how to invoke Mentat II, ask a current Mentat II 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 II program is based on X-Windows™; 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 II 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 II screen which consists of a main menu, a blank graphics area and a dialogue area. Figure 4.1 shows you the initial Mentat II display.

If the Mentat II 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.

4.2 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.

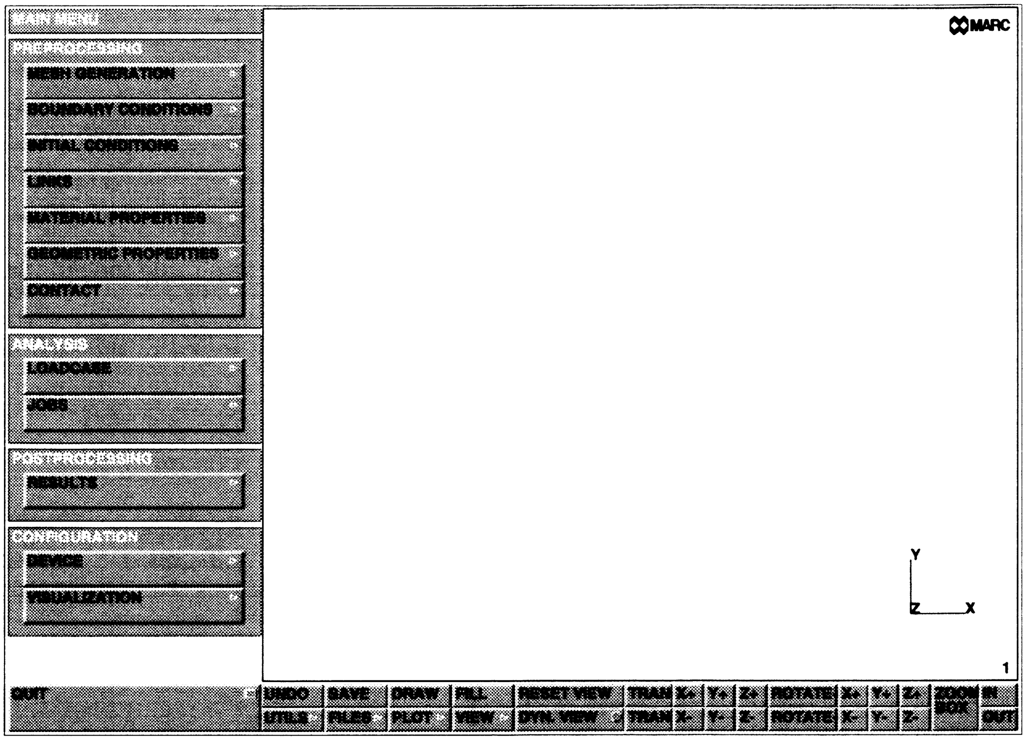


Figure 4.1 Initial Mentat II Display

The PROCEDURE command has two modes of operation:

1- Creates a procedure file (record mode)
2- Executes a procedure file (playback mode).

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

MAIN	*(main menu)*
UTILS	*(located at the bottom of the static menu)*
EXECUTE	*(located on the PROCEDURE panel)*
`path/filename`	

Remember to use <**ML**> to click on a button. After you click on the EXECUTE button, enter the file name of the procedure file you want to execute. Once you have done this, watch the program as it executes the information stored in the procedure file.

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 <**MR**> with <↑> 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 *(in the static menu)*
  RESET PROGRAM

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.

4.3 Stopping the Mentat II Program

Always make sure to save your work before you stop the Mentat II program. Use the SAVE button to write a copy of the database in Mentat II 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 II 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.

4.4 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 II user interface.

The structure you are going to model has the dimensions shown in Figure 4.2.

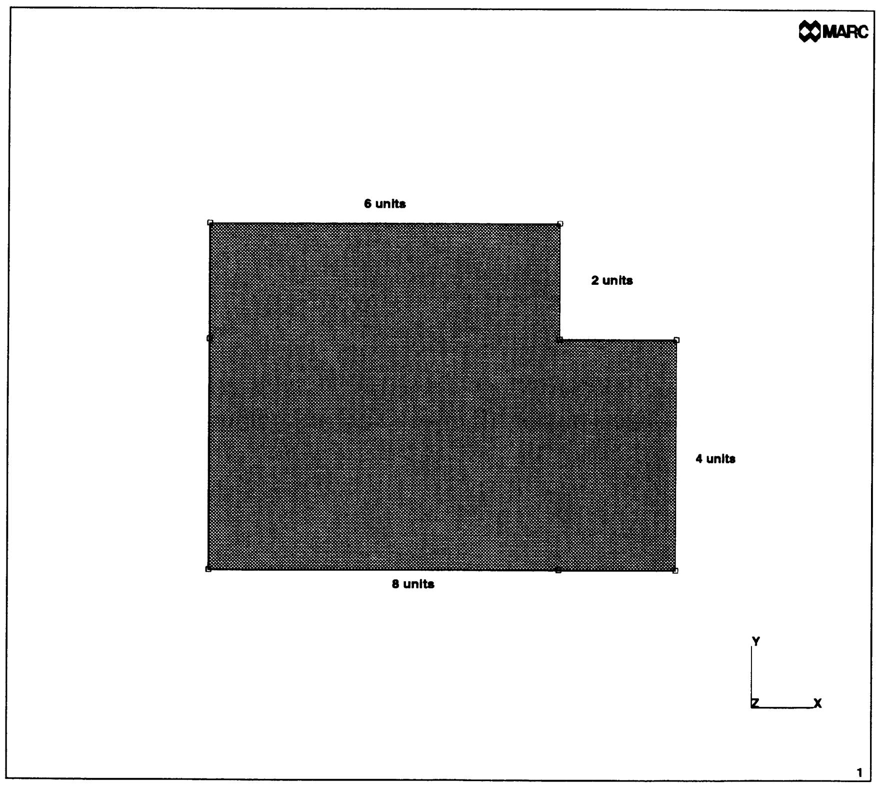


Figure 4.2 Dimensions of Structure to be Modeled

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.

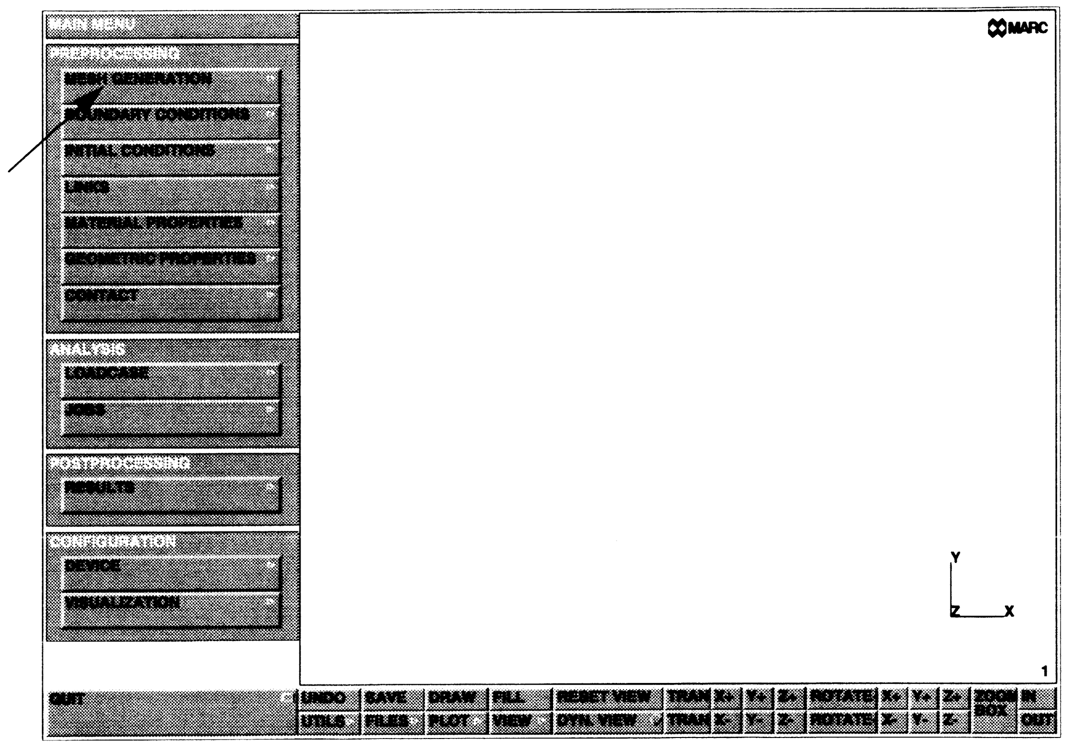


Figure 4.3 Accessing the Mesh Generation Menu

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.

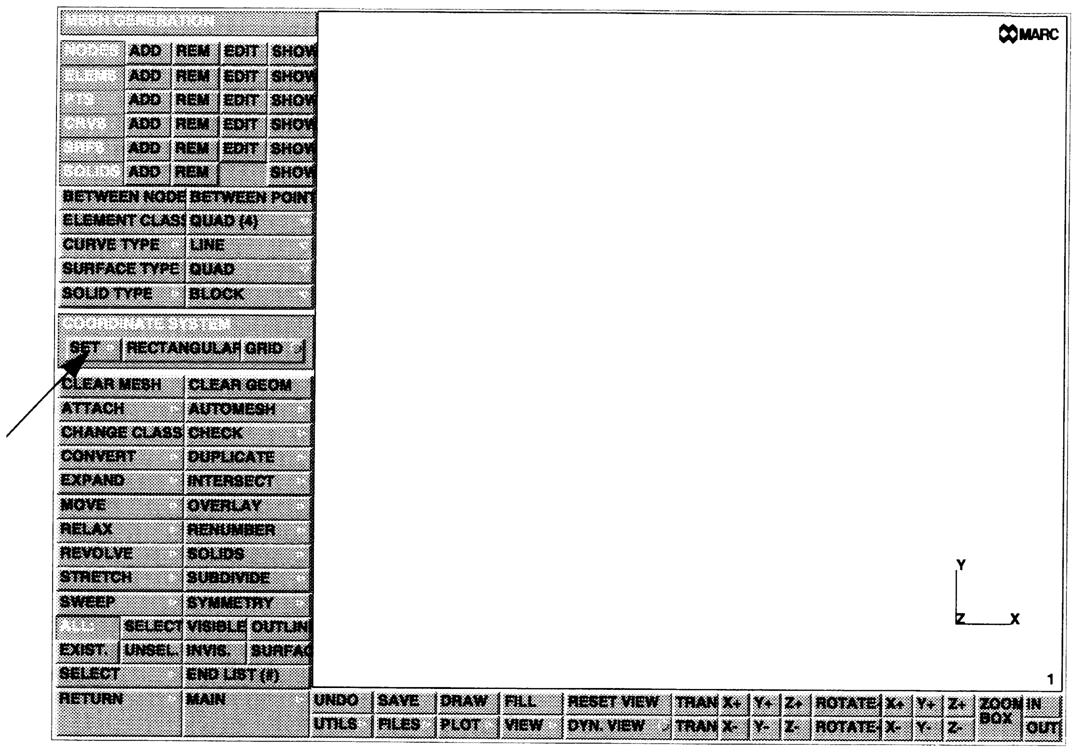


Figure 4.4 Mesh Generation Menu

The object you want to model has maximum dimensions 8 x 6 units. Click on the SIZE button and use the keyboard to enter `10 10` to set the grid size.

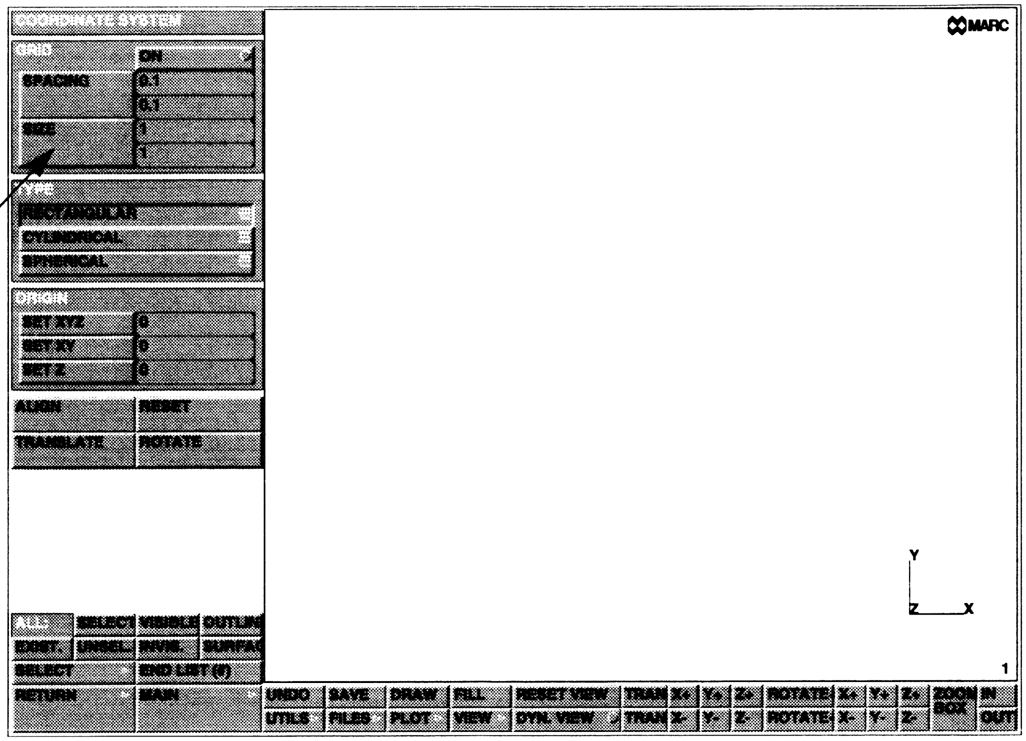


Figure 4.5 Coordinate System Menu

Figure 4.1 shows you the updated coordinate system menu with the numeral 10 appearing in the flat fields next to the SIZE 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 SPACING button and type in `1  1`. The program updates the menu accordingly as is shown in Figure 4.7.

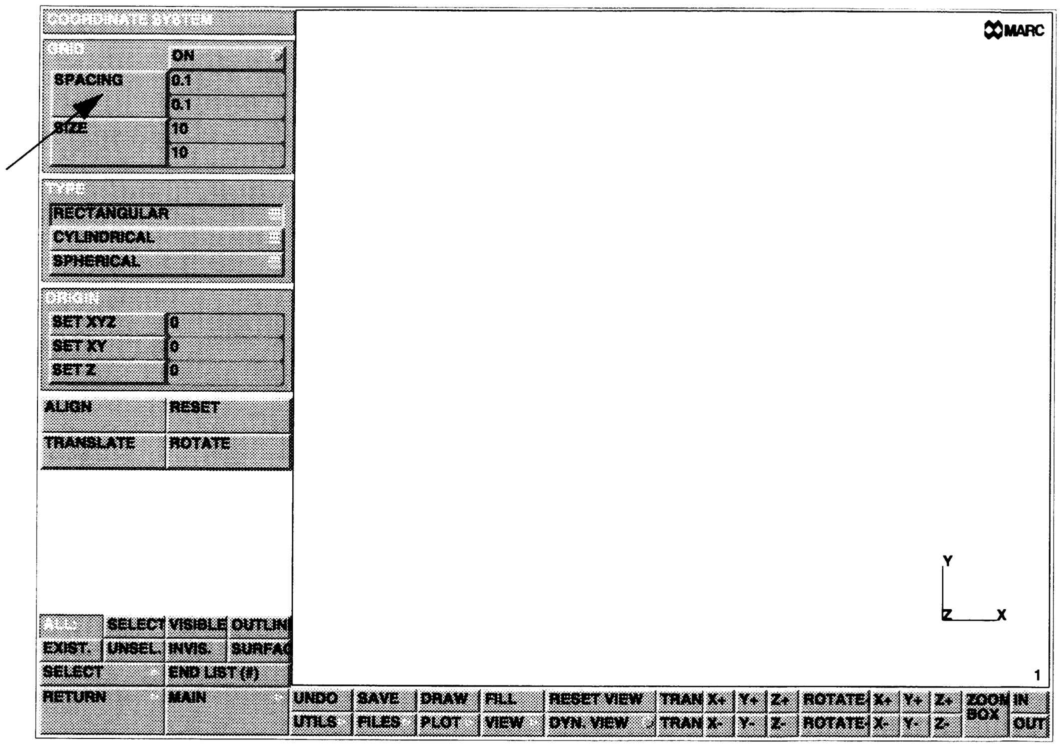


Figure 4.6 Setting the Grid Spacing

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

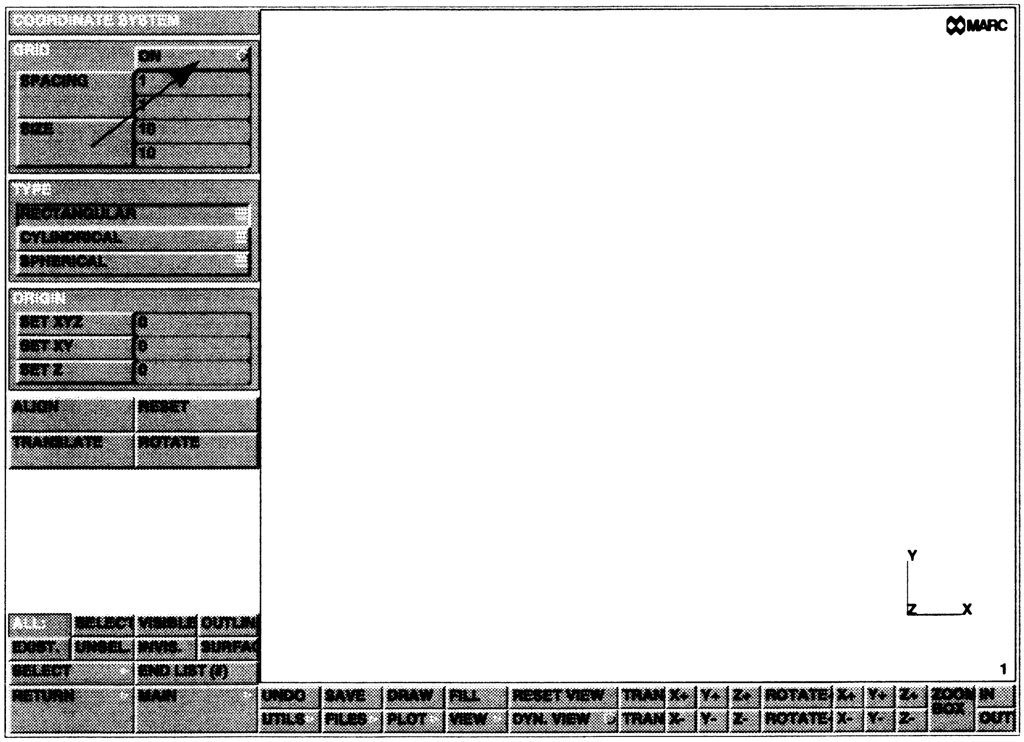


Figure 4.7 Activating the Grid Display

Figure 4.8 depicts the graphics area with the input grid displayed.

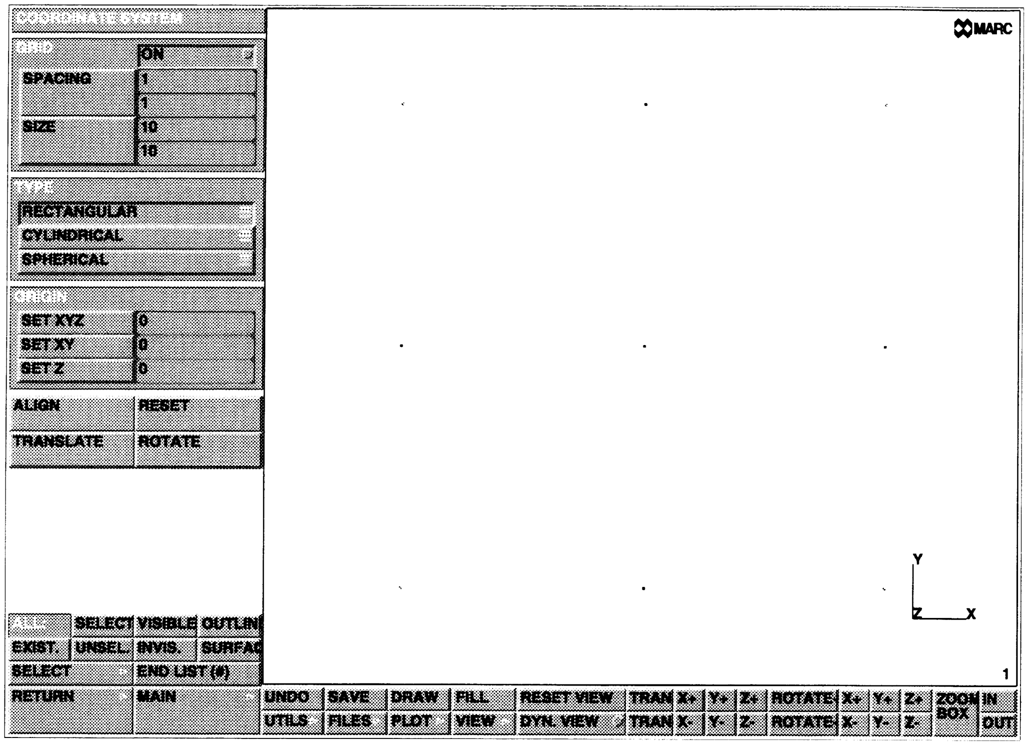


Figure 4.8 Input Grid Activated

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.

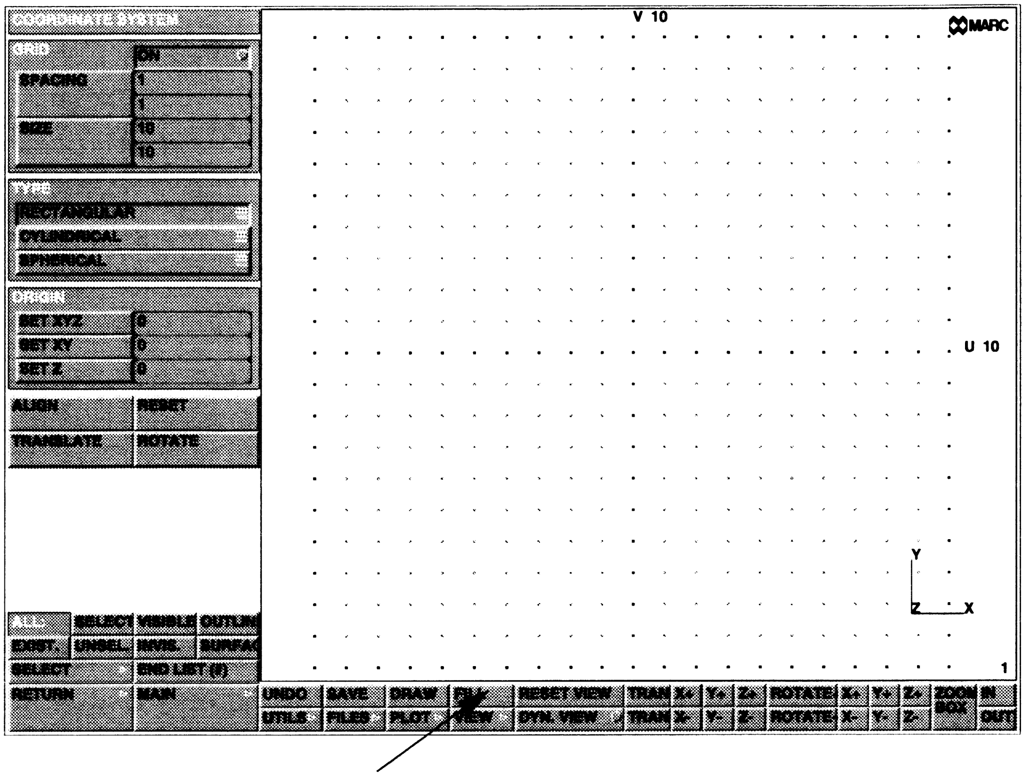


Figure 4.9 Scaling the picture

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 <**MR**> with the <↑> over the menu area.

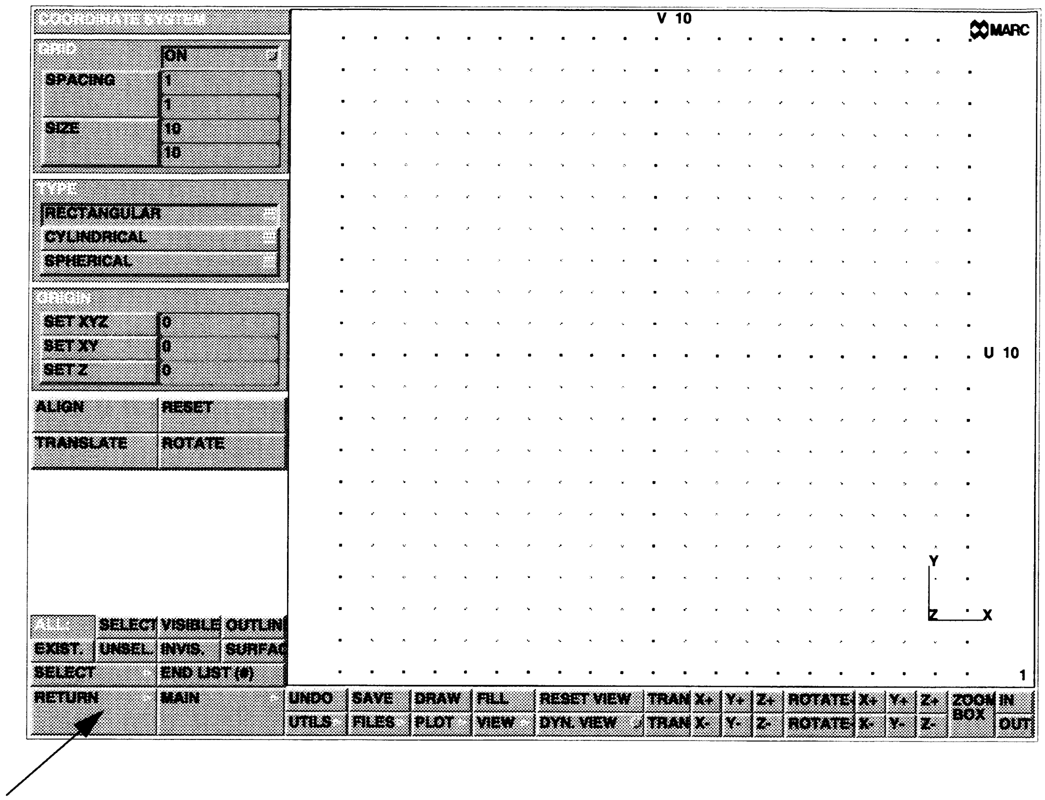


Figure 4.10 Returning to the Mesh Generation Menu

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
      SIZE
        `10 10`
      SPACING
        `1 1`
      grid ON *(on)*
      FILL
      RETURN

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.

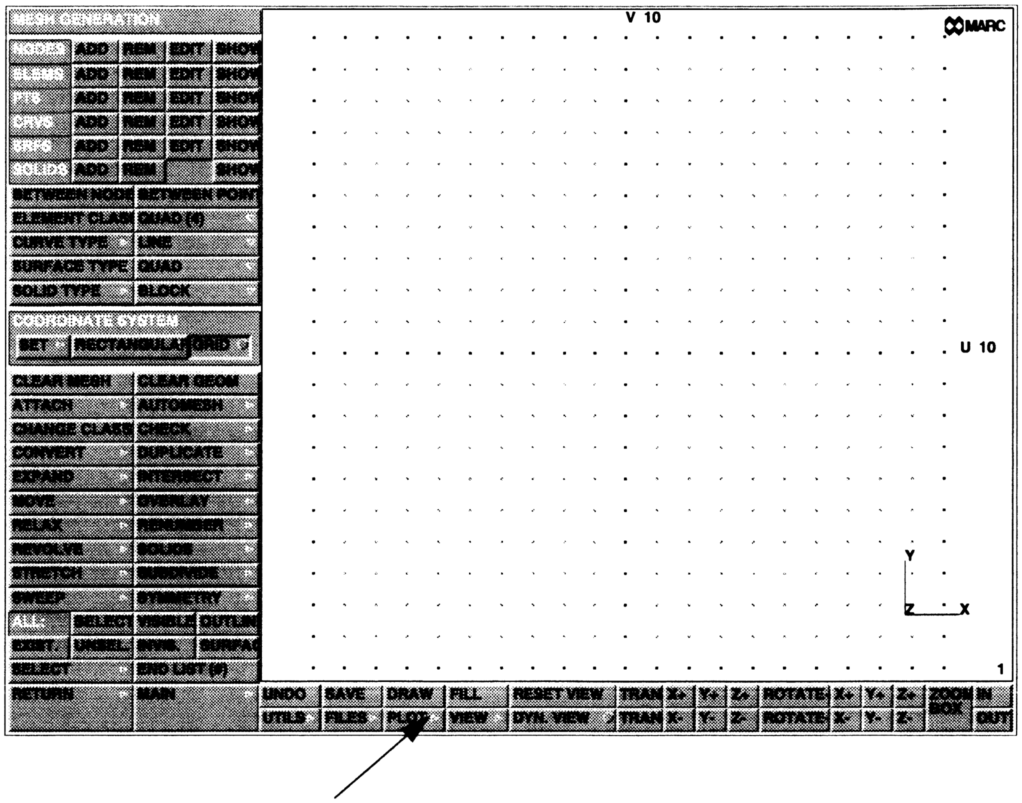


Figure 4.11 Accessing the Plot Menu

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.

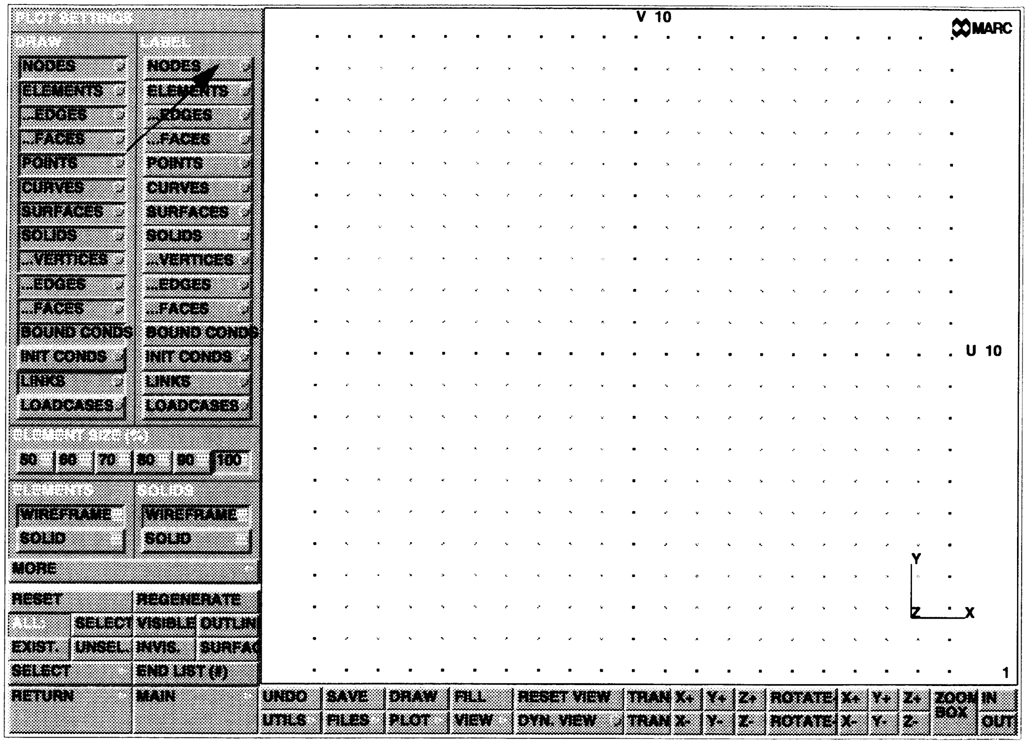


Figure 4.12 Activating the Node Labeling

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.

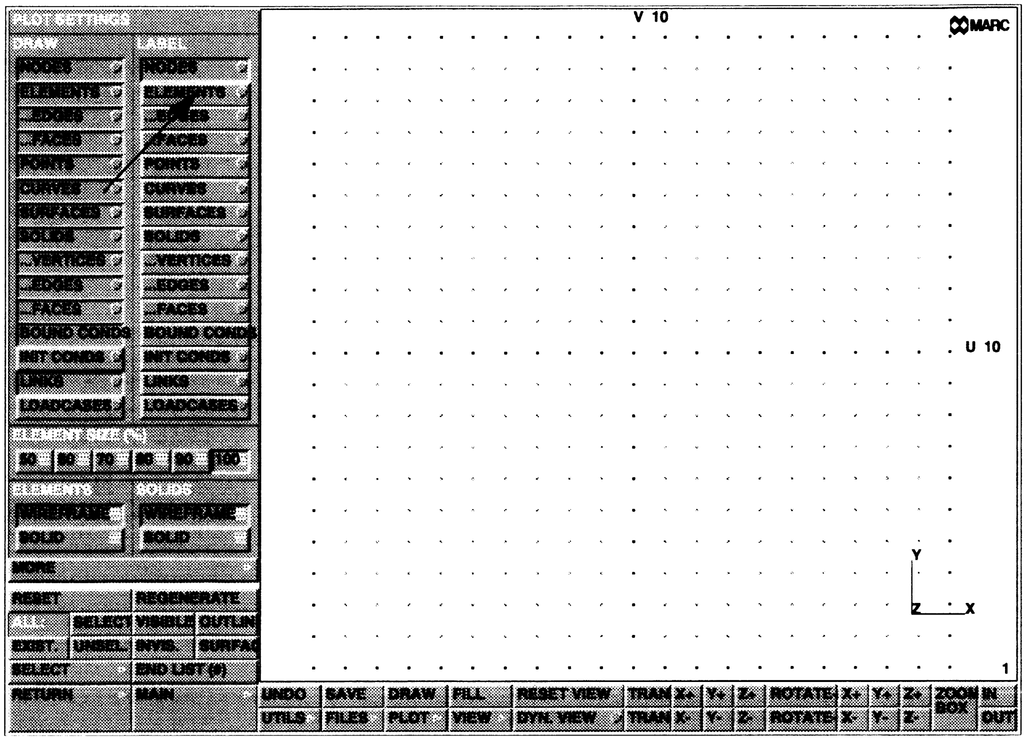


Figure 4.13 Activating the Element Labeling

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 <**MR**> with the <↑> over the menu area.

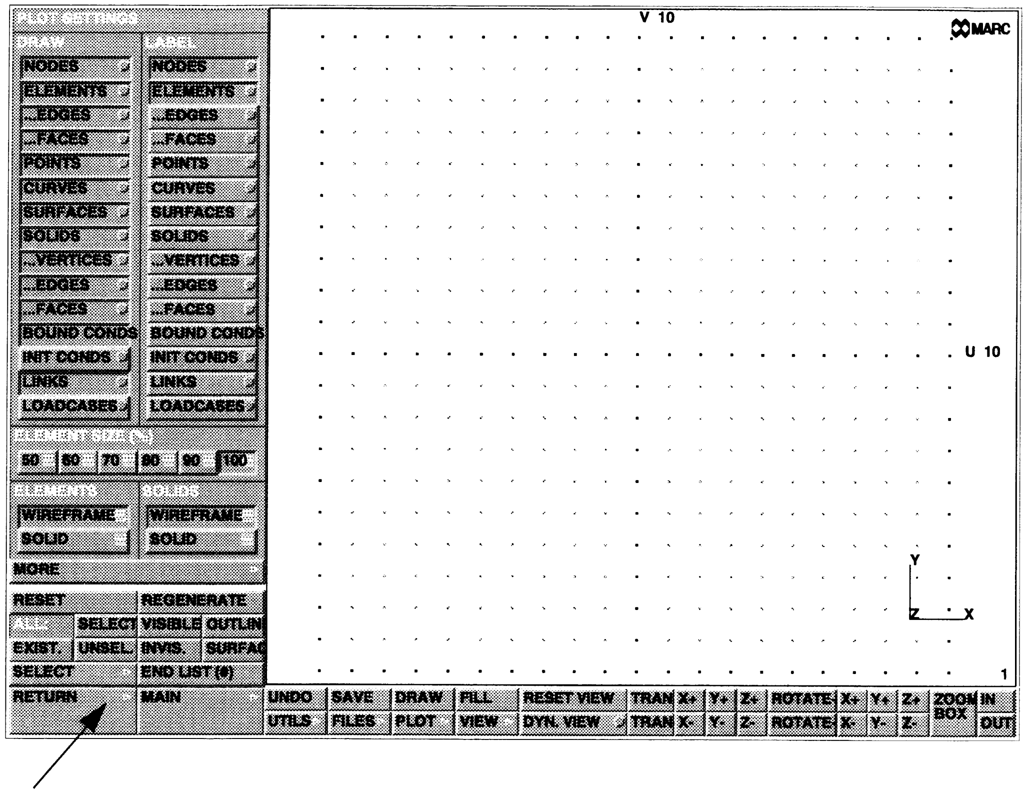


Figure 4.14 Returning to the Mesh Generation Menu

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.

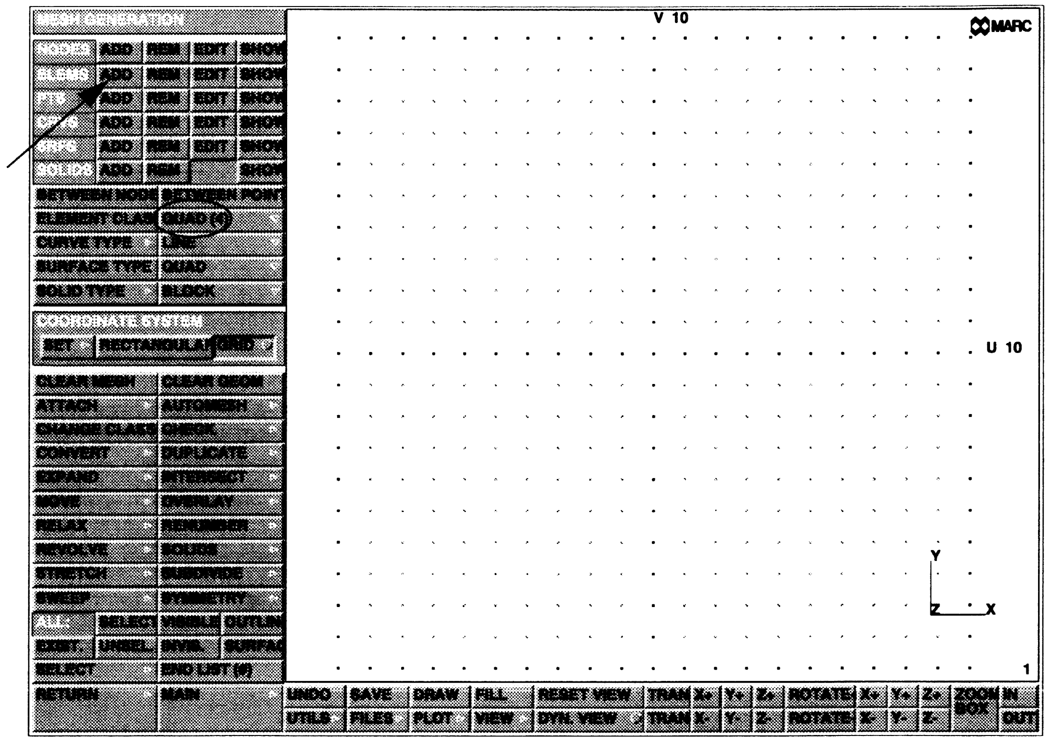


Figure 4.15 Adding an Element

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.

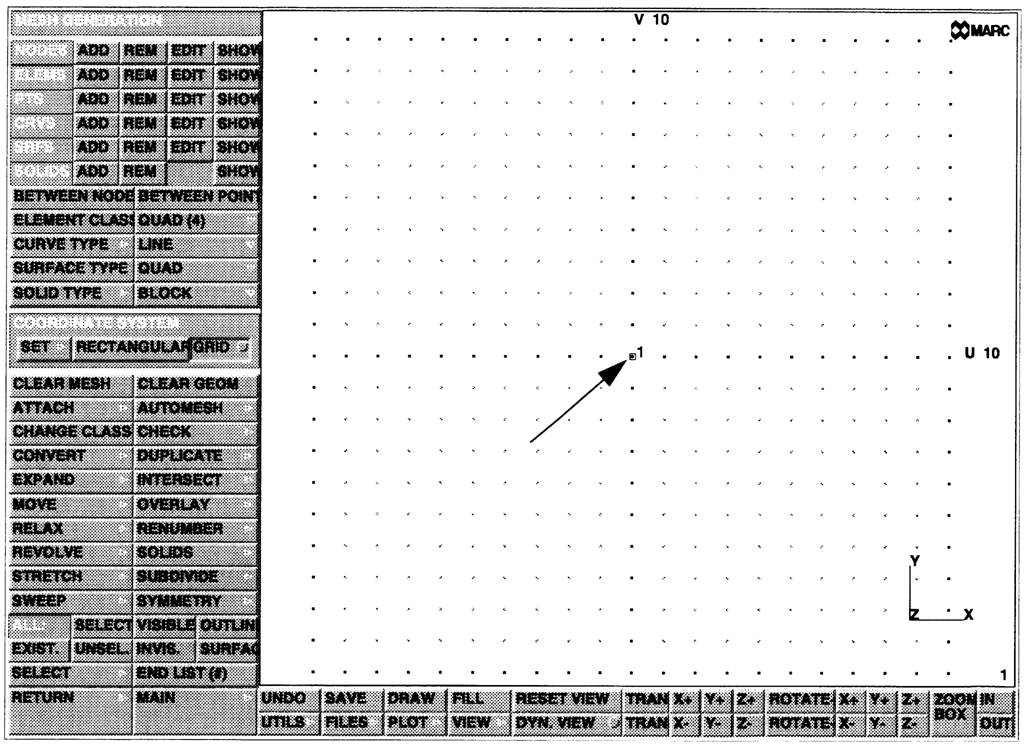


Figure 4.16 First Node of Element 1

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

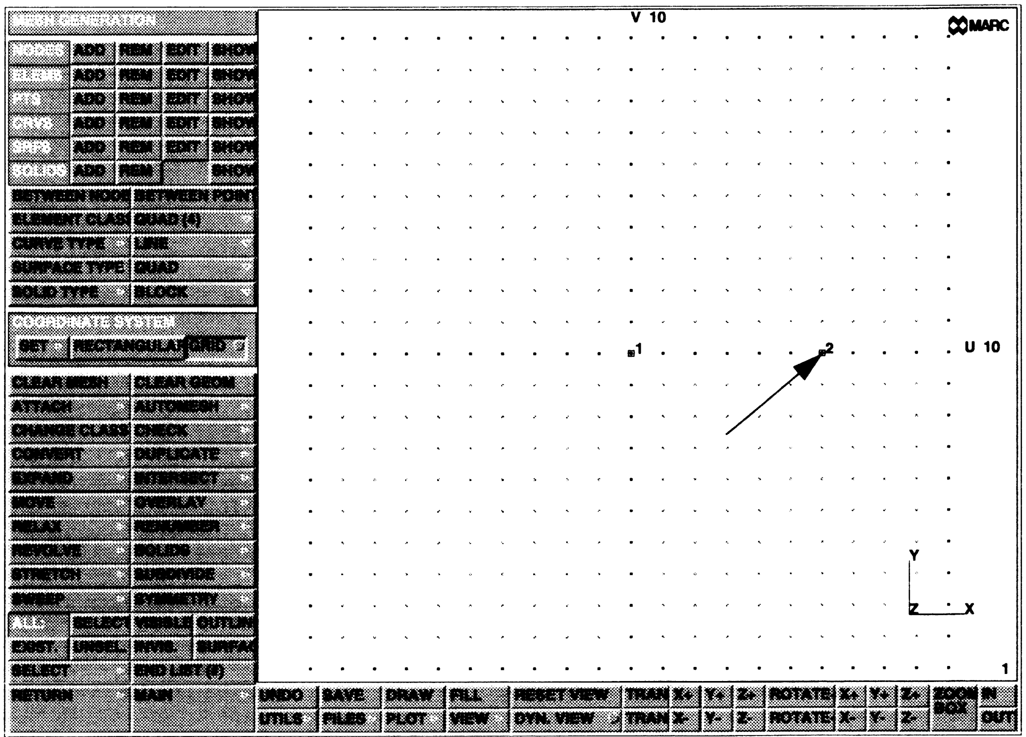


Figure 4.17 Second Node of Element 1

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

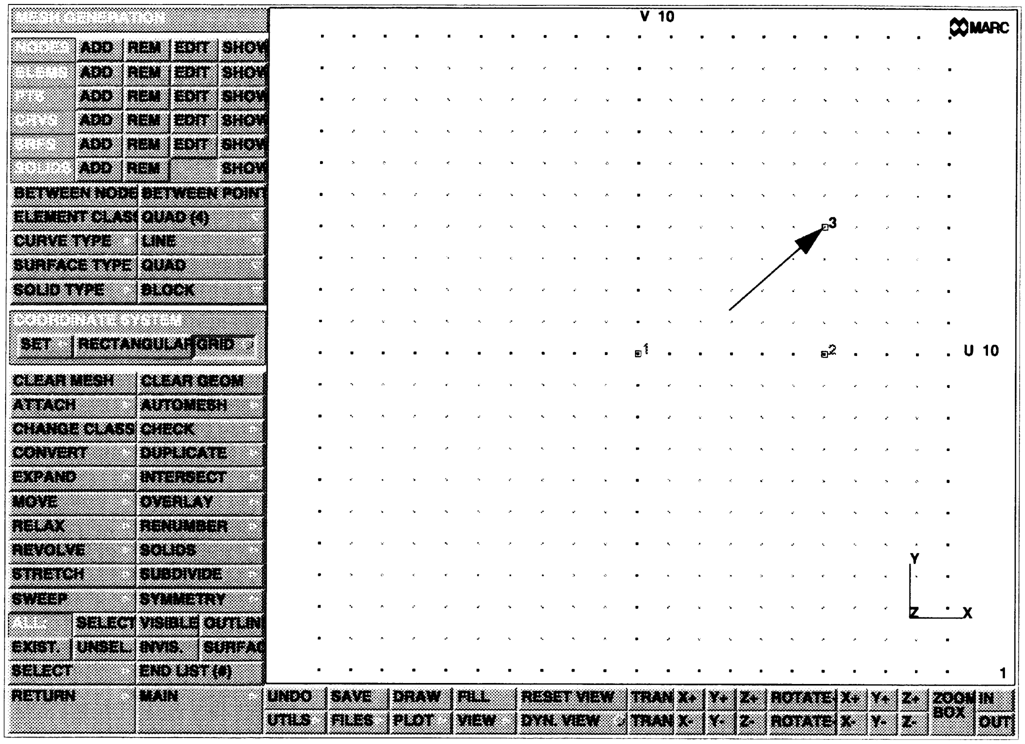


Figure 4.18 Third Node of Element 1

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 2-D, the face is the element itself. If you need to pick this element, click the <↑> in the center of the element.

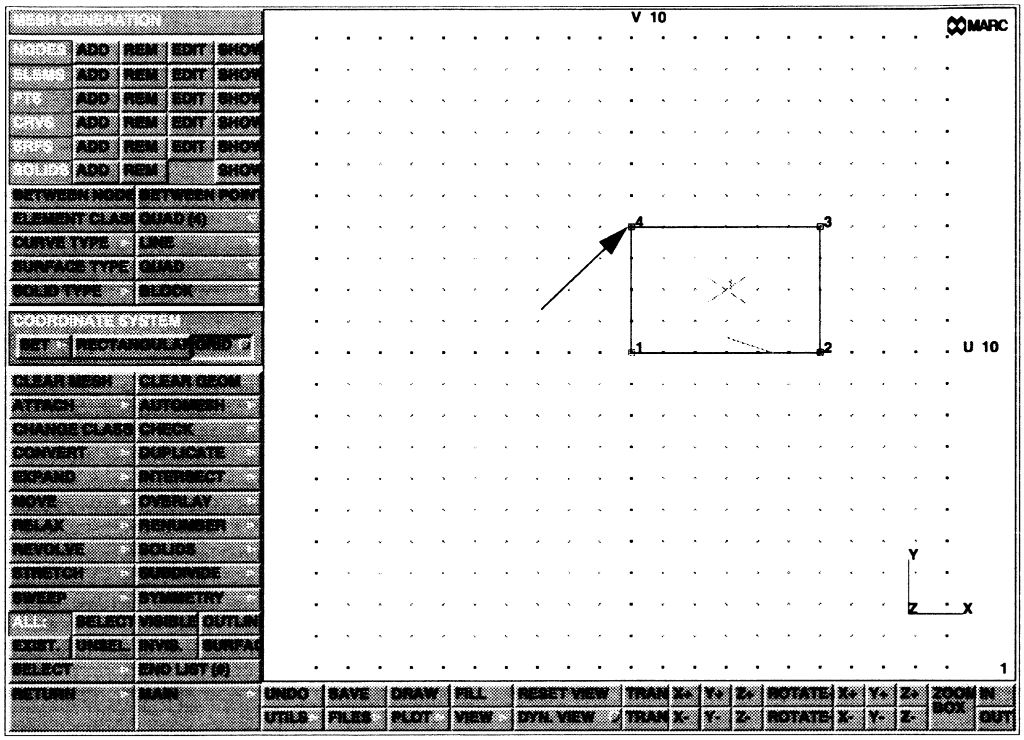


Figure 4.19 Element 1 Completed

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, Section 2.2, 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.

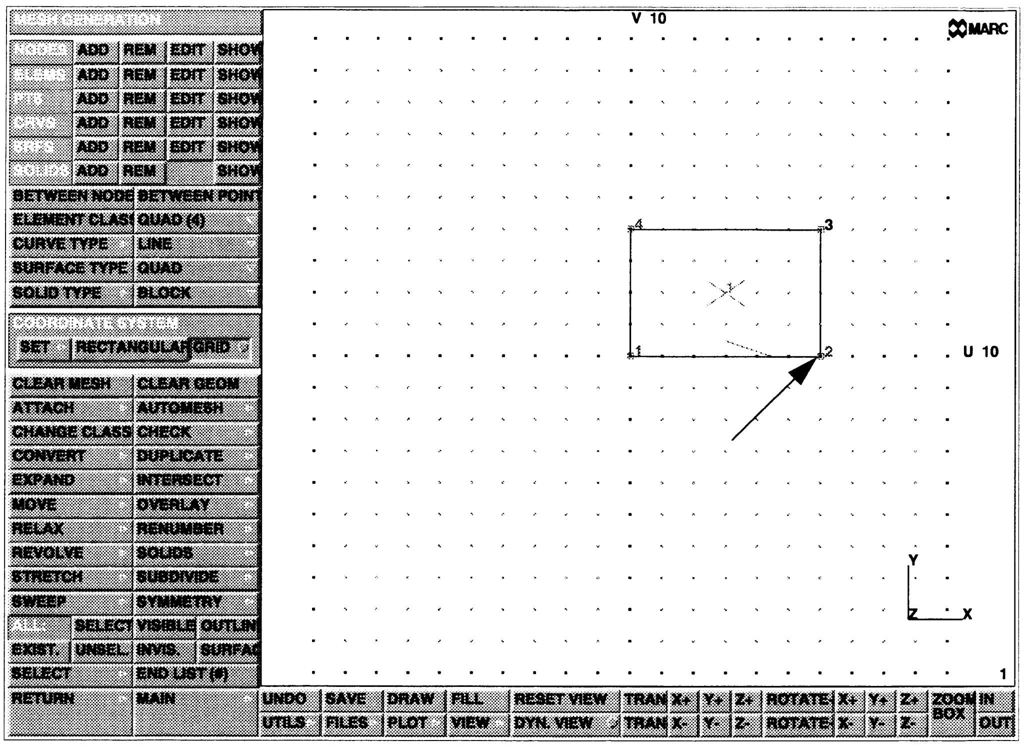


Figure 4.20 First Node of Element 2

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.

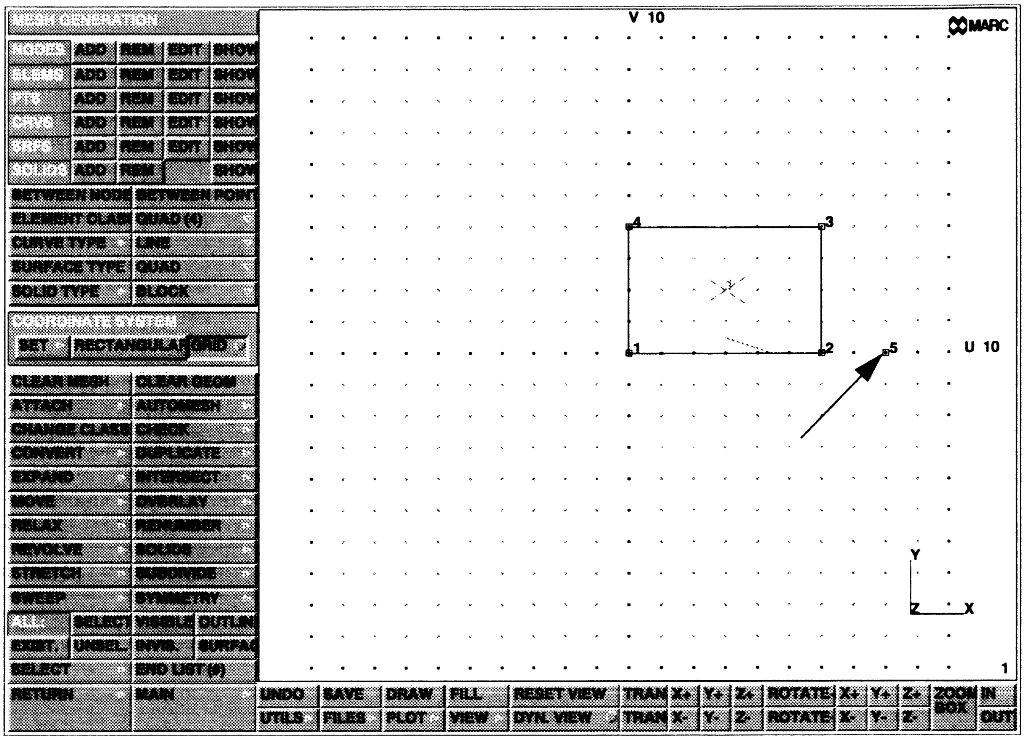


Figure 4.21 Second Node of Element 2

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

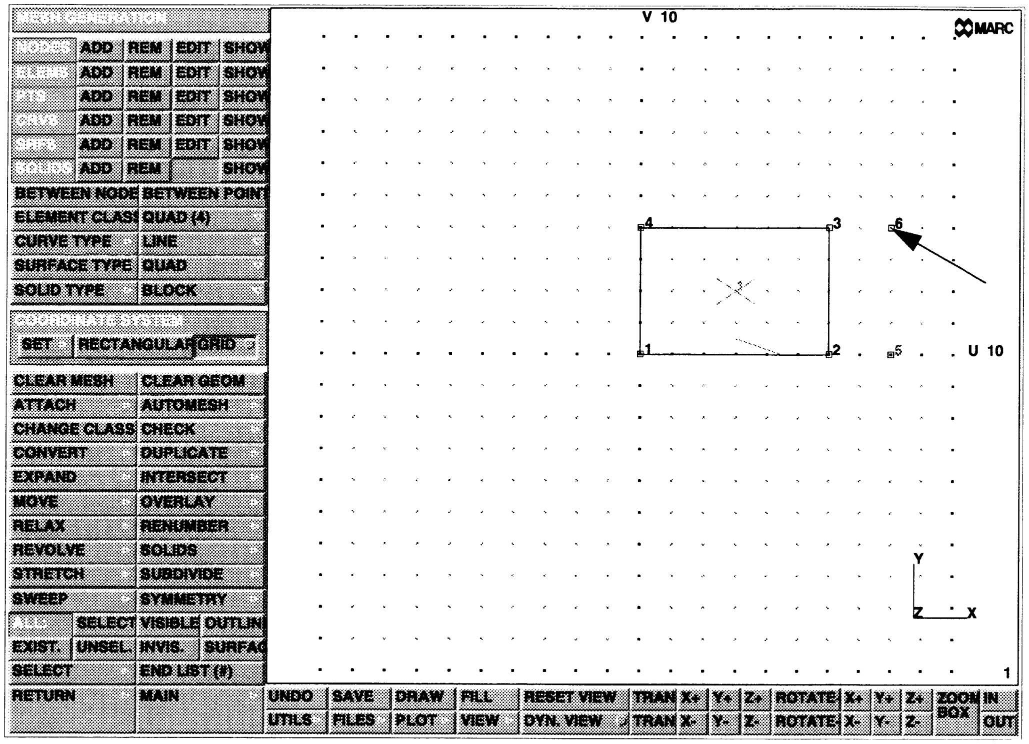


Figure 4.22 Third Node of Element 2

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.

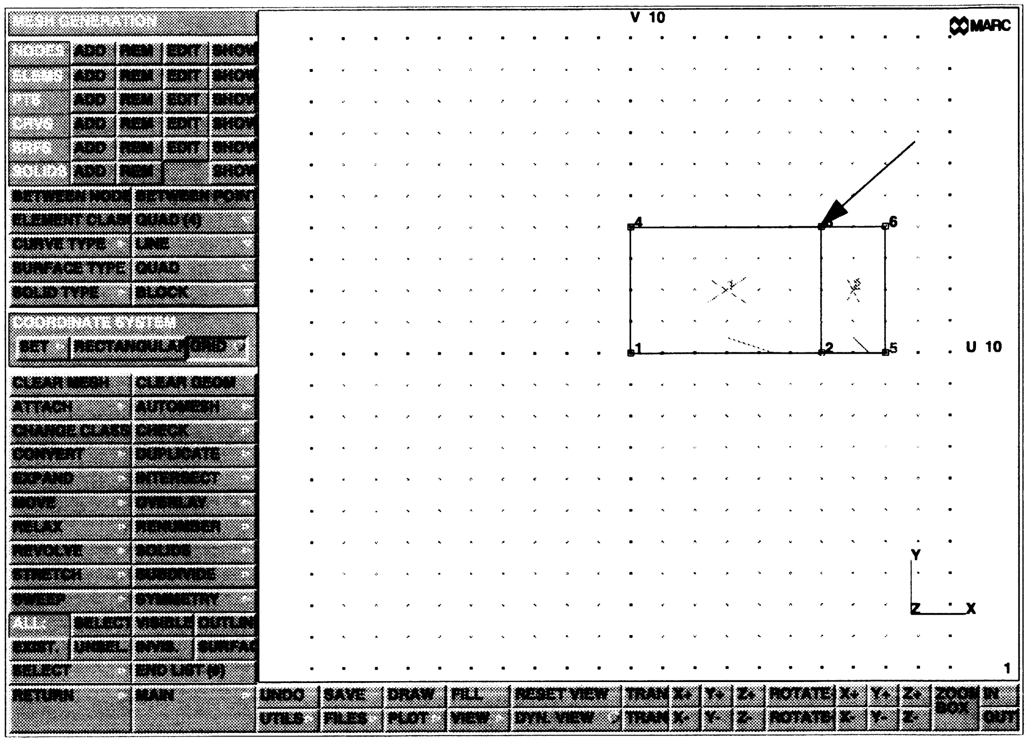


Figure 4.23 Element 2 Completed

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.

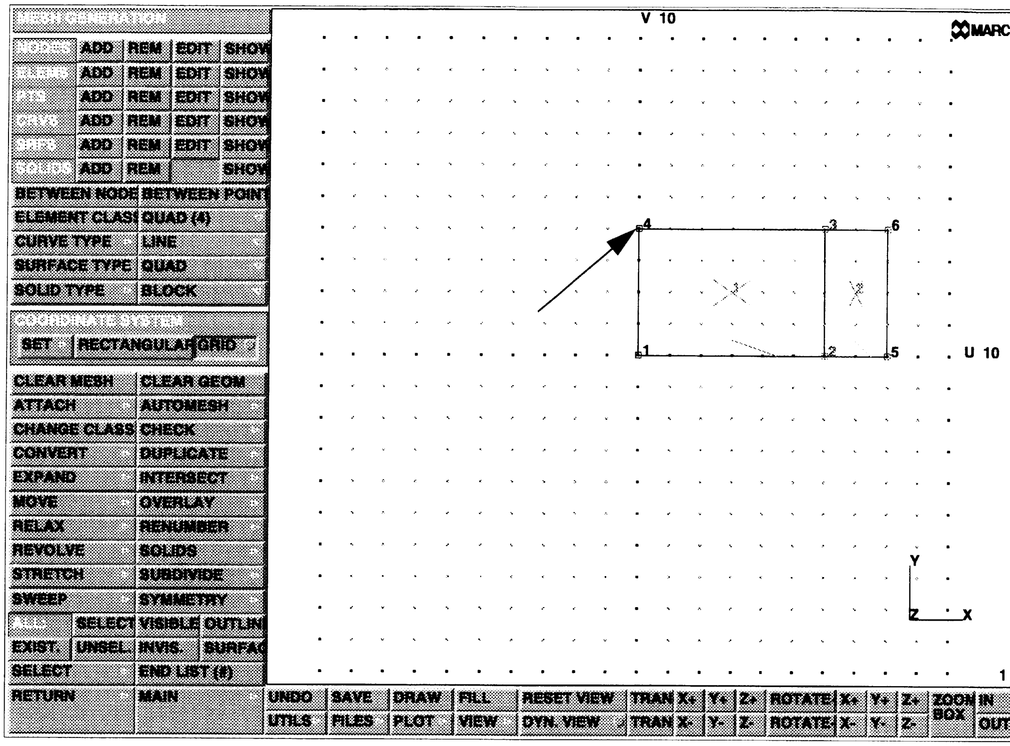


Figure 4.24 First Node of Element 3

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.

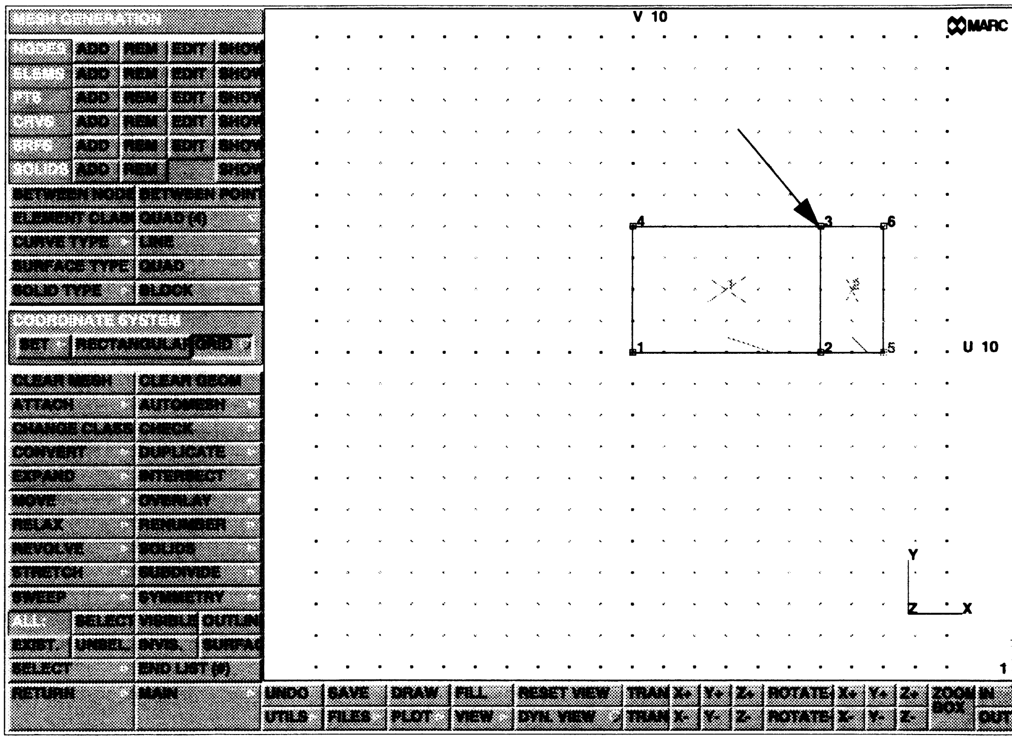


Figure 4.25 Second Node of Element 3

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.

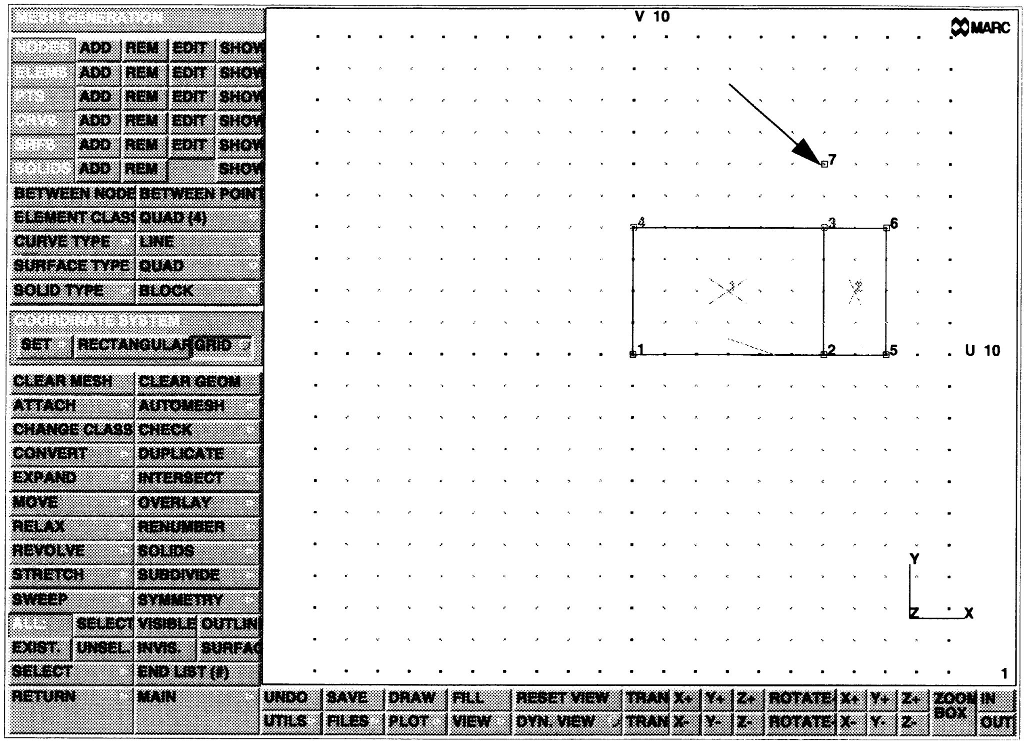


Figure 4.26 Third Node of Element 3

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

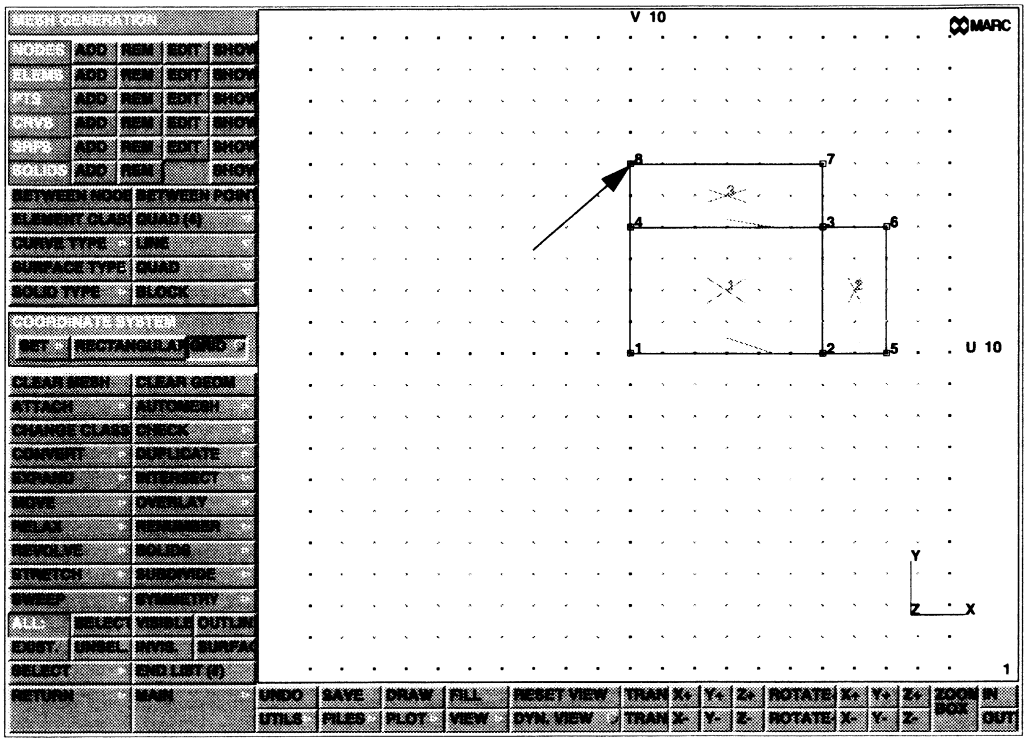


Figure 4.27 Element 3 Completed

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

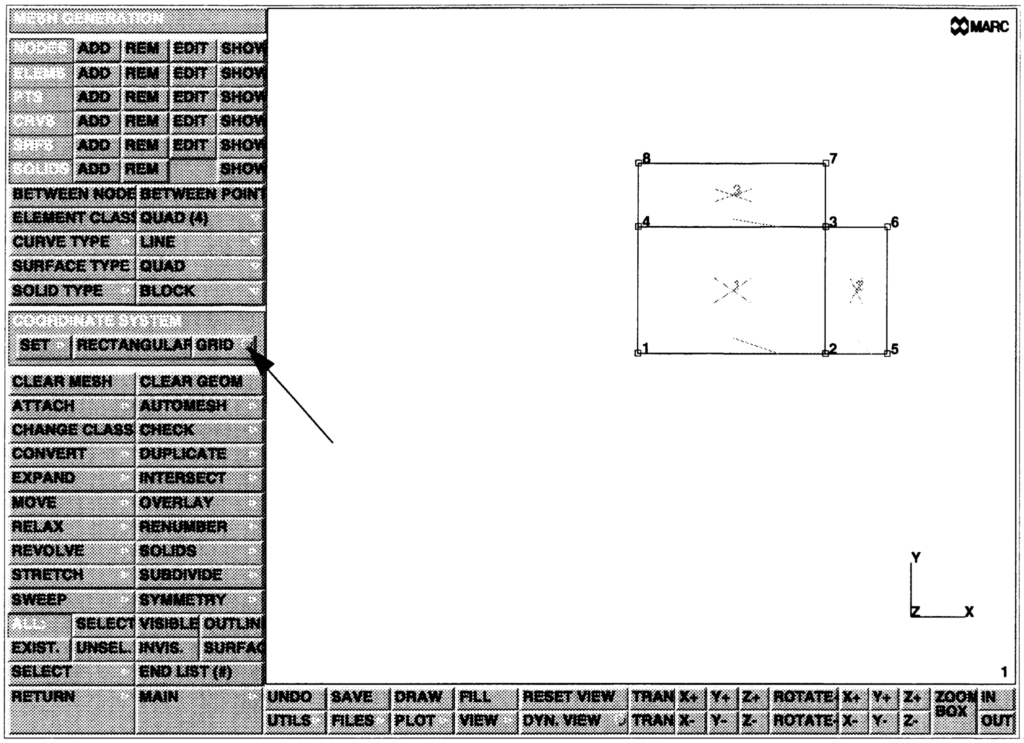


Figure 4.28 Turning Off the Input Grid

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.

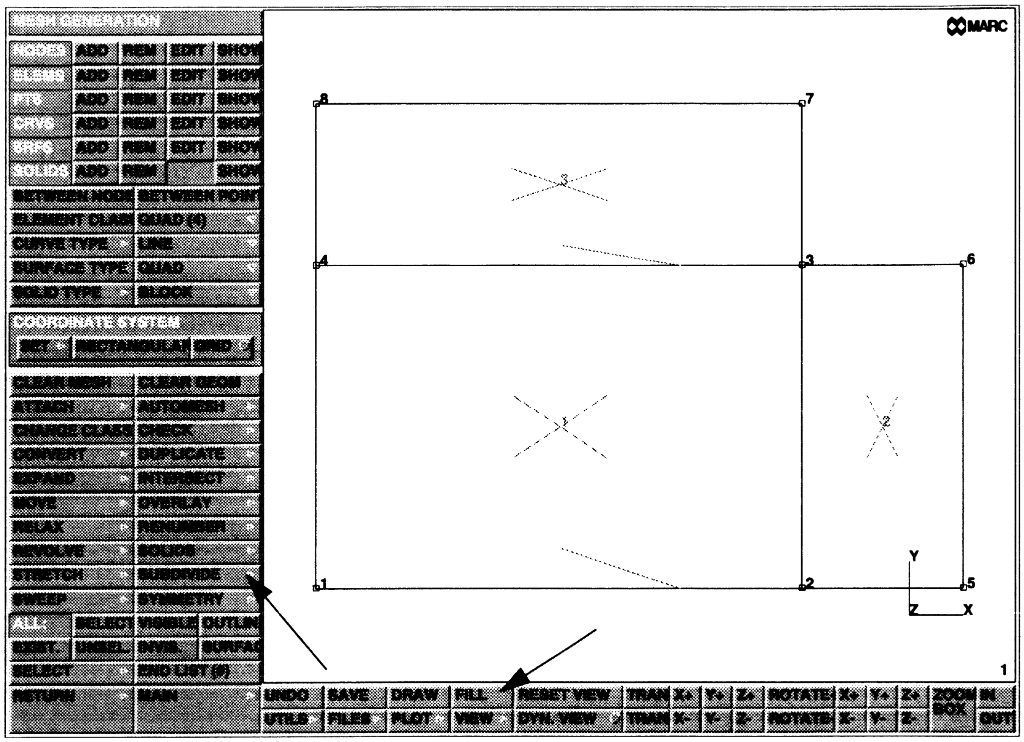


Figure 4.29 Picture Scaled to Fill Screen; Locating Subdivide

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`.

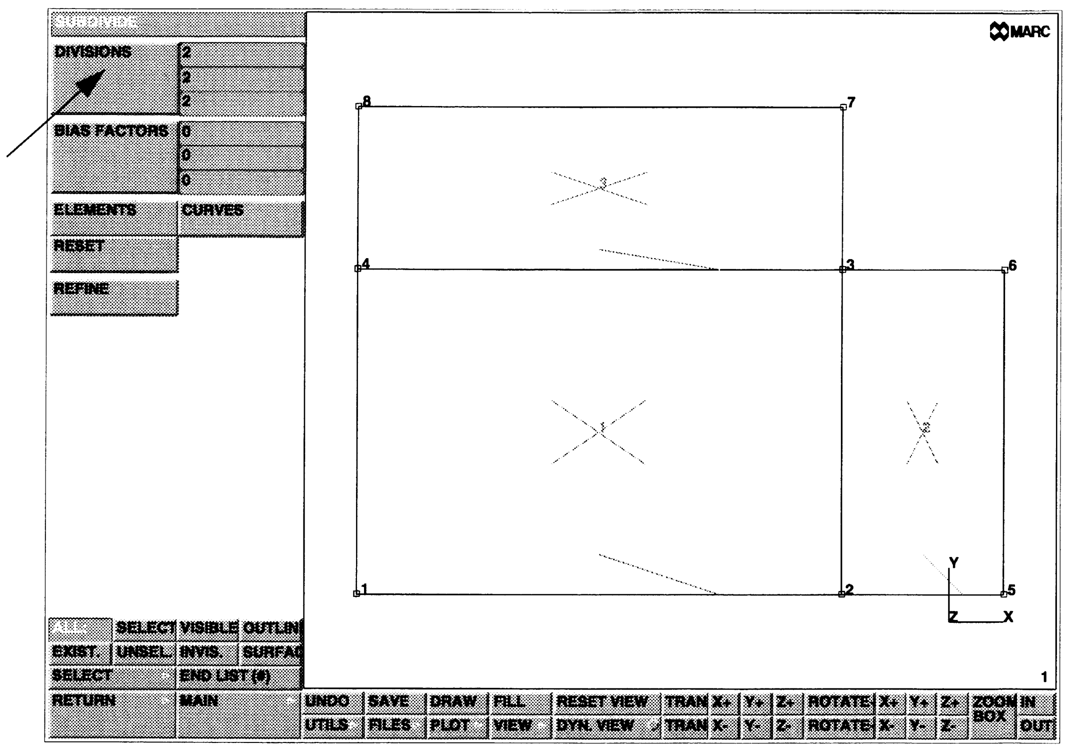


Figure 4.30 Setting the Subdivide Parameters

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:

```
Enter subdivide element list:
```

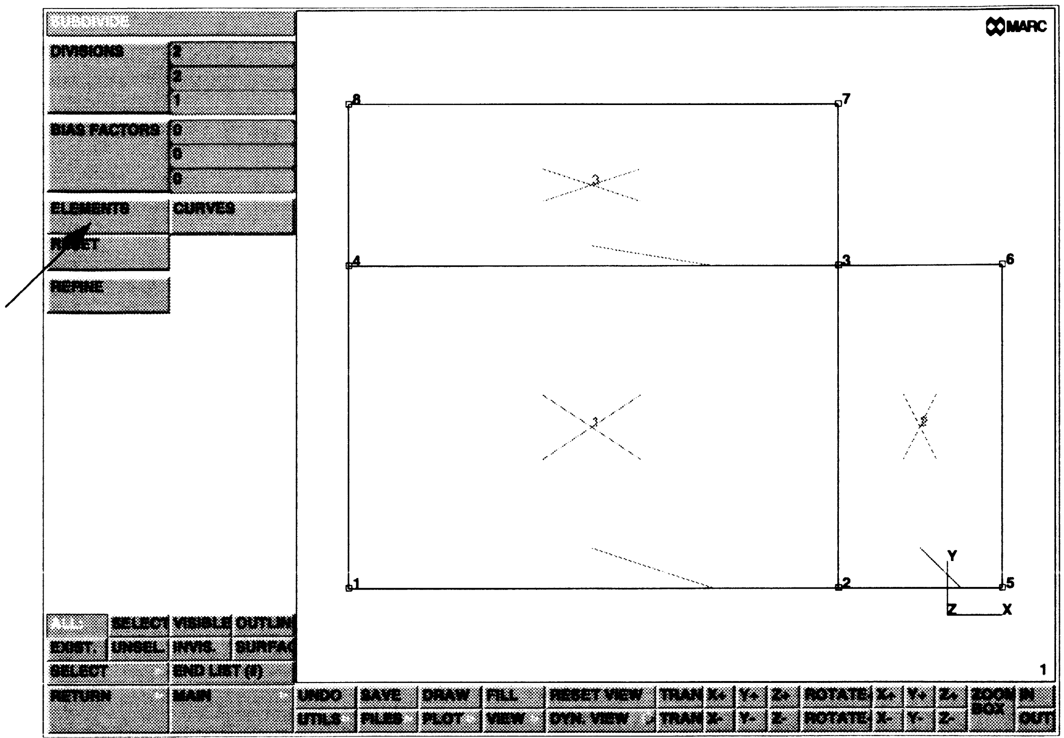


Figure 4.31 Activating the Subdivide Processor

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 <**MR**> with <↑> 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.

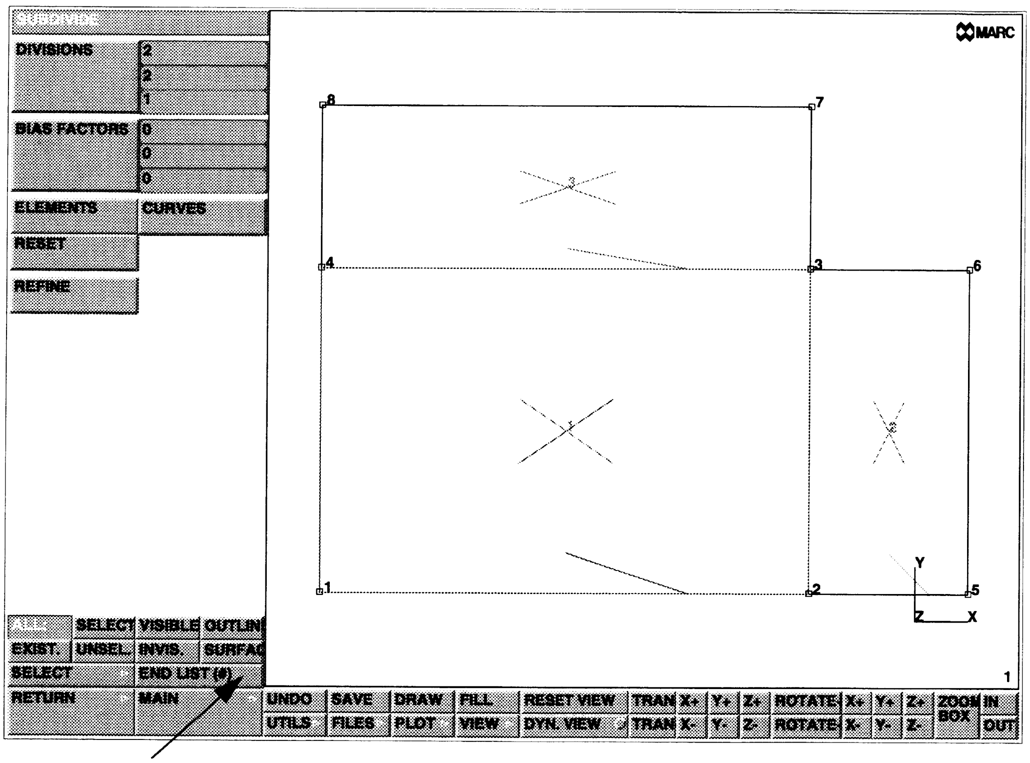


Figure 4.32 Indicating End of List

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.

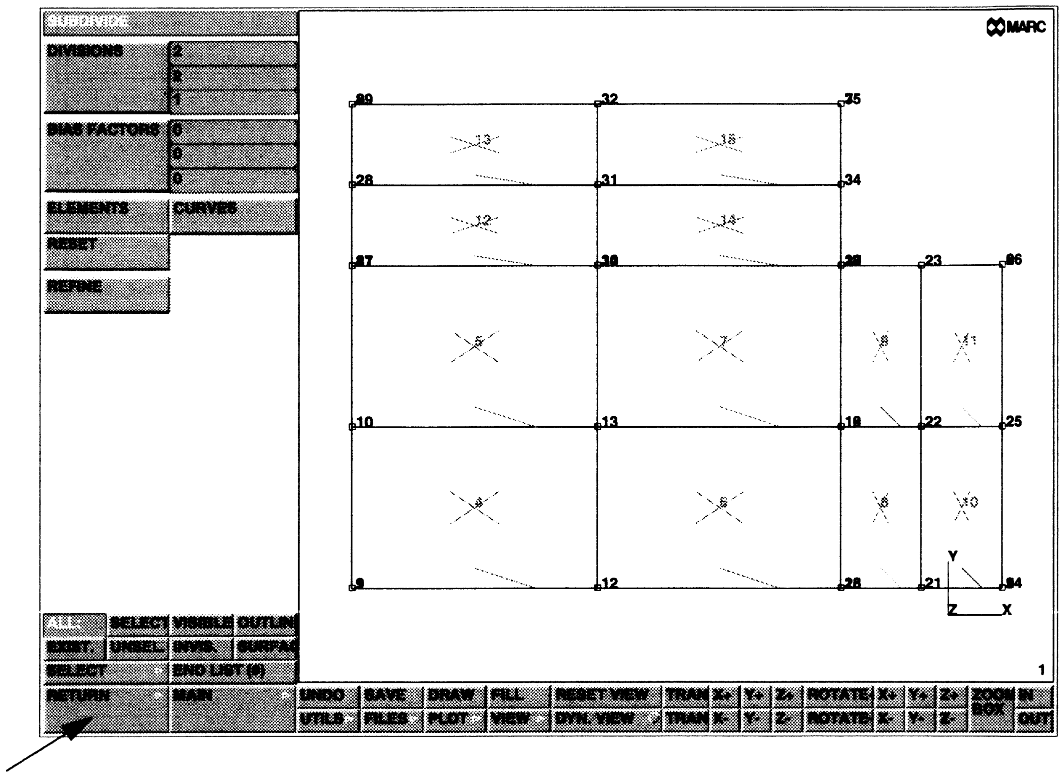


Figure 4.33 Returning to the Mesh Generation Menu

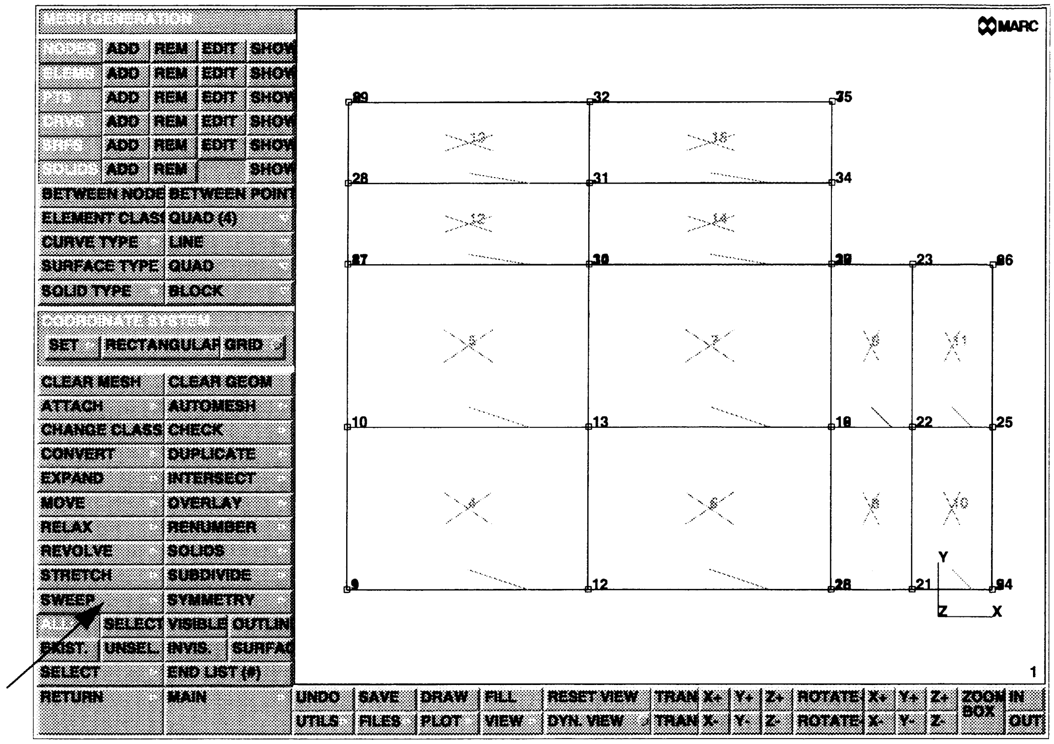


Figure 4.34 Accessing the Sweep Processor

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.

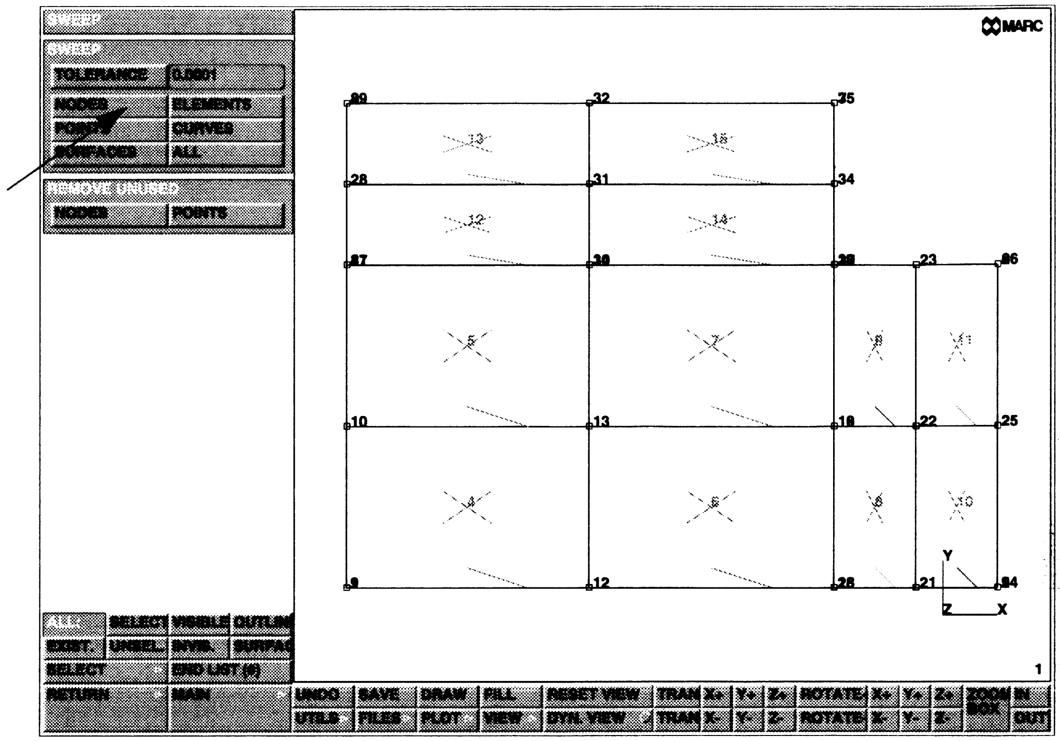


Figure 4.35 Activating the Sweep Processor

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.

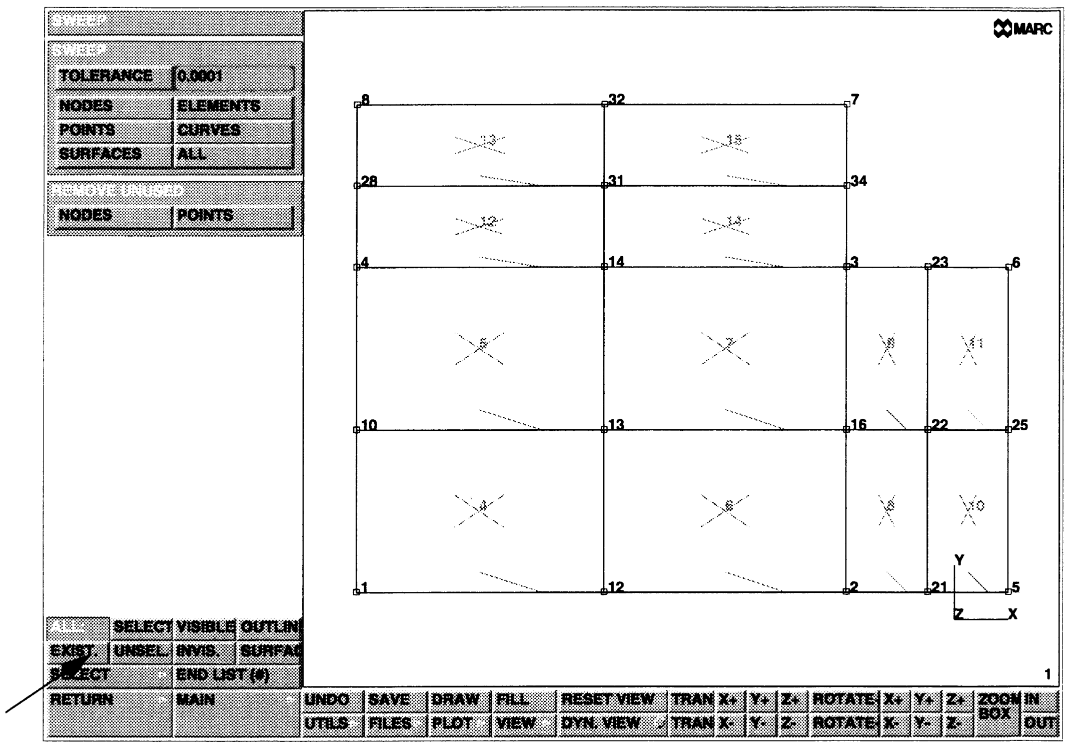


Figure 4.36 Identifying the Nodes to Sweep

Save the mesh by clicking on the SAVE button. The mesh is saved in Mentat II 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 II. 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.

Chapter 5: 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
- how to select 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

5.1 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 N/mm^2 and Poisson's ratio $\nu = 0.3$. A tensile load with magnitude p = 10 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.

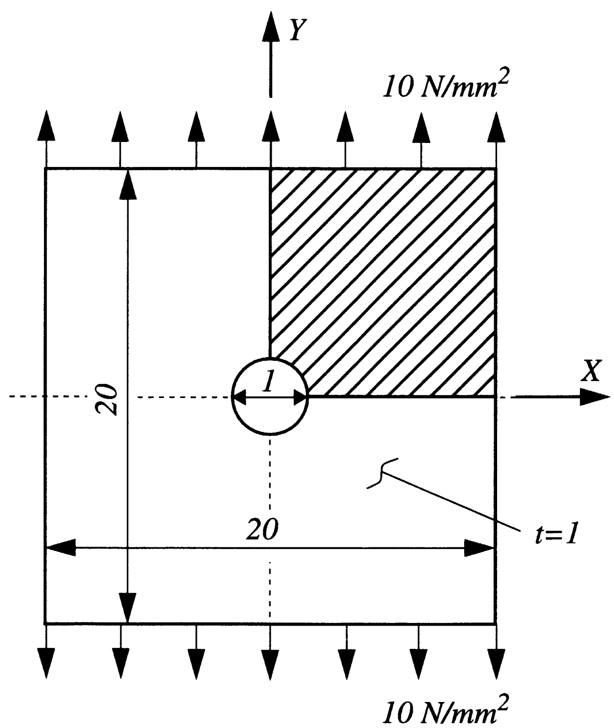


Figure 5.1 Plate with a Hole Subjected to Tension

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.

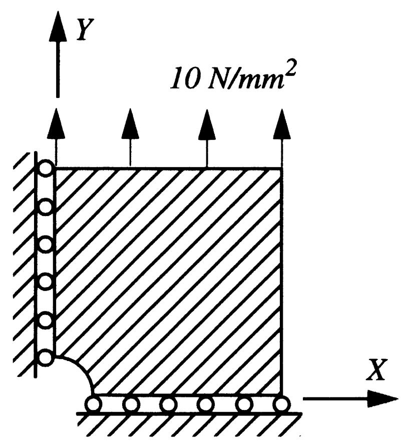


Figure 5.2 Quarter of the Plate with Symmetry Conditions and Tensile Load

5.1.1 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

5.2 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
            SPACING
                1  1
            SIZE
                10  10
            grid ON                                    (on)
            RETURN
        FILL
```

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          (Pick the following points from the grid)
            0  0  0                             (center point)
            1  0  0                           (starting point)
            0  1  0                             (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 *(Pick the following points from the grid)*
`point(10,0,0)`
`point(10,10,0)`
`point(0,10,0)`
END LIST (#)

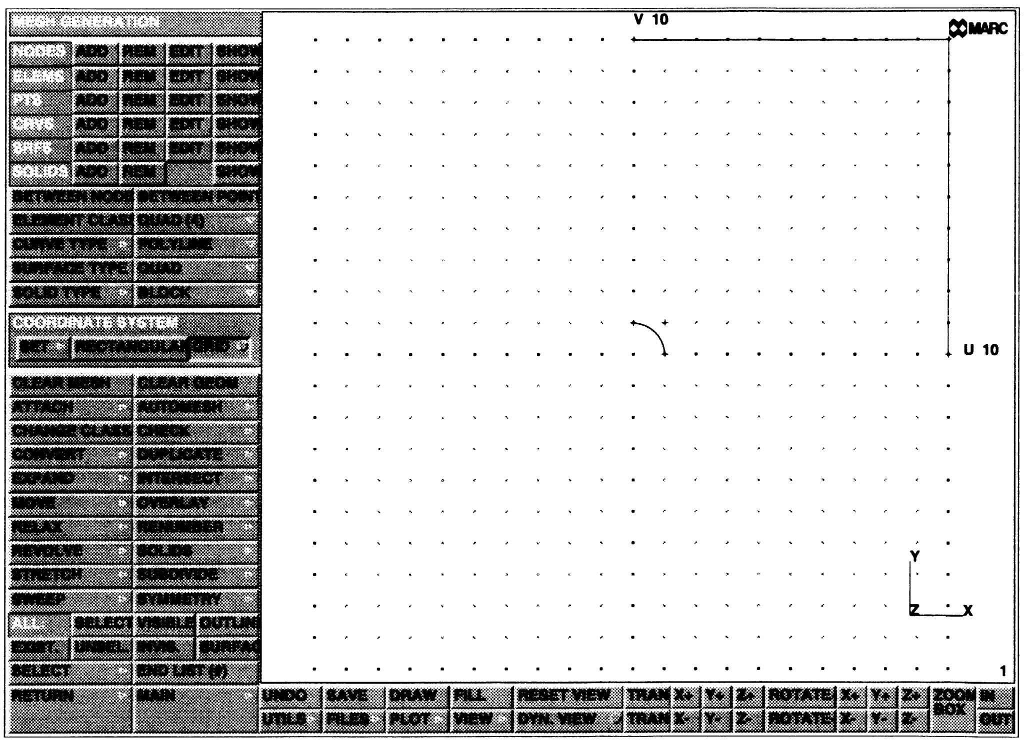


Figure 5.3 Boundary Curves of Quarter of the Plate

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 *(pick the arc)*
2 *(pick the polyline)*

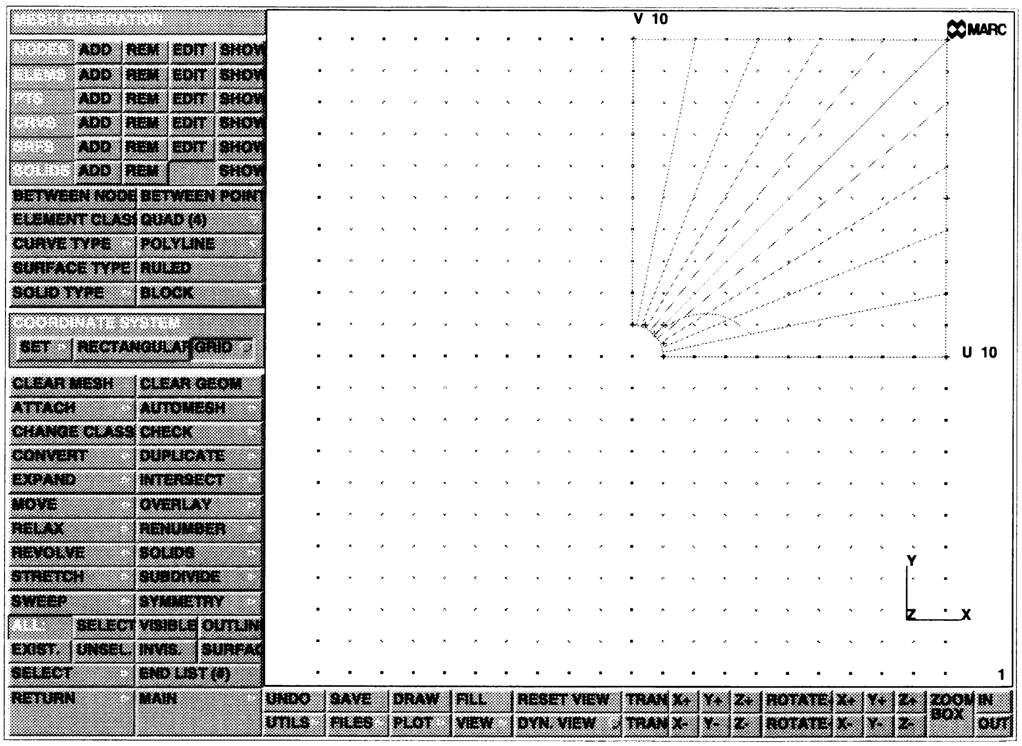


Figure 5.4 Surface Definition

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.

MAIN
MESH GENERATION
CONVERT
BIAS FACTORS
`0 -0.5`
SURFACES TO ELEMENTS
`1` *(pick the surface)*
END LIST (#)
RETURN
GRID *(off)*
FILL

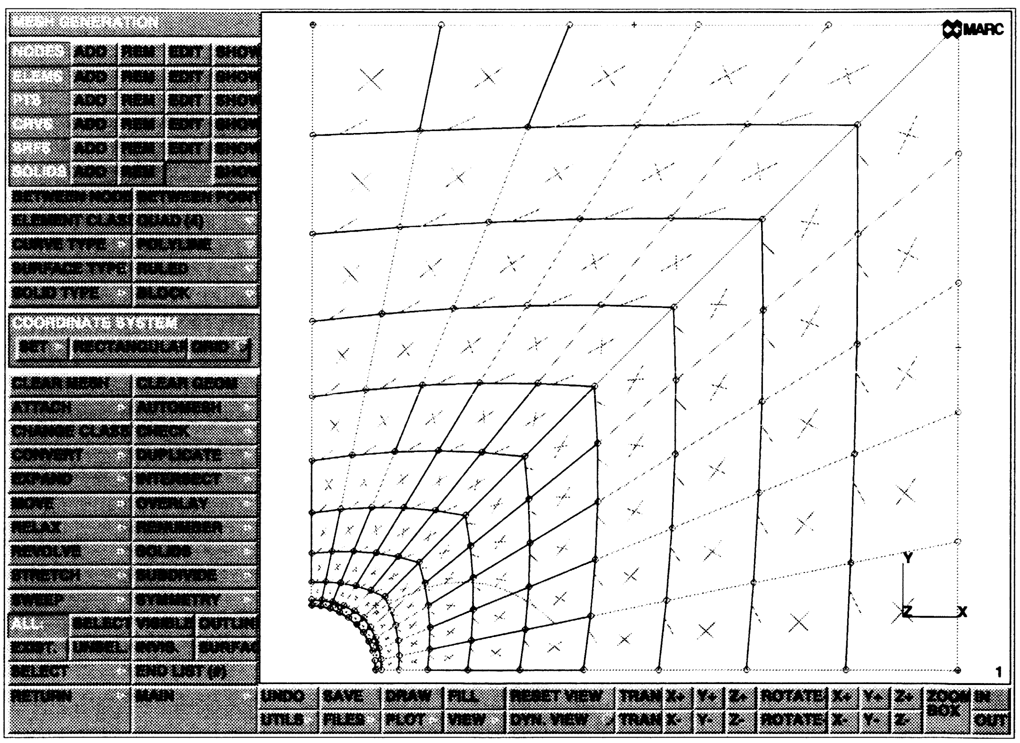


Figure 5.5 Generated Element Mesh

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. Observe that the arrows are now pointing in the correct direction.

```
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
```

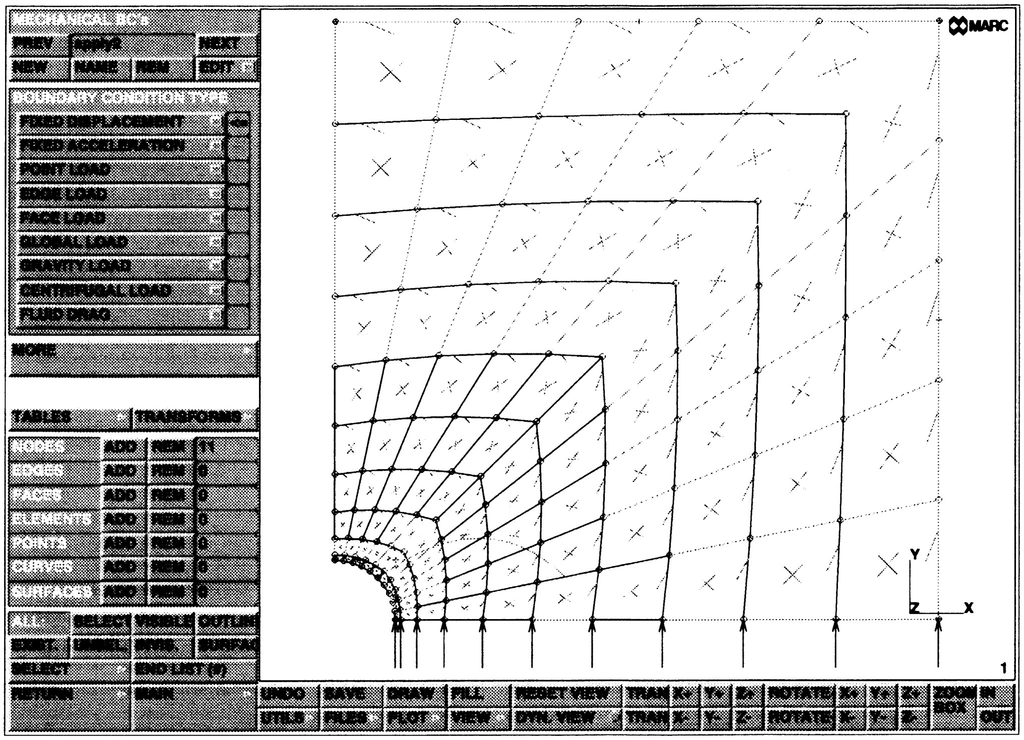


Figure 5.6 Applied Boundary Conditions at the line y=0

The applied loading is a tensile edge load with magnitude 10. Mentat II 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                          (specify pressure)
                OK
            edges ADD       (box pick the edges on the line y = 10)
                END LIST (#)
```

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

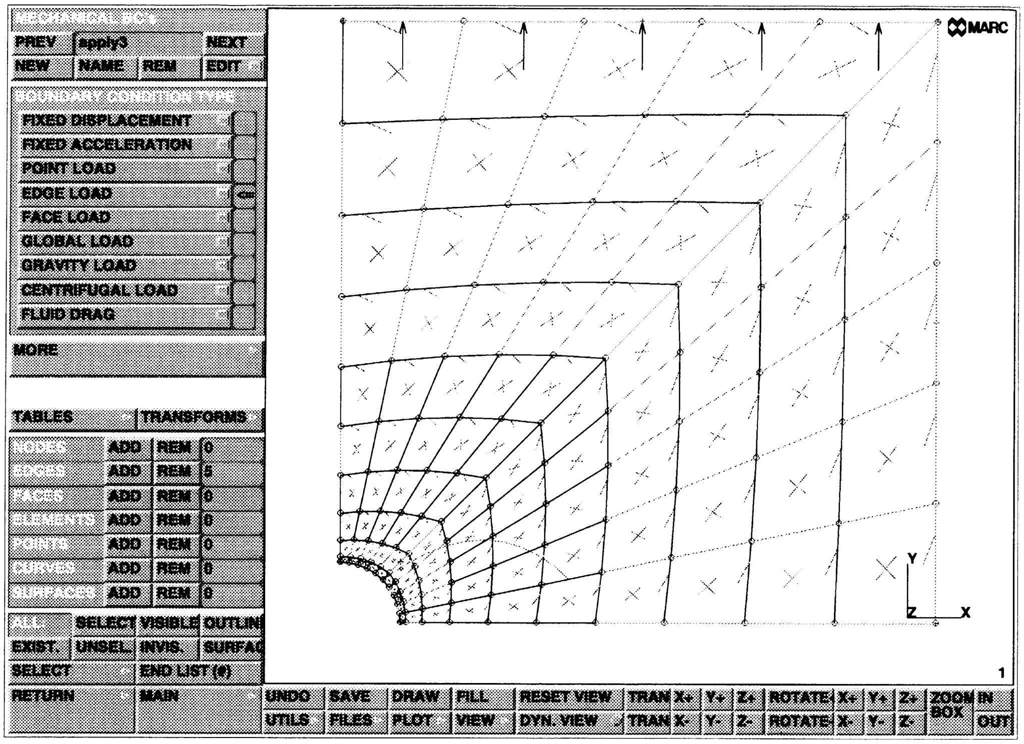


Figure 5.7 Edge Loading

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 II automatically requests for the Poisson's ratio, which can subsequently be entered. After entering this value, the mass density is requested. By entering a <CR> 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.

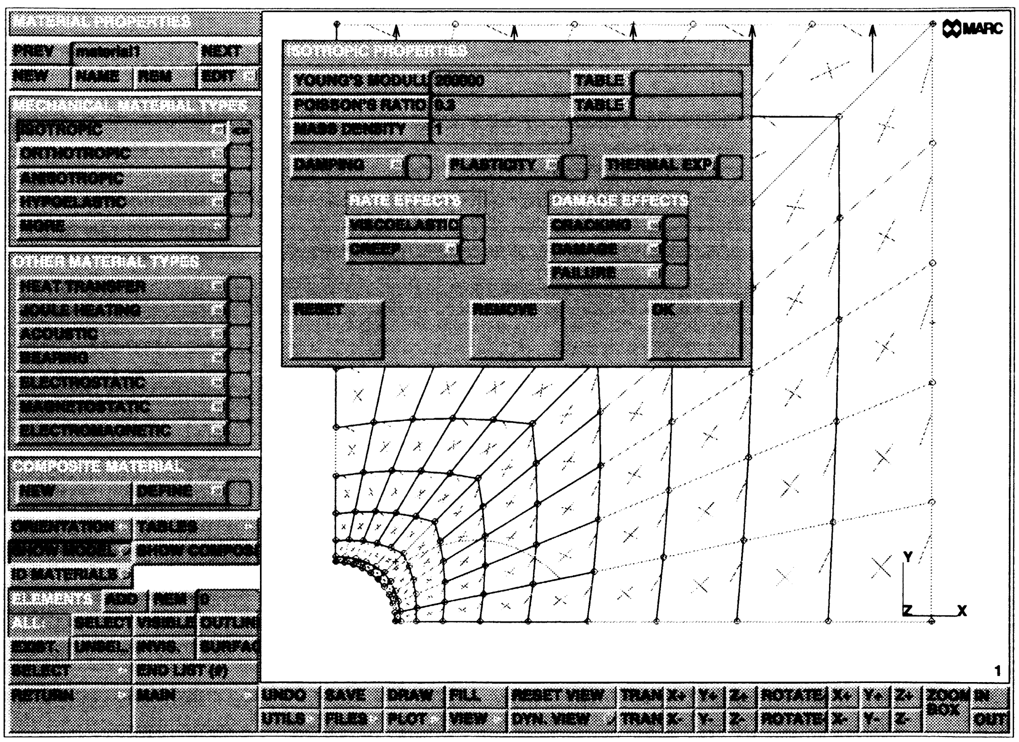


Figure 5.8 Material Properties

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:

MAIN
    JOBS
        MECHANICAL
            JOB RESULTS
                SELECT TENSORS
                    stress
                OK
            INITIAL LOADS
                OK

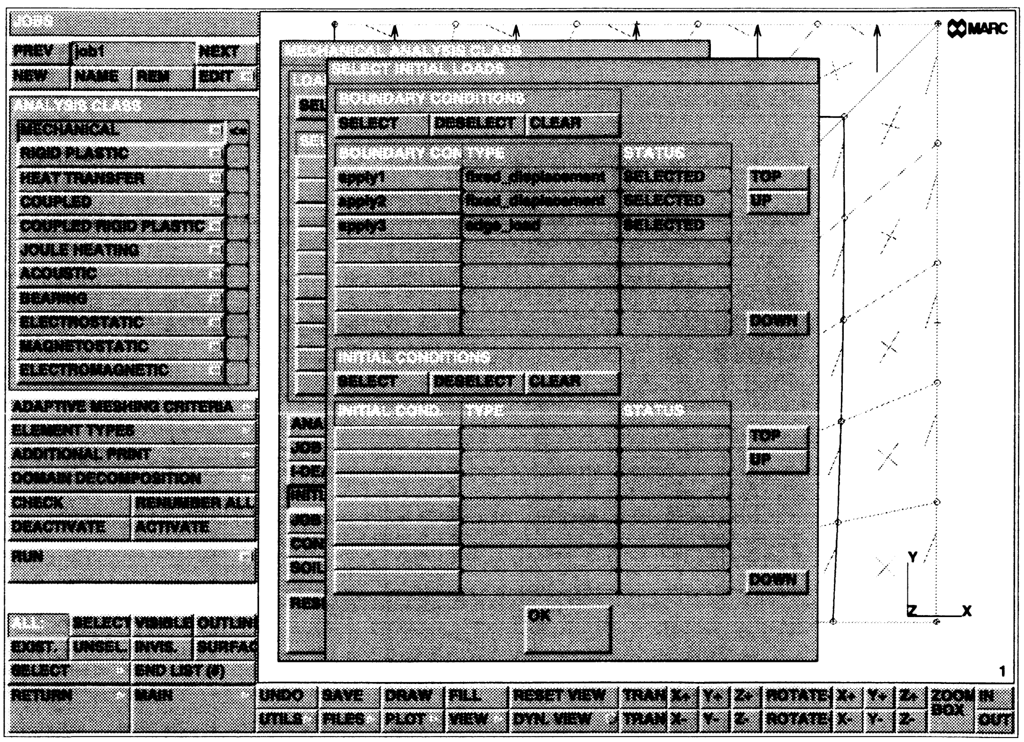


Figure 5.9 Initial Loads

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 *(full integration, QUAD(4))*
OK
all: EXIST.

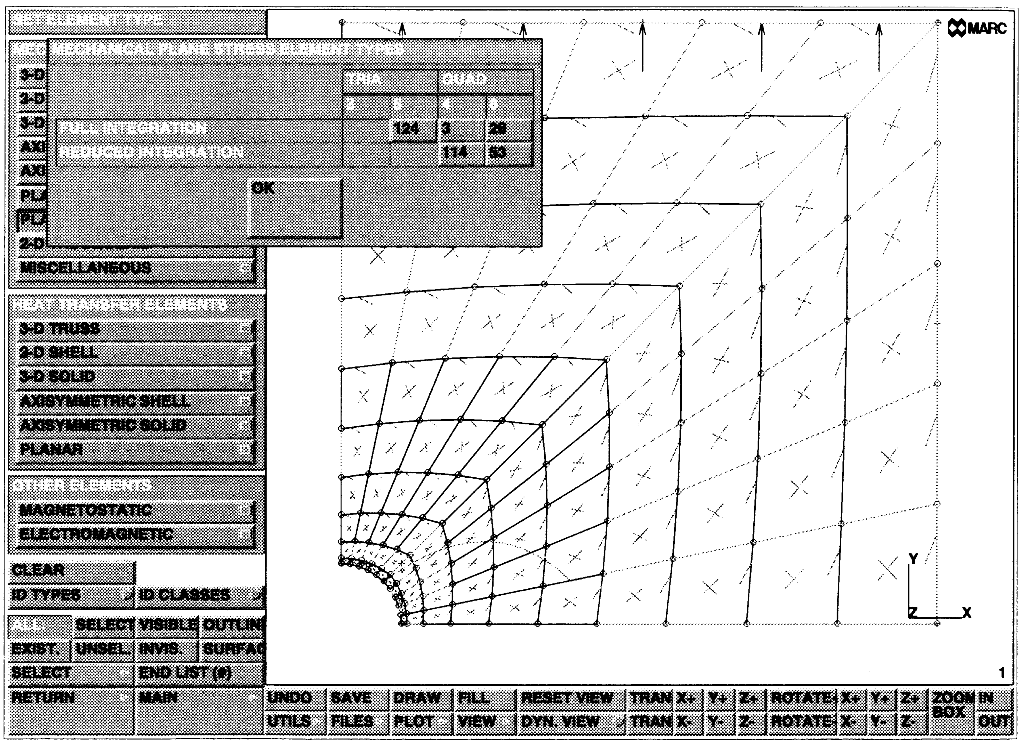


Figure 5.10 Selecting the Element Type

After a CHECK command to verify the consistency of the input, the database is saved and the job will be submitted. The analysis starts and with the MONITOR option the current status of the job can be observed.

MAIN
JOBS
CHECK
SAVE
RUN
SUBMIT 1
MONITOR

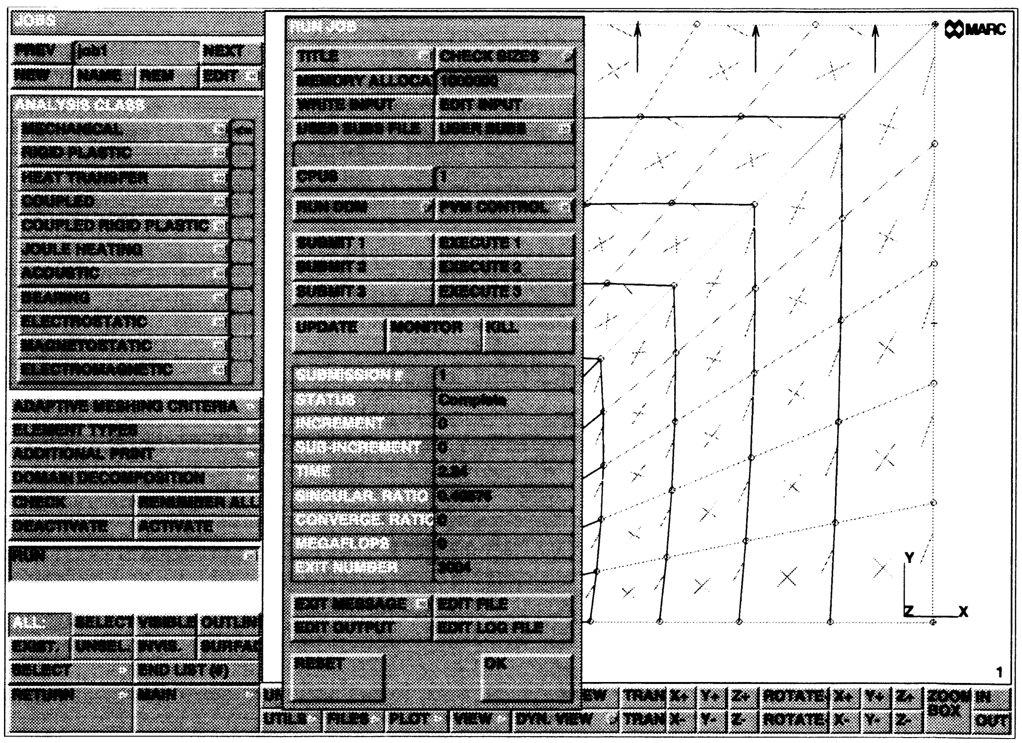


Figure 5.11 Screen After Job Completion

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 II 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
```

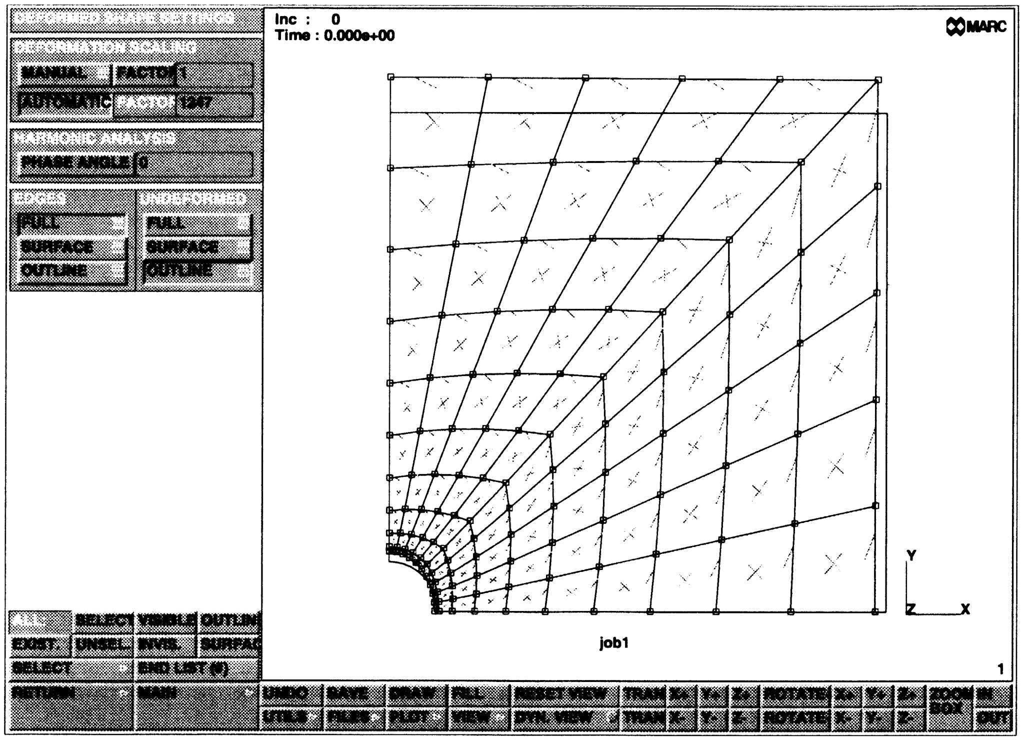


Figure 5.12 Deformed Structure Plot

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.

MAIN
RESULTS
SCALAR
Comp 22 of Stress
OK
CONTOUR BANDS

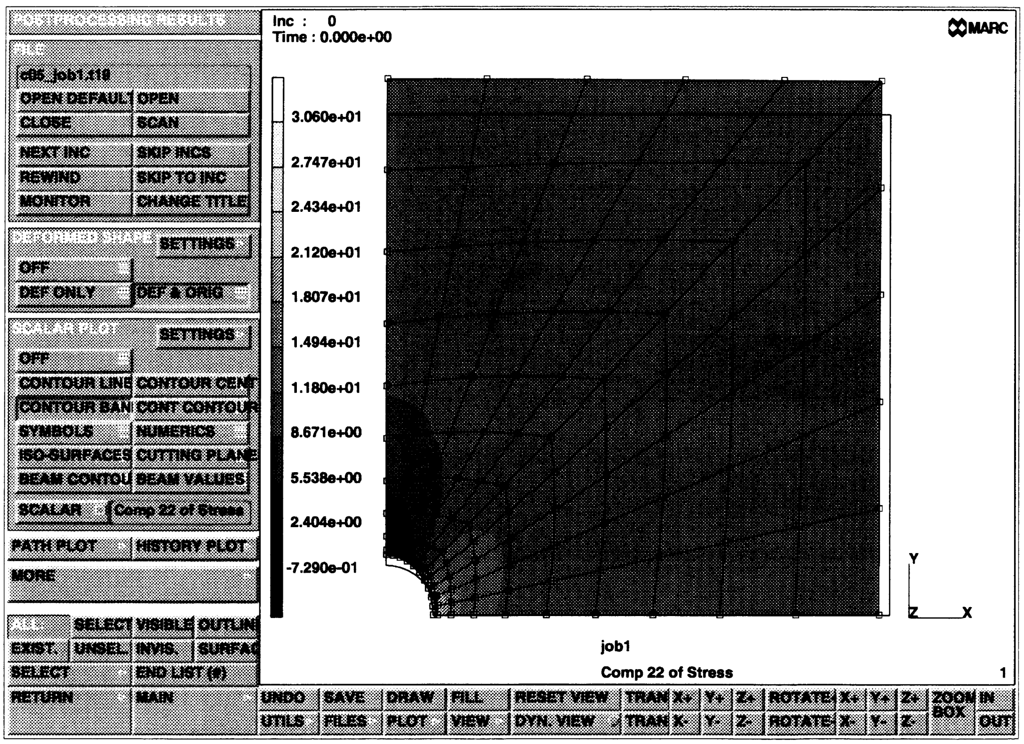


Figure 5.13 Contour Plot of yy Component of Stress

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 *(pick the first and last node at the line y = 0)*
END LIST (#)
SET VARIABLES
ADD CURVE
Arc Length
Comp 22 of Stress
FIT
RETURN
SHOW MODEL
RETURN

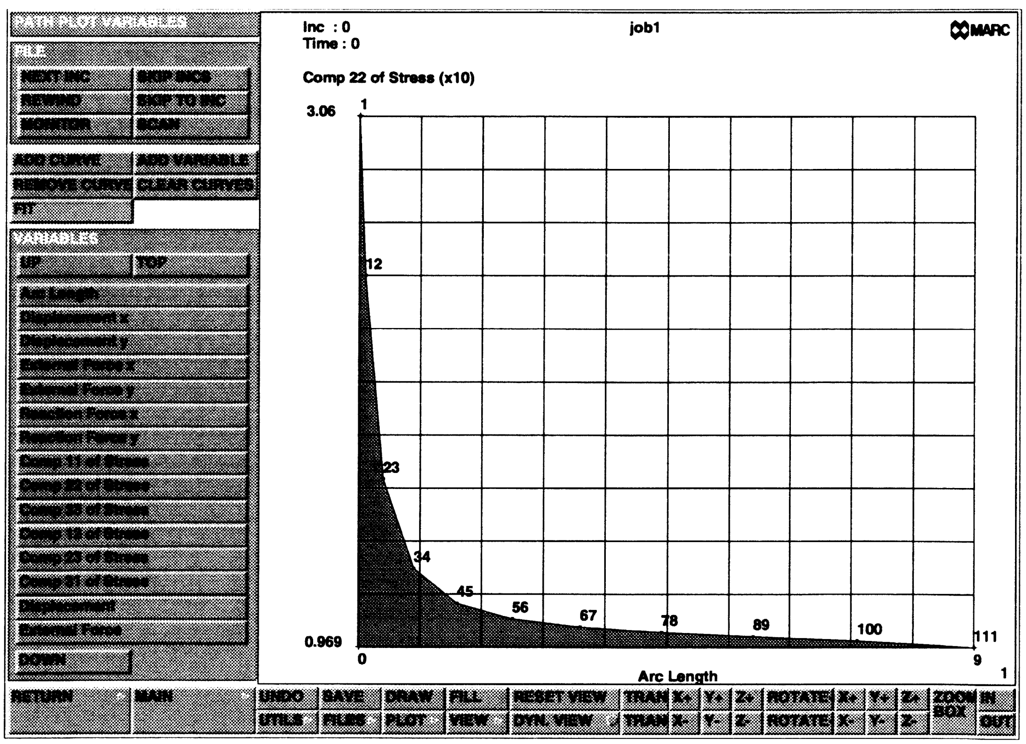


Figure 5.14 Path Plot

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 II.

5.3 Procedure File

```
| Version : MENTAT II 2.3
|
| step 1 - mesh generation
|
*set_grid_spacing
1 1
*set_grid_size
10 10
*set_grid on
*fill_view
*set_curve_type arc_cpp
*add_curves
0 0 0
1 0 0
0 1 0
*set_curve_type polyline
*add_curves
point(10,0,0)
point(10,10,0)
point(0,10,0)
# | End of List
*set_surface_type ruled
*add_surfaces
 1
 2
*set_convert_bias_factors
0 -0.5
*convert_surfaces
 1
# | End of List
*set_grid off
*fill_view
*check_upside_down
*flip_elements
all_selected
*check_upside_down
|
| step 2 - boundary conditions
|
*apply_type fixed_displacement
*apply_dof x
*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
```

Chapter 6: 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.
- Use of the automatic tetrahedral mesh generator to generate the mesh.
- Use of the symmetry and duplicate options to complete the model.

6.1 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.

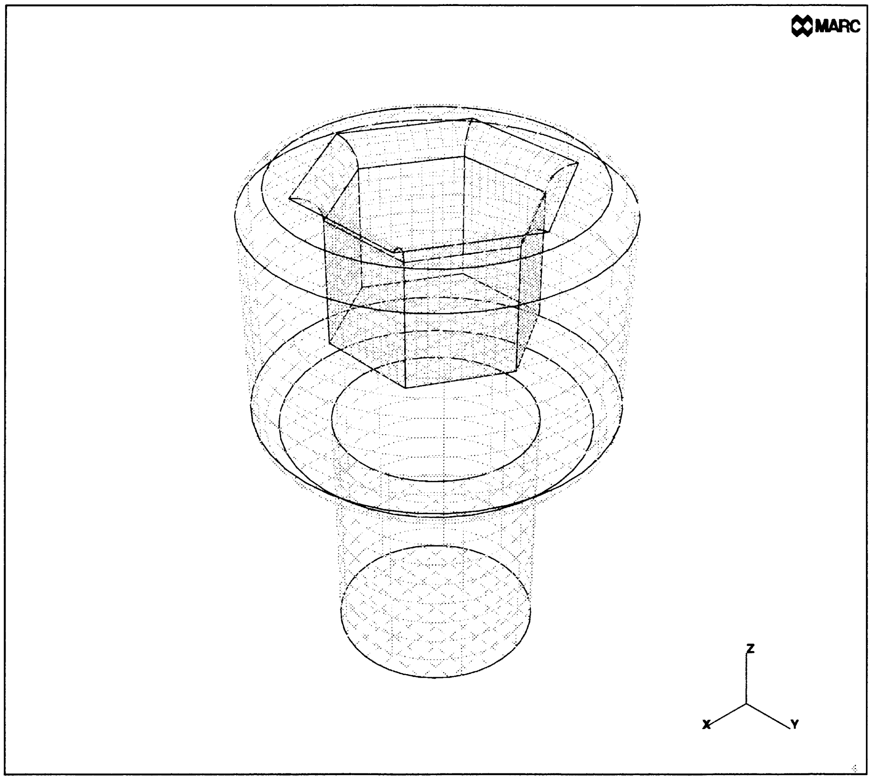


Figure 6.1 Simple Bolt Structure

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 option.

Once the solid model is completed, the automatic mesh generator will be used to generate the mesh. Before using the mesh generator 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.

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.

6.1.1 Overview 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 Meshing on the reduced solid model

Step 5 Use of symmetry and duplication operations to complete the mesh

Before beginning this session, it is necessary to use the Mentat II version including the ACIS based solid modeling capability. This is accessed by typing `mentatS`, `mentatSX`, or `mentatSOGL`.

6.2 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
            FILL
            show 4
            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.

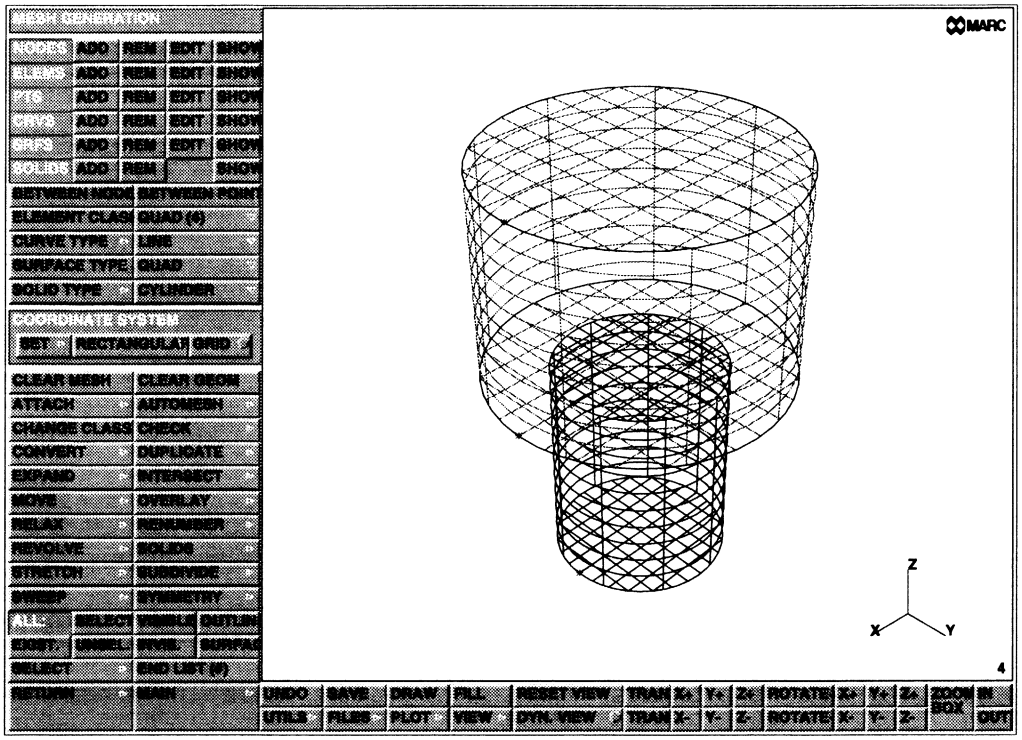


Figure 6.2 Two Cylindrical Solids

The two basic cylinders will now be united.

MAIN
    MESH GENERATION
        SOLIDS
            UNITE
                1 2 *(Pick solid 1 and 2)*
                END LIST (#)

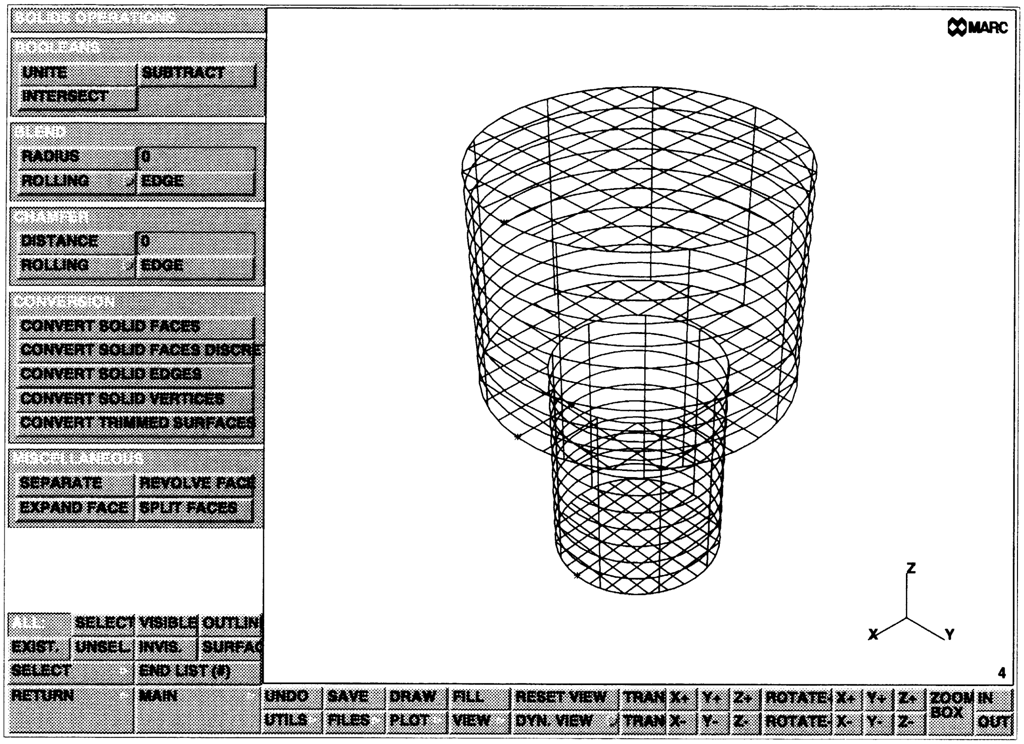


Figure 6.3 Result of UNITE Operation

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` *(solid prism base coordinates)*
`0 0 2.1` *(solid prism axis coordinates)*
`0.4` *(solid prism radius)*
`6` *(number of solid prism faces)*
SOLIDS
SUBTRACT
`1 2` *(Pick solids)*
END LIST (#)

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 Boolean operations.

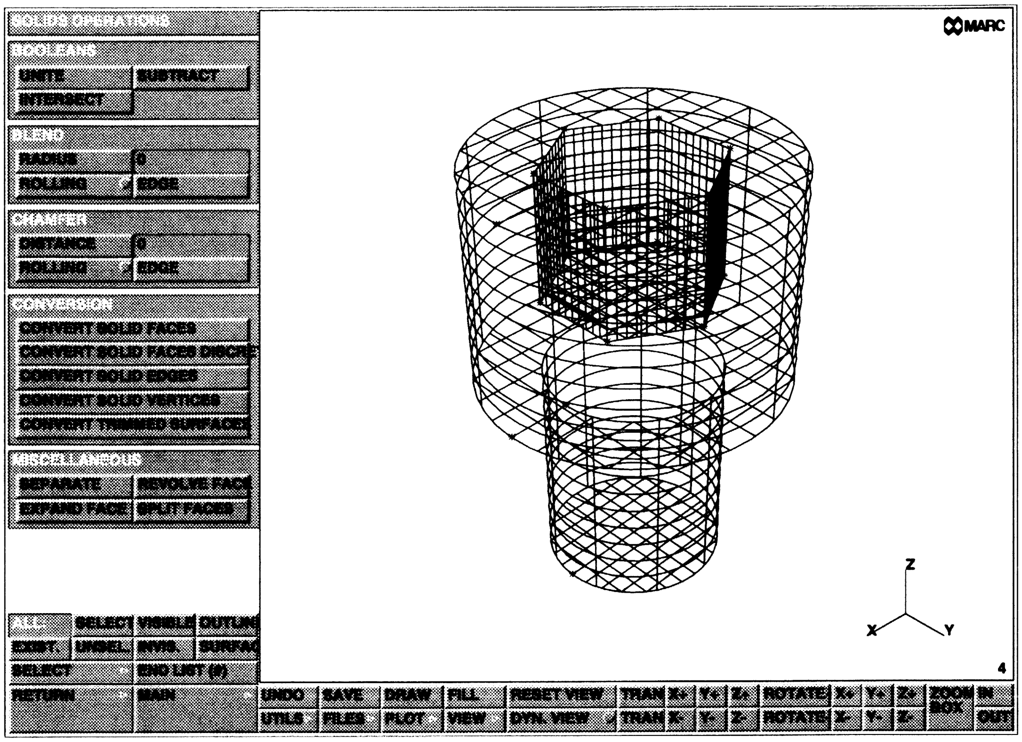


Figure 6.4 Result of SUBTRACT Operation

Step 2

In this step, a series of blending operations will be applied to some edges 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 below:

MAIN
MESH GENERATION
SOLIDS
blend RADIUS
`0.1`
PLOT
label solid EDGES *(on)*
REGENERATE
RETURN
blend EDGE
`1:12` *(Pick the solid edge)*

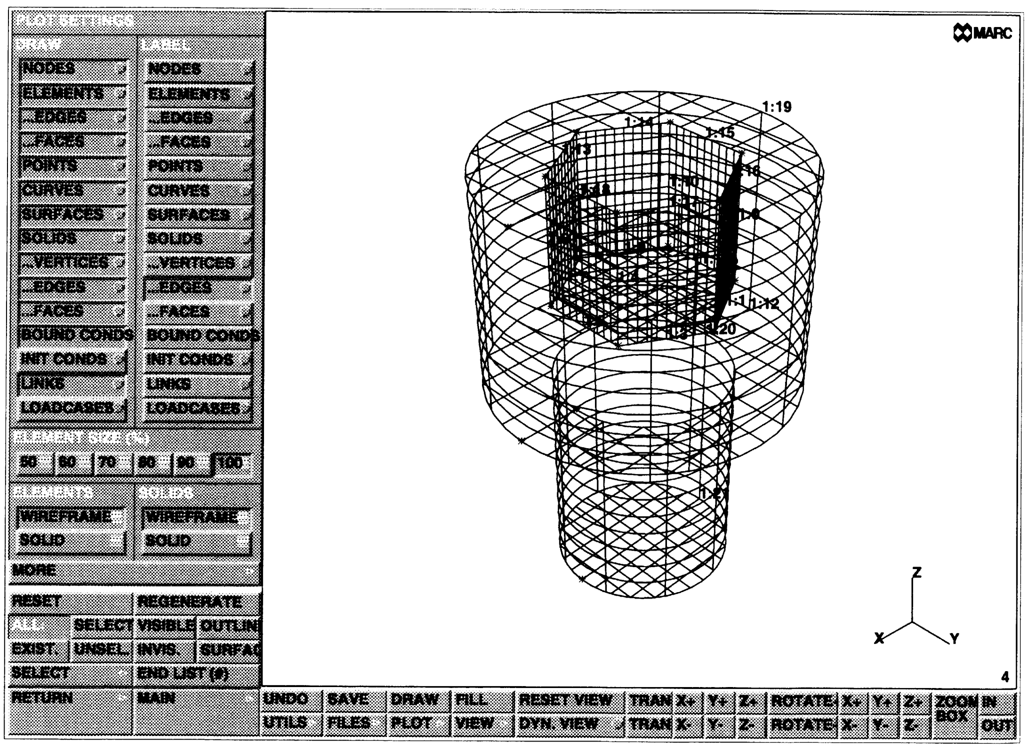


Figure 6.5 Activating the Labeling of Solid Edges

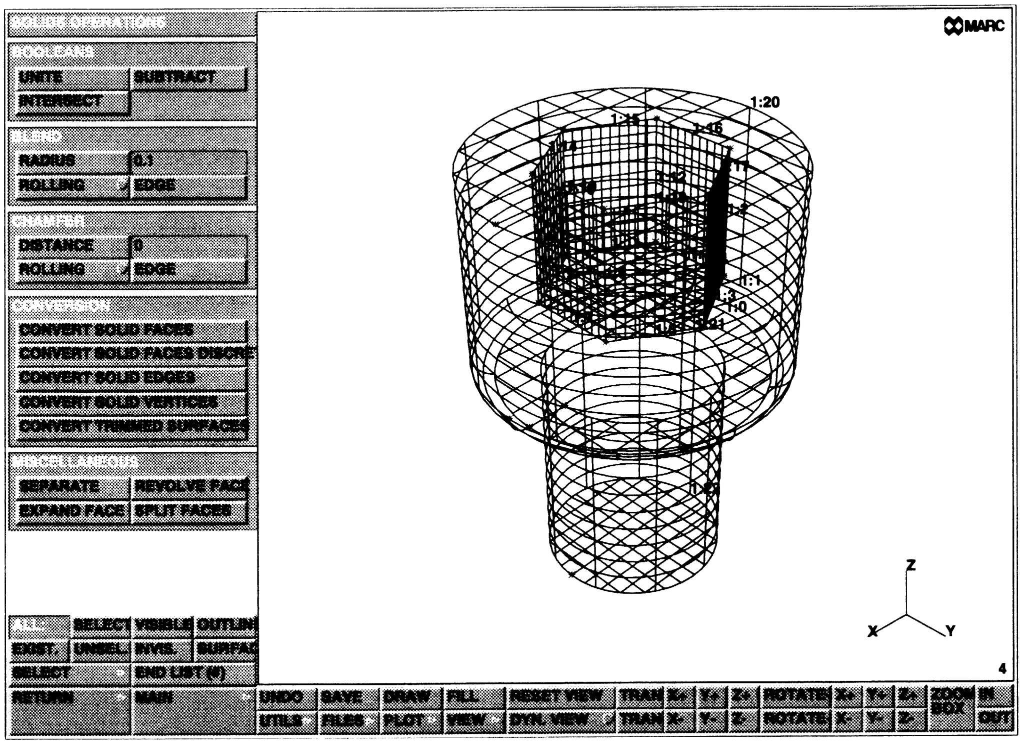


Figure 6.6 First Edge Blended

MAIN
    MESH GENERATION
        SOLIDS
            blend EDGE

```
1:20
1:6
1:9
1:14
1:18
1:23
1:27
```

            PLOT
                label solid EDGES *(off)*
                REGENERATE
                RETURN

The solid modeling phase is now completed as is shown in Figure 6.1.

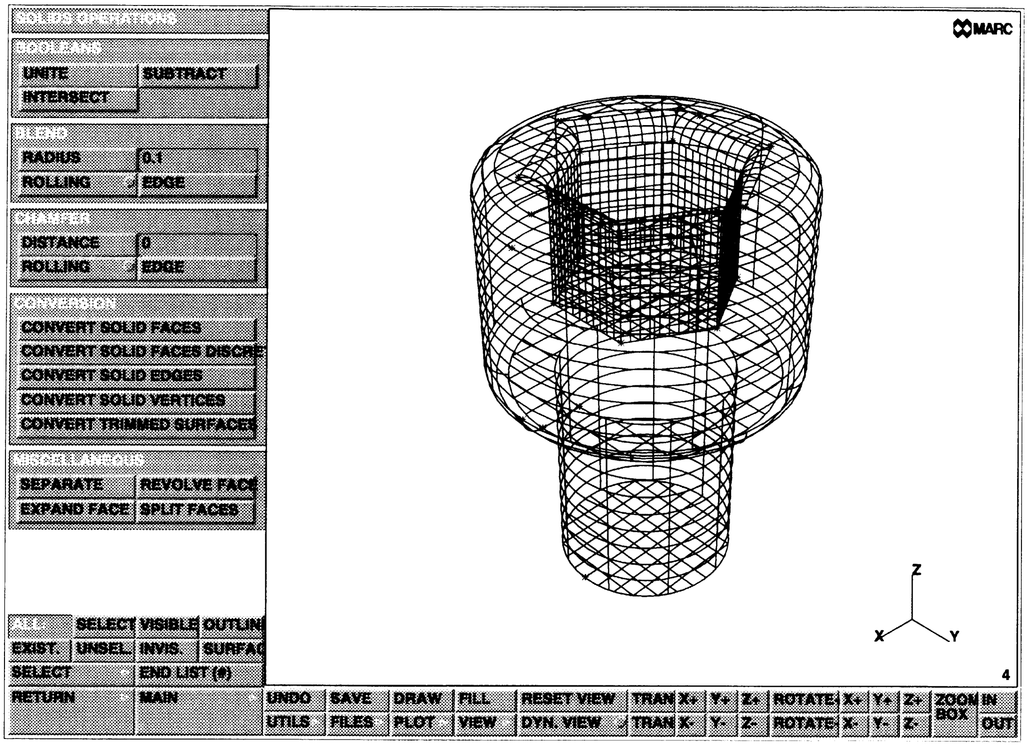


Figure 6.7 Completed Solid Model

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.

MAIN
  MESH GENERATION
    VIEW
      activate 1 *(on)*
      show 1
      FILL
      RETURN
    SOLID TYPE
      BLOCK
      RETURN
    solids ADD
      `-1 0 -1` *(solid block origin coordinates)*
      `2 2 4` *(solid block X, Y and Z dimensions)*

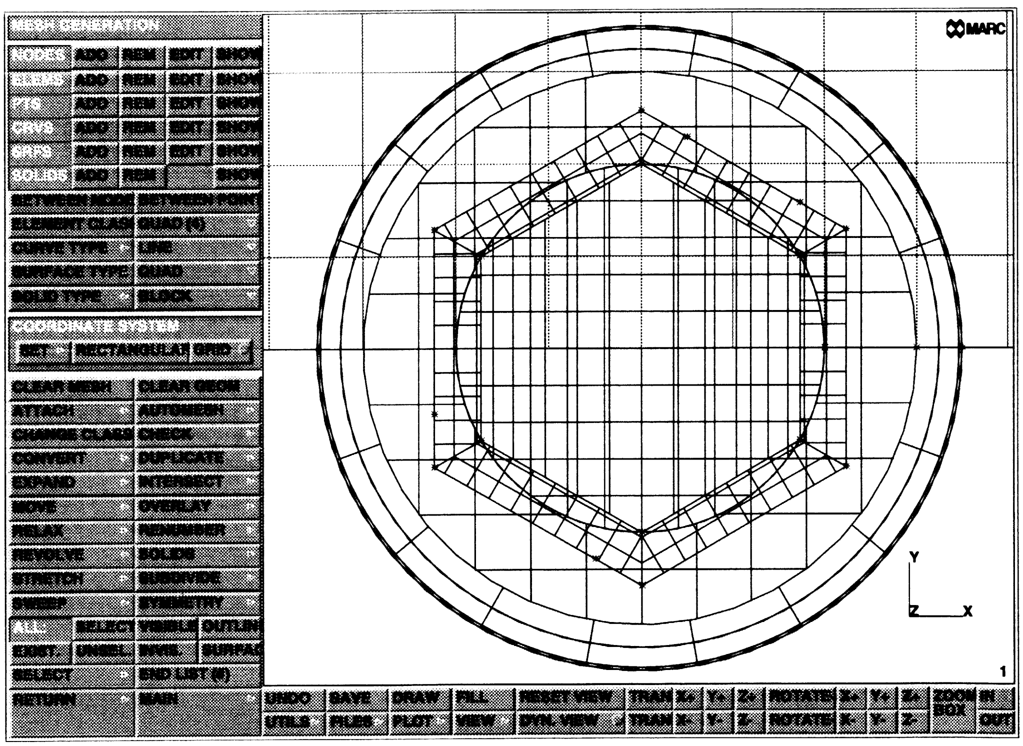


Figure 6.8 First Solid Block Added

MAIN
  MESH GENERATION
    DUPLICATE
      ROTATIONS
        0 0 150 *(duplicate rotations in X,Y and Z)*
      SOLIDS
        2 *(Pick the solid block)*
        END LIST (#)

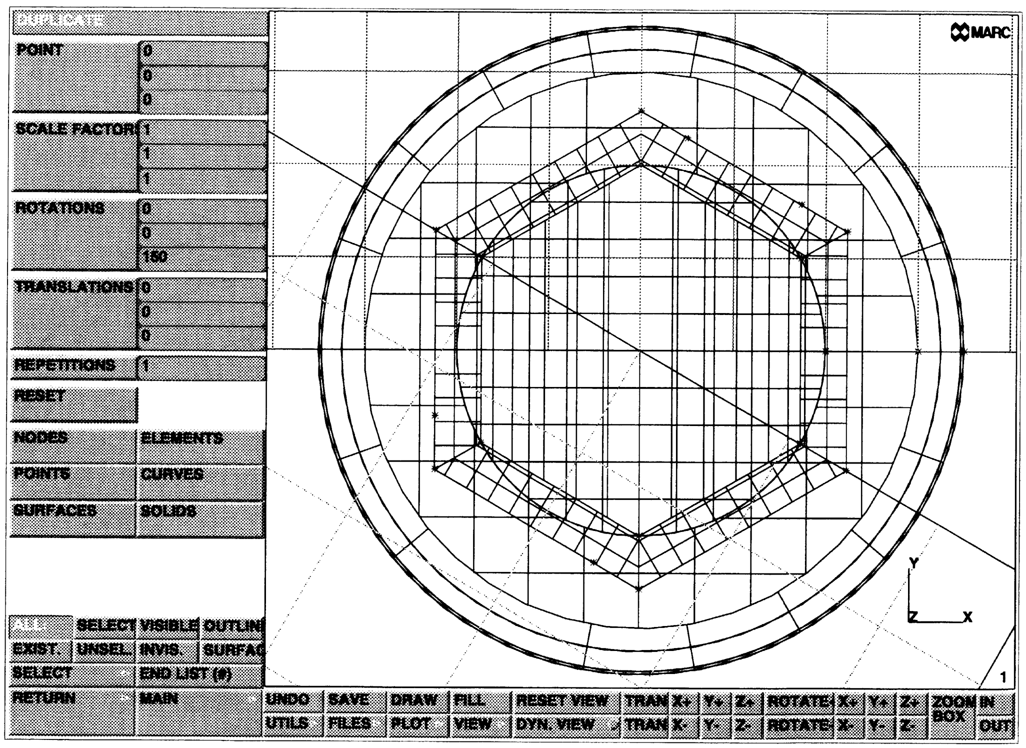


Figure 6.9 Duplicating the Solid Block

MAIN
- MESH GENERATION
  - SOLIDS
    - SUBTRACT
      - 1 *(Pick solid to subtract from)*
      - 2 *(Pick solid to be subtracted)*
      - 3 *(Pick solid to be subtracted)*
      - END LIST (#)
    - FILL
    - VIEW
      - show 4
      - RETURN

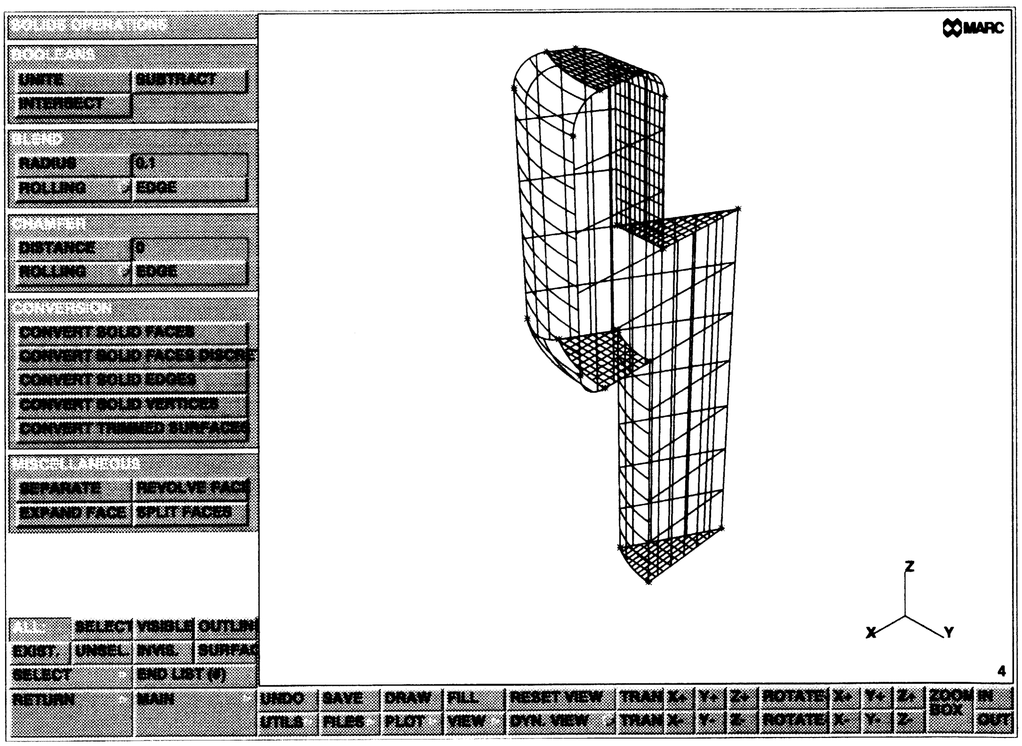


Figure 6.10 Completed Solid Segment

Step 4

In this step, the reduced model will be meshed automatically using the tetrahedral mesh generator. First a target element edge length will be specified on the edges of the solid. Here, for all solid edges an average element edge length of 0.1 is used.

MAIN
  MESH GENERATION
    AUTOMESH
      solids AVG EDGE LEN.
        0.1 *(enter element edge length)*
        all: EXIST.

In Figure 6.11, the resulting *seed points* are shown.

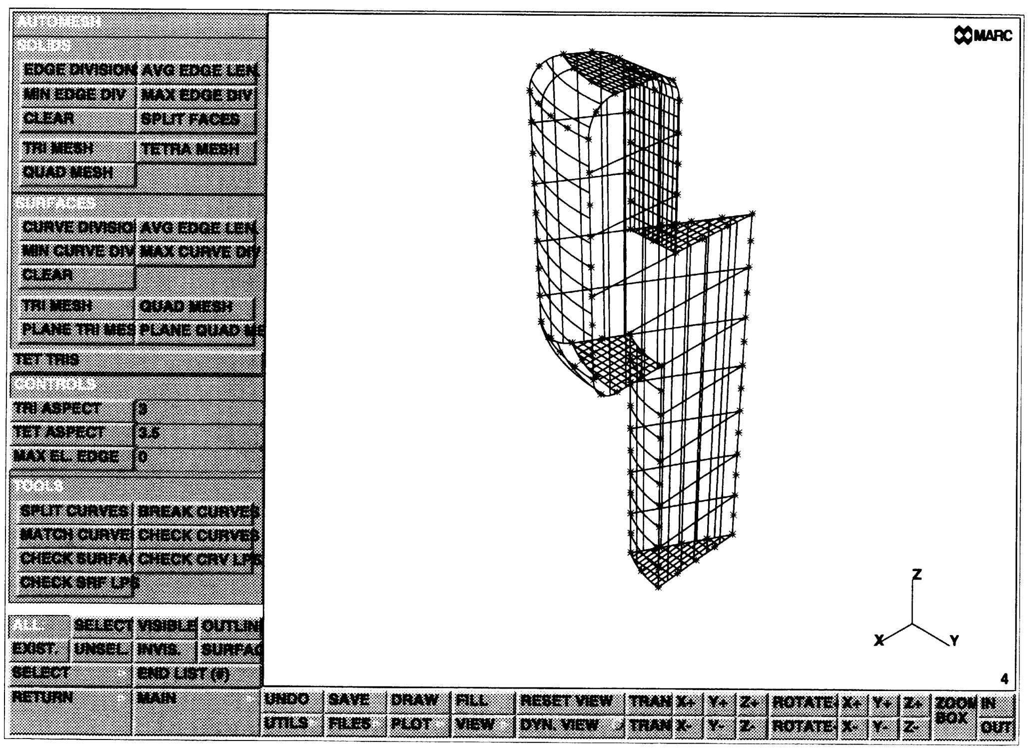


Figure 6.11 Seed Points for Automatic Meshing

Next, the automatic mesh generator will be used to generate a mesh. After the mesh generation, the drawing of solids 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
            TETRA MESH
                1                                   (Pick the solid)
                END LIST (#)
            PLOT
                draw SOLIDS                         (off)
                draw NODES                          (off)
                elements SOLID
                REGENERATE
                RETURN
```

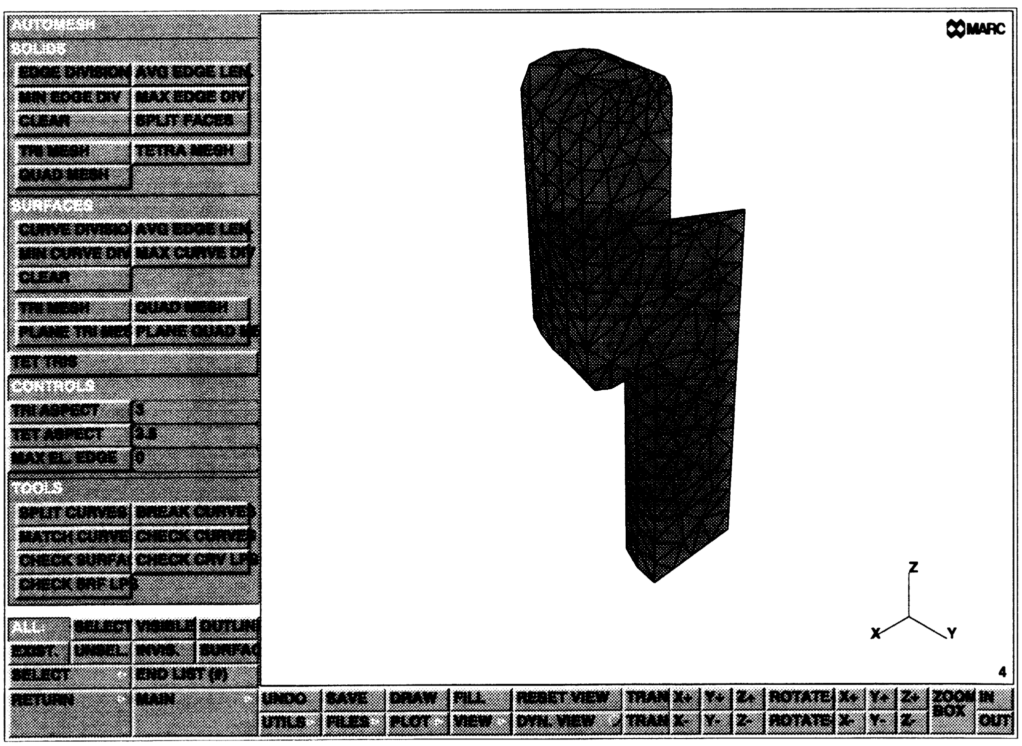


Figure 6.12 Completed Mesh for the Segment

Step 5

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.
```

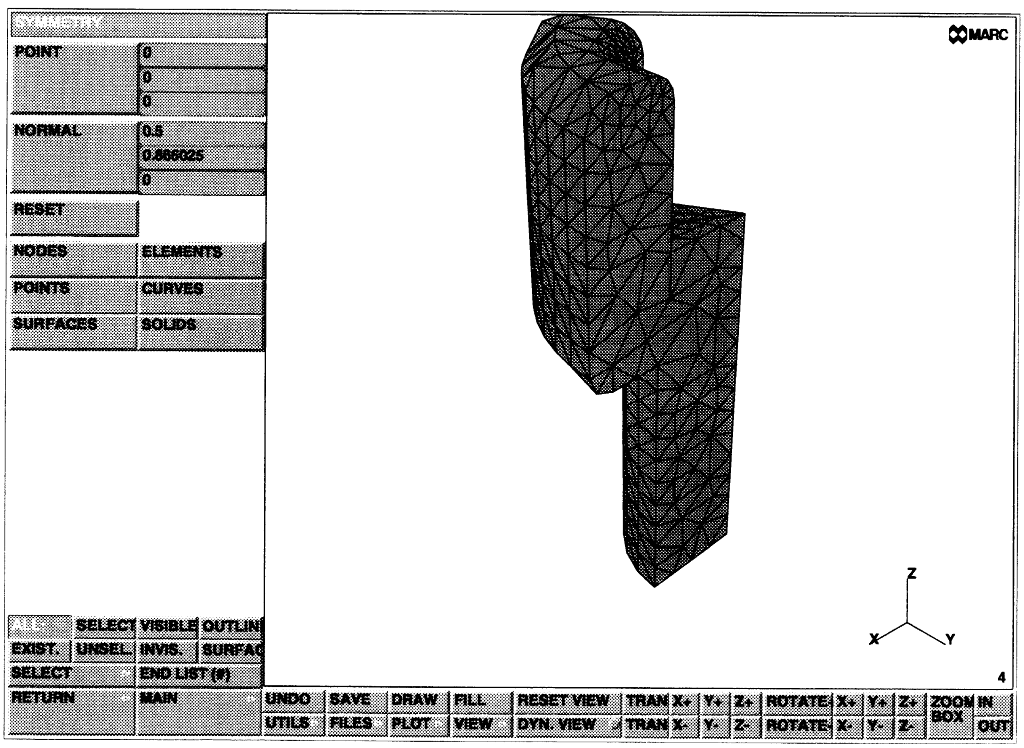


Figure 6.13 Mesh After Use of the SYMMETRY Operator

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

```
MAIN
    MESH GENERATION
        DUPLICATE
            ROTATIONS
                0  0  60
            REPETITIONS
                5
            ELEMENTS
                all: EXIST.
            FILL
```

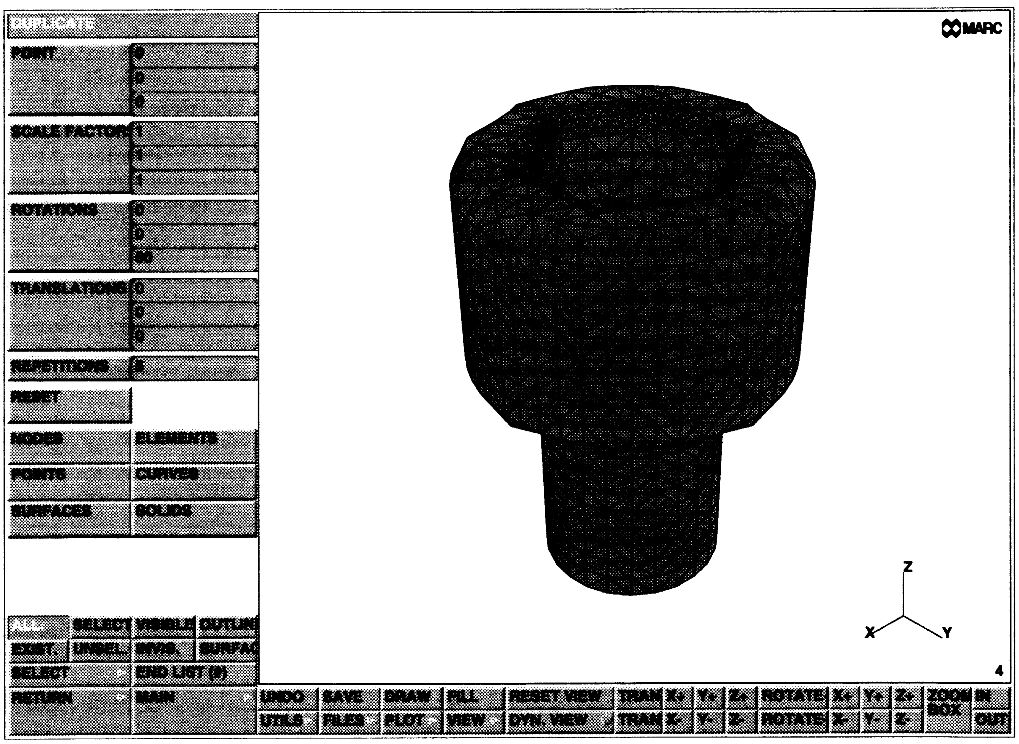


Figure 6.14 Element Mesh After the DUPLICATE Process

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 default tolerance of 0.0001.

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

MAIN
    MESH GENERATION
        SWEEP
            sweep NODES
                all: EXIST.
            SAVE

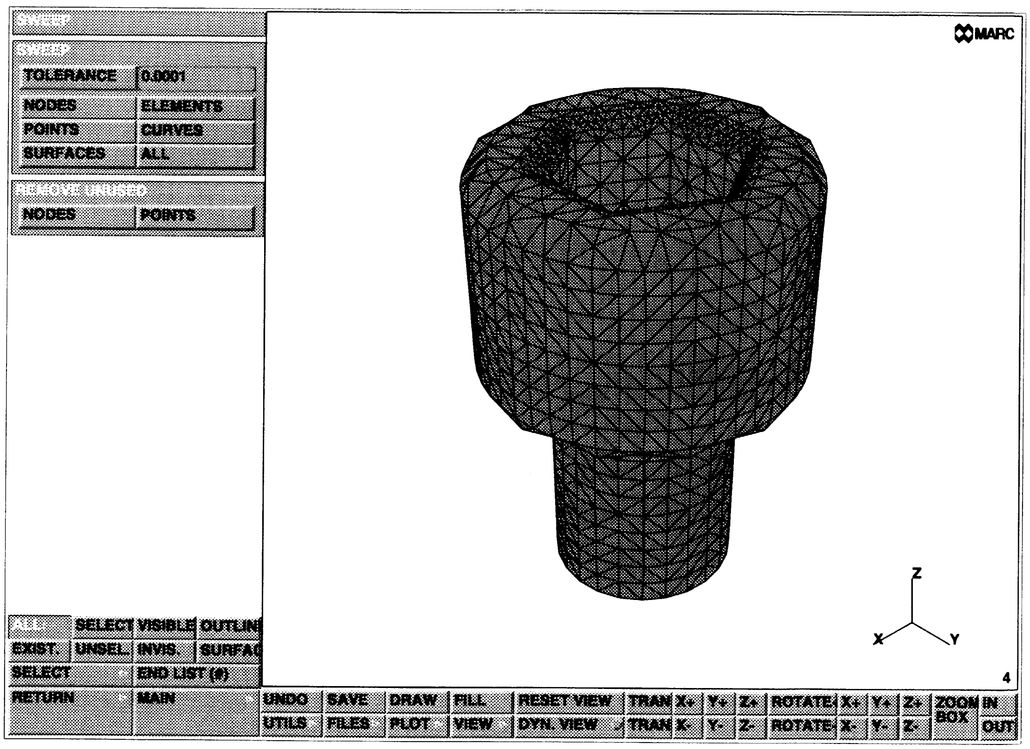


Figure 6.15 The SWEEP Processor

6.3 Procedure File

```
| Version : MENTAT II 2.3
|
| step 1 add the basic solid building blocks
|
*set_solid_type cylinder
*add_solids
0 0 0
0 0 1
0.4 0.4
*add_solids
0 0 1
0 0 2
0.7 0.7
*activate_view 4
*deactivate_view 1
*view_perspective
*fill_view
*show_view 4
*solids_unite
 1
 2
# | End of List
*set_solid_type prism
*add_solids
0 0 1.4
0 0 2.1
0.4
6
*solids_subtract
 1
 2
# | End of List
|
| step 2 modify the solid model by including blending
|
*solid_blend_radius
0.1
*set_solid_edge_labels on
*regen
*solid_blend_edge
 1:12
*solid_blend_edge
 1:20
 1:6
 1:9
 1:14
 1:18
```

```
 1:23
 1:27
*set_solid_edge_labels off
*regen
|
| step 3 prepare for the meshing by reducing the model to
|        the smallest symmetrical structure
|
*activate_view 1
*show_view 1
*fill_view
*set_solid_type block
*add_solids
-1 0 -1
2 2 4
*set_duplicate_rotations
0 0 150
*duplicate_solids
 2
# | End of List
*solids_subtract
 1
 2
 3
# | End of List
*fill_view
*show_view 4
|
| step 4 start the meshing on the reduced model
|        the smallest symmetrical structure
|
*solid_edge_length
0.1
all_existing
*solid_tetmesh_solids
 1
# | End of List
 *set_solids off
 *set_nodes off
*elements_solid
*regen
|
| step 5 use symmetry and duplicate to complete
| the mesh. Finally sweep double nodes
|
*set_symmetry_normal
sin(30*pi/180)
cos(30*pi/180)
0
*symmetry_elements
```

```
all_existing
*set_duplicate_rotations
0 0 60
*set_duplicate_repetitions
5
*duplicate_elements
all_existing
*fill_view
*sweep_nodes
all_existing
*save_as_model
c06
y
```

Chapter 7: Tube Flaring

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)

7.1 Background Information

7.1.1 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.

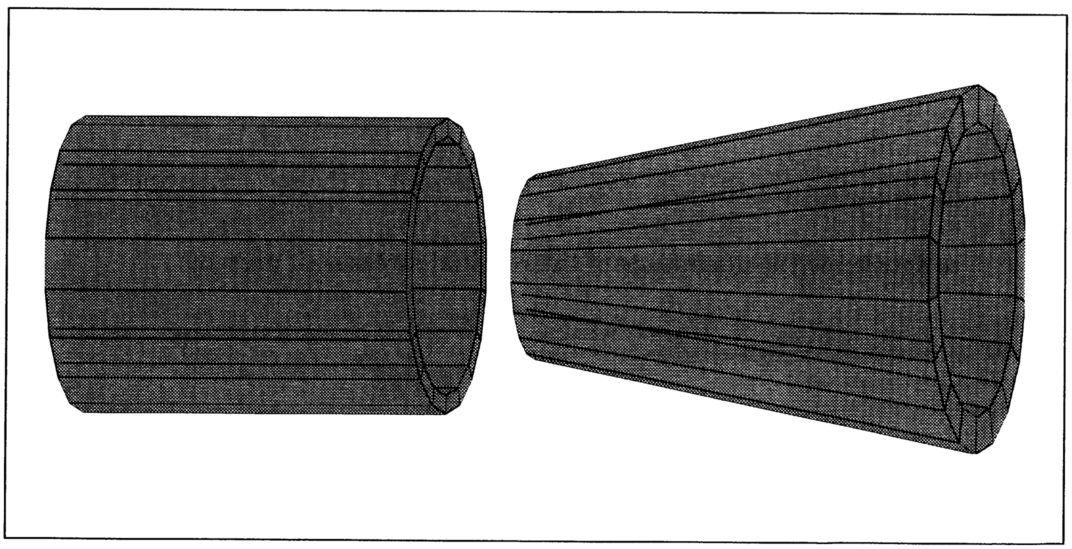


Figure 7.1 Cylindrical Tube and Flaring Tool

7.1.2 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.

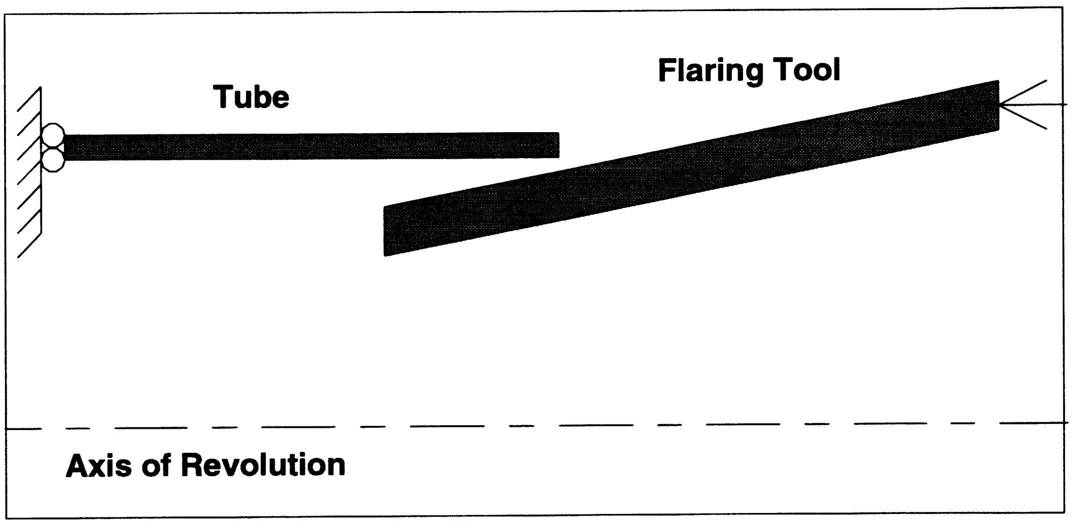


Figure 7.2 Axisymmetric Model of Tube and Flaring Tool

7.1.3 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.

7.1.4 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.

7.1.5 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 UPDATED LAGRANGE PROCEDURE, FINITE STRAIN PLASTICITY and LARGE DISPLACEMENT.

Step 6 Submit the job.

Step 7 Post-process the results by looking at the deformed structure and a history plot of the tip deflection of the tube.

7.2 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 the horizontal grid spacing to 1.0, the vertical grid spacing to 0.5, the horizontal size to 8.0 and the vertical size to 5.0 inches.

```
MAIN
    MESH GENERATION
        SET
            SPACING
                1  0.5
            SIZE
                8  5
            ON                                          (on)
            RETURN
        FILL
```

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                          (Pick top 2 points)
                END LIST (#)
```

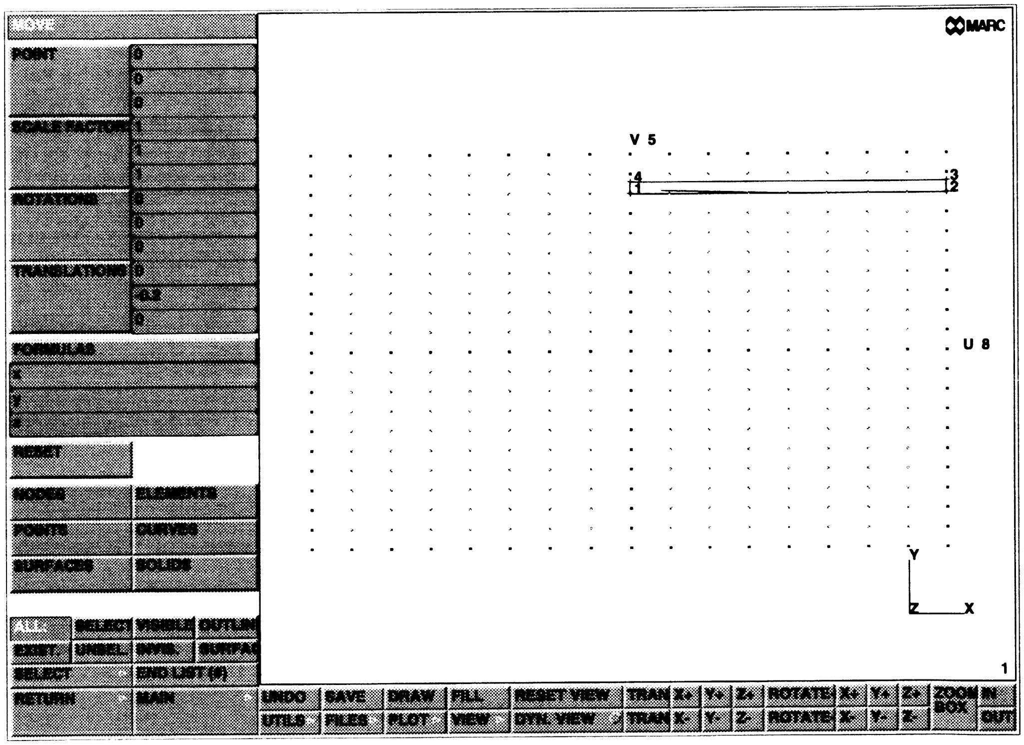


Figure 7.3 First Patch (Cross Section of Cylindrical Tube)

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

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 *(to switch to SPHERICAL)*
        SPHERICAL *(to switch to RECTANGULAR)*
        DUPLICATE
            TRANSLATIONS
                `0 0.6 0`
            POINTS
                `5 6` *(Pick the two points generated above)*
                END LIST (#)

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

MAIN
    MESH GENERATION
        srfs ADD
            `5 6 8 7` *(Pick points in counter-clockwise direction)*

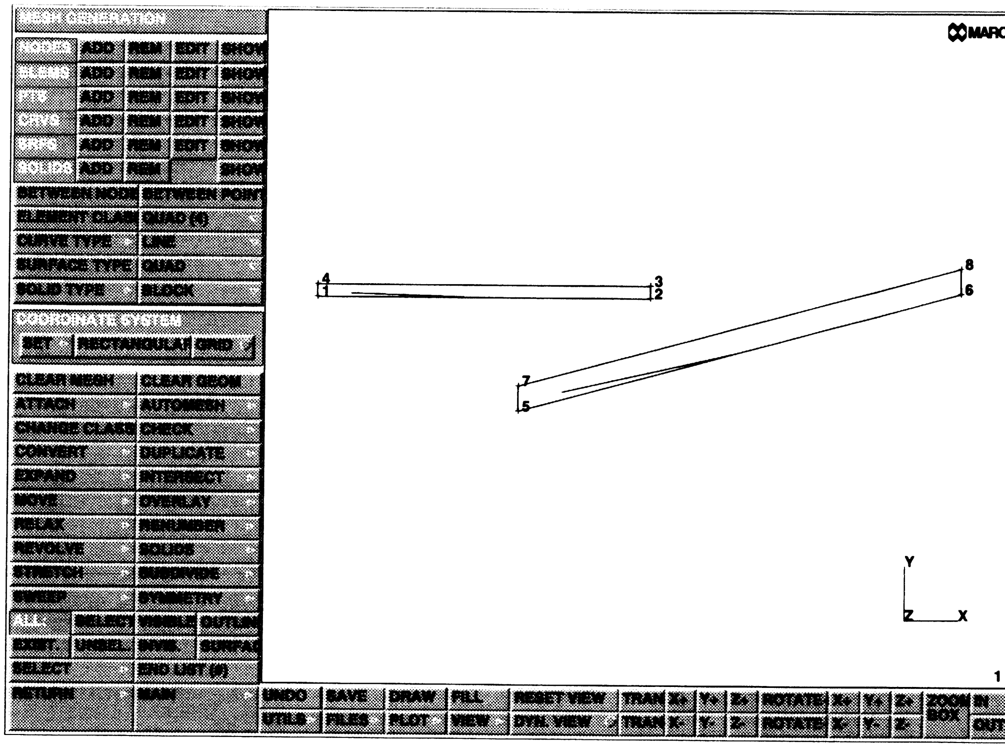


Figure 7.4 Second Patch (Cross Section of Flaring Tool)

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` *(Pick the surface to move)*
                END LIST (#)

The two patches that outline the cylindrical tube and conical flaring tool respectively are shown in Figure 7.5.

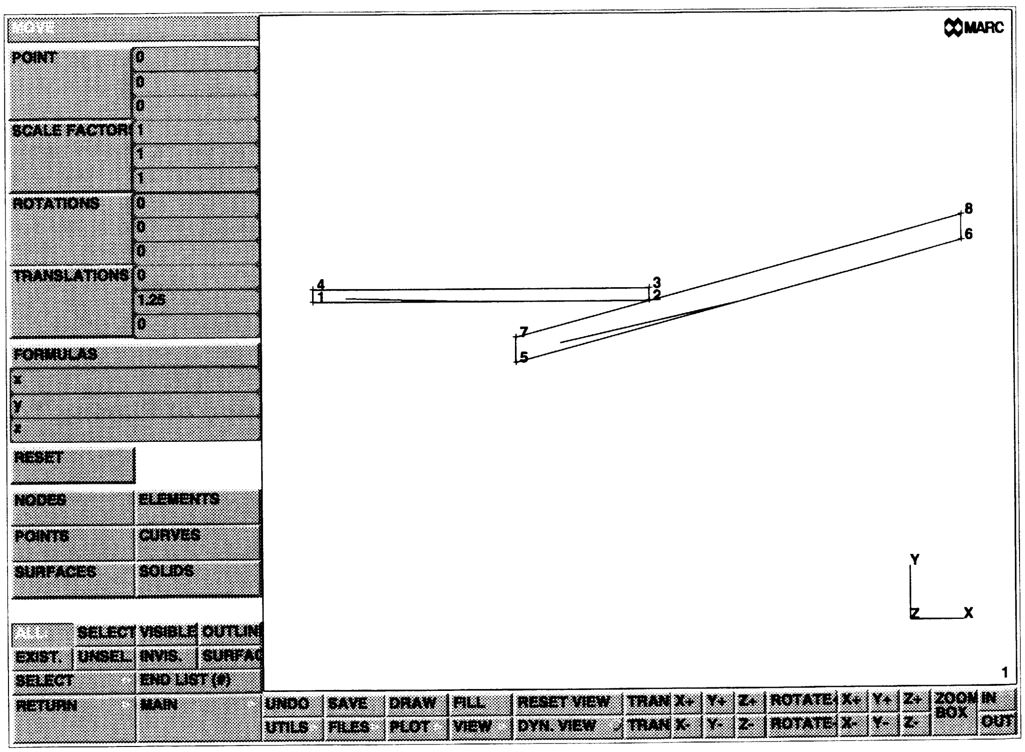


Figure 7.5 Tube and Flaring Tool Patches Defined

The two patches are converted to elements. The number of subdivisions is set to 8 x 3 for the cylindrical tube and to 14 x 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
`12` *(Pick the surface to convert)*
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.

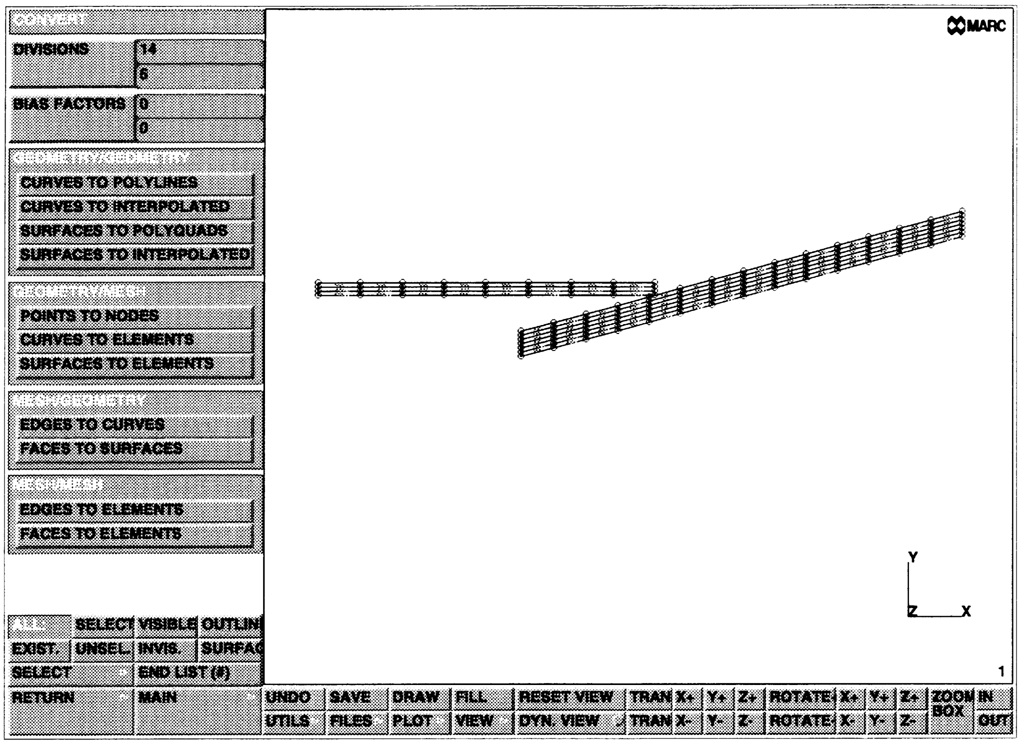


Figure 7.6 Tube and Flaring Tool Patches Converted

Once you have converted the two patches to elements, you can assign a MARC element type to the elements. Although Mentat II 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.

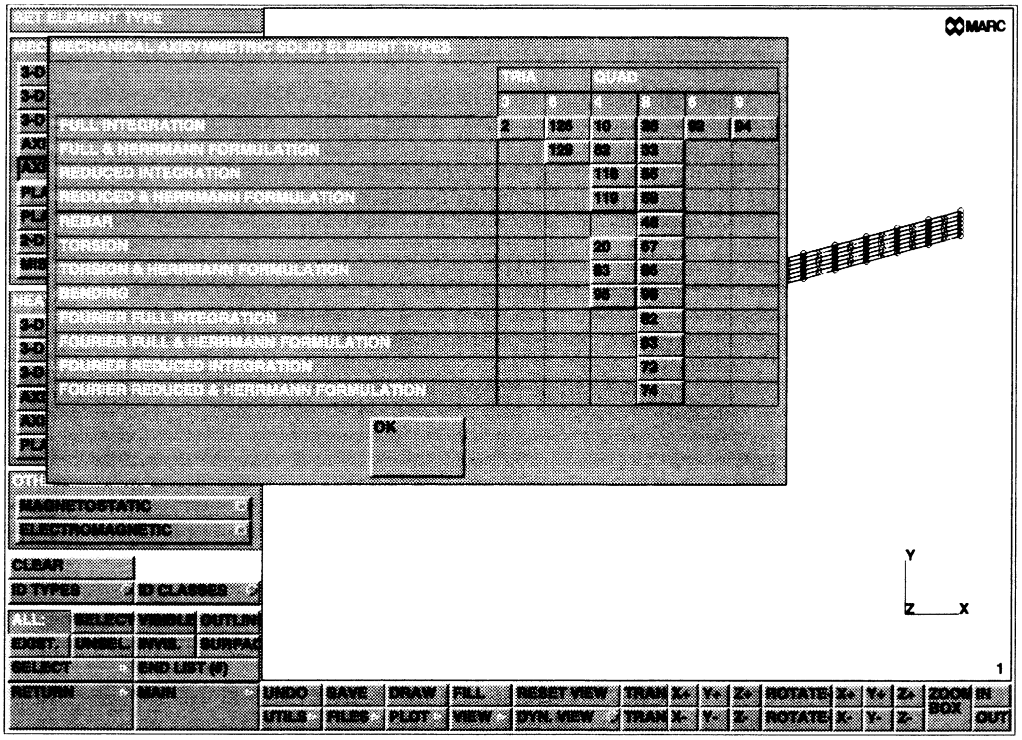


Figure 7.7 Select MARC Element Type

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 *(on)*
OK
nodes ADD
`1 10 19 28` *(Pick left row of nodes)*
END LIST (#)

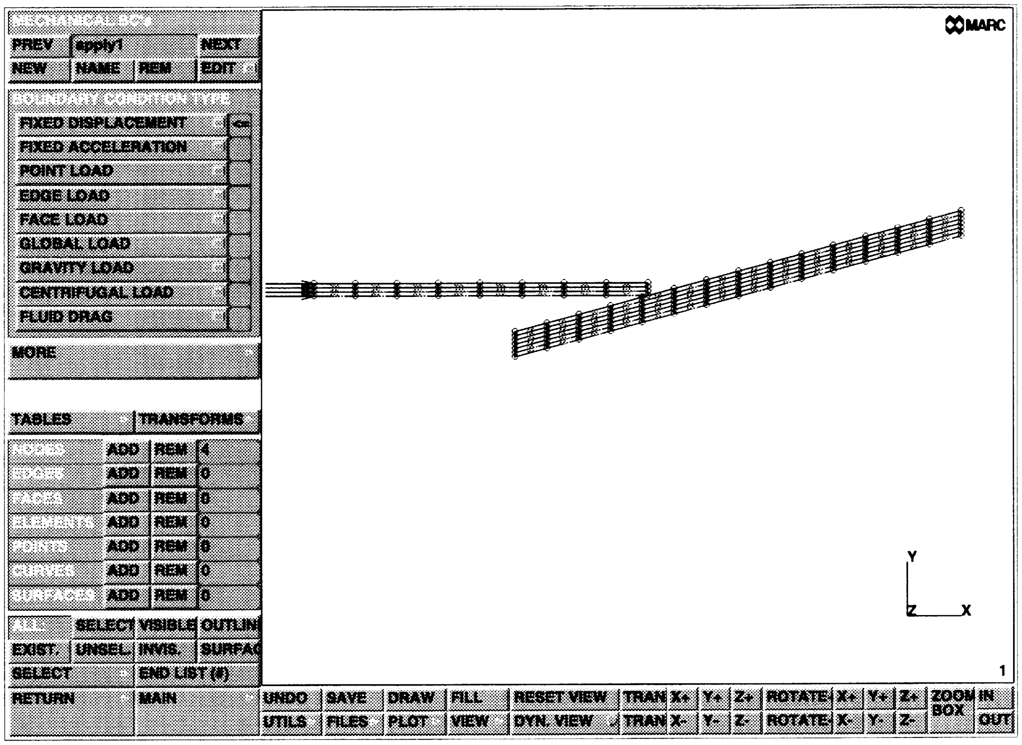


Figure 7.8 Fixed Nodes of Cylindrical Tube in X-Direction

It is often necessary to place (weak) springs between deformable bodies that come into contact to prevent rigid body motion if there is an inadequate amount of kinematic constraints (or boundary conditions). Enter a spring by using the following button sequence:

MAIN
LINKS
SPRING/DASHPOT
ZOOM BOX

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

STIFFNESS
`10.0e3`
NODE 1
`131`
DOF 1
`1`
NODE 2
`9`
DOF 2
`1`

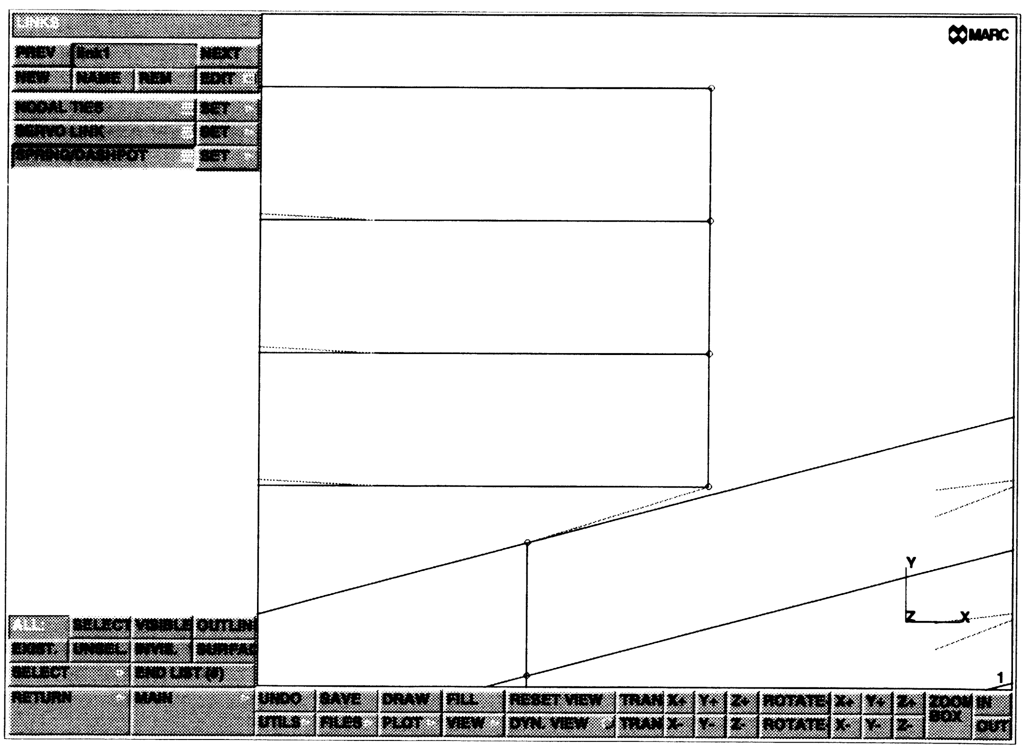


Figure 7.9 Specifying the Spring Between the Deformable Bodies

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
    MATERIAL PROPERTIES
        FILL
        ISOTROPIC
            YOUNG'S MODULUS
                30.0e6
            POISSON'S RATIO
                0.3
            PLASTICITY
                INITIAL YIELD STRESS
                    3.6e4
                OK
            OK
        elements ADD                    (use the Box Pick Method to
                                         select all tube elements)
            END LIST (#)
        PLOT
            elements SOLID
            RETURN
        ID MATERIALS                    (on)
```

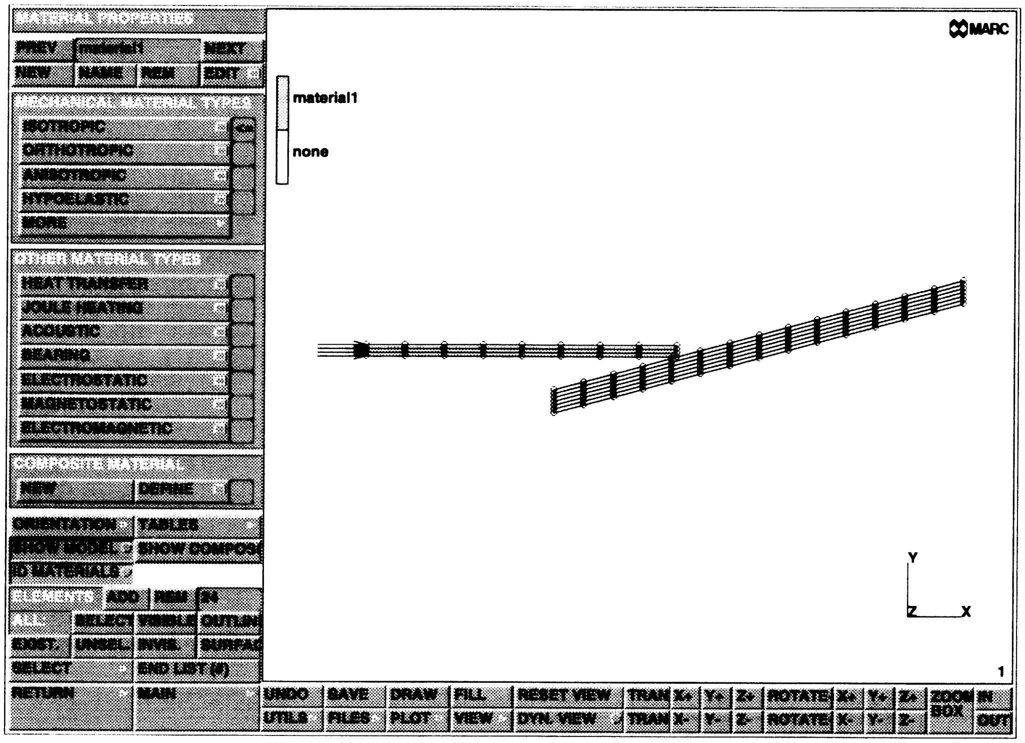


Figure 7.10 Material Properties Applied to All Tube Elements

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

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)*

END LIST (#)

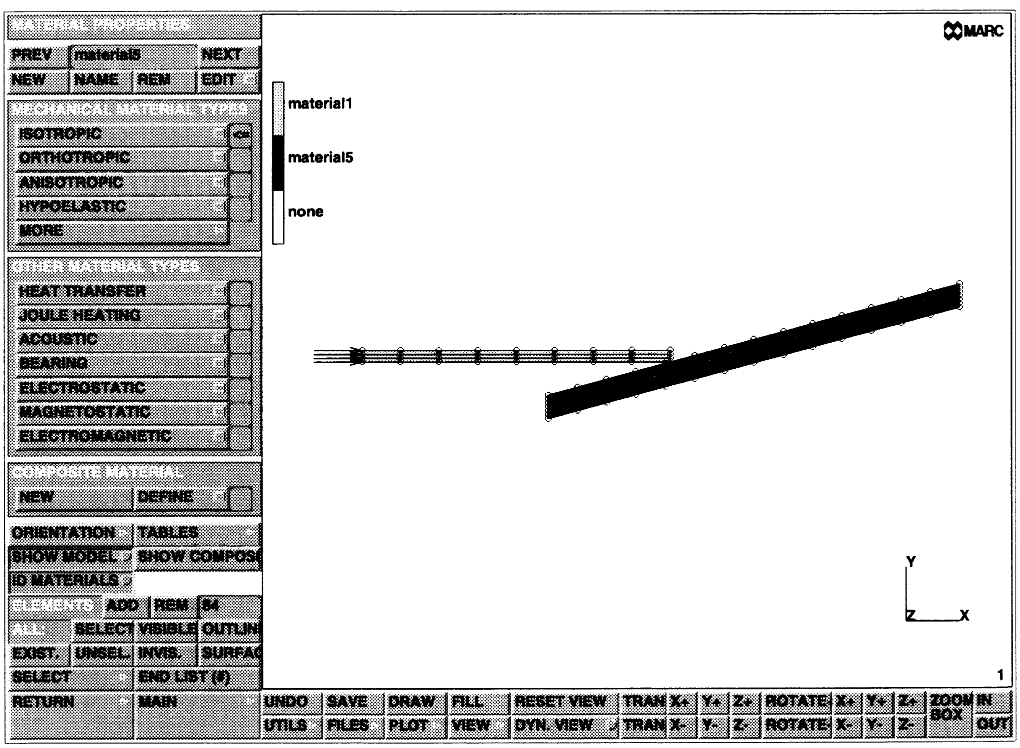


Figure 7.11 Material Properties Applied to All Elements

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
            DEFORMABLE
            NAME
                tube
            elements ADD              (Use the Box Pick Method to
                                          select all tube elements)
                END LIST (#)
            NEW
            DEFORMABLE
            NAME
                tool
            elements ADD              (Use the Polygon Pick Method to
                                          select all tool elements)
                END LIST (#)
```

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)
```

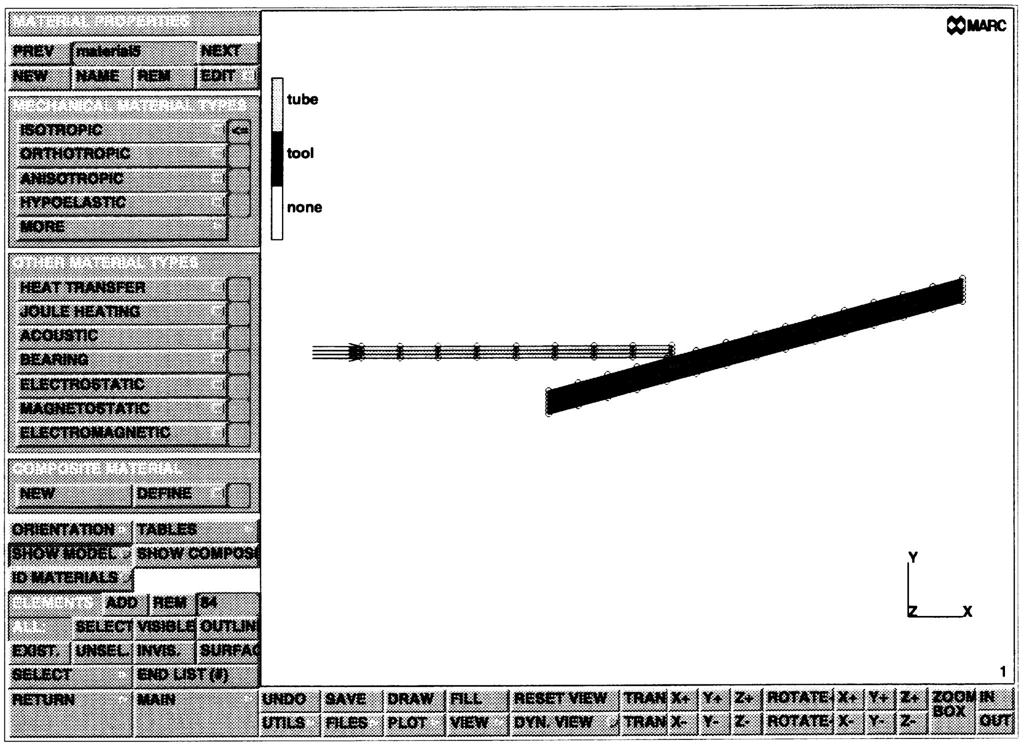


Figure 7.12 Identifying the Tube and Flaring Tool by Color

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

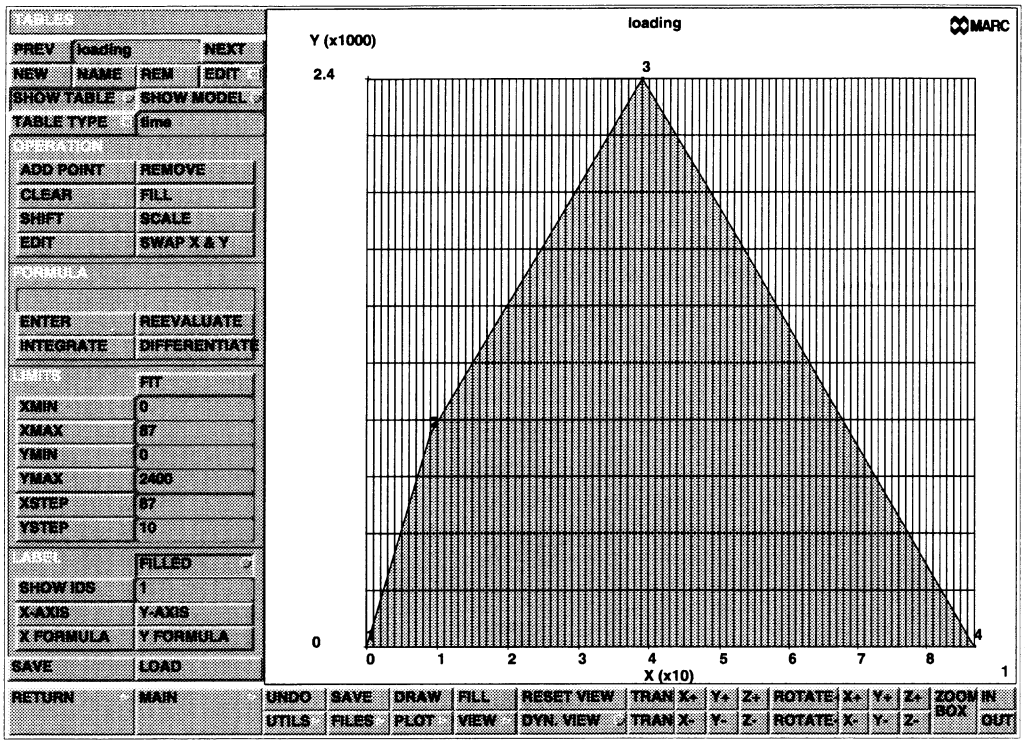


Figure 7.13 Loading of Flaring Tool

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 (#)

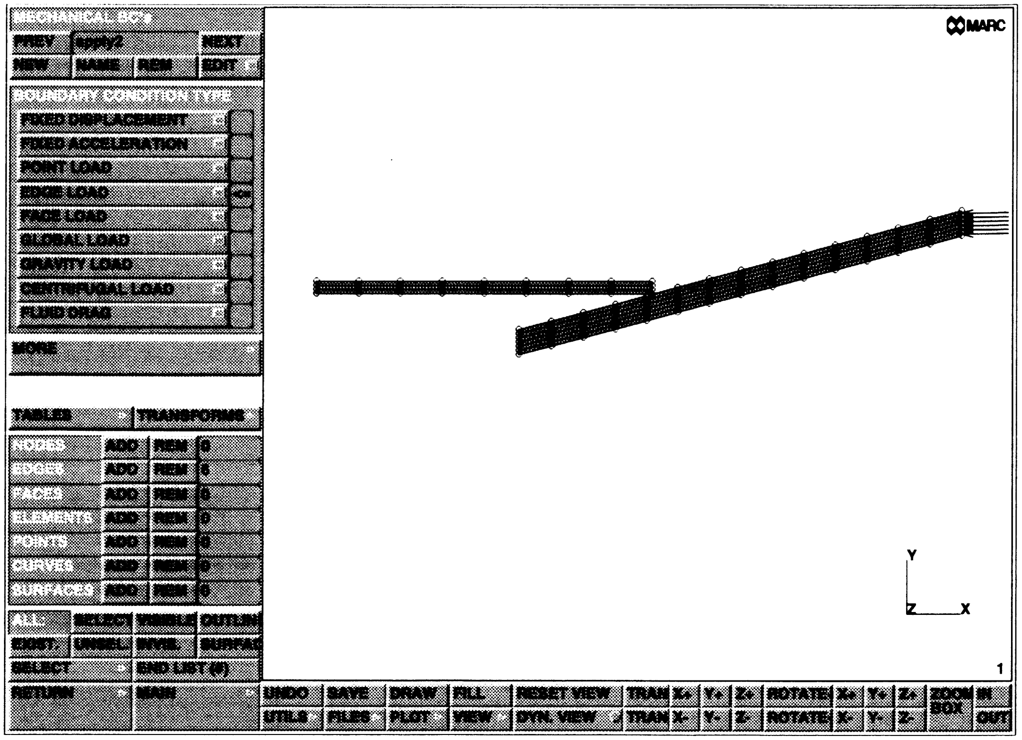


Figure 7.14 Loading Applied to Far Right End of Flaring Tool

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

MAIN
  LOADCASE
    mechanical analyses STATIC
      LOADS *(select all loads)*
        OK
      TOTAL LOADCASE TIME
        `87`
      # STEPS
        `87`
      OK

Step 5 & 6

Once you have defined the loadcase, activate the LARGE DISPLACEMENT, UPDATED LAGRANGE PROCEDURE, and FINITE STRAIN PLASTICITY options, and prepare the job for submission using the following button sequence:

MAIN
  JOBS
    MECHANICAL
      loadcases SELECT
        lcase1
      ANALYSIS OPTIONS
        LARGE DISPLACEMENT *(on)*
        UPDATED LAGRANGE PROCEDURE *(on)*
        FINITE STRAIN PLASTICITY *(on)*
        OK
      JOB RESULTS
        SELECT TENSORS
          stress
          pl_strain
        SELECT VARIABLES
          von_mises
          epl_strain
        OK
      AXISYMMETRIC
      OK
    CHECK
    SAVE
    RUN
      SUBMIT 1
      MONITOR

Step 7

The final phase of the analysis cycle (shown in Figure 1.1 of Chapter 1) is post-processing. Postprocessing involves viewing and evaluating the results of an analysis.

In order to evaluate analysis results with Mentat II, 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:

```
MAIN
    RESULTS
        OPEN DEFAULT
        FILL
```

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

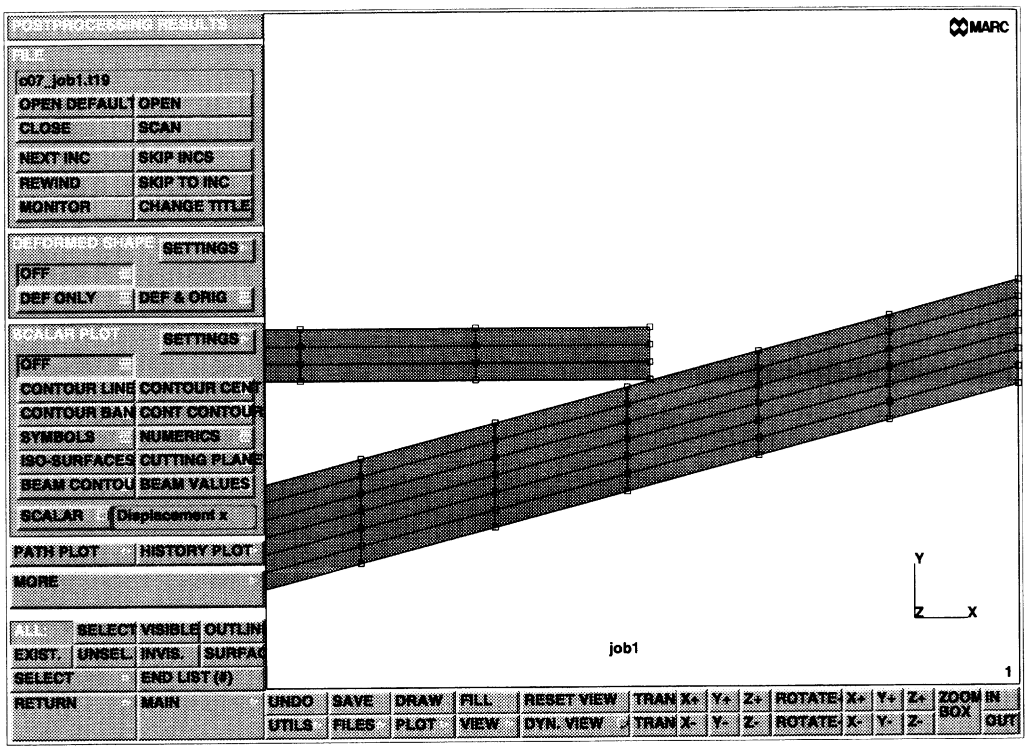


Figure 7.15 Close up of Contact Area to Better View Tip Deflection of Tube

The objective of the analysis, stated in Section 7.1.3, 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 *(from the NODES panel)*
                    Increment
                    Displacement y
                FIT

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.

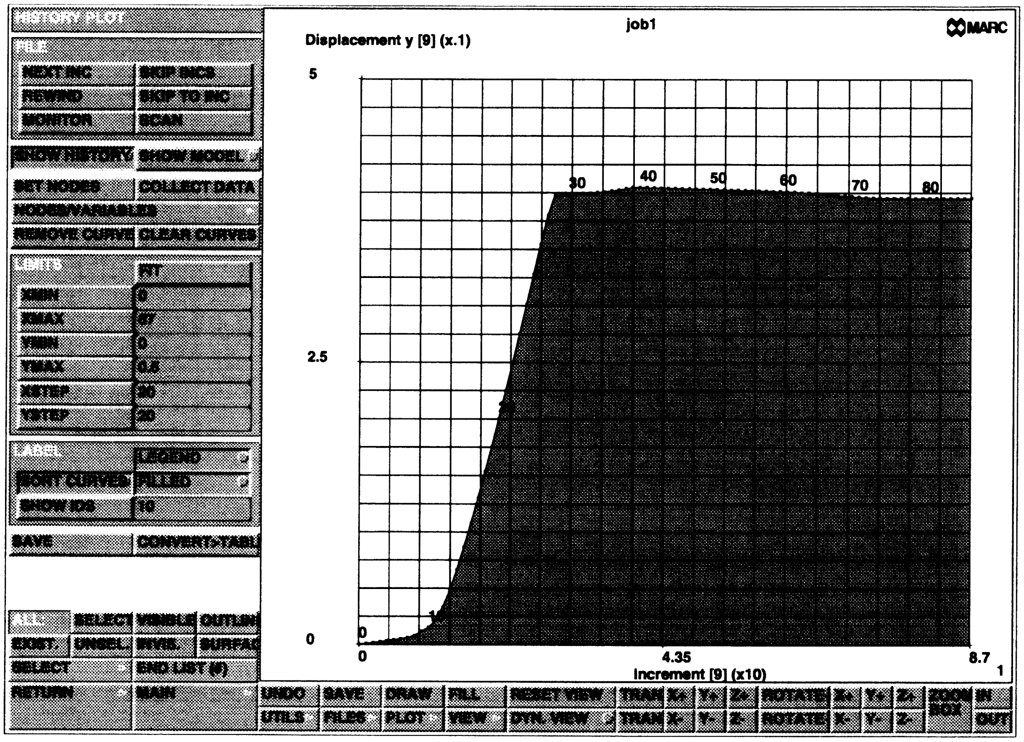


Figure 7.16 History Plot of Node 9 Over 87 Increments

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 to change the plot settings so that only the outline edges of the model are displayed.

MAIN
PLOT
MORE
OUTLINE
PREVIOUS
draw element FACES *(off)*
draw NODES *(off)*
REGENERATE
FILL

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
INCREMENTS
`100 1`
ANIMATION
PLAY

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.

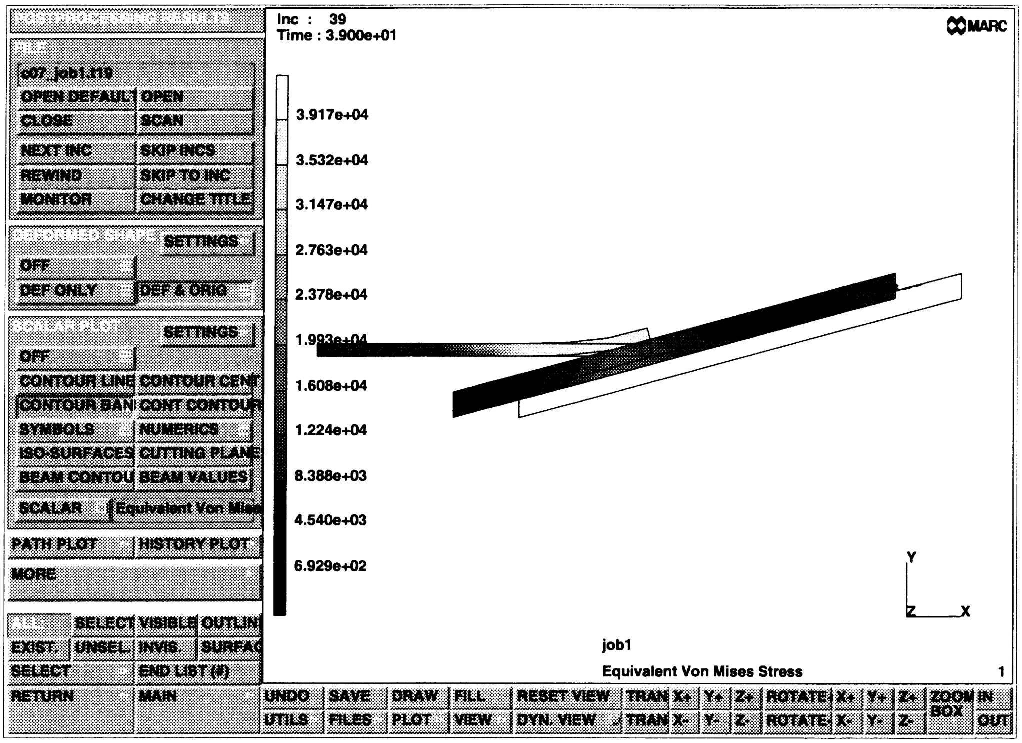


Figure 7.17 Plot of Original & Deformed Tube Showing von Mises Stresses at Increment 39

Next you can check the model for plastic strain. Since you have already specified the increment and have contour bands selected, you only need to click on SCALAR and 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.

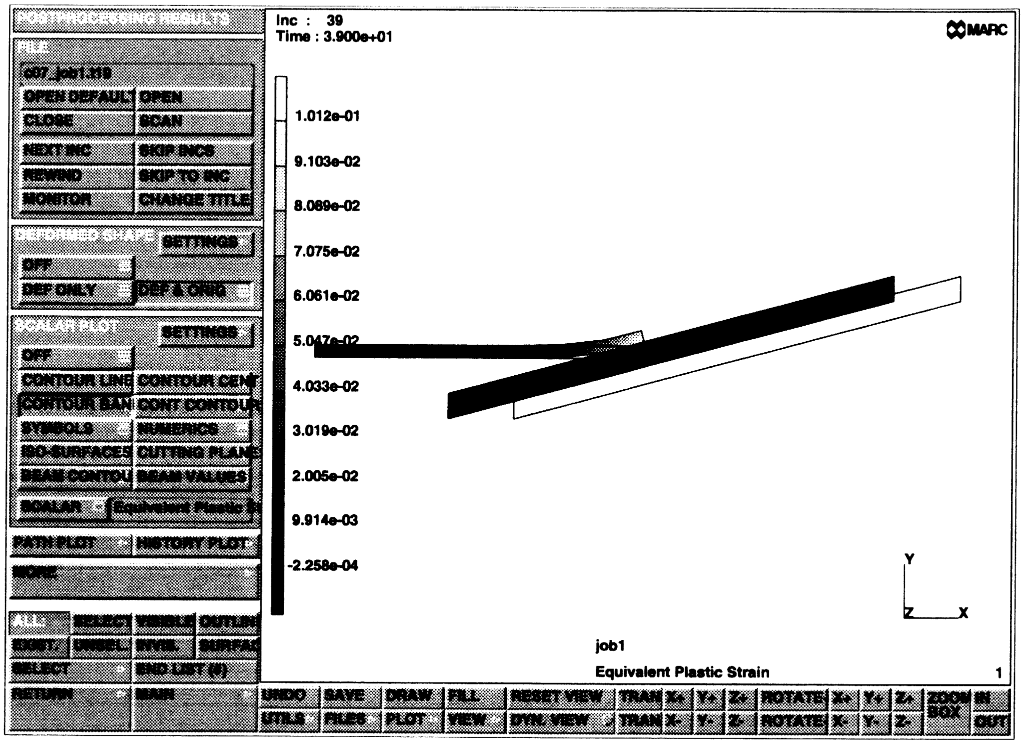


Figure 7.18 Plot of Original & Deformed Tube Showing Plastic Strain at Increment 39

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.

7.3 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.

7.4 Procedure File

```
| Version : MENTAT II 2.3
|
| 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.
|
| Step 1
|
| Create a 8 x 5 grid with dx=1, dy=.5
|
*set_grid_spacing
1 0.5
*set_grid_size
8 5
*set_grid on
*fill_view
*set_point_labels on
|
| Create a surface corresponding to the tube
| length=8.0, inner diam=4.0 outer diam=4.3
|
*add_surfaces
point(0,4,0)
point(8,4,0)
point(8,4.5,0)
point(0,4.5,0)
*set_move_translations
0 -0.2 0
*move_points
 4
 3
# | End of List
|
| Create a surface corresponding to the conical tool
| and translate it into position
|
*system_cylindrical
*add_points
5 15 0
16 15 0
 *set_grid off
*fill_view
*system_spherical
*system_rectangular
```

```
*set_duplicate_translations
0 0.6 0
*duplicate_points
 5
 6
# | End of List
*add_surfaces
 5
 6
 8
 7
*set_move_translations
0 1.25 0
*move_surfaces
 2
# | End of List
|
| Mesh the tube 8 x 3 elements
| Mesh the tool 14 x 6 elements
|
*set_convert_divisions
8 3
*convert_surfaces
 1
# | End of List
*set_convert_divisions
14 6
*convert_surfaces
 2
# | End of List
 *set_points off
 *set_point_labels off
 *set_surfaces off
*regen
|
| Select the MARC element type
|
*element_type 10
all_existing
|
| Fix the far left side of the tube in axial direction
|
*apply_type fixed_displacement
*apply_dof x
*add_apply_nodes
 1 10 19 28
# | End of List
|
| Create an artificial connection between the tube and
| the tool by means of a spring
```

```
|
*link_class spring
*zoom_box
*zoom_box(1,0.530682,0.522843,0.555682,0.496193)
*link_value stiffness
10e3
*spring_node 0
131
*spring_dof 0
1
*spring_node 1
9
*spring_dof 1
1
|
| Step 2
|
| Define and assign material properties to both
| tube and tool
|
*fill_view
*material_type mechanical:isotropic
*material_value isotropic:youngs_modulus
30e6
*material_value isotropic:poissons_ratio
.3
*material_type plasticity
*material_value plasticity:yield_stress
3.6e4
*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
*elements_solid
*identify_materials
*regen
*new_material
*material_type mechanical:isotropic
*material_value isotropic:youngs_modulus
40e6
*material_value isotropic:poissons_ratio
.3
*material_type plasticity
*material_value plasticity:yield_stress
6.0e4
*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
```

```
| Step 7
|
| Open the post file
|
*post_open_default
*fill_view
*zoom_box
*zoom_box(1,0.409762,0.559645,0.617480,0.451777)
|
| make a history plot of y-displacement of node 9 vs. time
|
*set_history_nodes
 9
# | End of List
*history_collect
0 100 1
*history_add
9
Increment
Displacement y
*history_fit
*set_history_increment_id
10
*set_history_xstep
20
*set_history_ystep
20
*set_history_ymax
.5
|
| generate and replay an animation sequence of
| the deformations
|
*edges_outline
 *set_faces off
 *set_nodes off
*regen
*fill_view
*post_rewind
*set_deformed both
*post_animate_increments
100
1
*animation_play
|
| Generate contour plots of the Mises stress and the
| equivalent plastic strain
|
*post_skip_to
39
```

```
*post_value
Equivalent Von Mises Stress
*post_contour_bands
*post_value
Equivalent Plastic Strain
```

Chapter 8: Container

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, temperature, plastic strain, etc.
- Animating the results of an analysis

8.1 Background Information

8.1.1 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

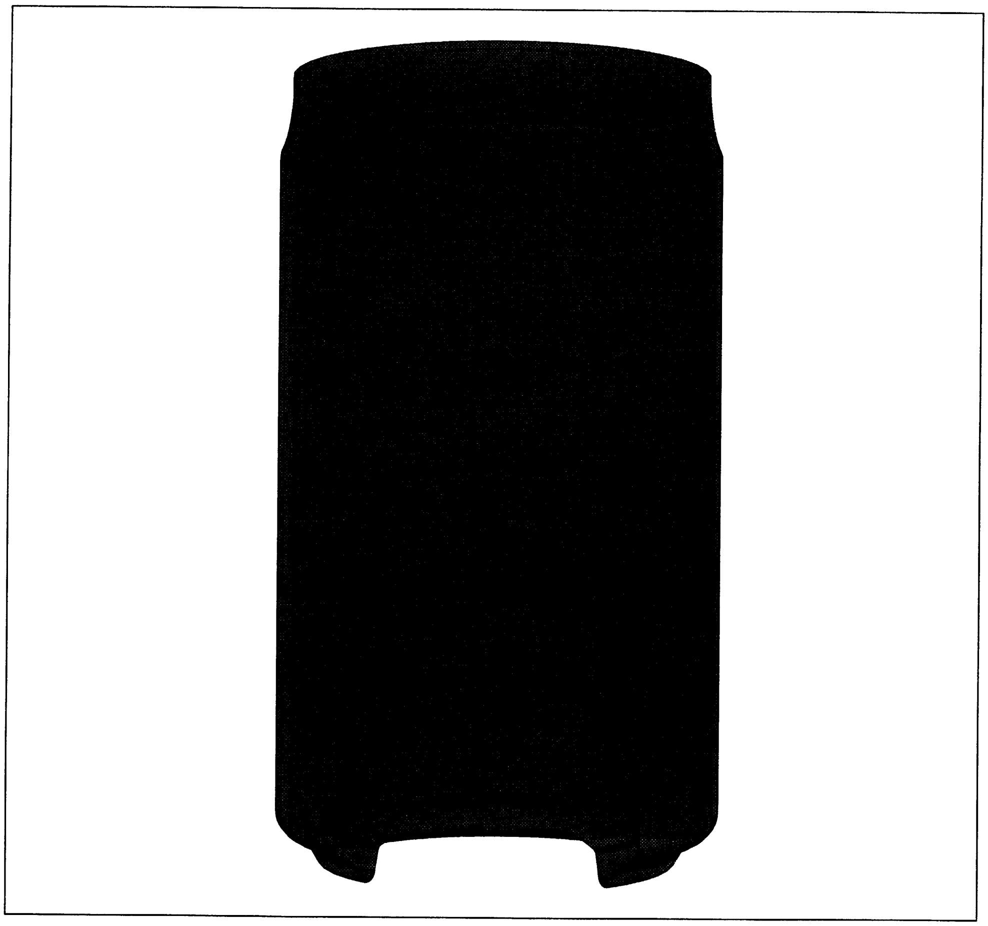


Figure 8.1 Aluminum Container

---

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

8.1.2 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:

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

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.

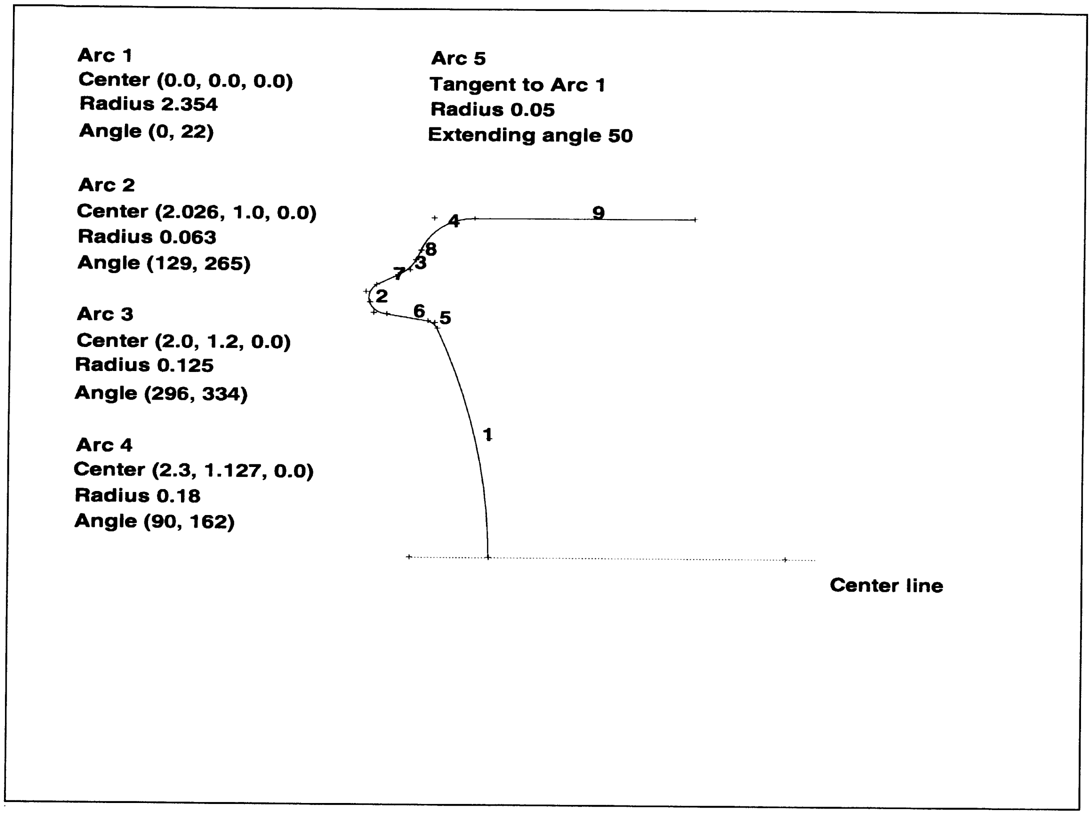


Figure 8.2 Section of Container to be Analyzed

8.1.3 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.

8.1.4 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.

Table 8.1 Stress Strain Data

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

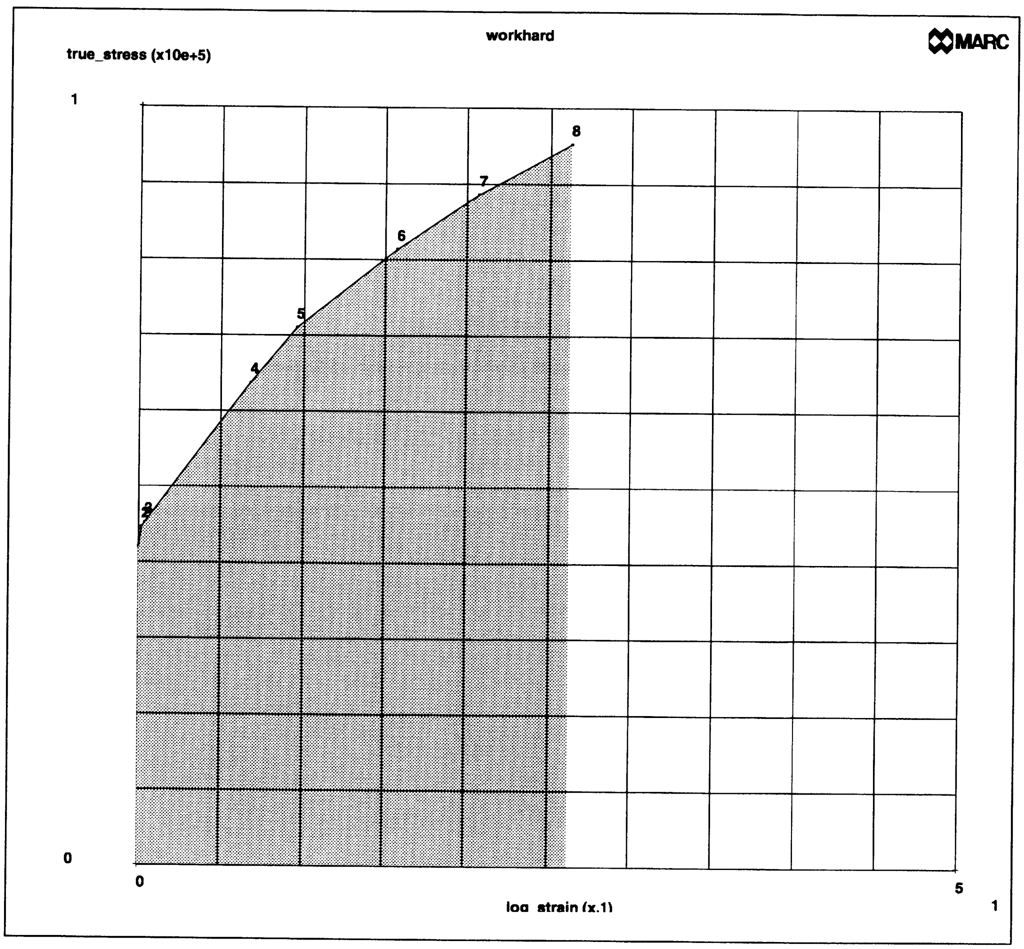


Figure 8.3 Graphical Representation of Cauchy Stress vs. Logarithmic Plastic Strain Data

8.1.5 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 Post-process the results by looking at the deformed shape and the load - displacement curve of the node located on the symmetry axis.

8.2 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 *(on)*
REGENERATE

Figure 8.4 shows the four arcs.

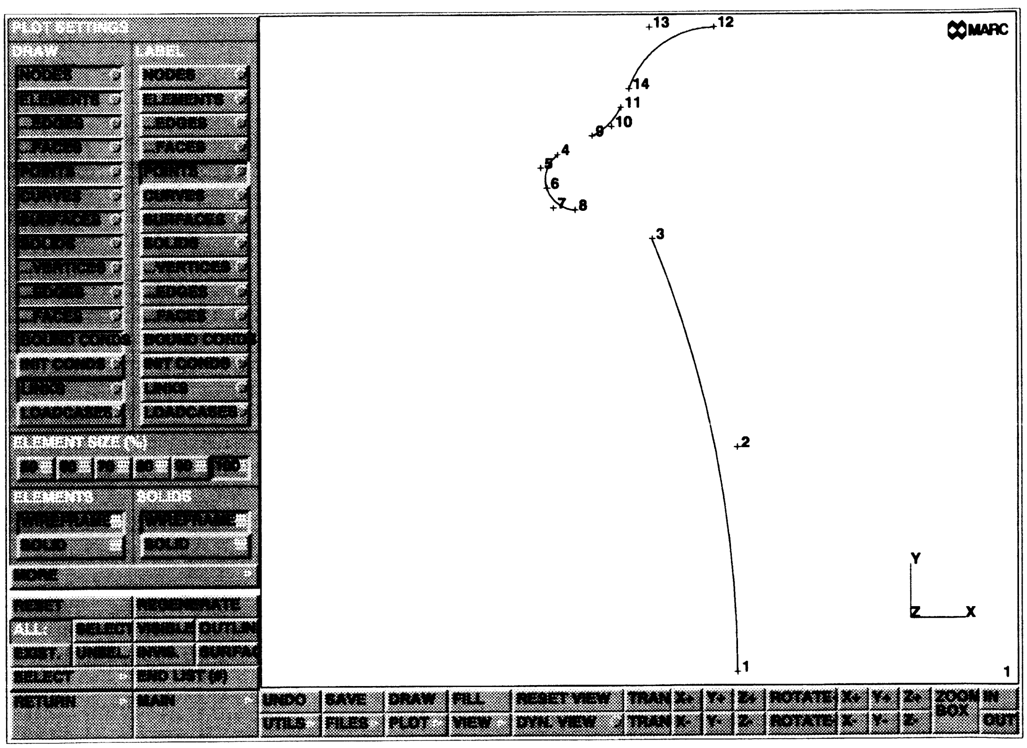


Figure 8.4 Using CRAA Type Arcs to Create First Four Curves

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:

MAIN
    MESH GENERATION
        CURVE TYPE
            TANGENT/RADIUS/ANGLE
                RETURN
            crvs ADD

`3`	*(click end point of arc #1)*
`0.05`	*(radius)*
`50.0`	*(arc angle)*

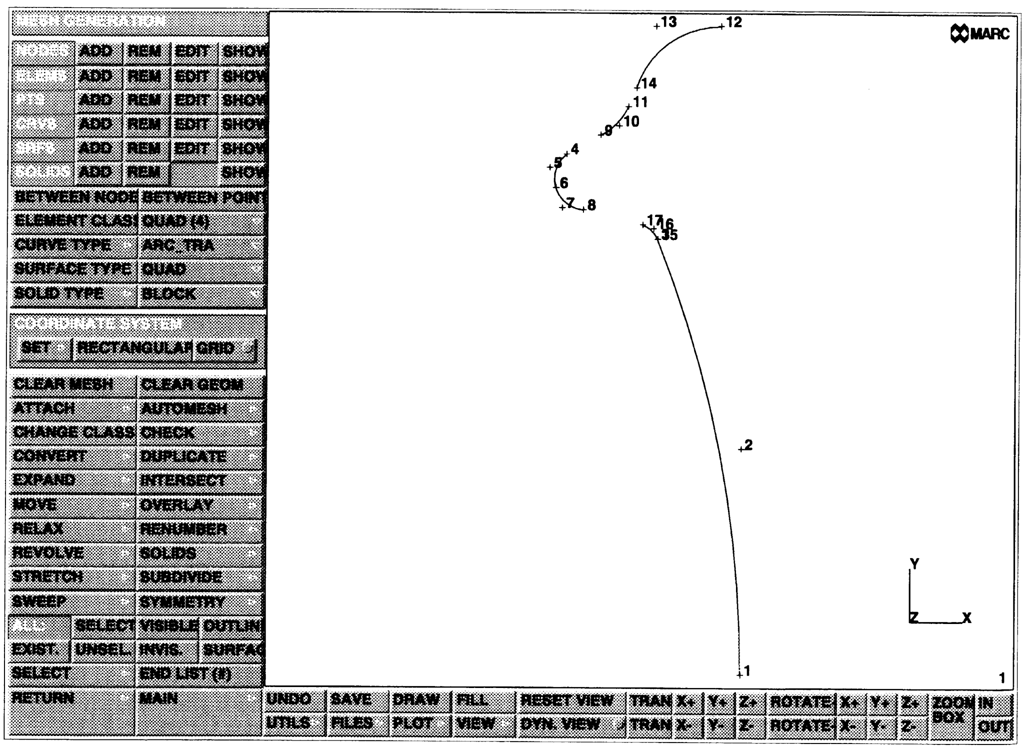


Figure 8.5 Using TRA Type Arc to Create Arc 5

Step 2

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 Section 8.1.2 on Idealization, 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` *(Click on points to connect)*
`4 9` *(Click on points to connect)*
`8 17` *(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)*

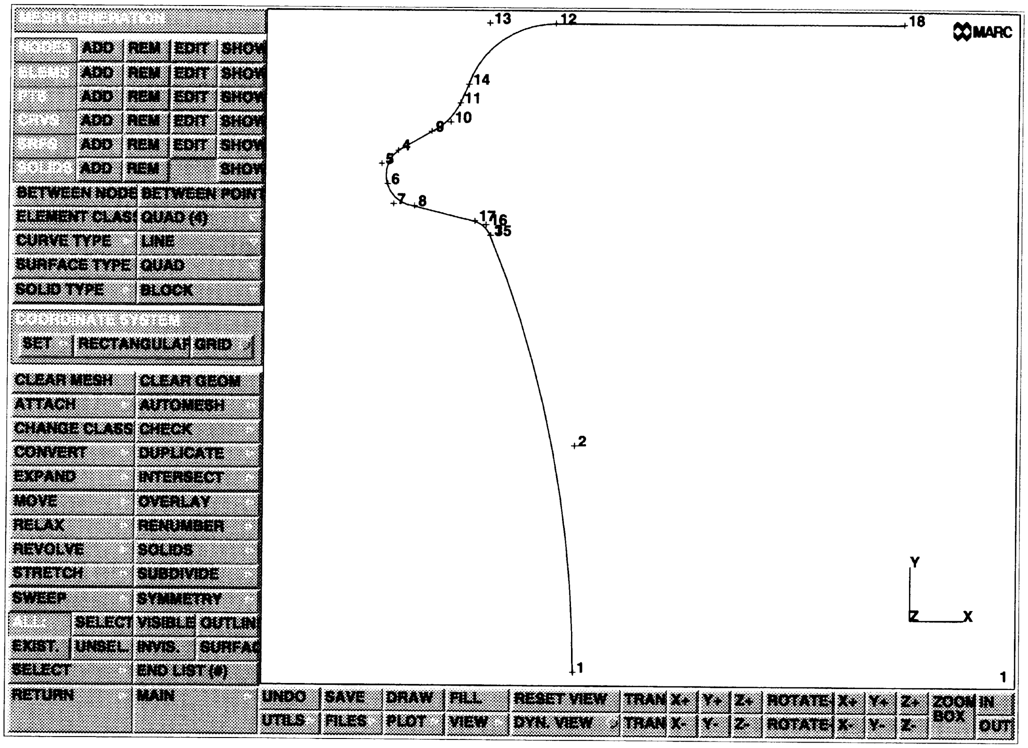


Figure 8.6 Connect the Arcs by Lines

Step 3

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 (#)
```

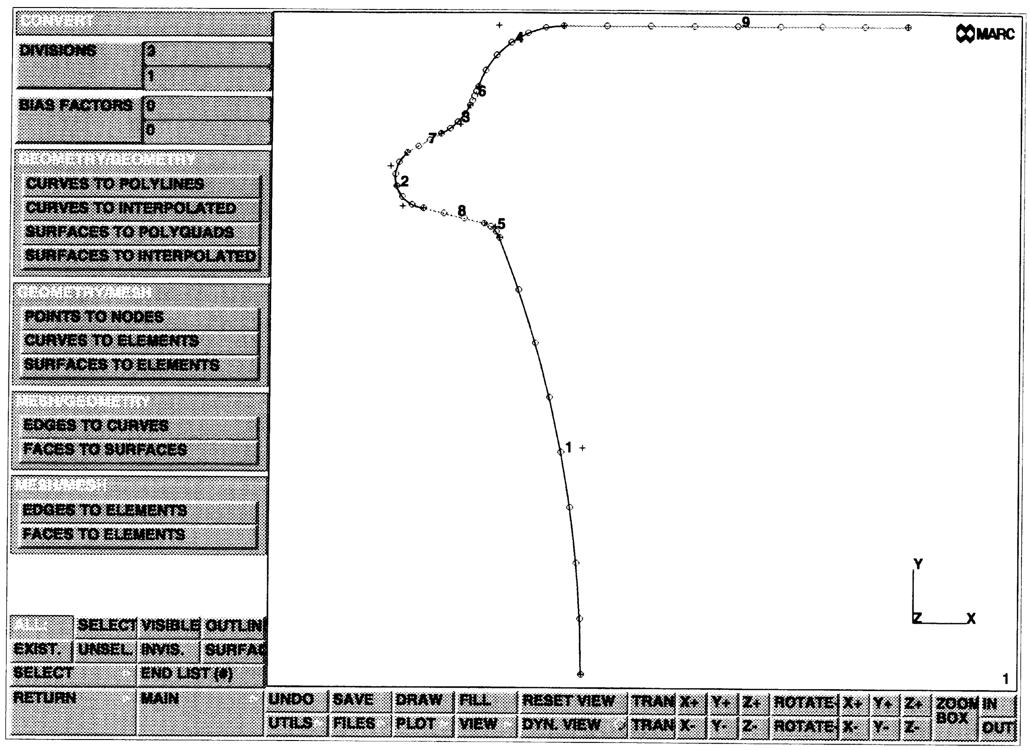


Figure 8.7 Model after Converting Curves to LINE(2) Elements

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.

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 GENERATION
        ATTACH
            attach nodes CURVE

`1`	*(Pick curve)*
`54 55 56 57 58 59 60 61`	*(Pick nodes)* *(these are the mid-side nodes in the definition area of curve #1)*
END LIST (#)	
`5`	*(Pick curve)*
`92 91 90`	*(Pick nodes)*
END LIST (#)	
`2`	*(Pick curve)*
`70 71 72 73 74 75`	*(Pick nodes)*
END LIST (#)	
`3`	*(Pick curve)*
`82 83 84 85`	*(Pick nodes)*
END LIST (#)	
`4`	*(Pick curve)*
`81 80 79 78 77 76`	*(Pick nodes)*
END LIST (#)	

The situation prior to and after the attach operation on curve 2 are shown in Figure 8.8 and Figure 8.9.

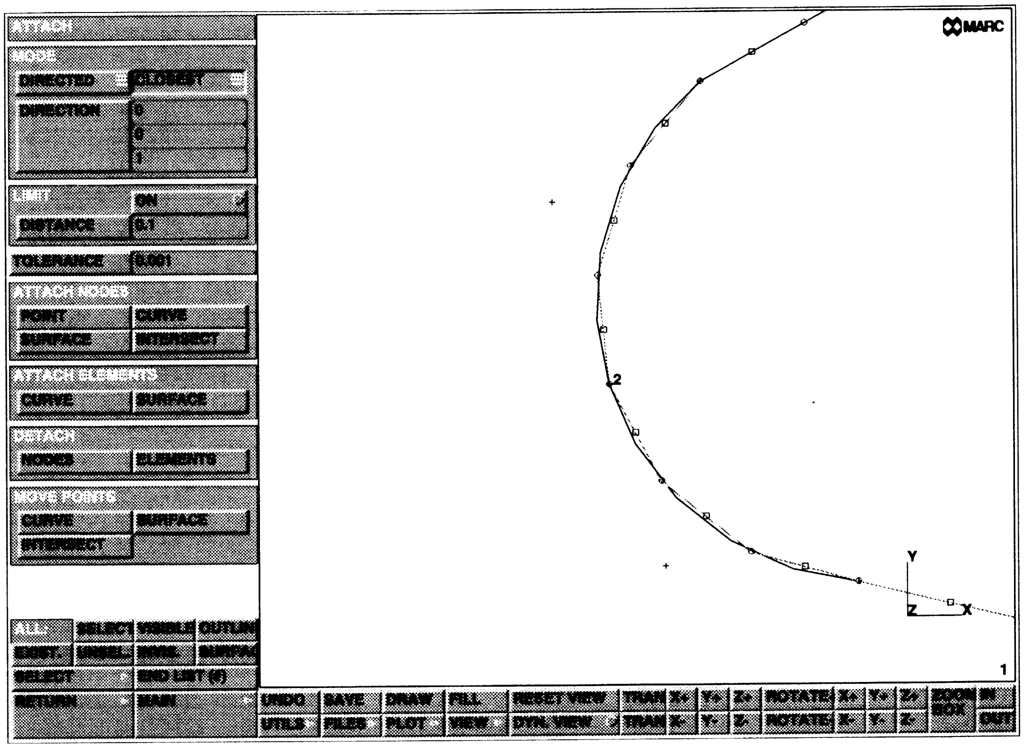


Figure 8.8 Situation Prior to Attach to Curve 2

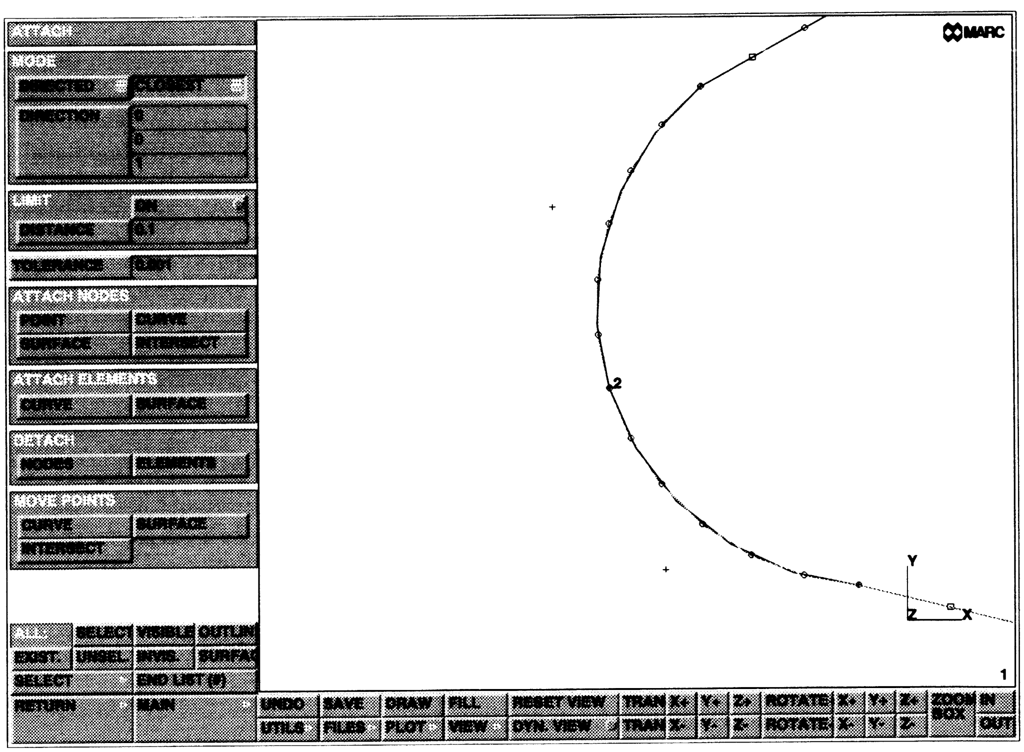


Figure 8.9 Situation After Attach to Curve 2

Step 5

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	*(on)*
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.

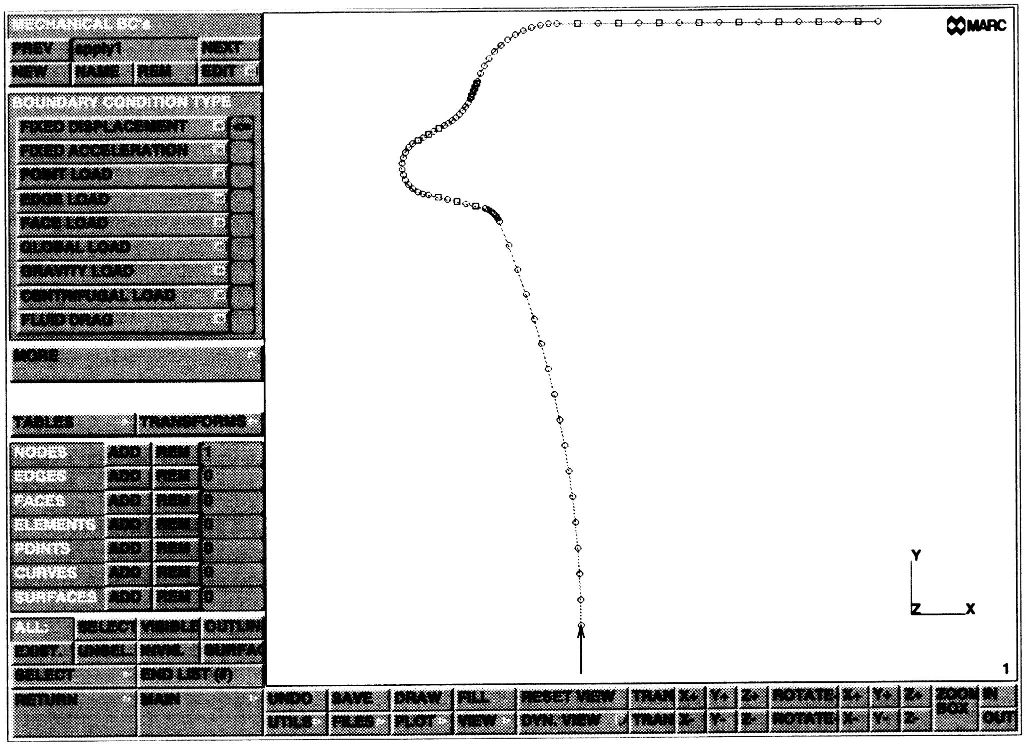


Figure 8.10 Fixed Displacement at Axis of Rotation

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 (#)

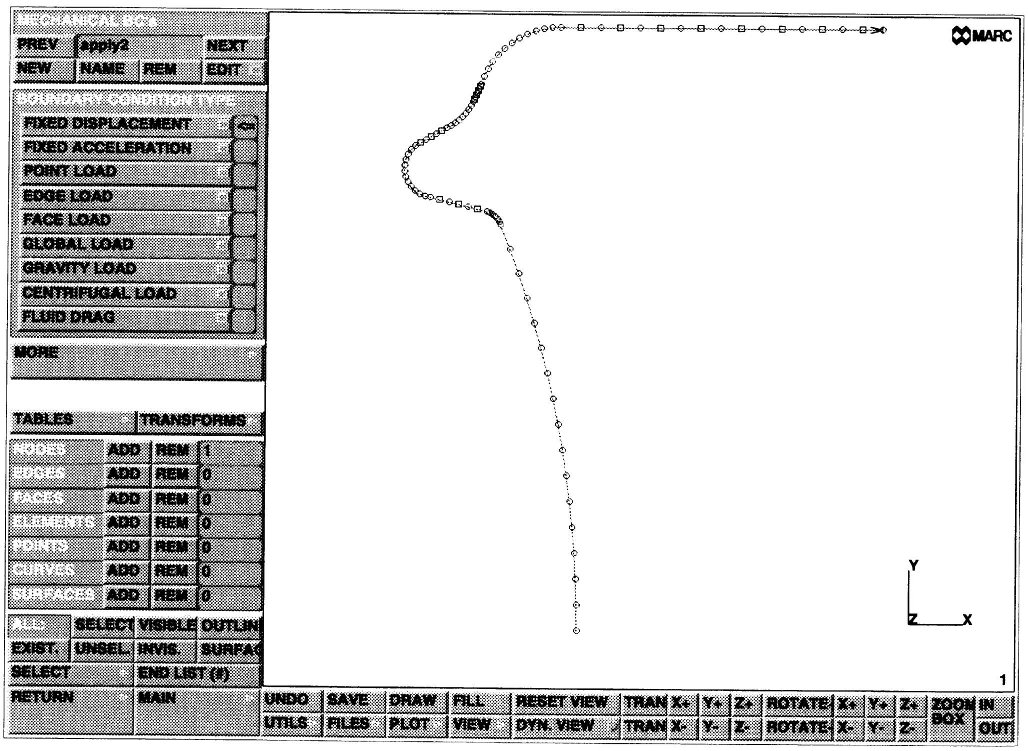


Figure 8.11 Boundary Conditions at the Circumference of Container

Step 6

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
                    850
                NAME
                    loading
                ADD POINT
                    0  0
                    1  850
                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 850 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 850 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 850 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, 850.0) point.

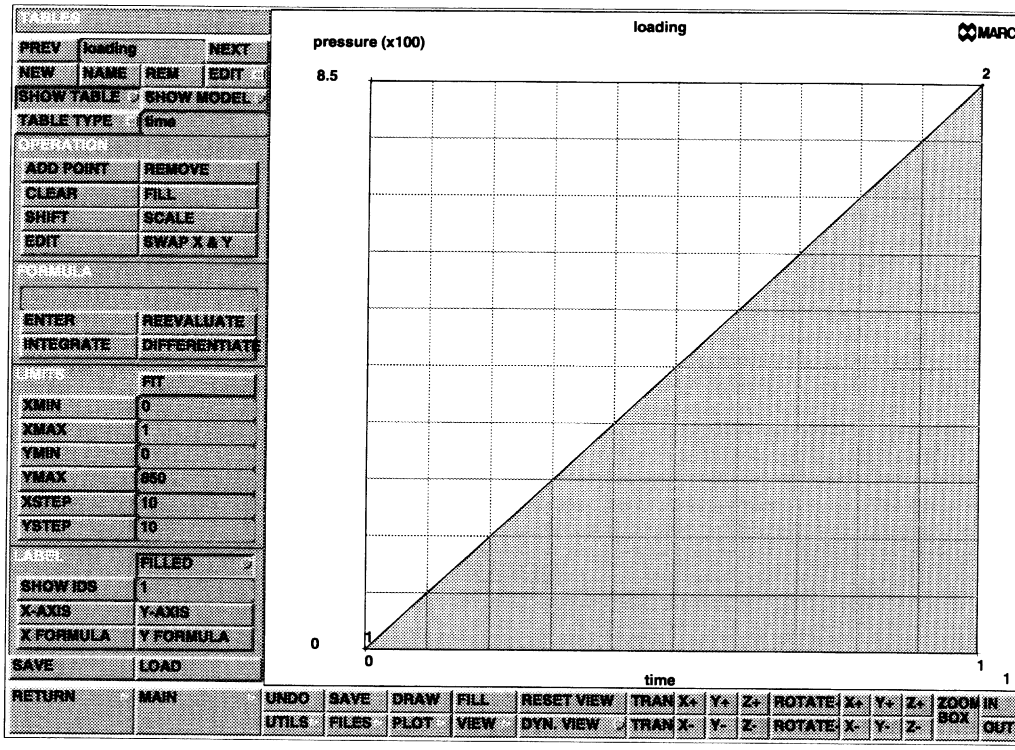


Figure 8.12 Load History Table

As mentioned in Section 8.1.3, 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 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.

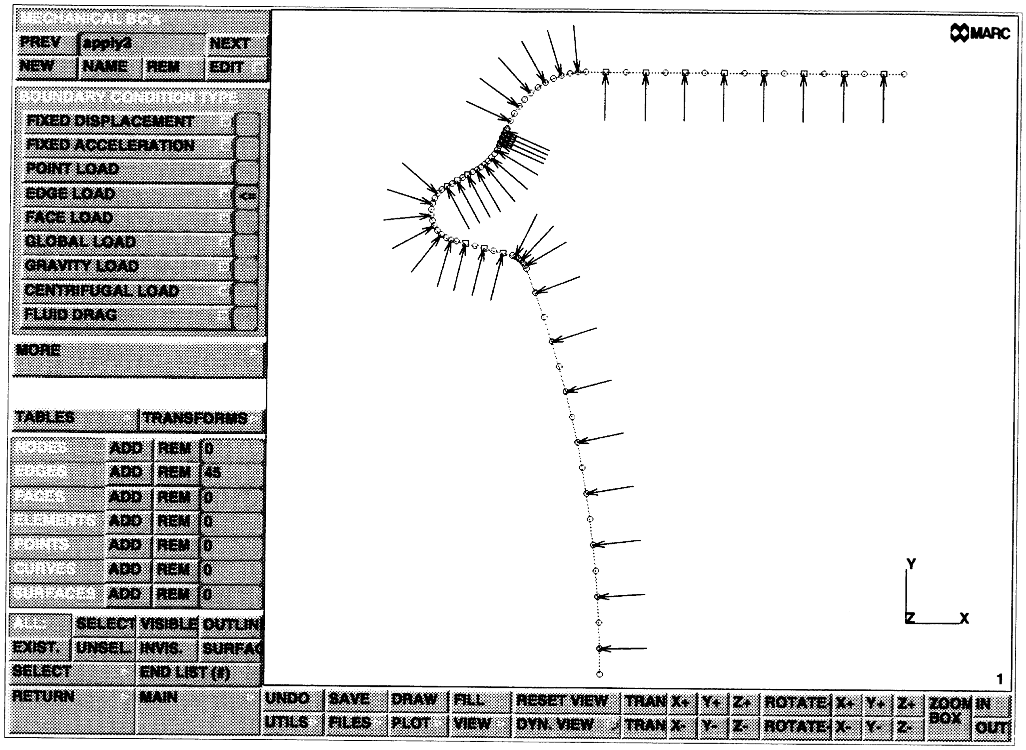


Figure 8.13 Pressure Load Applied

Step 7

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.

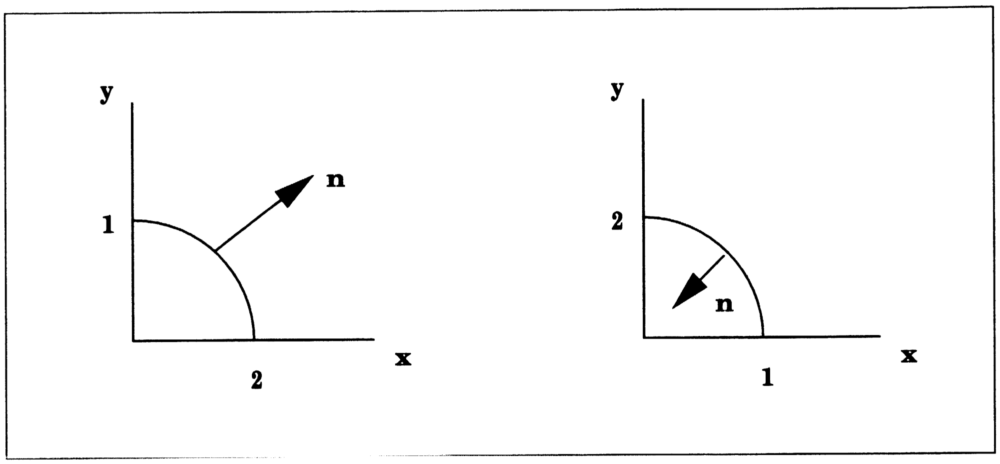


Figure 8.14 The Outward Normal in Arc Definition

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

MAIN
MESH GENERATION
CHECK
FLIP ELEMENTS
(Pick the elements to be flipped)
END LIST (#)

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 **<ML>**. Don't forget to specify end of list by clicking **<MR>** 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.

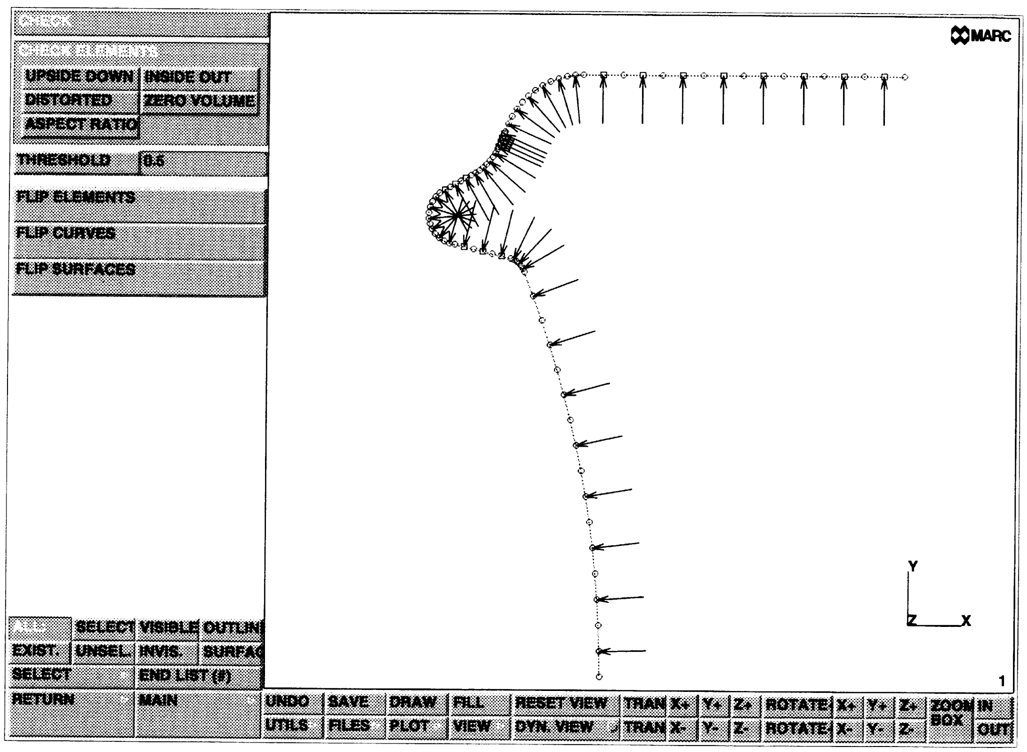


Figure 8.15 Correctly Directed Loads for all Elements

Step 8

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 Displacements + Updated Lagrange	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 *(Refer to plastic strain stress data in Table 8.1 on page 8.6)*
SHOW MODEL
RETURN

Plasticity may occur due to the extreme loading. Click on the PLASTICITY button which shows the plasticity properties panel. Choose the default von Mises yield criteria and the isotropic hardening rule. The initial yield stress is set to 1.

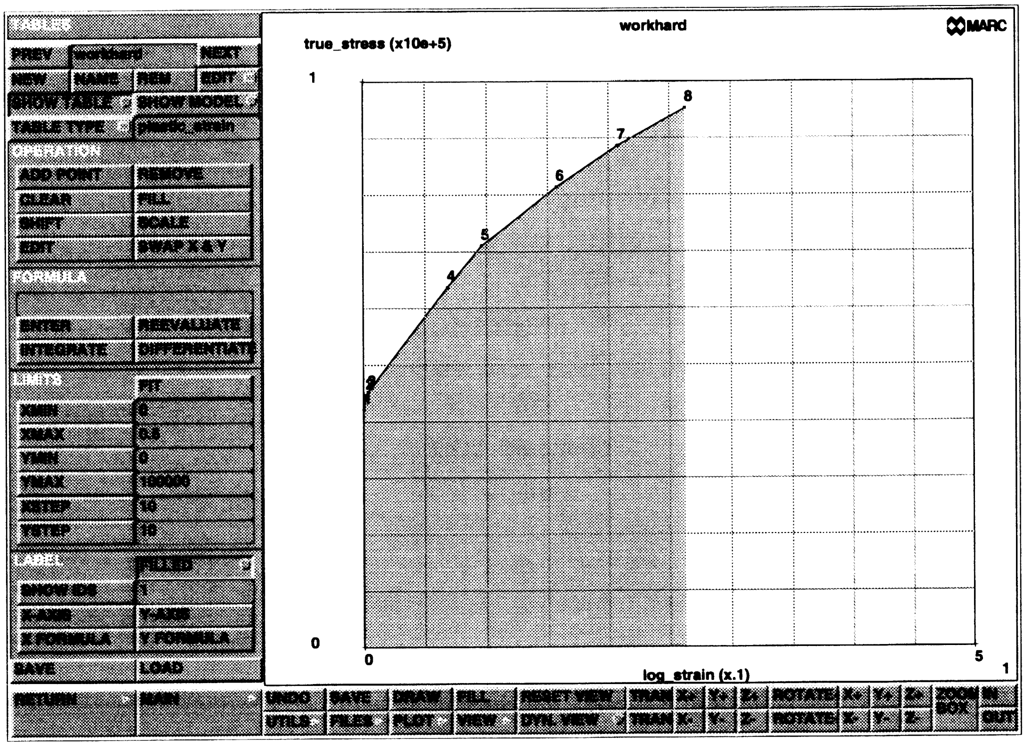


Figure 8.16 Workhardening Slope

MAIN
    MATERIAL PROPERTIES
        ISOTROPIC
            PLASTICITY
                INITIAL YIELD STRESS
                    1
                initial yield stress TABLE 1
                    workhard
                    OK
                OK
            OK
        elements ADD
            all: EXIST.

Step 9

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 *(on)*
                OK

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 one needs to specify that the arc length will never exceed the value used in the first increment.

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
lcase1

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
LARGE DISPLACEMENT *(on)*
UPDATE LAGRANGE PROCEDURE *(on)*
FINITE STRAIN PLASTICITY *(on)*
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
SELECT VARIABLES
von_mises
epl_strain
select variables OUTER&MIDPLANE
von_mises
epl_strain
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 II 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.

Step 12

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
RESET
draw NODES *(off)*
REGENERATE

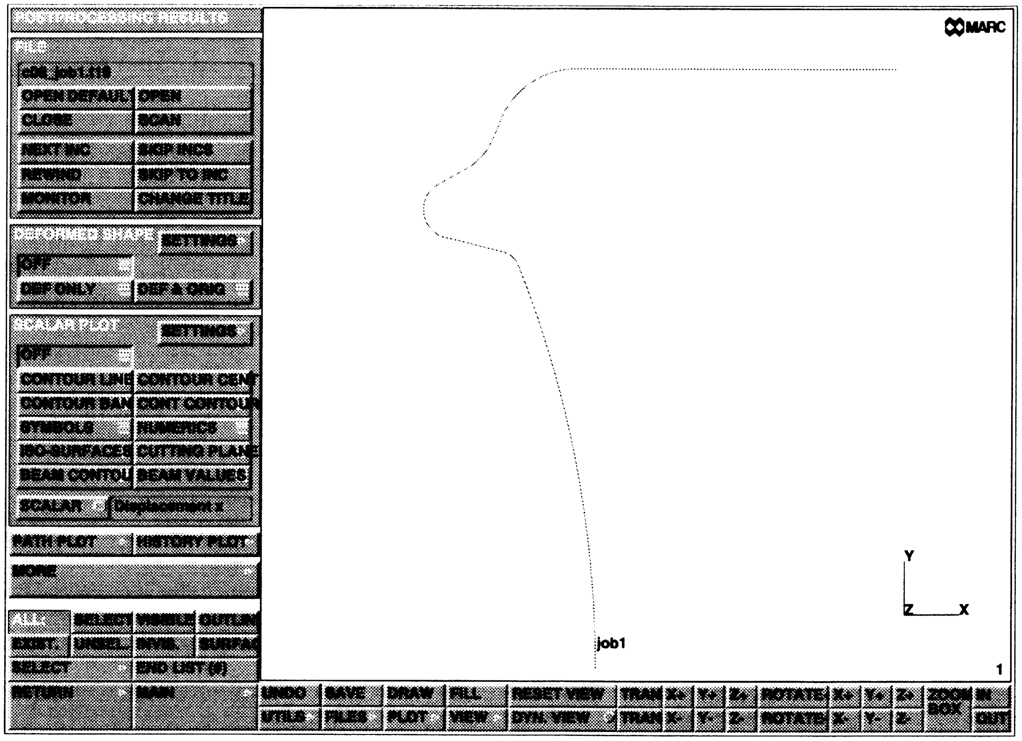


Figure 8.17 Resulting Post File of the Container

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.

MAIN
RESULTS
NEXT INC
DEF & ORIG
MORE
animate INCREMENTS
`25` *(Number of frames to save)*
`10` *(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
PLAY

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`

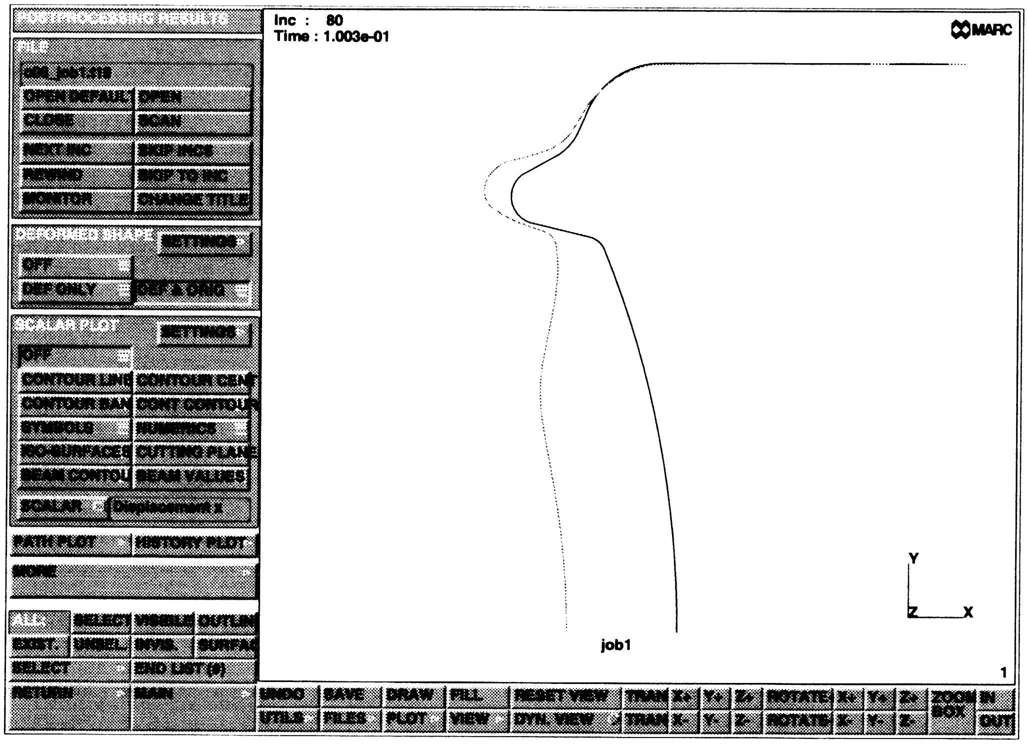


Figure 8.18 Original and Deformed Model at Increment 80

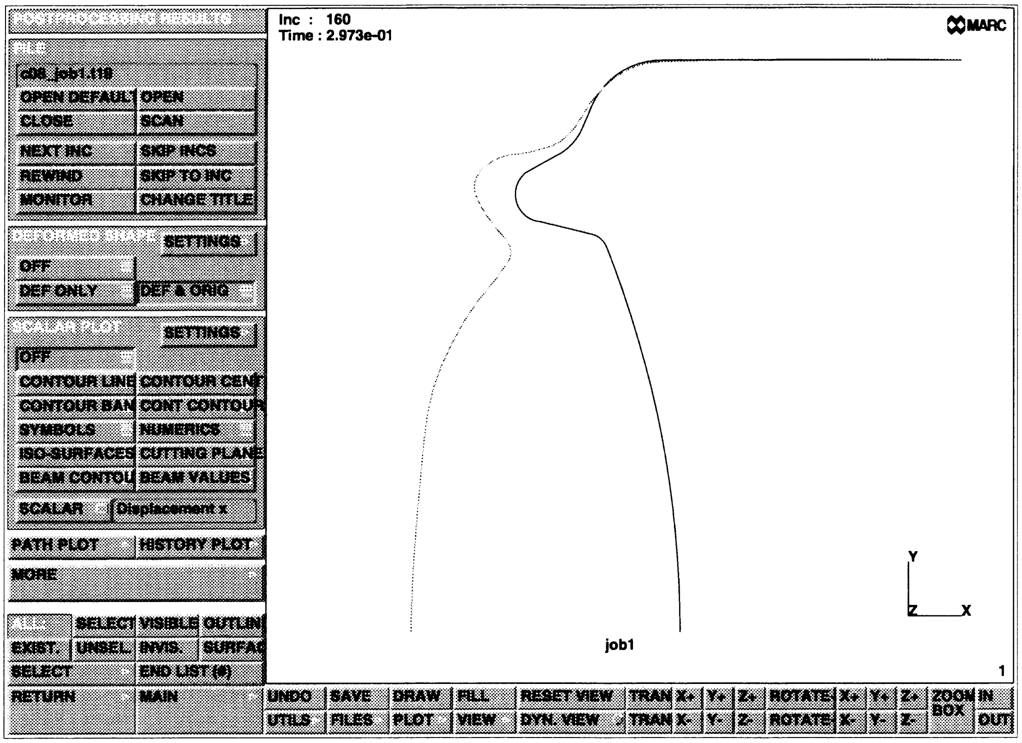


Figure 8.19 Original and Deformed Model at Increment 160

Figure 8.18 and Figure 8.19 show you that the bottom ridge first un-rolls and is followed by a snap-though of the arch. Ultimately, the shape of the bottom of the can will become spherical as is shown in Figure 8.20.

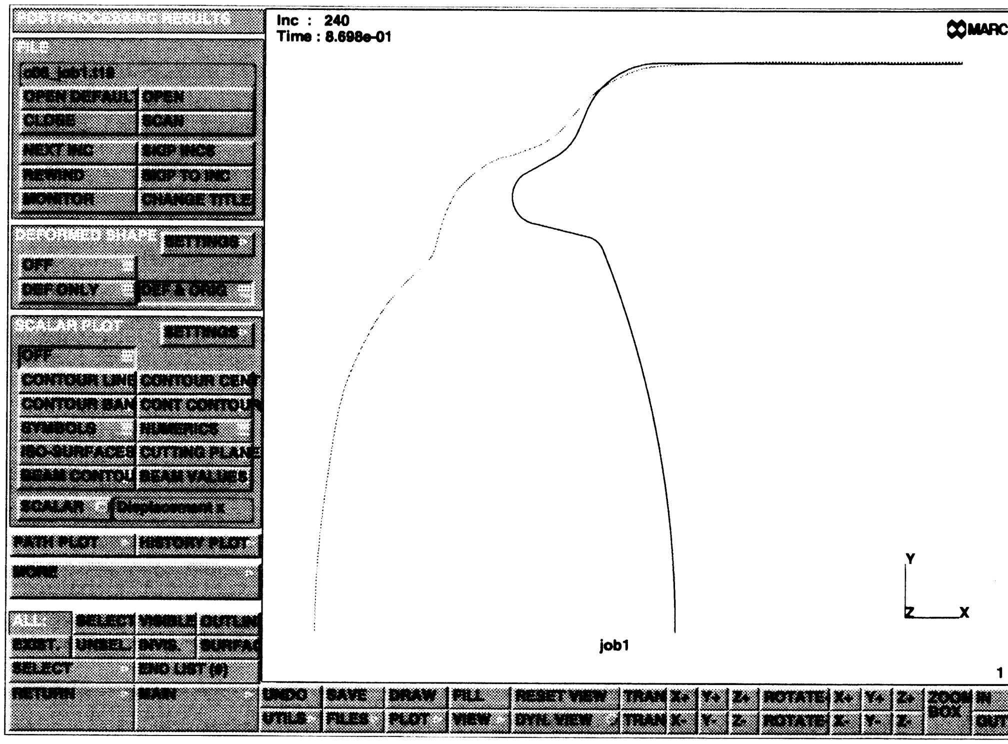


Figure 8.20 Original and Deformed Model at Increment 240

One 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 to plot the equivalent plastic strain.

- 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
          - Equivalent Plastic Strain Layer 3
        - FIT

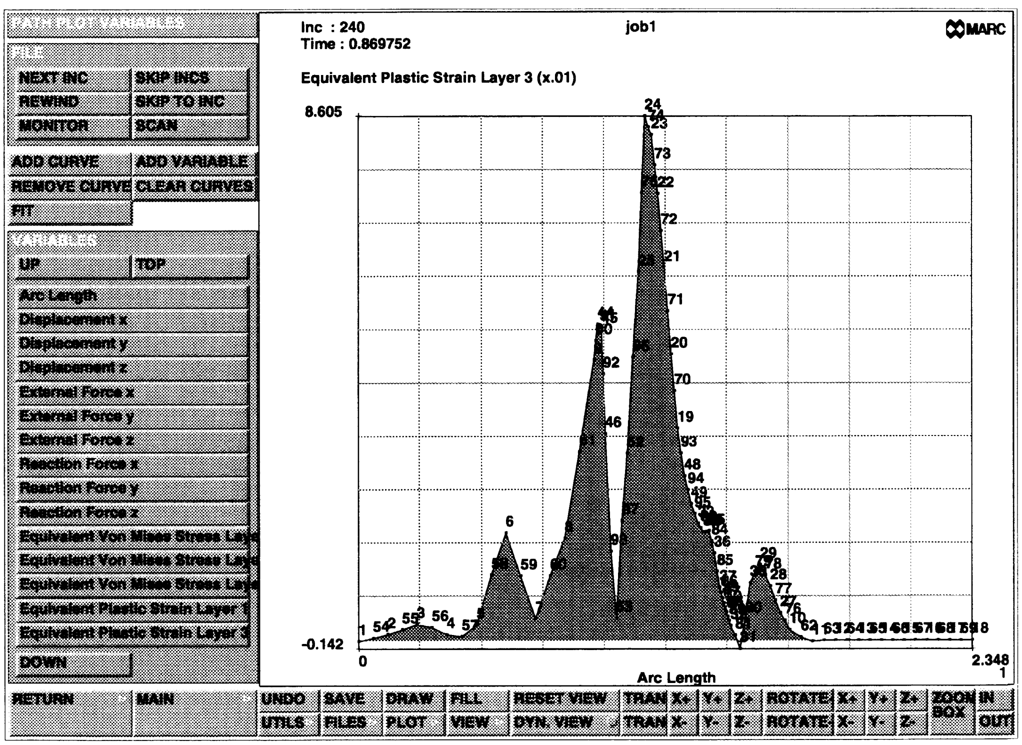


Figure 8.21 Path Plot of Equivalent Plastic Strain

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.

One can use the following button sequence to generate this figure:

```
MAIN
    RESULTS
        HISTORY PLOT
            SHOW MODEL
            SET NODES
                18                                  (upper right)
                1                                   (lower left)
                END LIST (#)
            COLLECT DATA
                0  250  1
            NODES/VARIABLES
                ADD 2-NODE CRV
                    1
                    Displacement x
                    18
                    DOWN
                    Reaction Force x
                FIT
                RETURN
            SHOW IDS
                10
            YMAX
                5000.0
```

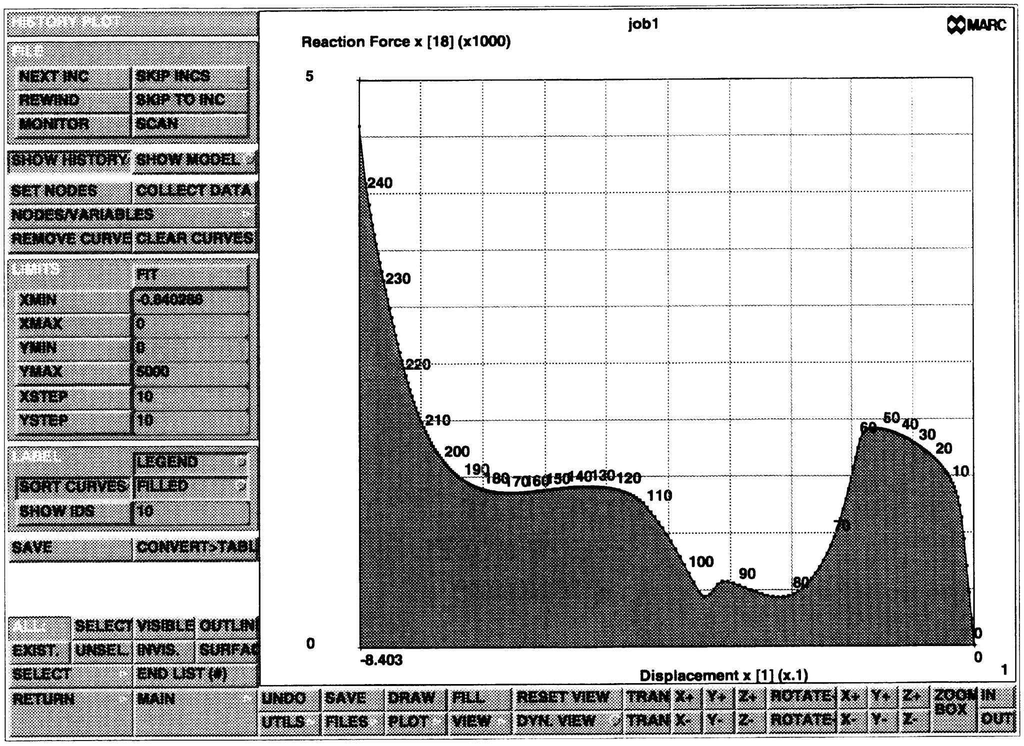


Figure 8.22 Axial Displacement of Bottom versus Reaction Force Diagram

8.3 Conclusion

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

8.4 Procedure File

```
| Version: MENTAT II 2.3
|
| This procedure file demonstrates the modeling of the
| bottom of an aluminum container under internal pressure.
| The particular configuration of this container leads to
| a snap through problem. Accurate modeling of the geometry
| is essential since it dramatically influences the
| snap-through process.
|
*reset
|
| Step 1
|
| Create the arcs as defined in the drawing using
| arc/radius/angle/angle
|
*set_curve_type arc_craa
*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
|
| label the points and switch the generation method for curves
| to arc/tangent/radius/angle
|
*set_point_labels on
*regen
*set_curve_type arc_tra
*add_curves
3
0.05
50.0
|
| Step 2
|
| close the contour using straight line segments and
| extend the cylindrical wall of the can with a line segment
```

```
|
*set_curve_type line
*add_curves
 11
 14
 4
 9
 8
 17
*add_points
3.0 1.307 0
*fill_view
*add_curves
 12
 18
|
| Step 3
|
| generate elements based on the geometric entities
|
 *set_point_labels off
*set_curve_labels on
*regen
*convert_curves
*set_convert_divisions
8 1
*convert_curves
1 9
# | End of List
*set_convert_divisions
6 1
*convert_curves
 4
 2
# | End of List
*set_convert_divisions
4 1
*convert_curves
 6
 3
# | End of List
*set_convert_divisions
3 1
*convert_curves
7 5 8
# | End of List
|
| Step 4
|
| Sweep duplicate nodes, convert line(2) elements
```

```
| to line(3) elements and attach the mid-nodes to
| the curves
|
*sweep_nodes
all_existing
*set_change_class line3
*change_elements
all_existing
*attach_nodes_curve
 1
 54
 55
 56
 57
 58
 59
 60
 61
# | End of List
*zoom_box
*zoom_box(1,0.260227,0.351523,0.335227,0.293147)
*attach_nodes_curve
 5
 92
 91
 90
# | End of List
*fill_view
*zoom_box
*zoom_box(1,0.128409,0.308376,0.206818,0.197970)
*attach_nodes_curve
 2
 70
 71
 72
 73
 74
 75
# | End of List
*fill_view
*zoom_box
*zoom_box(1,0.165909,0.219543,0.392045,0.012690)
*attach_nodes_curve
 3
 82
 83
 84
 85
# | End of List
 4
```

```
 81
 80
 79
 78
 77
 76
# | End of List
*fill_view
|
| Step 5
|
| Add boundary conditions
|
 *set_curve_labels off
 *set_curves off
 *set_points off
*regen
*new_apply
*apply_type fixed_displacement
*apply_dof y
*apply_dof z
*add_apply_nodes
 1
# | End of List
*fill_view
*new_apply
*apply_type fixed_displacement
*apply_dof x
*apply_dof z
*add_apply_nodes
 18
# | End of List
|
| Step 6
|
| Create the load history as a function of time
|
*set_table_type
time
*set_table_xmin
0
*set_table_xmax
1
*set_table_ymin
0
*set_table_ymax
850
*table_name
loading
*table_add
```

```
0 0
1 850
*set_table_xname
time
*set_table_yname
pressure
*new_apply
*apply_type edge_load
*apply_value p
1
*apply_table p0
loading
*add_apply_edges
all_existing
*fill_view
|
| Step 7
|
|
| Correct the definition direction of elements that
| have pressures in opposite direction
|
*flip_elements
 62 63 64 65 66 67 88 89 90
# | End of List
 68 69 70 71 72 73
# | End of List
|
| Step 8
|
| Specify the material properties
|
*material_type mechanical:isotropic
*material_value isotropic:youngs_modulus
11.0e6
*material_value isotropic:poissons_ratio
0.3
*new_table
*set_table_type
plastic_strain
*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
0.1570 81315
0.2070 88560
0.2623 95216
*show_model
*material_type mechanical:isotropic
*material_type plasticity
*material_value plasticity:yield_stress
1
*material_table plasticity:yield_stress0
workhard
*add_material_elements
all_existing
|
| Step 9
|
|
| Specify the thickness of the shell elements
|
*geometry_type mech_axisym_shell
*geometry_value thick
0.0250
*add_geometry_elements
all_existing
|
| Step 10
|
|
| Specify the control values for an auto-increment job
|
*loadcase_type static
*loadcase_option nonpos:on
*loadcase_value force
0.05
*loadcase_option stepping:arclength
*loadcase_value maxinc
600
*loadcase_value initfraction
.05
*loadcase_value maxmultiplier
1.0
```

```
|
| Step 11
|
|
| Define a mechanical analysis and submit the job
|
|
| set the correct MARC element type
|
*element_type 89
all_existing
*job_class mechanical
*job_option dimen:axisym
*add_job_loadcases
lcase1
|
| switch on large disp, finite strain, updated Lagrange and follow force
|
*job_option large:on
*job_option update:on
*job_option finite:on
*job_option follow:on
*job_class mechanical
|
| specify the post-variables
|
*add_post_var
von_mises
epl_strain
*post_var_outer_layers
von_mises
epl_strain
*job_param layers
5
*check_job
*save_as_model
c08
y
*update_job
*submit_job 1
*update_job
*monitor_job
|
| Step 12
|
|
| post processing
|
*post_open_default
*plot_reset
```

```
 *set_nodes off
*regen
|
| generate the animation sequence
|
*post_next
*set_deformed both
*post_animate_increments
25
10
*fill_view
*animation_play
|
| generate deformed shape figures at three load levels
|
*post_skip_to
80
*post_skip_to
160
*post_skip_to
240
|
| Make a graph of Plastic_Strain vs. position for inc 240
|
*set_nodes on
*regen
*set_pathplot_path
 1
 6
 9
 75
 94
 89
 79
 11
 18
# | End of List
*pathplot_add
Arc Length
Equivalent Plastic Strain Layer 3
*pathplot_fit
|
| make a graph of deformation vs. load
|
*show_model
*set_history_nodes
 18
 1
# | End of List
*history_collect
```

```
0
250
1
*history_add_2nodes
1
Displacement x
18
*pick_list_next(history_variable,5)
Reaction Force x
*history_fit
*set_history_increment_id
10
*set_history_ymax
5000
```

Chapter 9: Manhole

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 II capabilities.

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

9.1 Background Information

9.1.1 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.

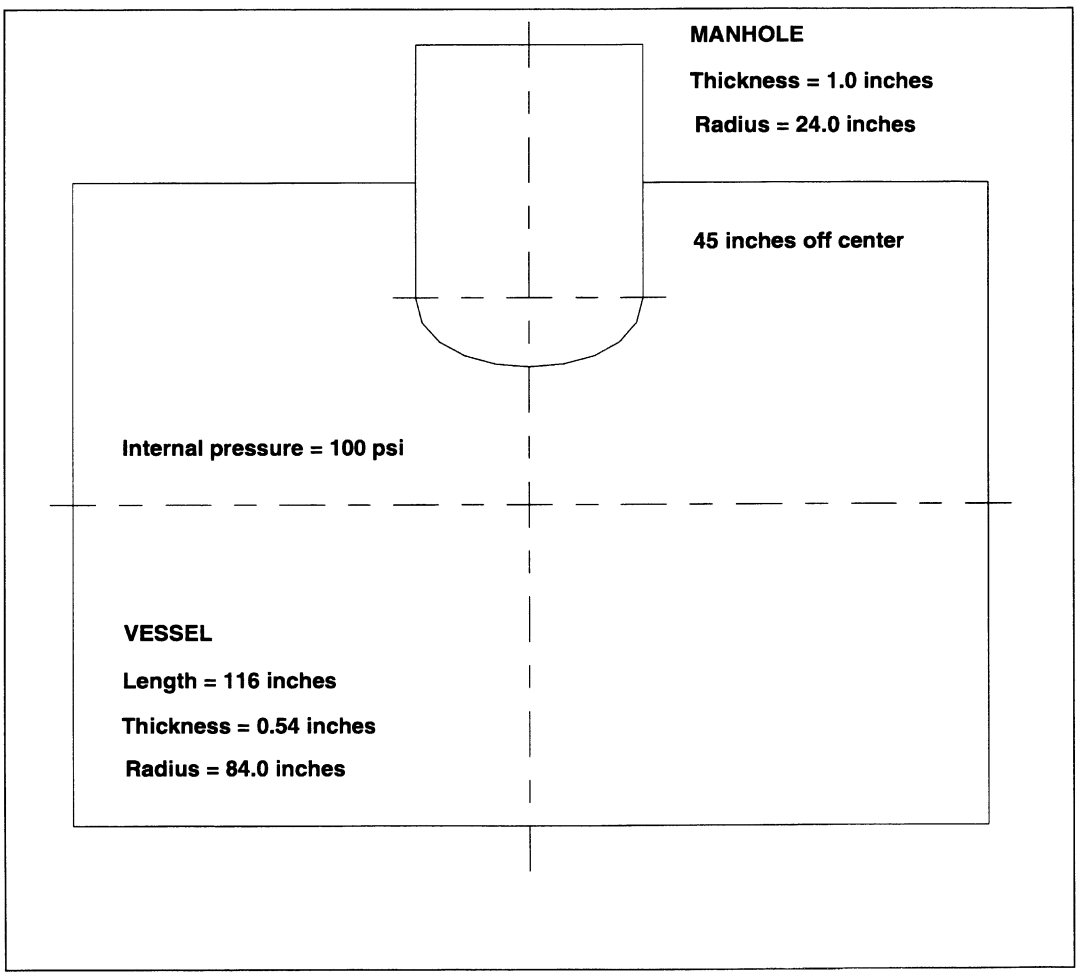


Figure 9.1 Vessel Dimensions

The manhole is positioned 45 inches from the center line as indicated by Figure 9.2.

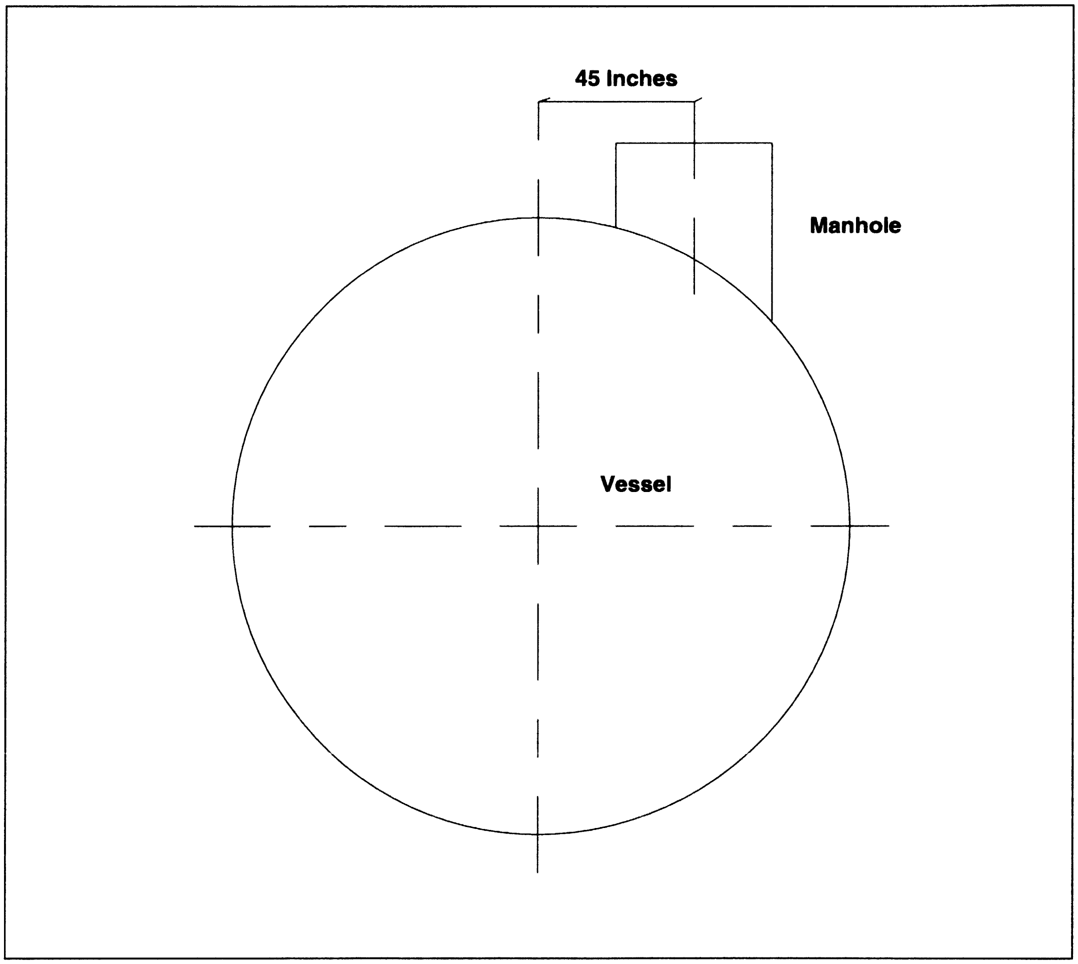


Figure 9.2 Side View of Vessel

9.1.2 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.

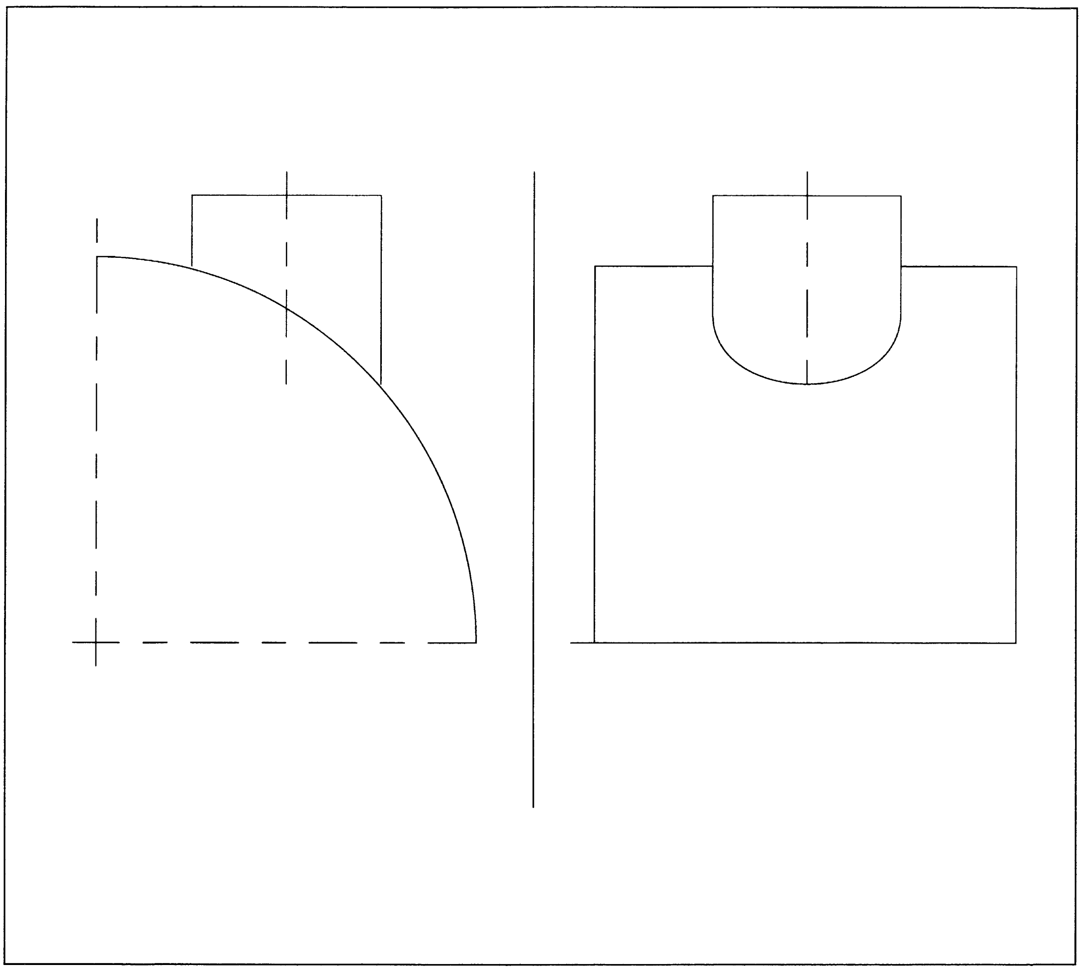


Figure 9.3 Section to be Analyzed

9.1.3 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.

9.1.4 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

9.1.5 Steps

Step 1 Create two cylindrical surfaces: one for the vessel 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.

9.2 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. Section 9.1.2 on Idealization 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 *(on)*
      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+ *(in the static menu next to ROTATE+)*
        FILL

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 **<ML>** with the **<↑>** 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` *(1st point on the axis of the cylinder)*
            `45 120 58` *(2nd point on the axis of the cylinder)*
            `24 24` *(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

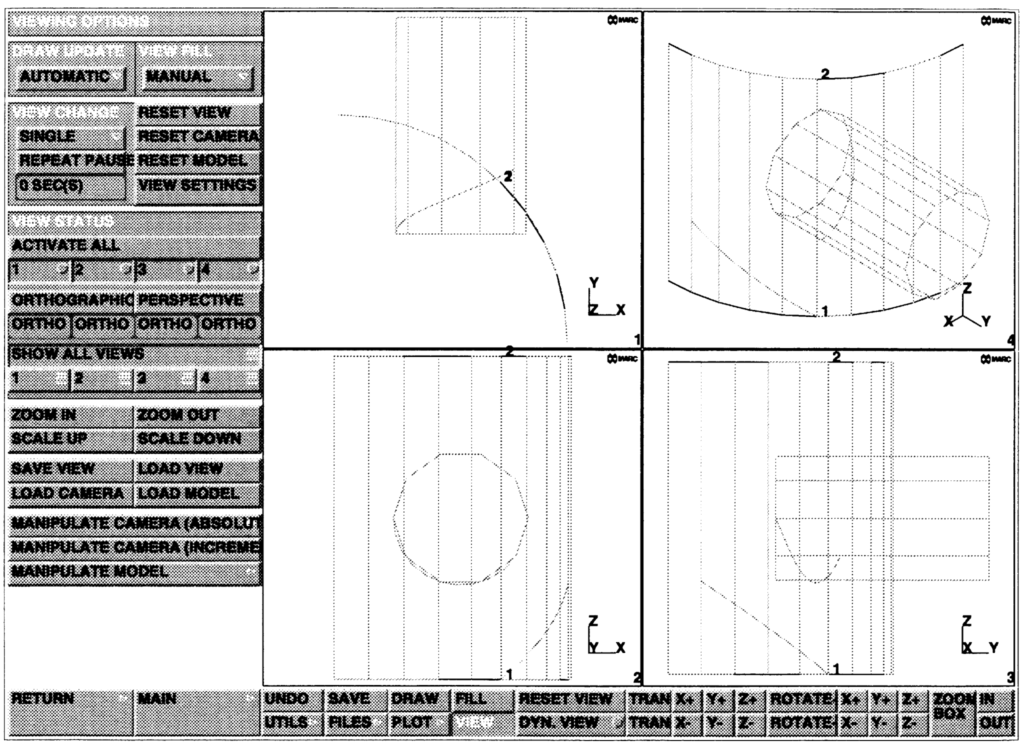


Figure 9.4 Four Views of Completed Model Geometry

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                              (Pick the ruled surface)
                END LIST (#)
```

To get a better overview of the mode, change the view option to 2 and deactivate the face identification option to unclutter the picture. Display curves and surfaces using 30 subdivisions. Figure 9.5 illustrates where the manhole cylinder penetrates the surface of the vessel.

```
PLOT
    draw element FACES                                   (off)
    MORE
        DIVISIONS
            CURVES
                30
            SURFACES
                30
            REGENERATE
            VIEW
                show 2                 (Below SHOW ALL VIEWS)
                RETURN
```

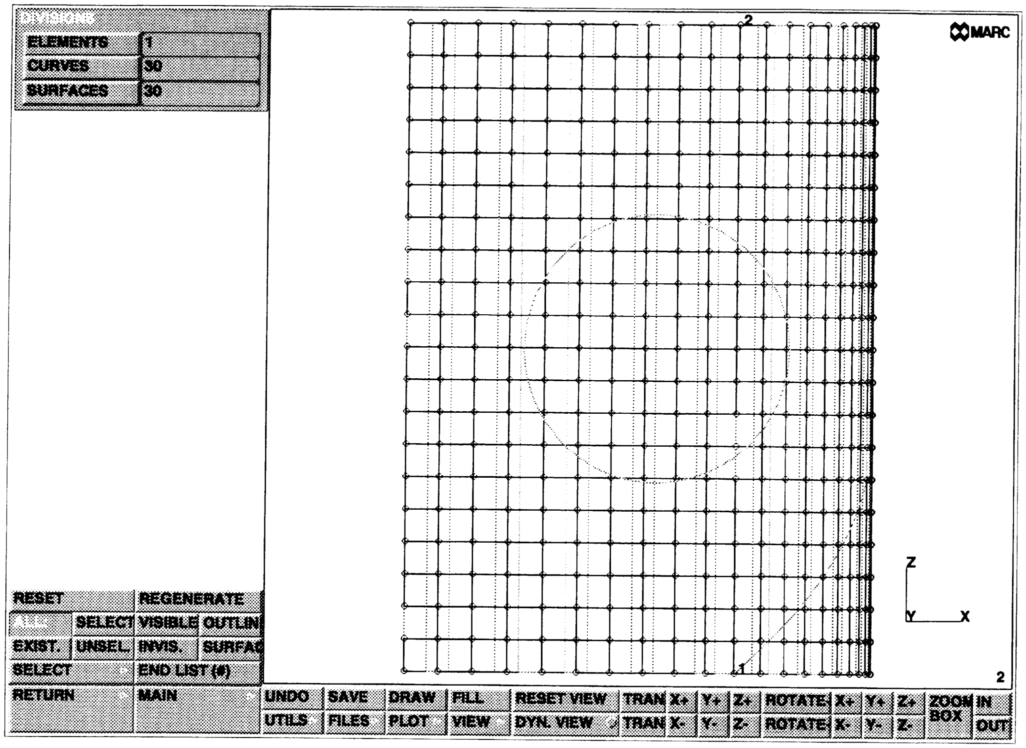


Figure 9.5 Elements Generated on Vessel Surface

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

(Box pick the elements)

            END LIST (#)
        SWEEP
            remove unused NODES

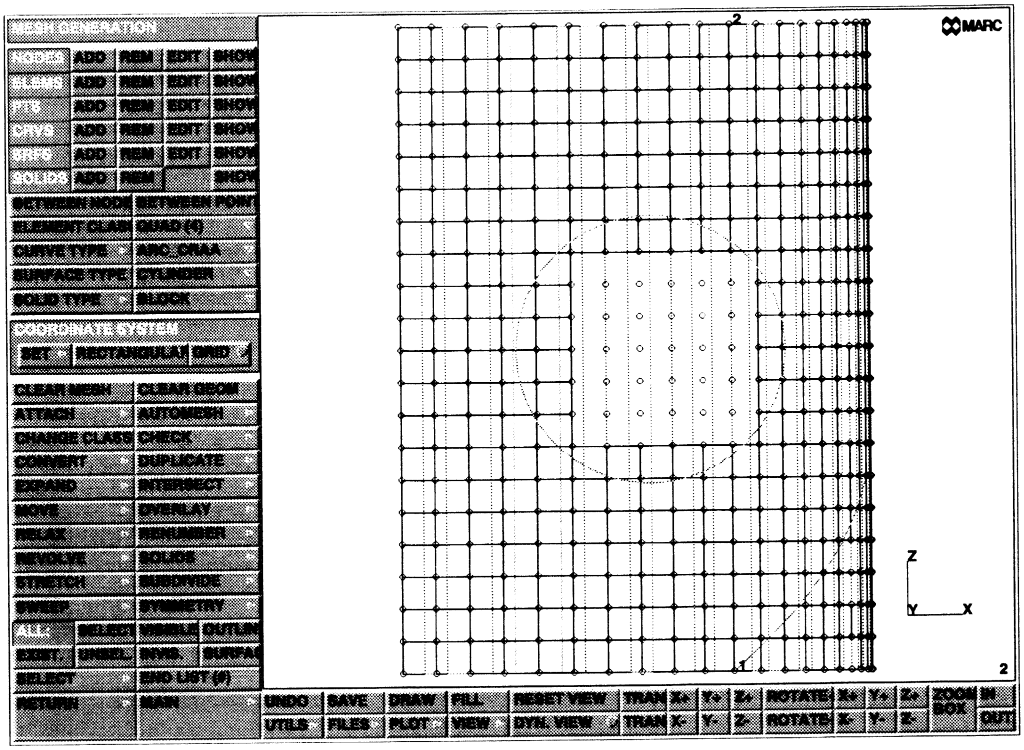


Figure 9.6 Vessel with Elements Removed

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.

MAIN
MESH GENERATION
RELAX
NODES
all: EXIST.

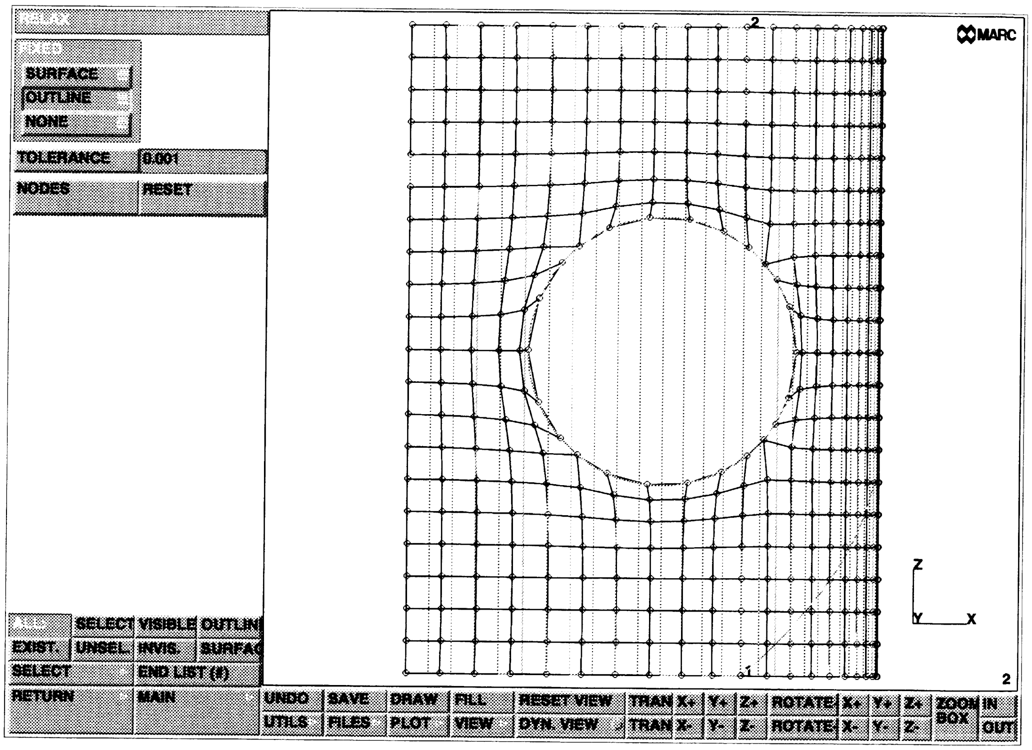


Figure 9.7 Peripheral Nodes Attached to Cylinder

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.

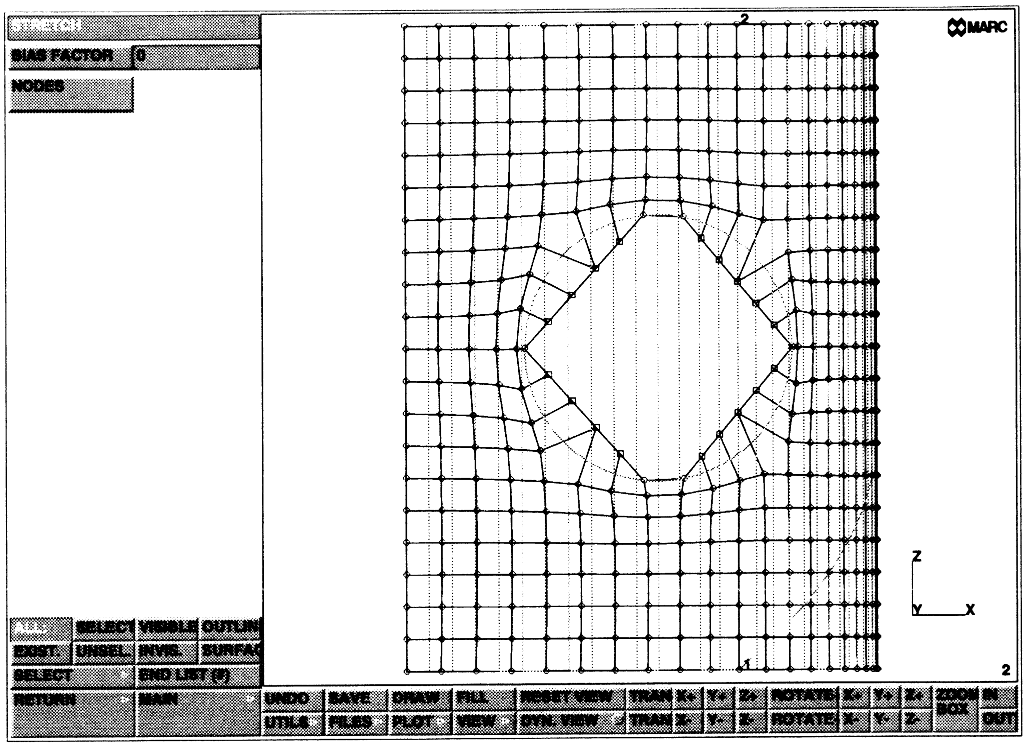


Figure 9.8 Evenly Distributed Nodes

The following button sequence is used to stretch the nodes:

```
MAIN
    MESH GENERATION
        STRETCH
            NODES
                226        (Pick the first node of the stretch node path)
                161        (Pick the last node of the stretch node path)
                END LIST (#)
```

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                          (Pick the surface)
                1                          (Pick the surface)
                247  268  289  288         (Pick the nodes)
                END LIST (#)
```

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.

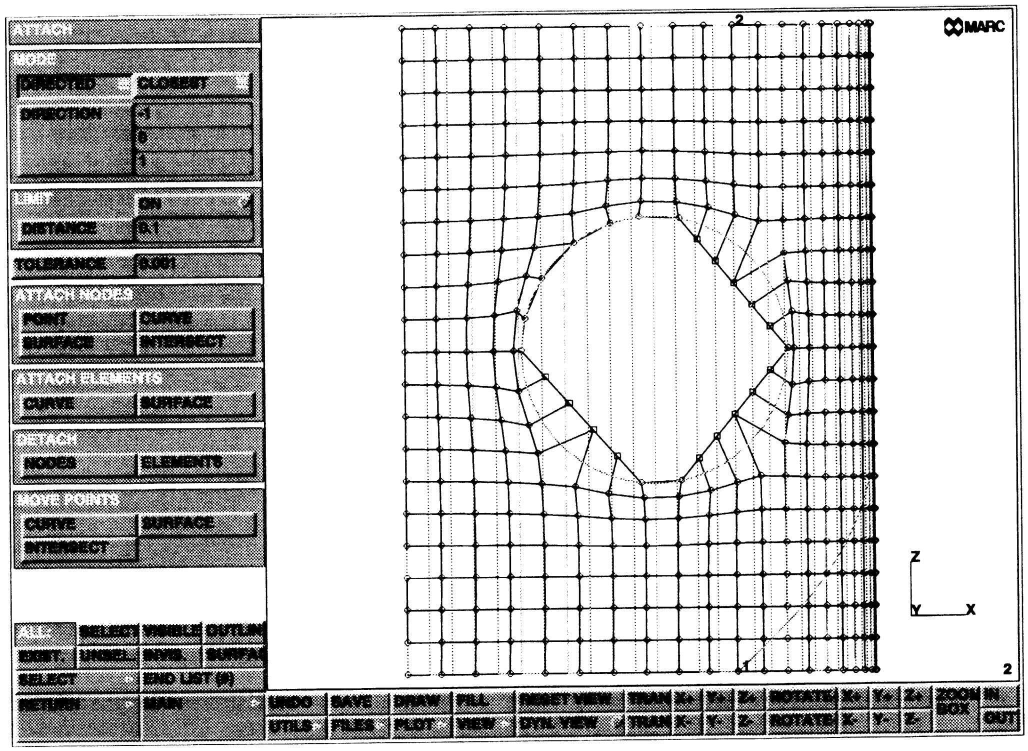


Figure 9.9 Using the Directed Attach Method to Re-Attach the Nodes

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
                (Box pick all elements below the symmetry line)
            END LIST (#)
        SWEEP
            remove unused NODES

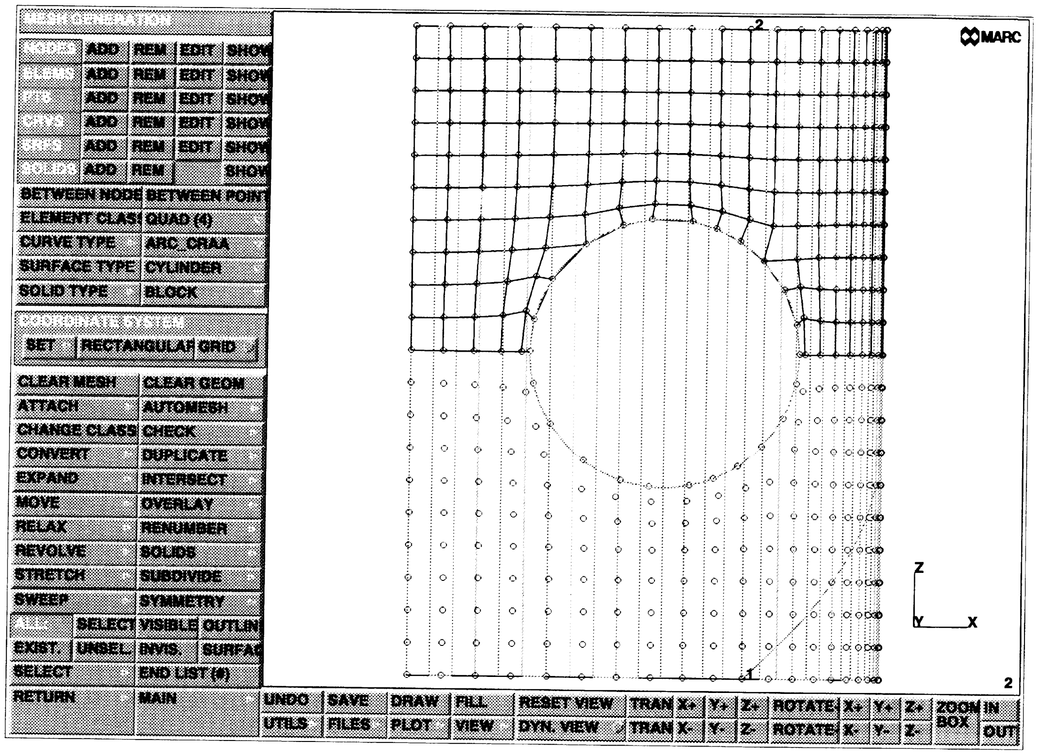


Figure 9.10 Removing Unused Nodes

Step 6

There are several ways to create the manhole. One 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
                (Pick the edges at the perimeter of the hole)
                END LIST (#)

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.

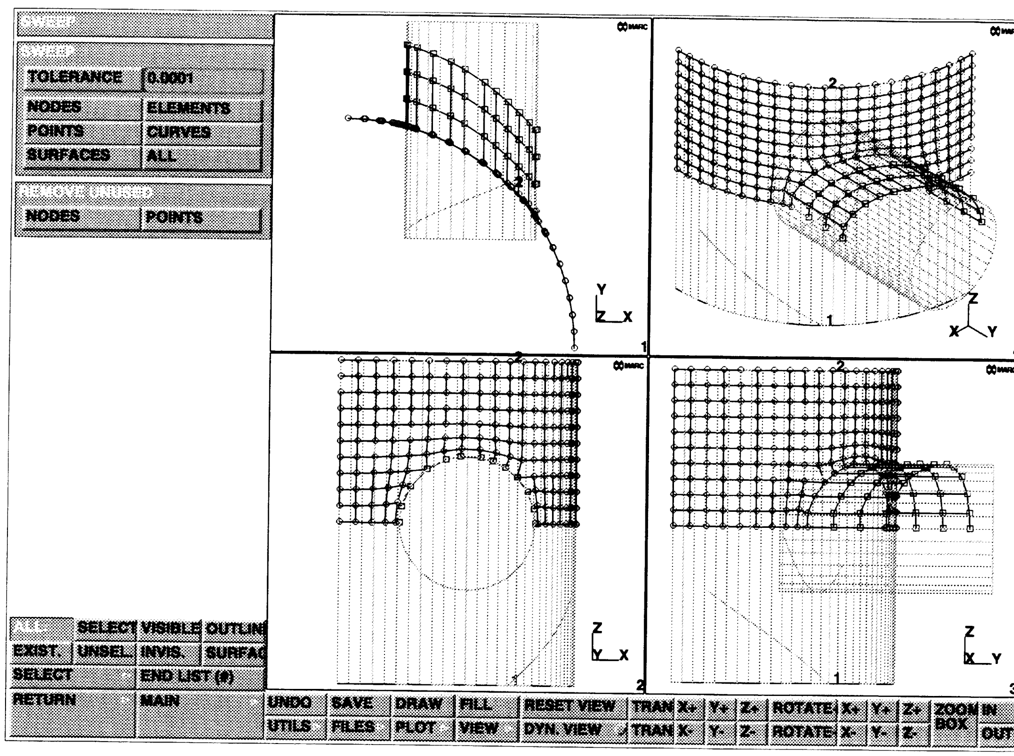


Figure 9.11 Manhole with Nearly Correct Coordinates

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
            SPACING
                10  10
            SIZE
                100  100
            grid ON                                          (on)
            ROTATE
                90  0  0
            TRANSLATE
                0  112  0
            RETURN
        PLOT
            draw POINTS                                      (on)
            RETURN
        pts ADD
            -10  0  0
            100  0  0
            100  100  0
            -10  100  0
        FILL
        srfs ADD
            29  30  31  32   (pick the four points generated above)
        GRID                                                 (off)
        FILL
```

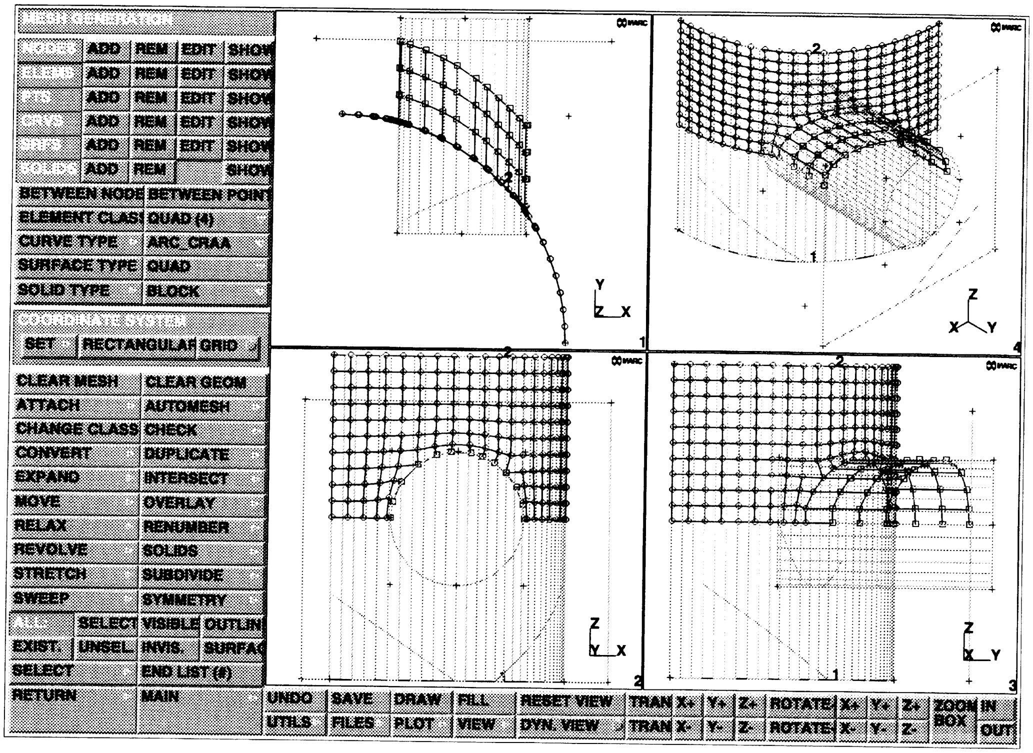


Figure 9.12 Creating the Patch

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 *(Pick the cylinder)*
3 *(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.

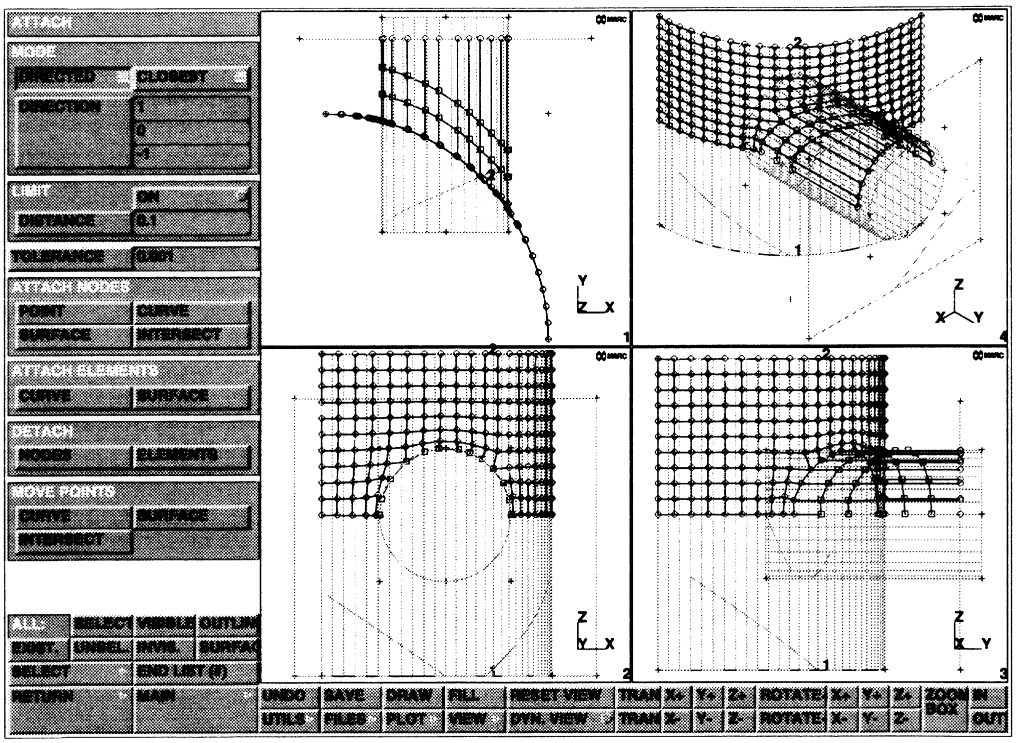


Figure 9.13 Attached Nodes to Patch

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

(Pick the top row of elements)

END LIST (#)

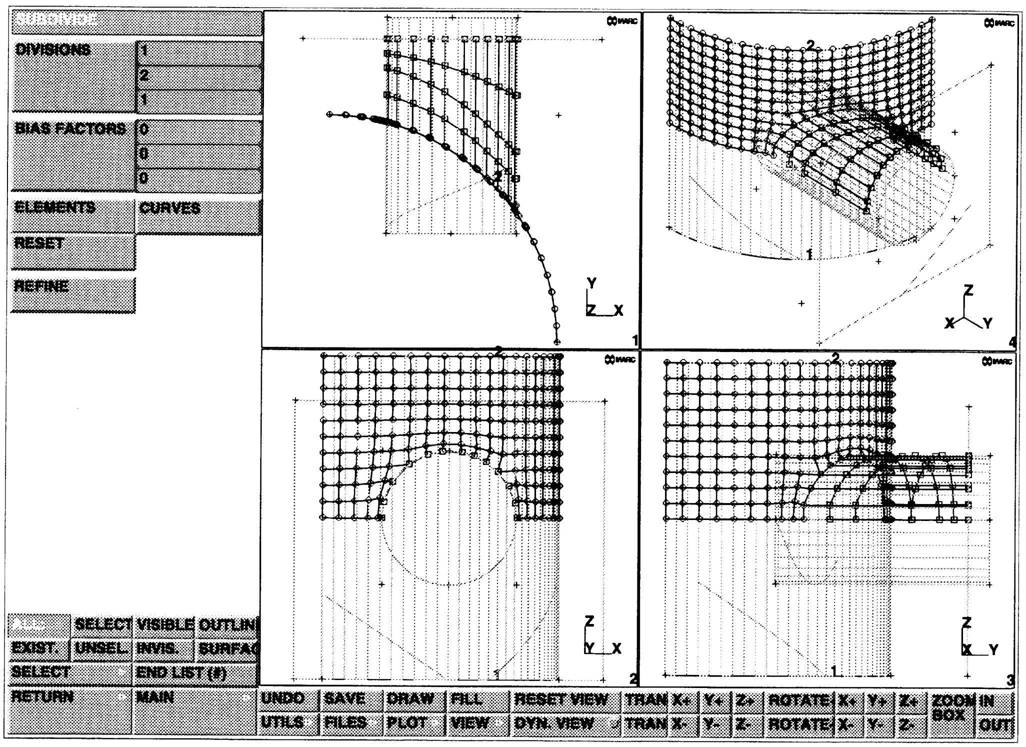


Figure 9.14 Improved Aspect Ratio for Top Row Elements

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 0 100` *(center line)*
(Pick the nodes along the curved and straight edges of the vessel; not those of the manhole)
END LIST (#)

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

MAIN
BOUNDARY CONDITIONS
MECHANICAL
FIXED DISPLACEMENT
ON z displace *(on)*
ON y rotation *(on)*
OK
nodes ADD
(Pick the nodes along the symmetry plane of the vessel and manhole)
END LIST (#)
VIEW
show 4
RETURN

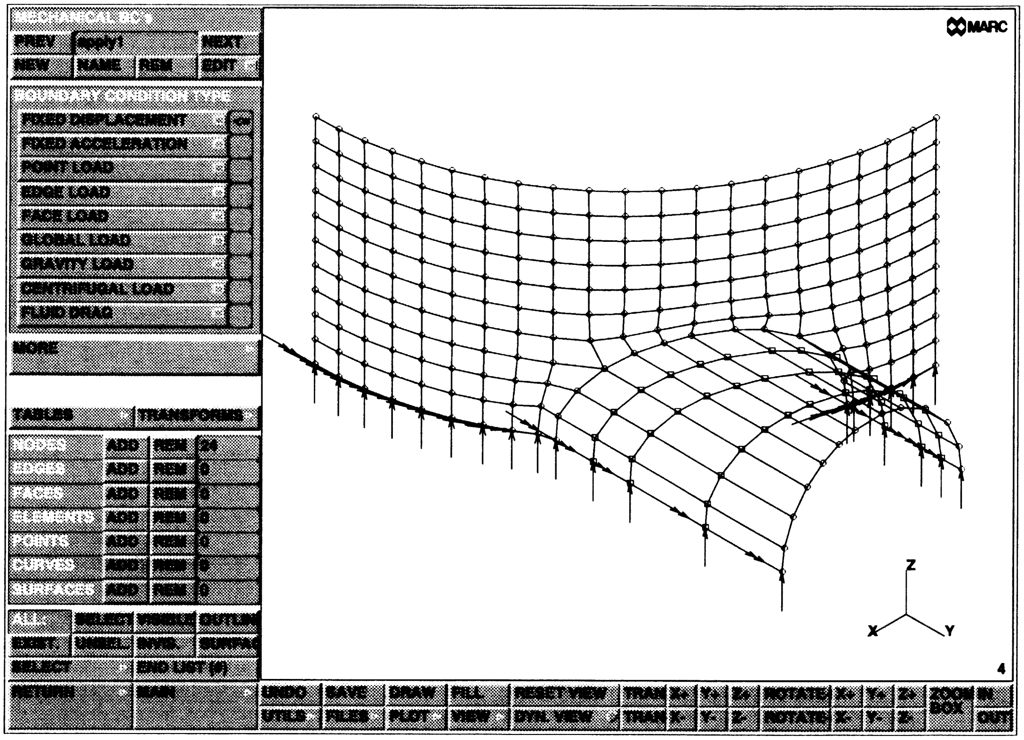


Figure 9.15 Boundary Conditions Applied

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

MAIN
BOUNDARY CONDITIONS
MECHANICAL
NEW
EDGE LOAD
PRESSURE
`-4200`
OK
edges ADD

(Pick the edges on the curved side of the vessel)

END LIST (#)

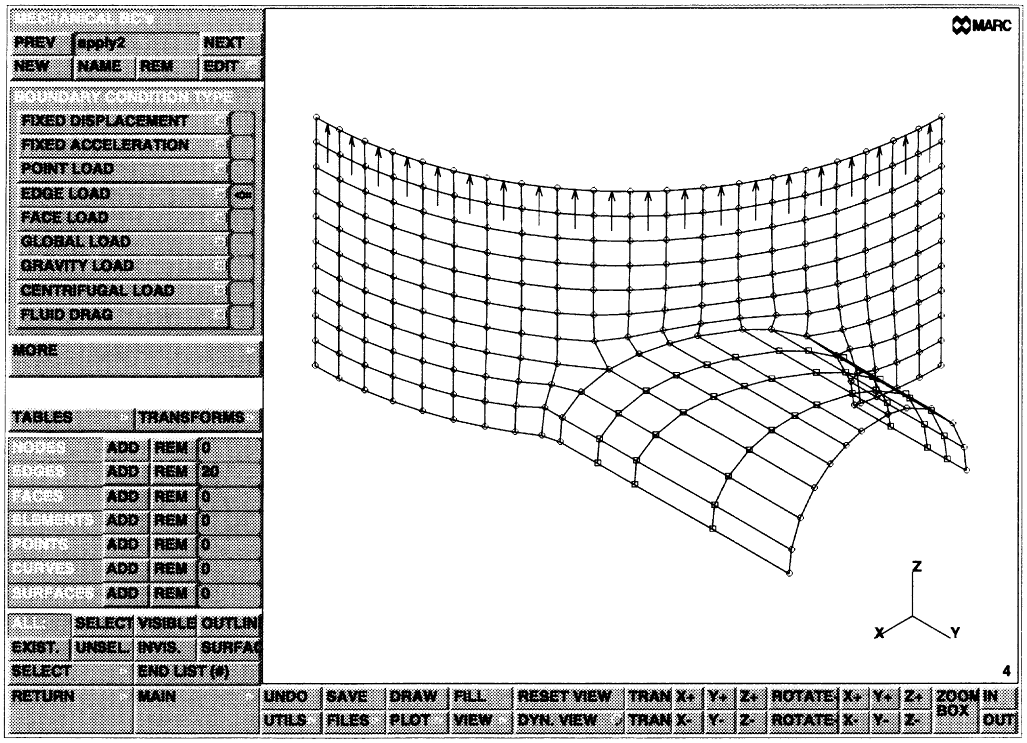


Figure 9.16 Edge Loads Applied in Direction of Center Line

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.

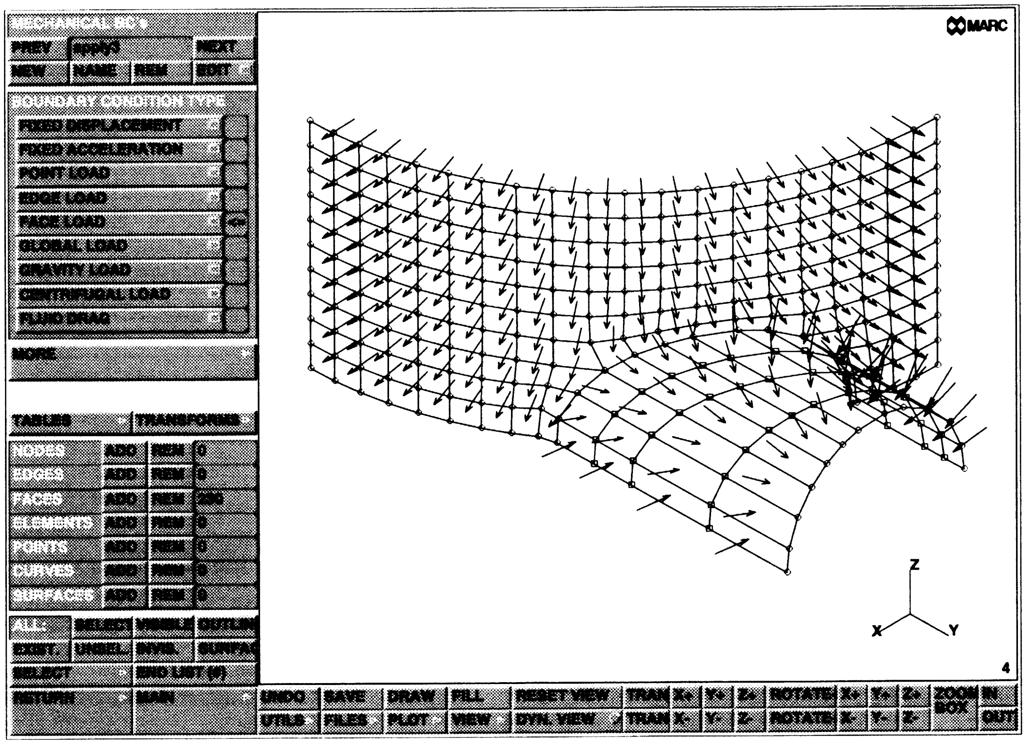


Figure 9.17 Internal Pressure Applied

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

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

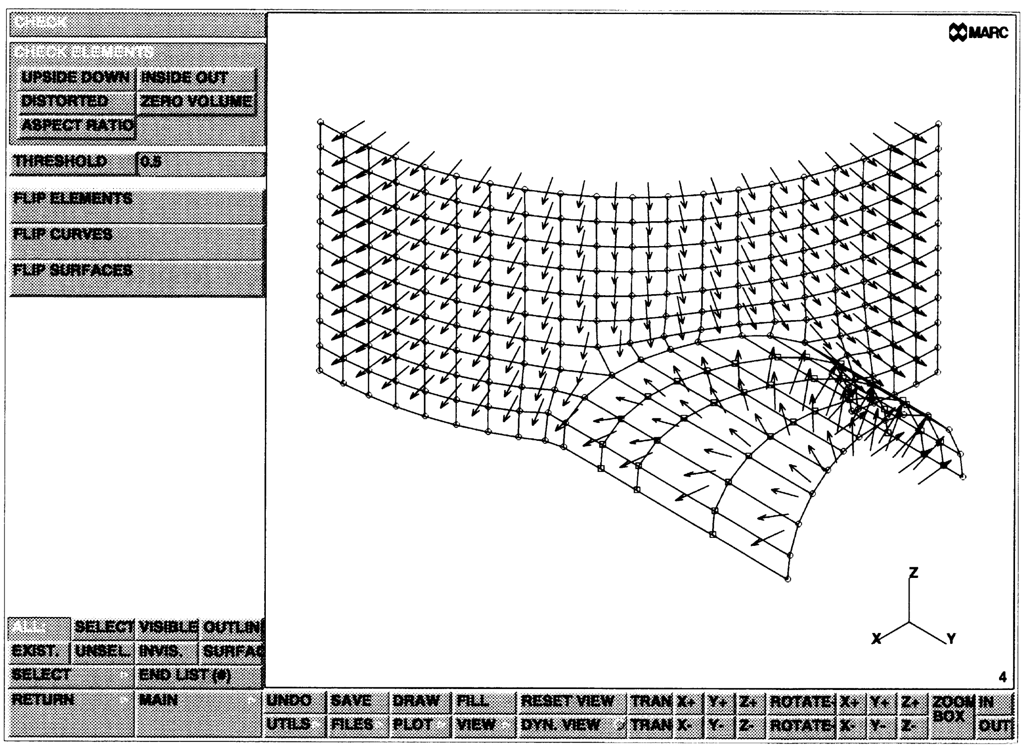


Figure 9.18 Corrected Load Direction

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)*
                ON z rotation *(on)*
                OK
            nodes ADD
(Pick the nodes on the straight edges of the vessel)
                END LIST (#)

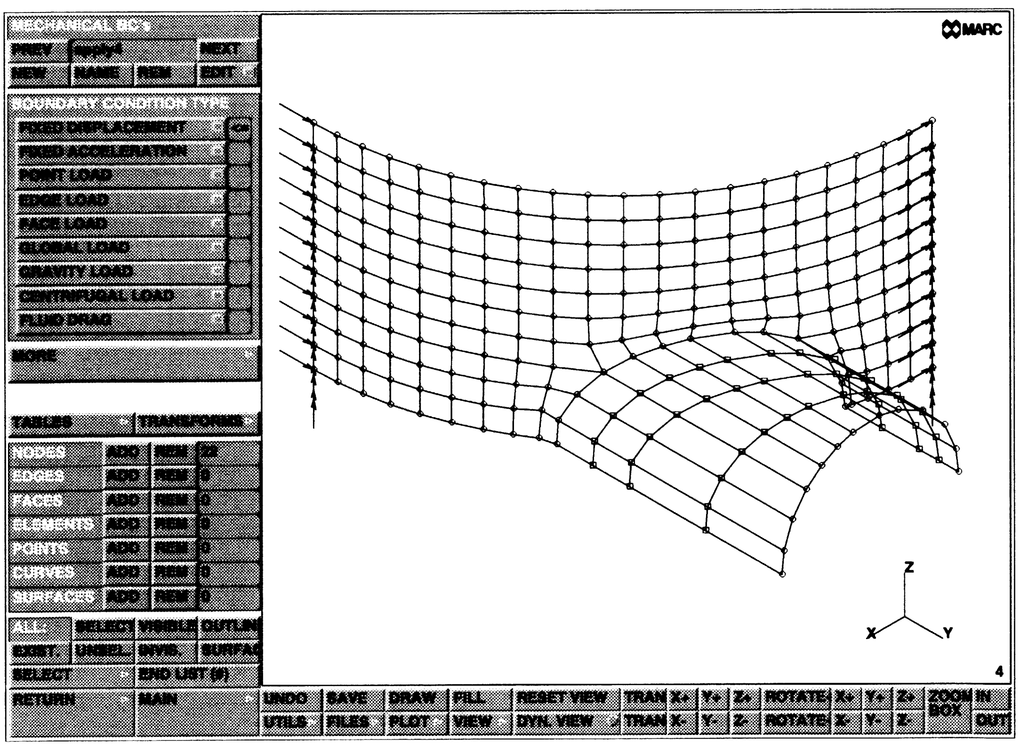


Figure 9.19 Boundary Conditions Applied to Vessel Edges

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
                (Pick the edges at the top rim of the manhole)
                END LIST (#)
            VIEW
                show 4
                RETURN

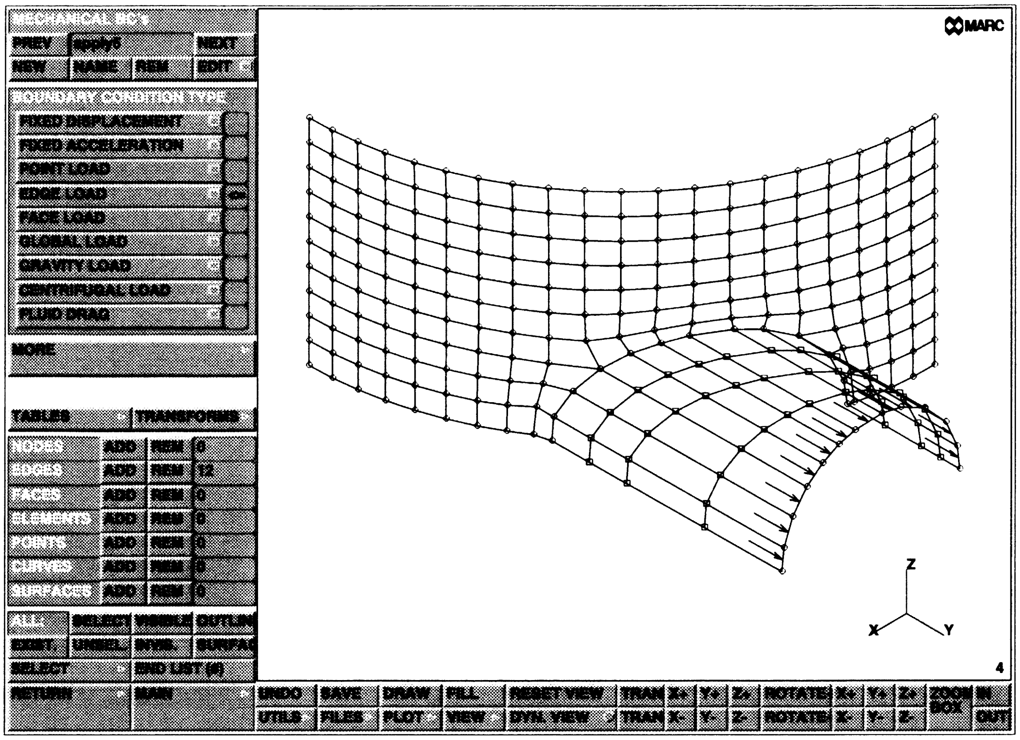


Figure 9.20 Edge Load Applied to Top Rim of Manhole

Step 12

The material properties for both the vessel and the manhole are specified in 9.1.3. 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
                                        (Pick the elements of the manhole)
                END LIST (#)
```

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
                SELECT TENSORS
                    stress
                select tensors OUTER & MIDPLANE
                    stress
                SELECT VARIABLES
                    von_mises
                select variables OUTER & MIDPLANE
                    von_mises
                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:

```
MAIN
    RESULTS
        OPEN DEFAULT
        NEXT INC
        PLOT
            draw NODES                                        (off)
            RETURN
        SCALAR
            DOWN
            DOWN
            Equivalent Von Mises Stress Layer 1
            OK
        CONTOUR BANDS
        DEF & ORIG
        FILL
```

Figure 9.21 shows the resulting model contoured with von Mises stresses.

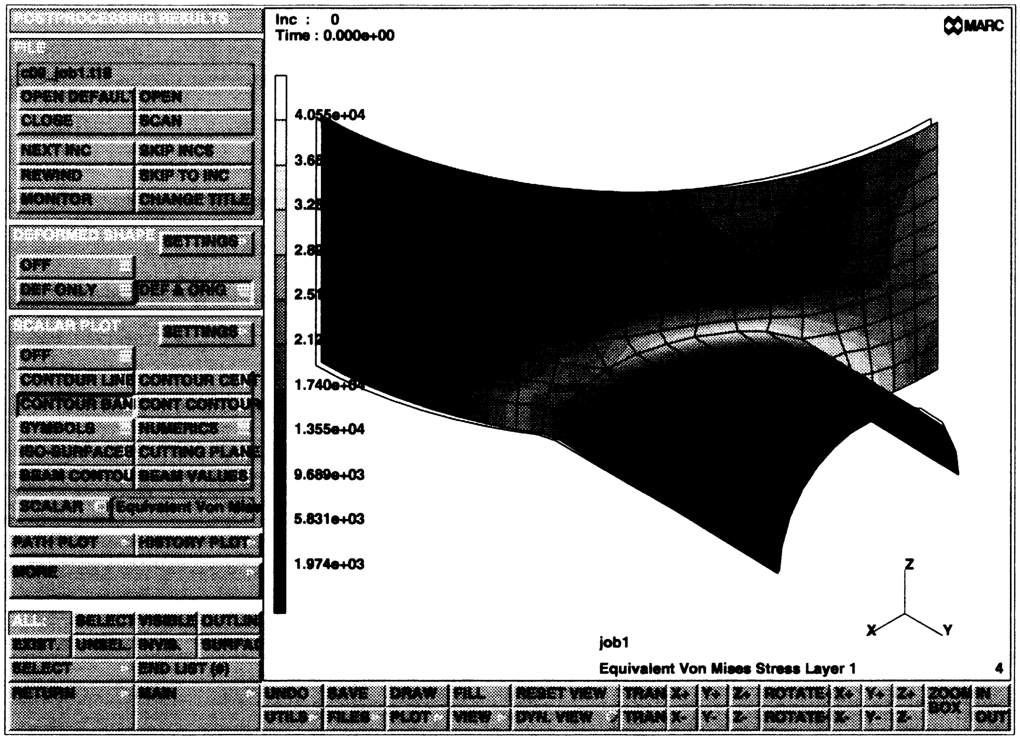


Figure 9.21 Model with von Mises Stress Contours

9.3 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.

9.4 Procedure file

```
| Version : MENTAT II 2.3
|
| This procedure file demonstrates the modeling of an eccentric hole in a
| tank of a larger diameter. The entire structure is loaded by an internal
| pressure.
|
*reset
|
| Step 1
|
| Create two curves of radius 84 over an angle of 90 degrees
|
*set_curve_type arc_craa
*add_curves
0 0 0
84
0 90
0 0 116
84
0
90
*set_curve_labels on
*fill_view
*rot_model_increment
45
*rot_model_cspace_y_for
*fill_view
|
| Create a ruled surface between the two curves
|
*set_surface_type ruled
*add_surfaces
 1
 2
|
| Switch to surface type cylinder and generate a cylinder surface
|
*set_surface_type cylinder
*add_surfaces
45 40 58
45 120 58
24
24
*rot_model_cspace_y_rev
|
| Display in all 4 views
|
```

```
 *set_points off
*show_all_views
*activate_all_views
*fill_view
|
| Step 2
|
| Convert this surface to a 20 x 20 mesh
|
*set_convert_divisions
20 20
*convert_surfaces
 1
# | End of List
 *set_faces off
*set_curve_divisions
30
*set_surface_divisions
30
*regen
*show_view 2
|
| Step 3
|
| Remove elements to create a hole
|
*remove_elements
 150 151 152 153 154 155 170 171 172 173 174 175 190 191 192 193 194
 195 210 211 212 213 214 215 230 231 232 233 234 235 250 251 252 253
 254 255
# | End of List
|
| Step 4
|
| Remove the unused nodes, left after deleting the elements and attach
| nodes on the perimeter of the hole to the intersection of surface 2 and 1.
|
*remove_unused_nodes
*attach_nodes_intersect
2
1
 157 158 159 160 161 162 163 178 184 199 205 220 226 241 247 262 268 283 284 285
 286 287 288 289
# | End of List
|
| Improve the element shape by relaxing the nodes.
|
*relax_nodes
all_existing
|
```

```
| Now stretch the four 'diagonal' sides of the hole
|
*stretch_nodes
 226
 161
# | End of List
 160
 220
# | End of List
 220
 286
# | End of List
 287
 226
# | End of List
|
| Step 5
|
| Perform a 'directed' attach in each quadrant
|
*attach_mode_directed
*set_attach_direction
-1 0 1
*attach_nodes_intersect
2
1
 247
 268
 289
 288
# | End of List
*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
|
| Cut 1/2 model away (symmetry)
|
*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
|
| Step 6
|
| Generate line elements on the edges on the perimeter of
| the hole
|
*remove_unused_nodes
*edge_lines
 209:1 216:3 229:1 236:3 249:1 256:3 270:0 271:0 272:0 273:0
 274:0 275:0
# | End of List
|
| Select the line elements and store them as set 'sticks'
|
*select_elements_class
line2
*store_elements
sticks
all_selected
*select_clear
*show_all_views
```

```
|
| Step 7
|
| Expand the line elements to form the foot of the manhole
|
*set_expand_translations
0 10 0
*set_expand_repetitions
3
*expand_elements
sticks
*sweep_nodes
all_existing
|
| Step 8
|
| Generate an additional surface at the future top location
| of the manhole
|
*set_surface_type quad
*set_grid_spacing
10
*set_grid_size
100
*set_grid on
*set_grid_spacing
10 10
*set_grid_size
100 100
*system_rotate
90 0 0
*system_translate
0 112 0
*set_points on
*add_points
-10 0 0
100 0 0
100 100 0
-10 100 0
*fill_view
|
| Add the 'quad' surface
|
*add_surfaces
 29
 30
 31
 32
 *set_grid off
*fill_view
```

```
|
| Attach the top nodes of the manhole to the
| surface
|
*attach_nodes_intersect
 2
 3
 448 455 462 469 476 483 490 497 504 511 518 525 532
# | End of List
|
| Step 9
|
| Split the top row of elements
|
*sub_divisions
1 2 1
*subdivide_elements
 415 418 421 424 427 430 433 436 439 442 445 448
# | End of List
|
| Step 10
|
|
| Clean the mesh by sweeping duplicate nodes
|
*sweep_nodes
all_existing
 *set_points off
 *set_curves off
 *set_surfaces off
*regen
*show_view 2
*fill_view
|
| Step 11
|
| Set up a transformed coordinate system B.C.'s applied
| to nodes on the perimeter of the structure
|
*add_transform_nodes
*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
|
| Apply fixed disp B.C.'s to nodes on the symmetry plane
```

```
|
*apply_type fixed_displacement
*apply_dof z
*apply_dof ry
*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
*show_view 4
|
| Apply an edge-load to the vessel
|
*new_apply
*apply_type edge_load
*apply_dof p
*apply_value p
-4200
*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
|
| Apply the internal pressure
|
*new_apply
*apply_type face_load
*apply_dof p
*apply_value p
-100
*add_apply_faces
all_existing
|
| Correct the definition direction for elements in the manhole
|
*show_view 2
*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
*show_view 4
|
| Apply fixed disp B.C.'s to the straight edges of the vessel
|
*new_apply
*apply_type fixed_displacement
*apply_dof y
*apply_dof rz
*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
|
| Apply an edge-load to the top of the manhole
|
*new_apply
*apply_type edge_load
*apply_dof p
*apply_value p
-1200
*show_view 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
*show_view 4
|
| Step 12
|
| Apply the 'steel' material properties
|
*material_type mechanical:isotropic
*material_value isotropic:youngs_modulus
30.0e+06
*material_value isotropic:poissons_ratio
.3
*add_material_elements
all_existing
|
| Step 13
|
| Apply the geometric properties (thickness) to elements
| belonging to the manhole and belonging to the vessel
|
*new_geometry
*geometry_type mech_three_shell
*geometry_value thick
1.0
*show_view 2
*add_geometry_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
*new_geometry
*geometry_type mech_three_shell
*geometry_value thick
0.54
*add_geometry_elements
```

```
 201 202 203 204 205 206 207 208 209 216 217 218 219 220 221
 222 223 224 225 226 227 228 229 236 237 238 239 240 241 242
 243 244 245 246 247 248 249 256 257 258 259 260 261 262 263
 264 265 266 267 268 269 270 271 272 273 274 275 276 277 278
 279 280 281 282 283 284 285 286 287 288 289 290 291 292 293
 294 295 296 297 298 299 300 301 302 303 304 305 306 307 308
 309 310 311 312 313 314 315 316 317 318 319 320 321 322 323
 324 325 326 327 328 329 330 331 332 333 334 335 336 337 338
 339 340 341 342 343 344 345 346 347 348 349 350 351 352 353
 354 355 356 357 358 359 360 361 362 363 364 365 366 367 368
 369 370 371 372 373 374 375 376 377 378 379 380 381 382 383
 384 385 386 387 388 389 390 391 392 393 394 395 396 397 398
 399 400
# | End of List
*show_view 4
*elements_solid
*identify_geometries
*regen
*identify_none
*regen
|
| Step 14
|
| Set MARC element type, prepare and submit the job
|
*element_type 75
all_existing
*job_class mechanical
*add_post_tensor
stress
*post_tensor_outer_layers
stress
*add_post_var
von_mises
*post_var_outer_layers
von_mises
*job_param layers
3
*check_job
*save_as_model
c09
y
*update_job
*submit_job 1
*monitor_job
|
| Step 15
|
| Post-process the results
|
```

```
*post_open_default
*post_next
 *set_nodes off
*post_value
*pick_list_next(post_value,14)
*pick_list_next(post_value,14)
Equivalent Von Mises Stress Layer 1
*post_contour_bands
*set_deformed both
*fill_view
```

Chapter 10: Tire

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.

10.1 Background Information

10.1.1 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.

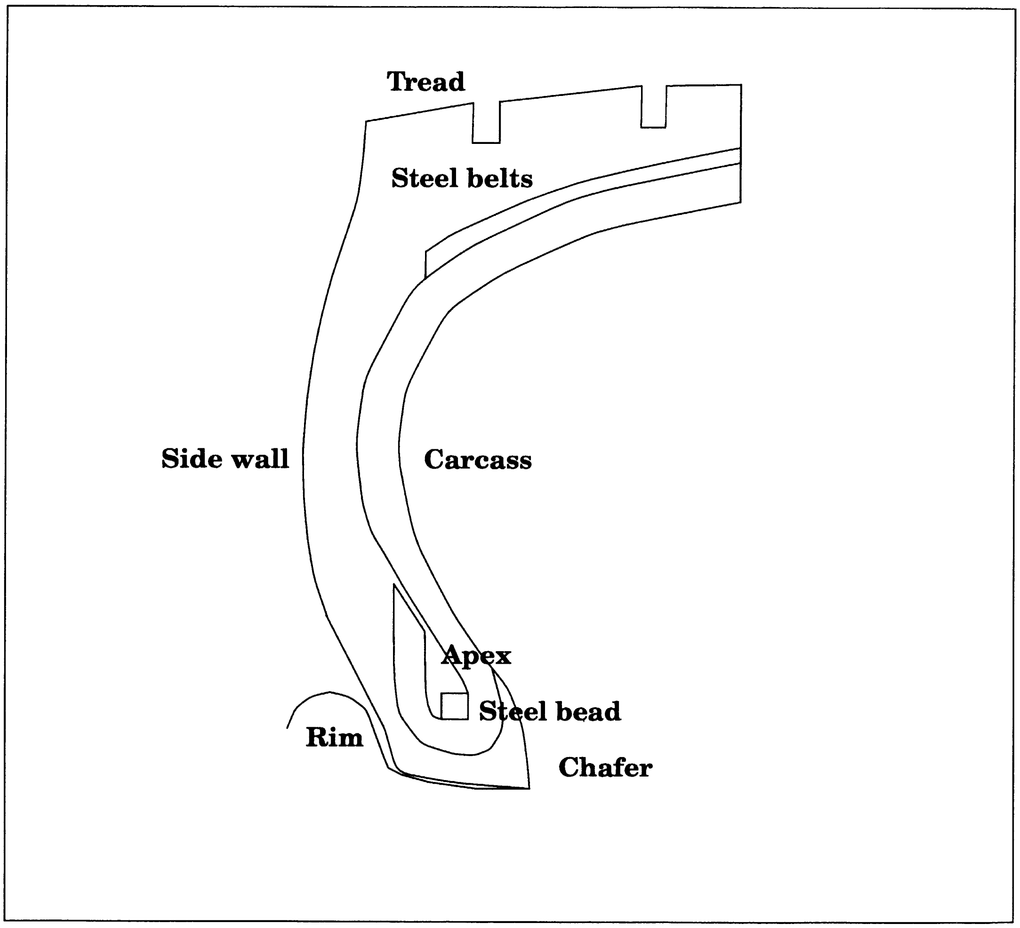


Figure 10.1 Cross-Section of Automobile Tire

10.1.2 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.

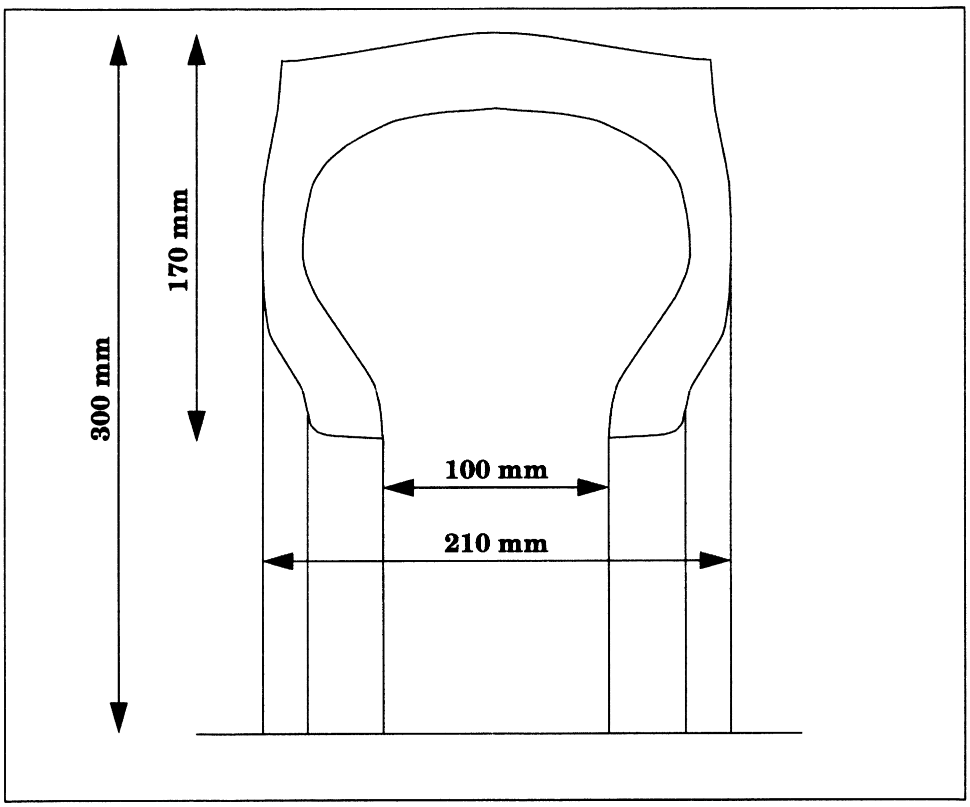


Figure 10.2 Overall Dimensions of the Tire

10.1.3 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.

10.2 Analysis

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

10.2.1 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.

10.2.2 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.

10.2.3 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}	C_{20}	C_{30}
965kPa	-193kPa	193kPa

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

10.2.4 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 Post-process the results.

10.2.5 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
            SPACING
                1  1
            SIZE
                17  17
            grid ON                                    (on)
            FILL
```

Observe that the dimension of the grid size is specified in centimeters. The material constants specified in paragraph 10.1.2 require a conversion from kPa into N/cm^2, in order to be consistent with the units used here. For a good resolution of the Bezier curve drawing, set the number of subdivisions to 20. 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
                CURVES
                    20
                RETURN
            RETURN
    MESH GENERATION
        CURVE TYPE
            BEZIER
```

Zoom in on the left-upper quadrant of the grid.

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 *(Pick the appropriate grid points)*
```
point(0,14,0)
point(-11,13,0)
point(-11,6,0)
point(-5,4,0)
point(-5,0,0)
```
END LIST (#)

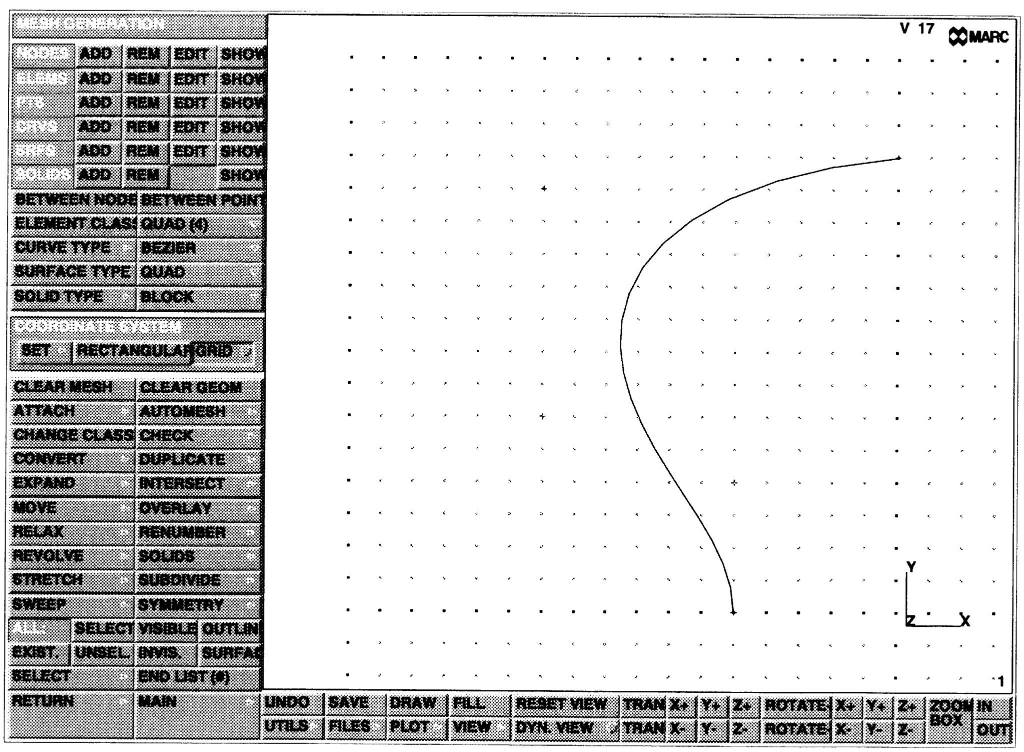


Figure 10.3 Interior Tire Wall

Increase the resolution of the grid to 0.5 units:

MAIN
MESH GENERATION
SET
SPACING
`0.5  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.

MAIN
MESH GENERATION
crvs ADD *(Pick the appropriate grid points)*
```
point(-9,16,0)
point(-9,14,0)
point(-10.5,13,0)
point(-10.5,8.5,0)
```
END LIST (#)

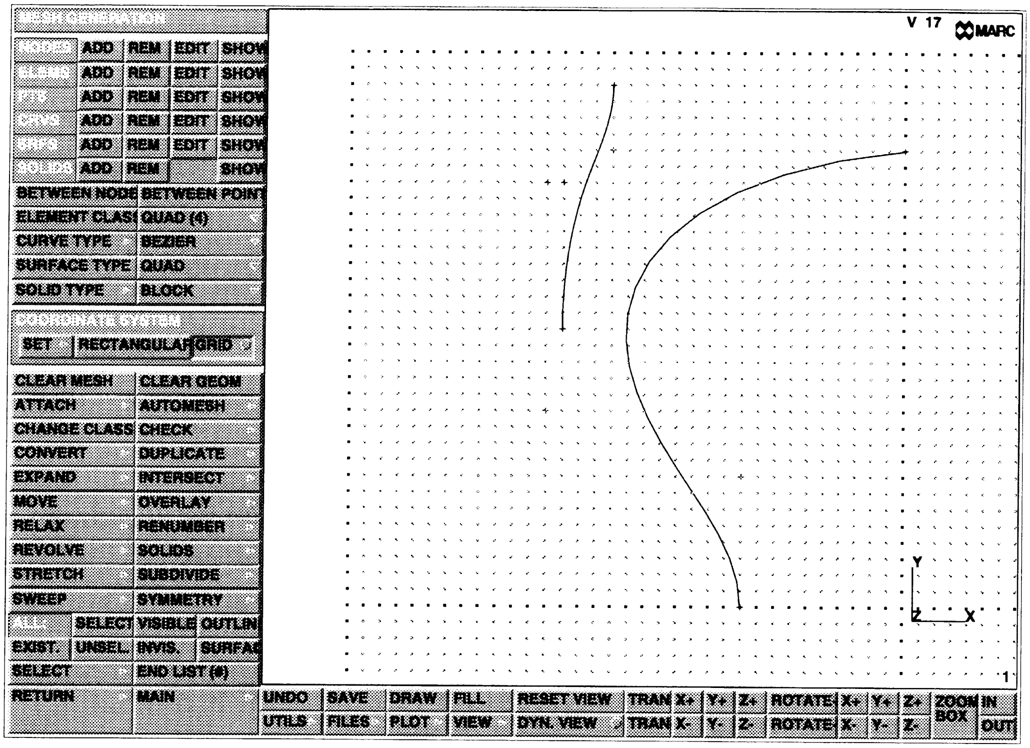


Figure 10.4 Part of Exterior Tire Wall Added

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

MAIN
    PLOT
        label POINTS *(on)*
        REGENERATE

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.

MAIN
MESH GENERATION
crvs ADD
`9` *(Pick point)*
`point(-10.5,4,0)` *(Pick grid point)*
`point(-9.5.3.5,0)` *(Pick grid point)*
`point(-8.5,1.5,0)` *(Pick grid point)*
`point(-8,0,0)` *(Pick grid point)*
`point(-7.5,0.5,0)` *(Pick grid point)*
`5` *(Pick point)*
END LIST (#)

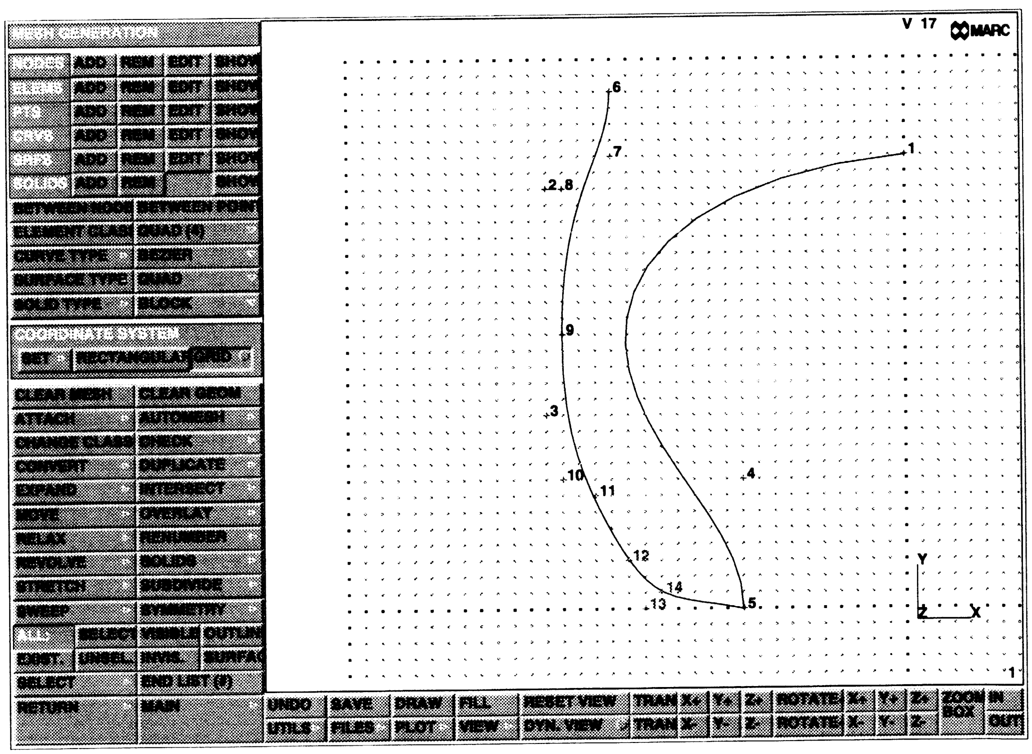


Figure 10.5 Exterior Tire Wall Completed

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`	*(Pick point)*
`-9.5  -0.5  0`	*(Pick grid point)*
`10`	*(Pick point)*
`-11.0  -0.5  0`	*(Pick grid point)*
`11`	*(Pick point)*
`-7  5  0`	*(Pick grid point)*
END LIST (#)	

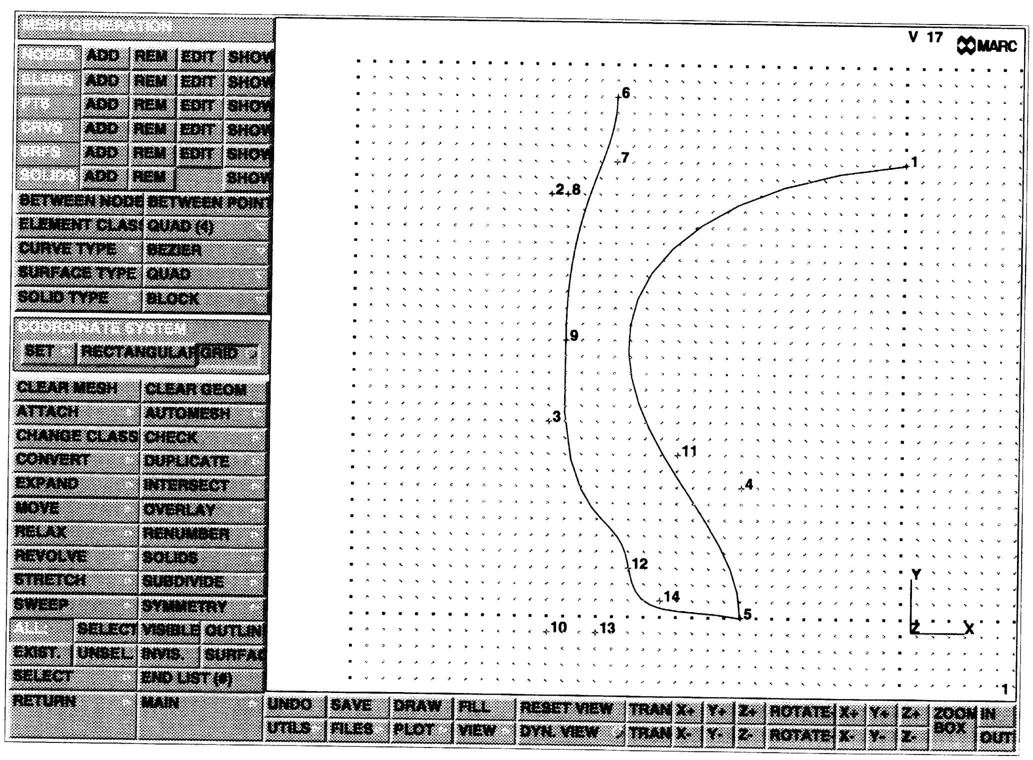


Figure 10.6 Correction of Tire Geometry

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` *(Pick point)*
            `point(-7.5,16,0)` *(Pick grid point)*
            `point(-1.5,17,0)` *(Pick grid point)*
            `point(0,17,0)` *(Pick grid point)*
            END LIST (#)

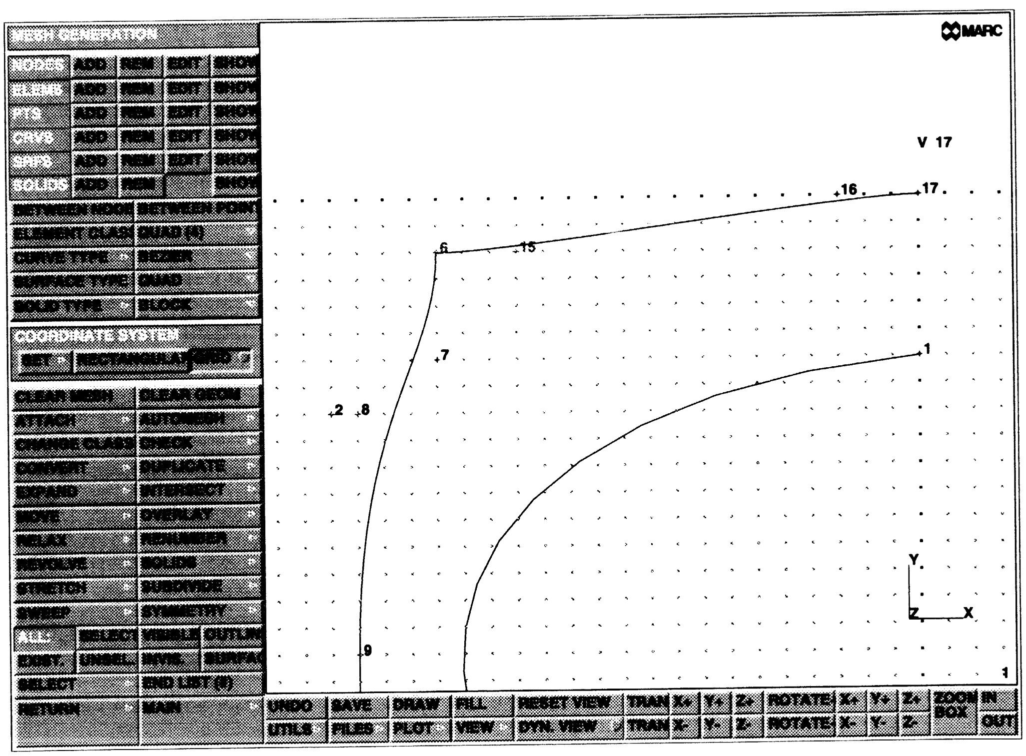


Figure 10.7 Tire Tread

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.

MAIN
MESH GENERATION
crvs ADD
`17` *(Pick point)*
`1` *(Pick point)*
END LIST (#)

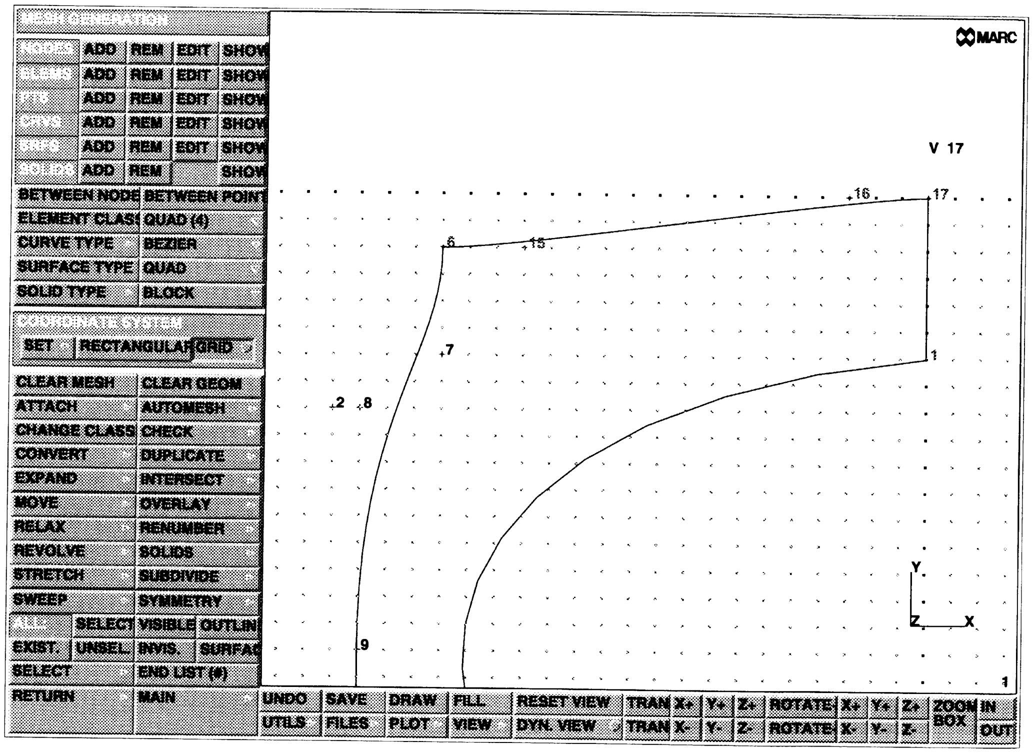


Figure 10.8 Completed Boundary

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.

MAIN
- MESH GENERATION
  - GRID *(off)*
  - FILL
  - OVERLAY
    - DIVISIONS
      - 20 20
    - CURVE MESH
      - all: EXIST.

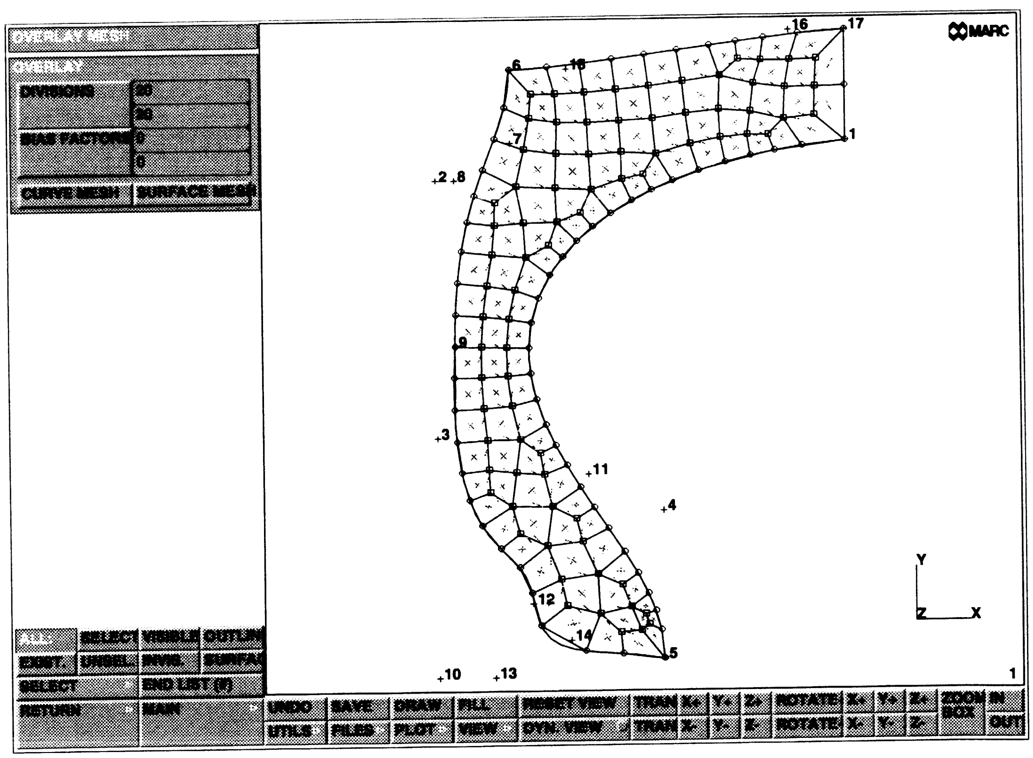


Figure 10.9 Mesh Generated by OVERLAY

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.

```
MAIN
    SUBDIVIDE
        DIVISIONS
            2  1  1
        ELEMENTS
            52  3  5  7  61                    (Pick elements)
            END LIST (#)
```

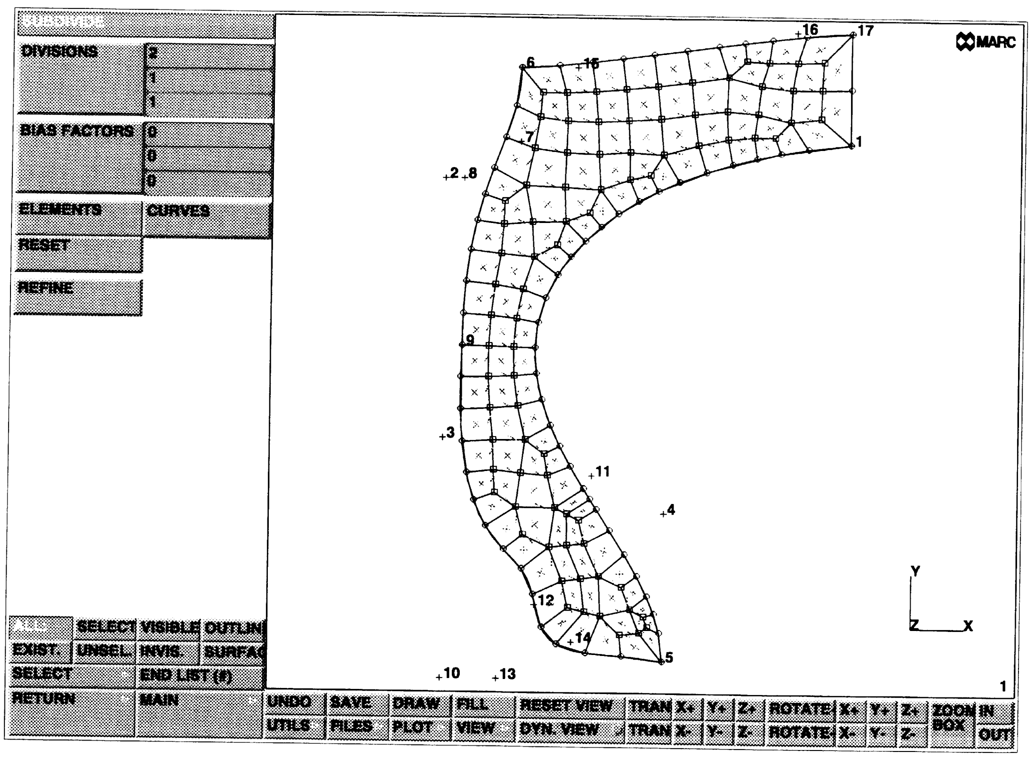


Figure 10.10 Step 1 of Mesh Refinement

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.

```
MAIN
    MESH GENERATION
        SUBDIVIDE
            REFINE
                1                          (Pick refine node)
                1                          (Pick element)
                END LIST (#)
```

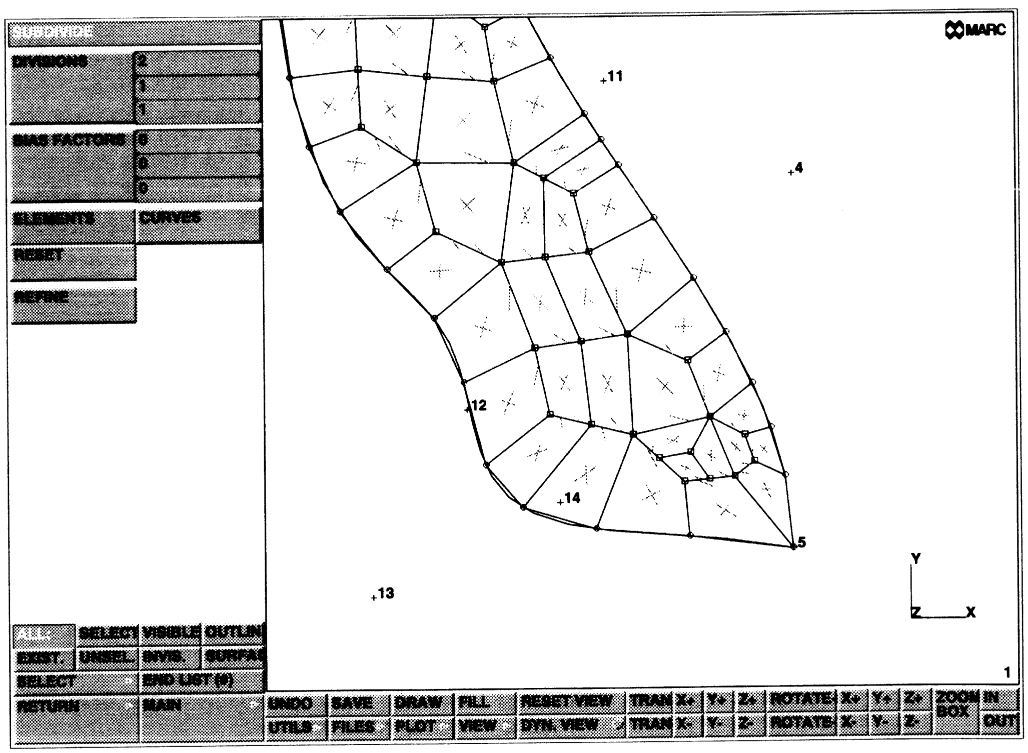


Figure 10.11 Local Refinement Around a Node

Complete this action by subdividing two more elements according to Figure 10.12.

MAIN
MESH GENERATION
SUBDIVIDE
48 47 *(Pick elements)*
END LIST (#)

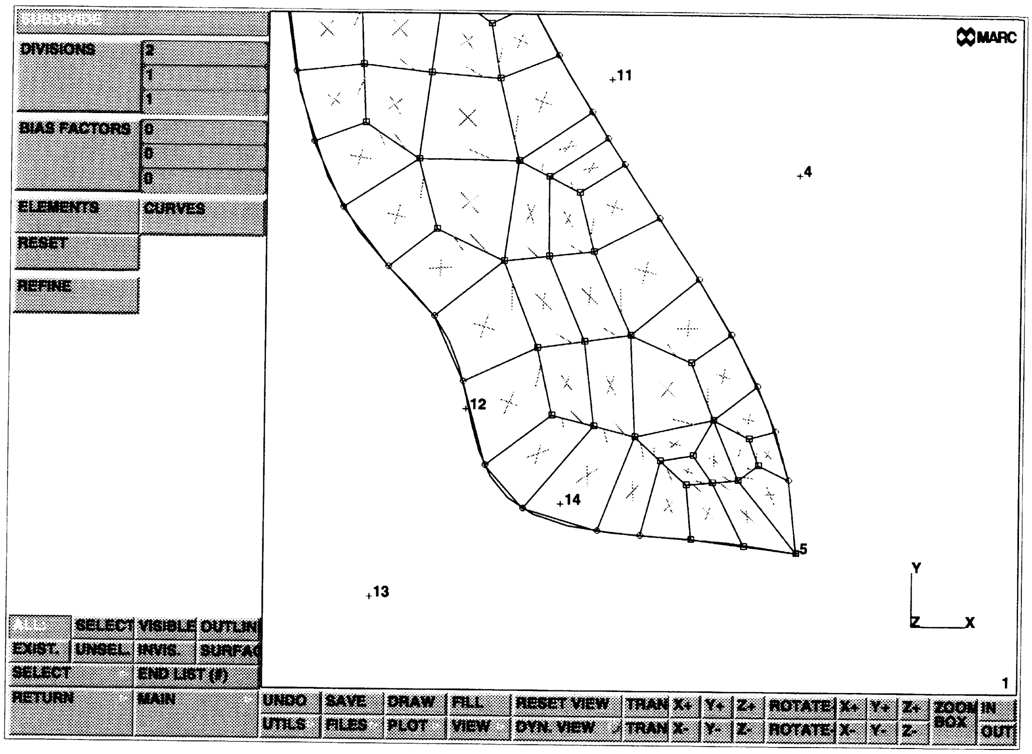


Figure 10.12 Completion of Local Mesh Refinement

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

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)*
`point(-12,0,0)`	*(Pick grid point)*
`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 (#)	

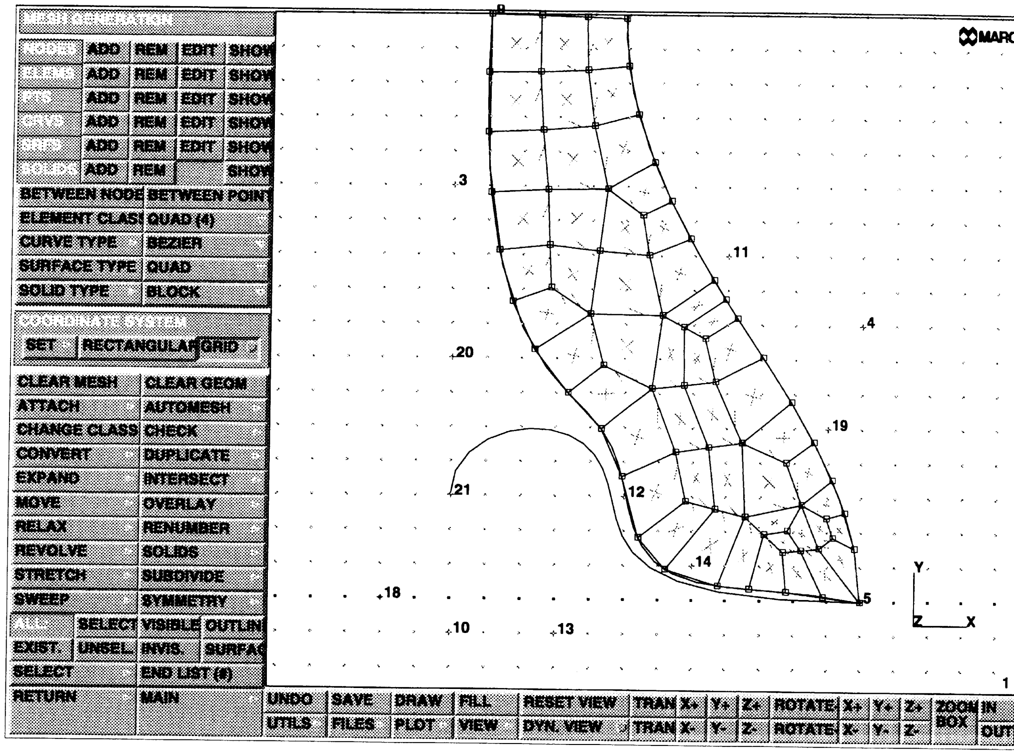


Figure 10.13 Wheel Rim Added

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                                        (off)
        MOVE
            TRANSLATIONS
                0 13 0
            ELEMENTS
                all: EXIST.
            CURVES
                all: EXIST.
            RETURN
        FILL
```

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
            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
            CURVES ADD
                6 *(Pick curve)*
                END LIST (#)
            PLOT
                label POINTS *(off)*
                draw POINTS *(off)*
                elements SOLID
                RETURN
            ID CONTACT *(on)*

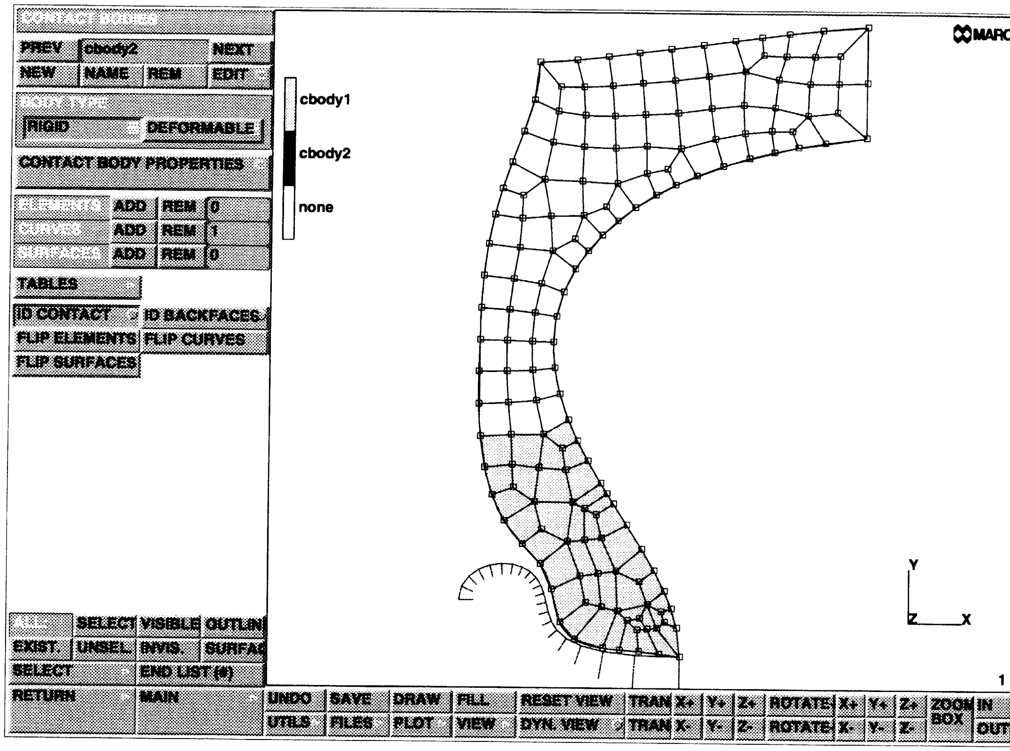


Figure 10.14 Identification of Contact Bodies

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 *(off)*
            PLOT
                elements WIREFRAME
                FILL

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 *(on)*
                OK
            nodes ADD
                `131 139 143` *(Pick the three nodes at the right)*
                END LIST (#)

The symmetry boundary conditions are displayed in Figure 10.15.

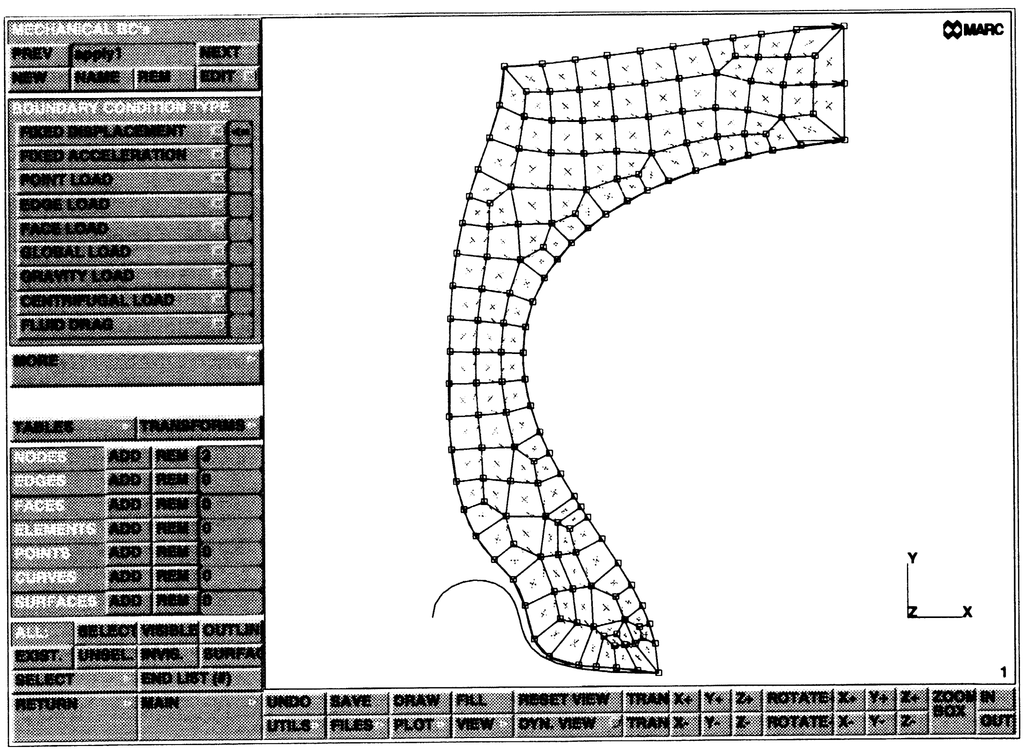


Figure 10.15 Symmetry Boundary Conditions Applied

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
        MECHANICAL
            NEW
            EDGE LOAD
                pressure TABLE
                loading
                OK
            edges ADD
                (Pick the edges on the inside)
                END LIST (#)

The results of the applied internal pressure are depicted in Figure 10.16.

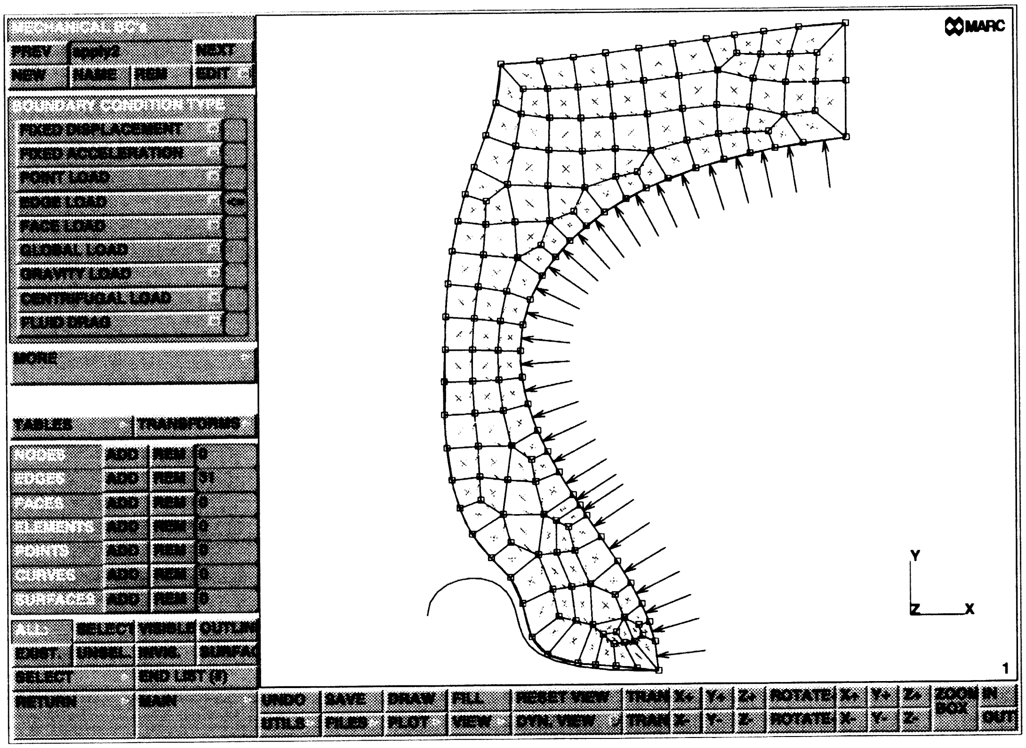


Figure 10.16 Internal Pressure Applied

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.

MAIN
  JOBS
    MECHANICAL
      loadcases SELECT
        lcase1
      ANALYSIS OPTIONS
        LARGE DISPLACEMENT *(on)*
        NO FOLLOWER FORCE
          (to switch to FOLLOWER FORCE)
        OK
      JOB RESULTS
        SELECT VARIABLES
        ecauchy
        OK
      AXISYMMETRIC
      OK
    ELEMENT TYPES
      AXISYMMETRIC SOLID
        82 *(FULL & HERRMANN FORMULATION / QUAD (4))*
        OK
      all: EXIST.

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
```

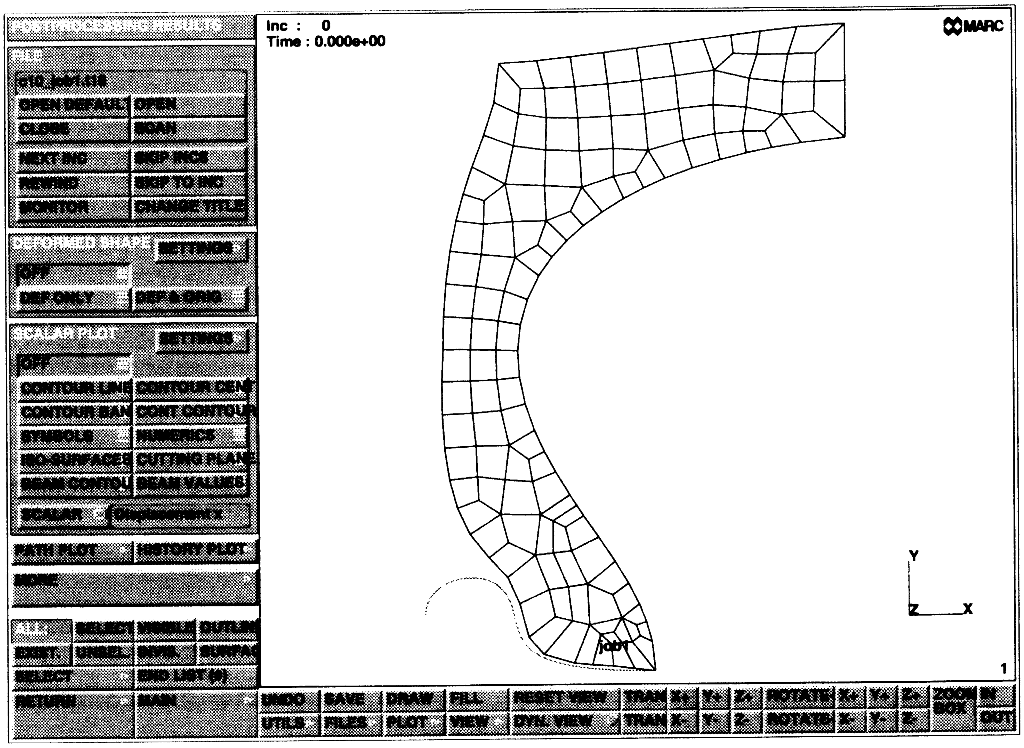


Figure 10.17 Mesh without Node Labeling and Face Identification

Click on DEF & ORIG to request the original and deformed structure to be shown. The animation buttons are in the second part of the post-processing 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.

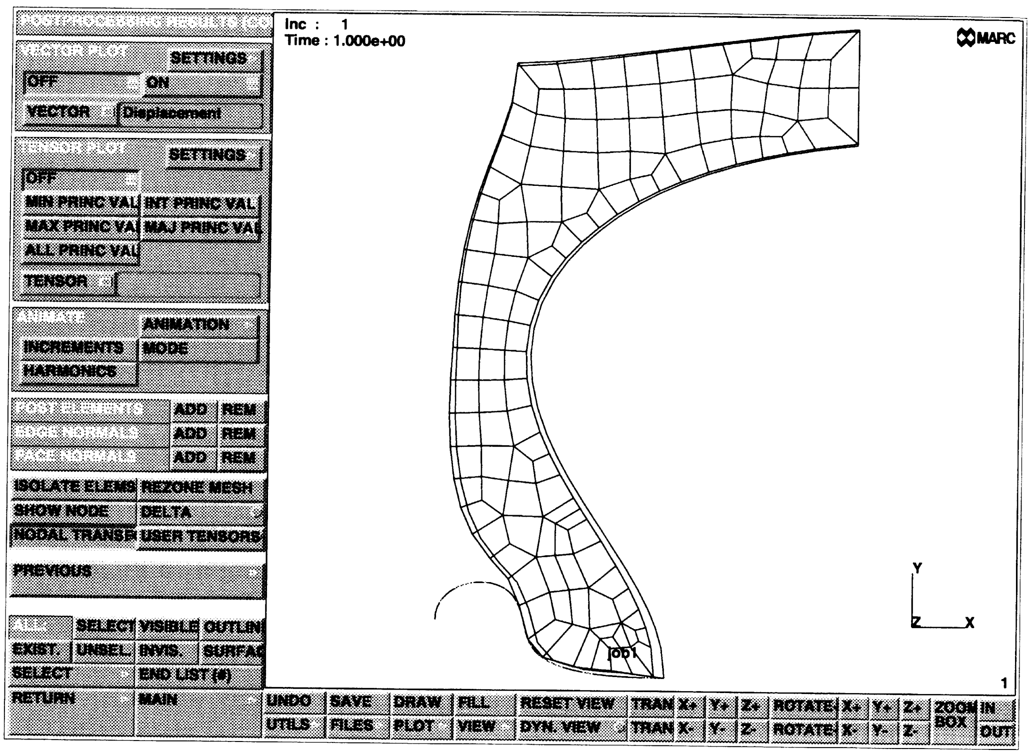


Figure 10.18 Second Animation Frame

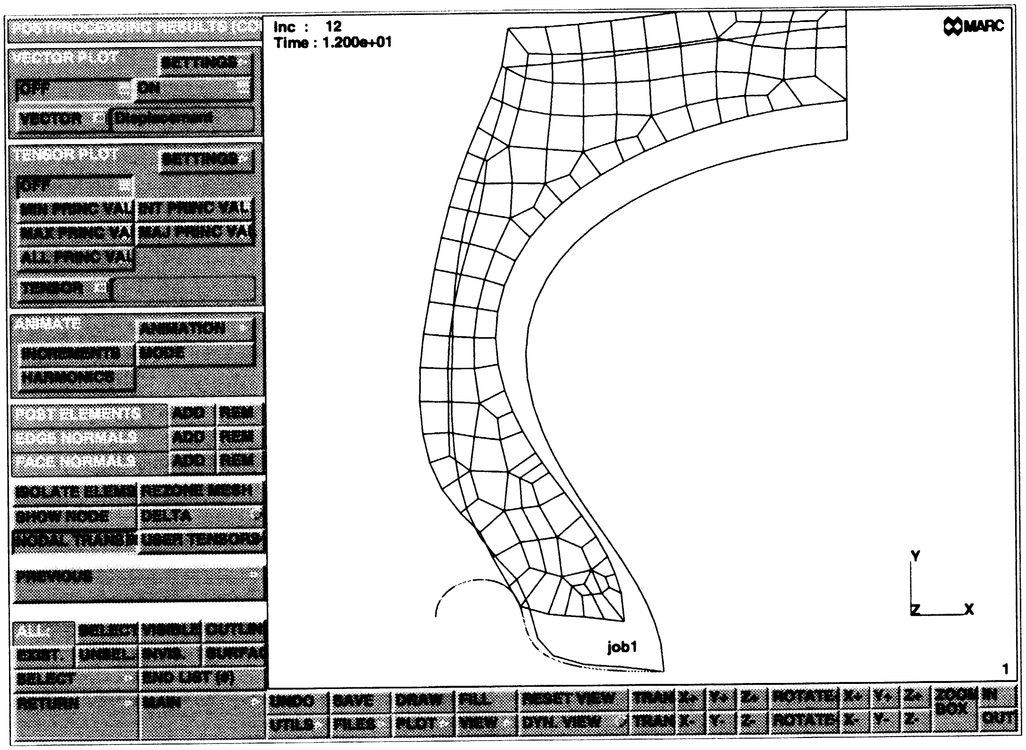


Figure 10.19 Last Animation Frame

To animate the sequence of frames use the following button sequence:

MAIN
 RESULTS
  FILL
  MORE
   ANIMATION
    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.

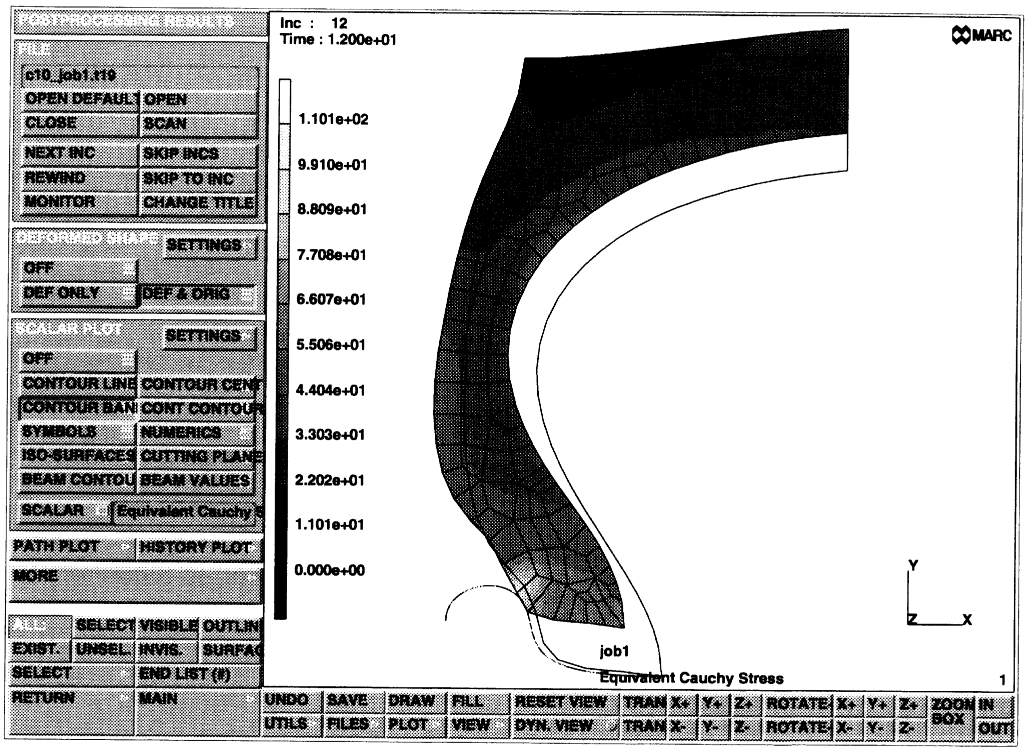


Figure 10.20 Mesh with Equivalent Cauchy Stress Contours

10.3 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.

10.4 Procedure File

```
| Version : MENTAT II 2.3
|
| This demo describes the analysis of the cross section
| of an automobile tire. The model is loaded by an
| internal pressure and contact with the rim
| is to be analyzed.
|
|
| Step 1: Create boundary using Bezier curves.
|
*set_grid_spacing
1 1
*set_grid_size
17 17
*set_grid on
*fill_view
*set_curve_divisions
20
*set_curve_type bezier
*zoom_box
*zoom_box(1,0.046591,0.567259,0.538636,0.019036)
*add_curves
point(0,14,0)
point(-11,13,0)
point(-11,6,0)
point(-5,4,0)
point(-5,0,0)
# | End of List
*set_grid_spacing
0.5
0.5
*add_curves
point(-9,16,0)
point(-9,14,0)
point(-10.5,13,0)
point(-10.5,8.5,0)
# | End of List
*set_point_labels on
*regen
*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
```

```
# | End of List
*edit_points
 13
-9.5 -0.5 0
*edit_nodes
10
*edit_points
10
-11 -0.5 0
*edit_points
11
-7 5 0
*add_curves
 6
point(-7.5,16,0)
point(-1.5,17,0)
point(0,17,0)
# | End of List
*zoom_box
*zoom_box(1,0.357955,0.383249,0.877273,0.025381)
*add_curves
 17
 1
# | End of List
|
| Step 2: automatic meshing (OVERLAY) to create mesh.
|
 *set_grid off
*fill_view
*set_overlay_divisions
20 20
*overlay_mesh
all_existing
*sub_divisions
2 1 1
*subdivide_elements
 52
 3
 5
 7
 61
# | End of List
*zoom_box
*zoom_box(1,0.275000,0.980964,0.613636,0.692893)
*refine_node
 1
 1
# | End of List
*subdivide_elements
 48
```

```
 47
# | End of List
*sweep_nodes
all_existing
*remove_unused_nodes
*renumber_all
|
| Step 3: Create the rim as a rigid body and identify
| the contact bodies
|
*fill_view
*zoom_box
*zoom_box(1,0.163636,0.989848,0.627273,0.520305)
*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)
# | End of List
|
 *set_grid off
*set_move_translations
0 13 0
*move_elements
all_existing
*move_curves
all_existing
*fill_view
*new_contact_body
*contact_deformable
*add_contact_body_elements
 1 2 3 4 5 6 7 43 44 45 46 47 48 49 50 51 52 53 54 55 56 57 58 59 103 104 105
 106 107 108 109 110 111 112 113 114 115 116 117 118 119
# | End of List
*new_contact_body
*add_contact_body_curves
 6
# | End of List
 *set_point_labels off
 *set_points off
*elements_solid
*identify_contact *regen
|
|
| Step 4: Add boundary conditions
|
 *identify_none *regen
*elements_wireframe
```

```
*fill_view
*apply_type fixed_displacement
*apply_dof x
*add_apply_nodes
 131 139 143
# | End of List
|
|
| Step 5: Apply internal pressure
|
*table_name
loading
*set_table_type
time
*set_table_xmax
300
*set_table_ymax
220
*table_add
0 0
300 214
*show_model
*new_apply
*apply_type edge_load
*apply_table p0
loading
*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
*regen
*material_type mechanical:mooney
*material_value mooney:c10
96.5
*material_value mooney:c20
-19.3
*material_value mooney:c30
19.3
*add_material_elements
all_existing
|
| Step 6: Submit the job
|
*loadcase_type static
*loadcase_value time
300
*loadcase_value nsteps
300
*job_class mechanical
*add_job_loadcases
```

```
lcase1
*job_option large:on
*job_option follow:on
*add_post_var
ecauchy
*job_option dimen:axisym
*element_type 82
all_existing
*save_as_model
c10
y
*update_job
*submit_job 1
*monitor_job
|
| Step 7: Post-process the results
|
*post_open_default
*post_next
 *set_nodes off
 *set_faces off
*regen
*set_deformed both
*post_animate_increments
100
1
*fill_view
*animation_play
*post_value
Equivalent Cauchy Stress
*post_contour_bands
```

Chapter 11: Transmission Tower

Chapter Overview

This chapter describes a sample session that illustrates the functionality of the Mentat II 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 II 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.

11.1 Background Information

11.1.1 Tower Description

The tower is 68 feet high, 18 feet square at the base, 4 feet square at the top, and has 6 cable-arms, 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.

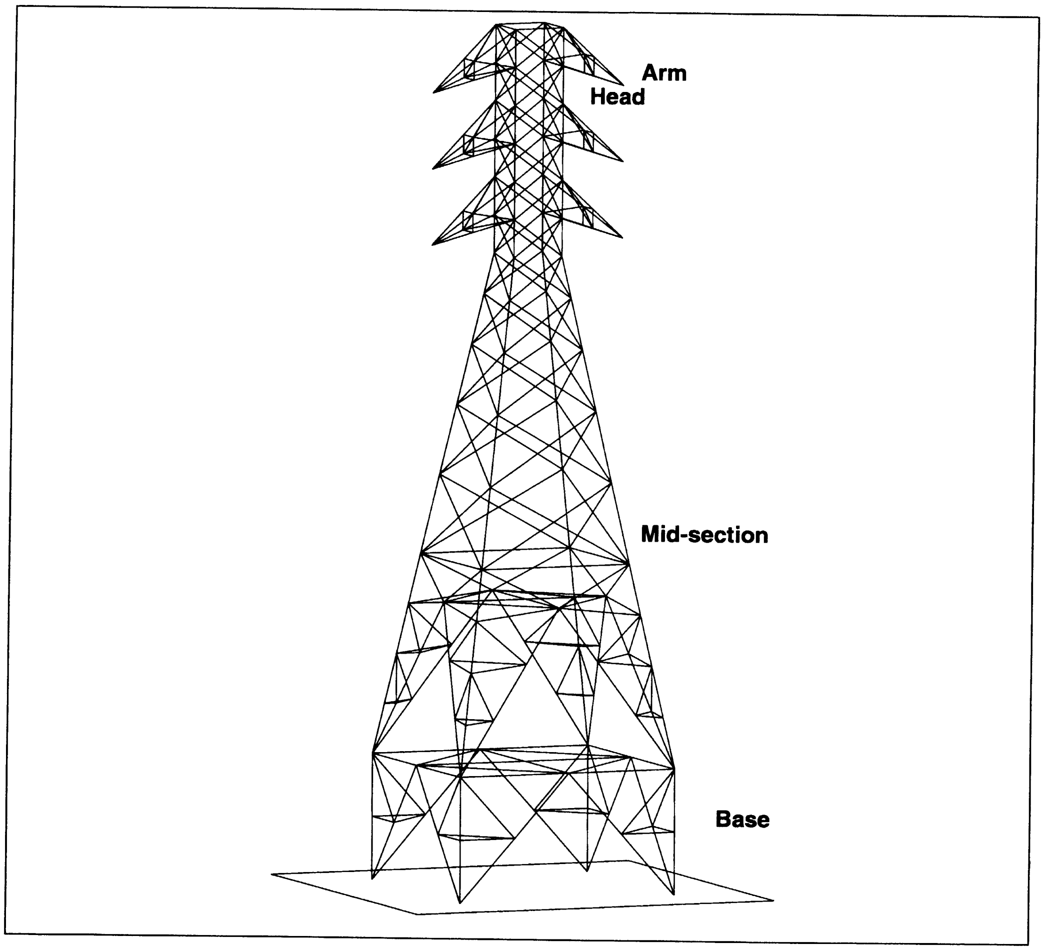


Figure 11.1 Transmission Tower

11.1.2 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.

11.1.3 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.

11.1.4 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/ft^3.

- **Elements**
  MARC Type 52, three-dimensional, two-noded beam element.

- **Element Properties**
  Those obtained from the AISC steel manual.
  L3x3x1/4
  Weight = 4.9 lbs/ft.
  Area = 0.01 ft^2.
  Ixx = Iyy = 6.0e-5 ft^4.

L2x2x1/4
Weight = 3.19 lbs/ft.
Area = 0.00651 ft^2.
Ixx = Iyy = 2.0e-5 ft^4.

11.1.5 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.

11.2 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 II 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 two-dimensional 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.

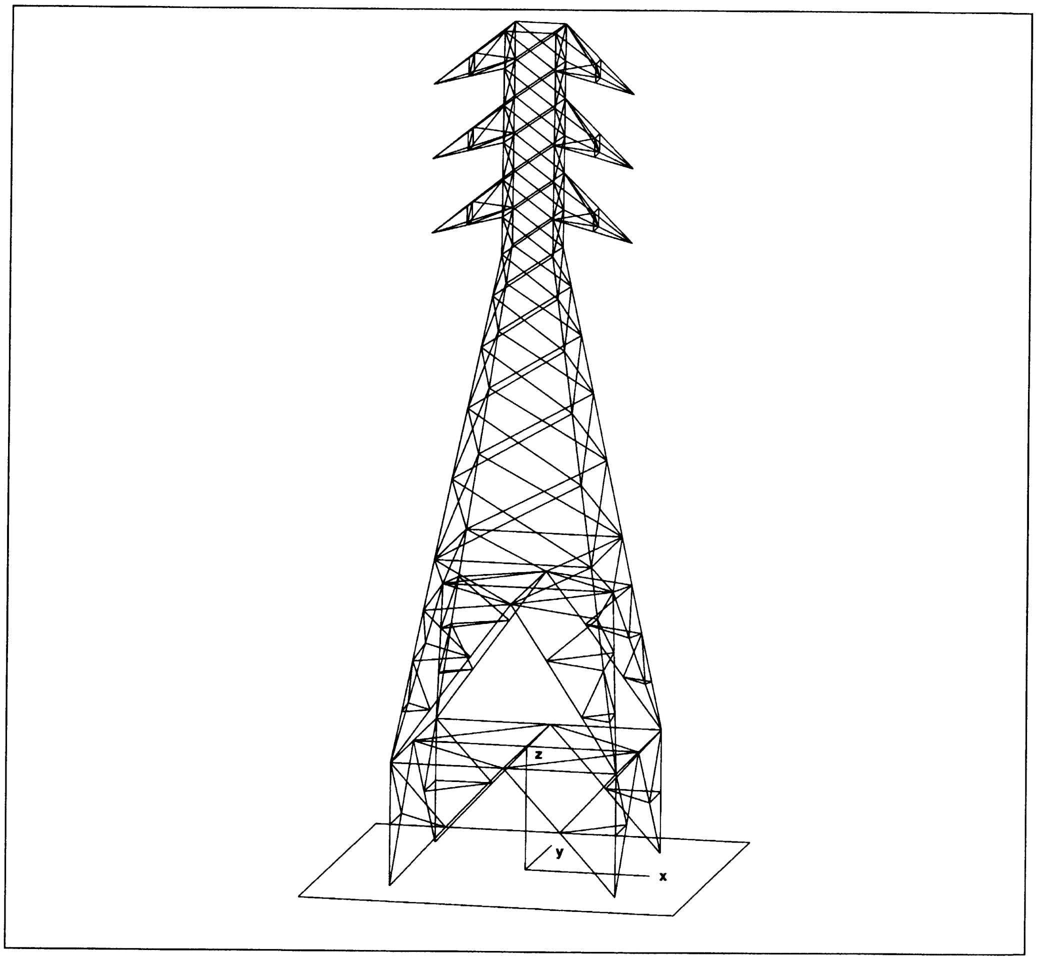


Figure 11.2 Local Coordinate System (u-v-w)

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 II 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
            SPACING
                1  1
            SIZE
                10  10
            grid 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)`	*(pick grid point)*
`node(-9,10,0)`	*(pick grid point)*
`node(0,10,0)`	*(pick grid point)*
`1`	*(pick node)*
`2`	*(pick node)*
`3`	*(pick node)*

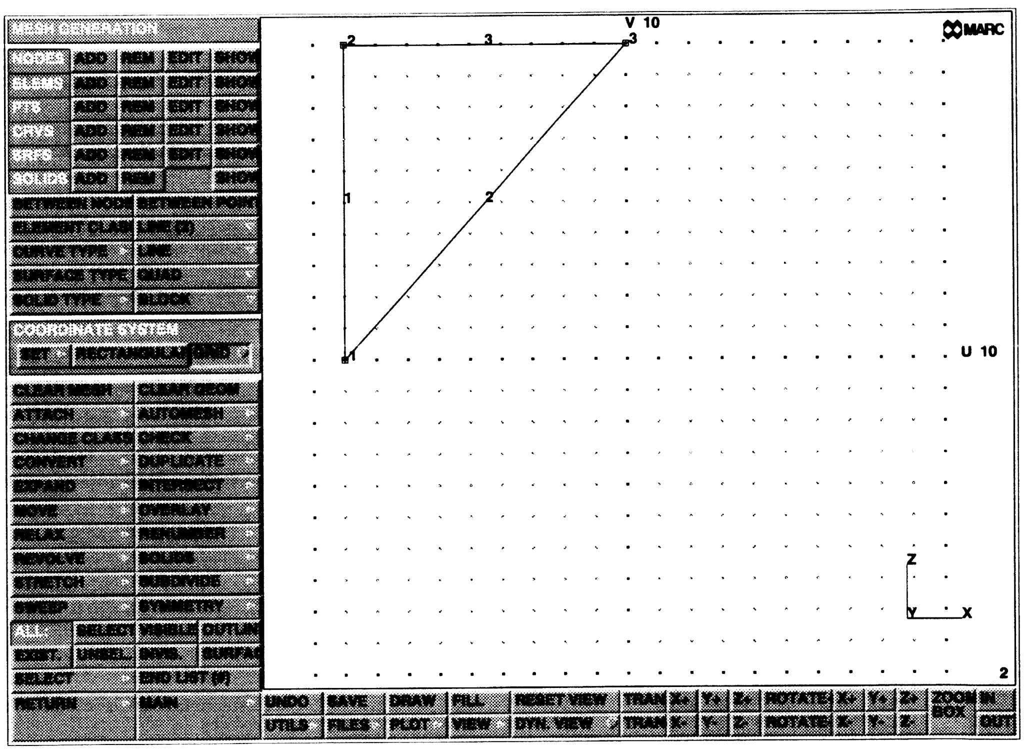


Figure 11.3 First Three Elements of Tower Base

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	*(pick node)*
8	*(pick node)*
8	*(pick node)*
6	*(pick node at upper left position)*

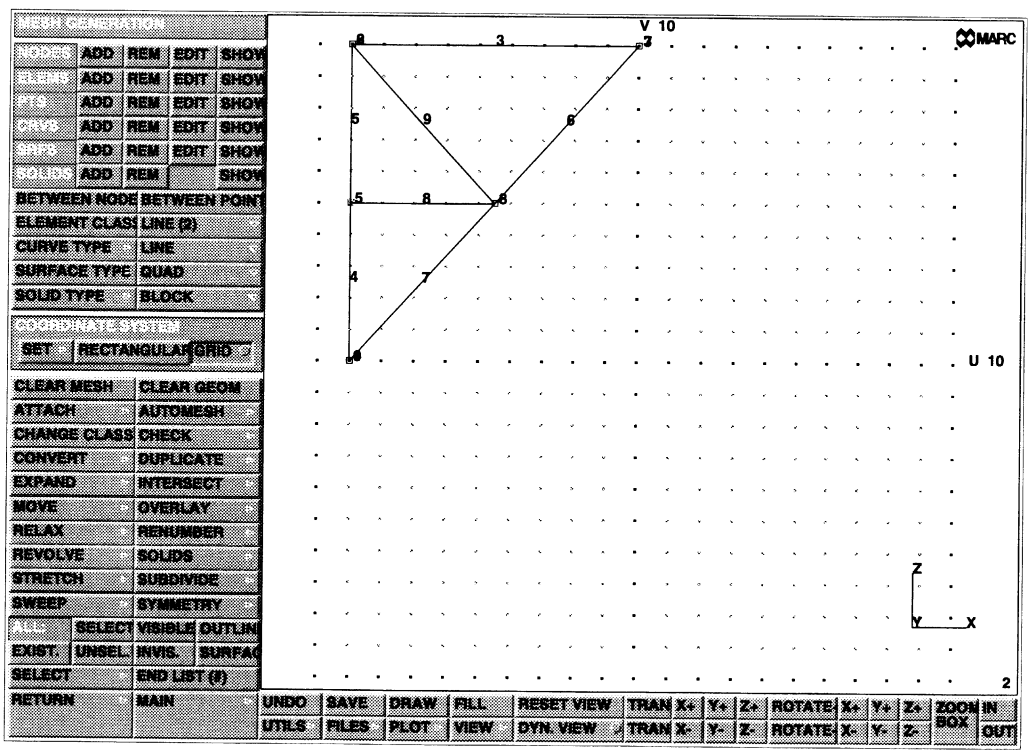


Figure 11.4 Left Base of Tower

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
SPACING
`2 2`
SIZE
`20 20`
TRANSLATE
`0 -7 50`
VIEW
activate 1 *(off)*
activate 2 *(off)*
activate 4 *(on)*
PERSPECTIVE
ACTIVATE ALL
SHOW ALL VIEWS
FILL
show 2
RETURN

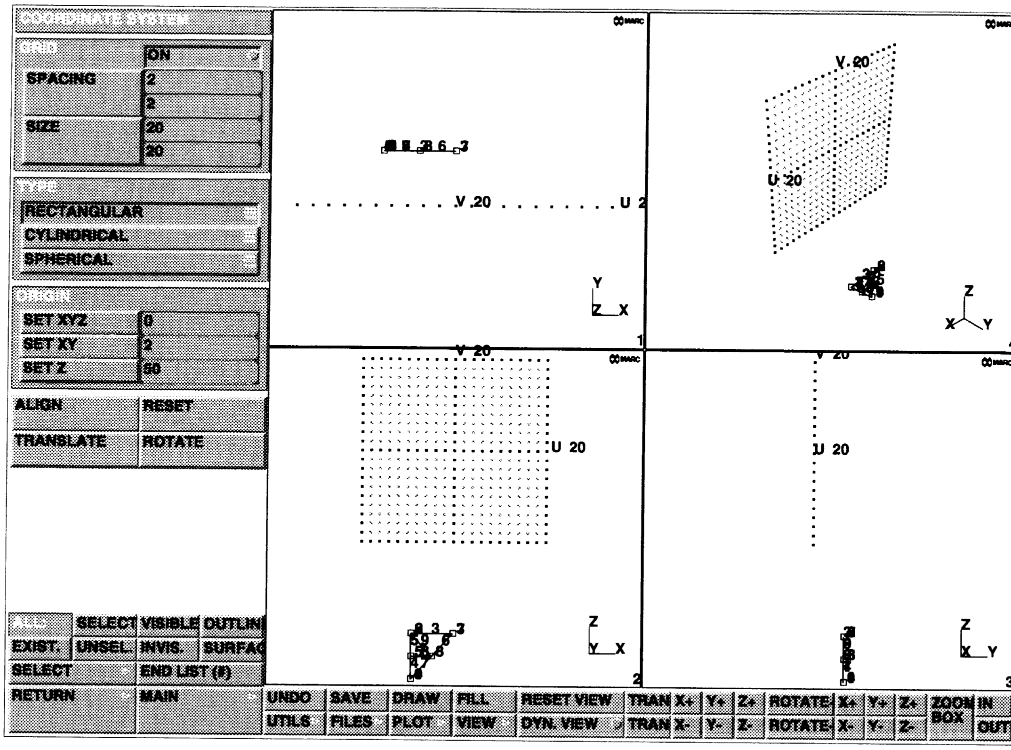


Figure 11.5 Four Views of New Local Coordinate System

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.

- MAIN
  - MESH GENERATION
    - elems ADD *(Pick from the grid)*
      - `node(-2,0,0)`
      - `node(-2,18,0)`
    - SUBDIVIDE
      - DIVISIONS
        - `6 2 2`
      - ELEMENTS
        - `10` *(the newly generated element)*
        - END LIST (#)

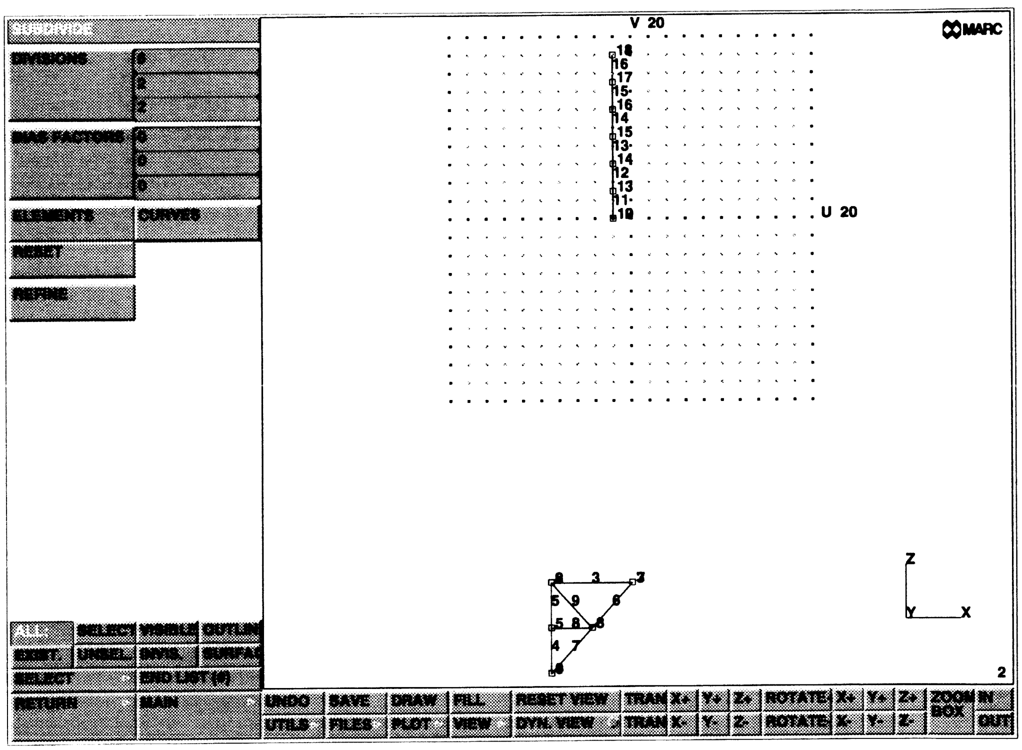


Figure 11.6 Creating the Left Side of the Tower Head

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
```

(upper right of lower section)
(upper left of lower section)
(bottom node of top section)

SPACING

```
1  1
```

SIZE

```
50  50
```

VIEW
show 3
RETURN

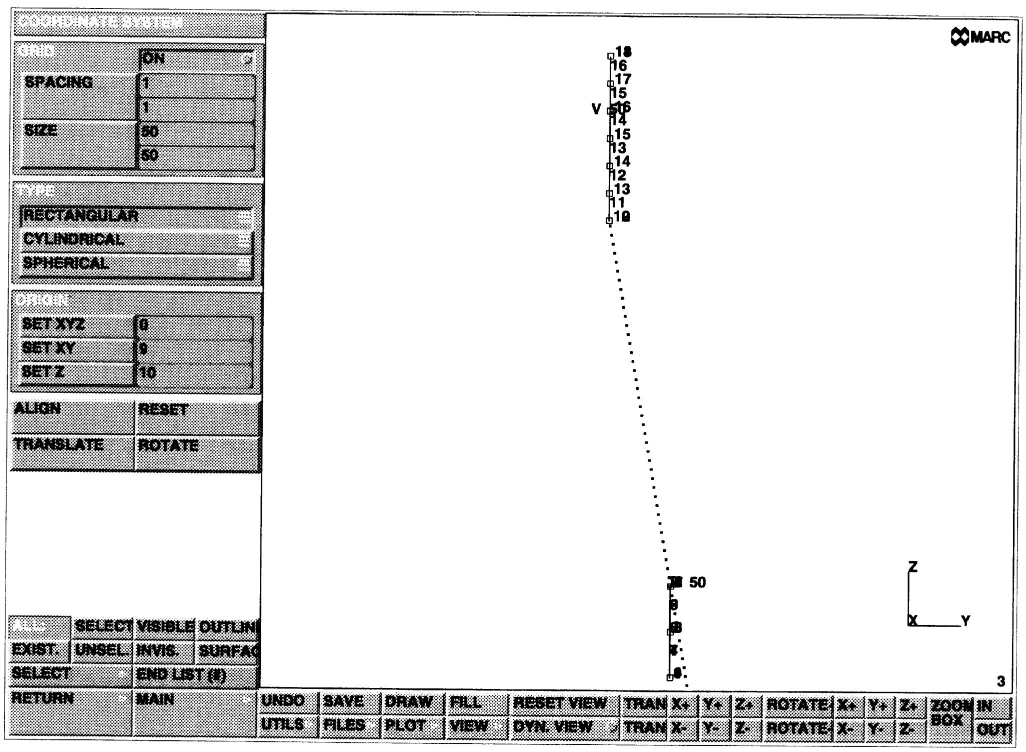


Figure 11.7 Local u and v Axes Aligned

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 $-0.5 \le b \le +0.5$.

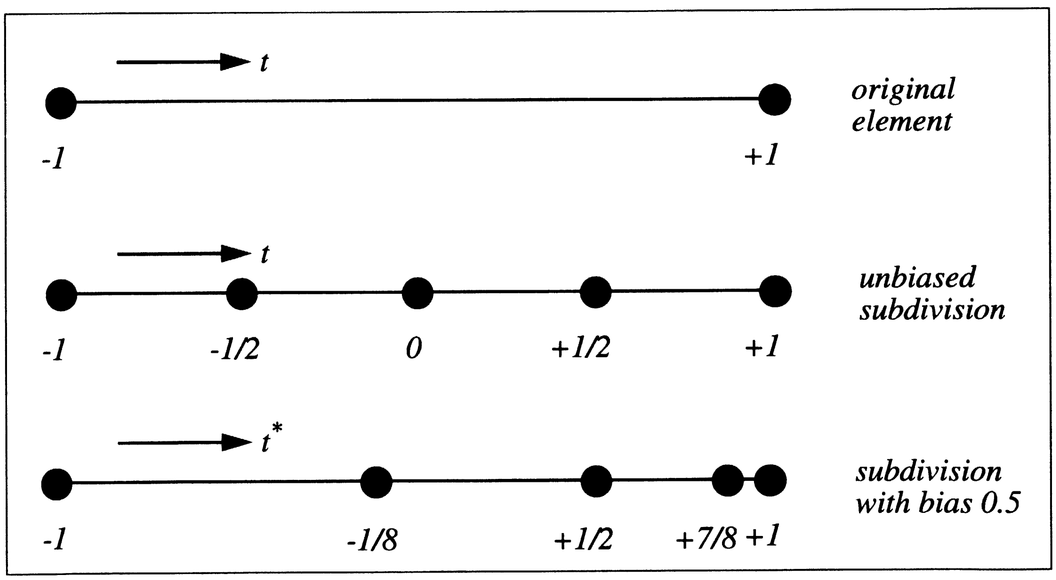


Figure 11.8 Subdivision of Line Element (Biased to the Right)

The mid-section consists of a biased and an unbiased part (LO-mid-section and HI-mid-section). 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` *(pick grid point)*
elems ADD
`6` *(pick upper left node of lower section)*
`12` *(pick bottom node of top section)*
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 (#)

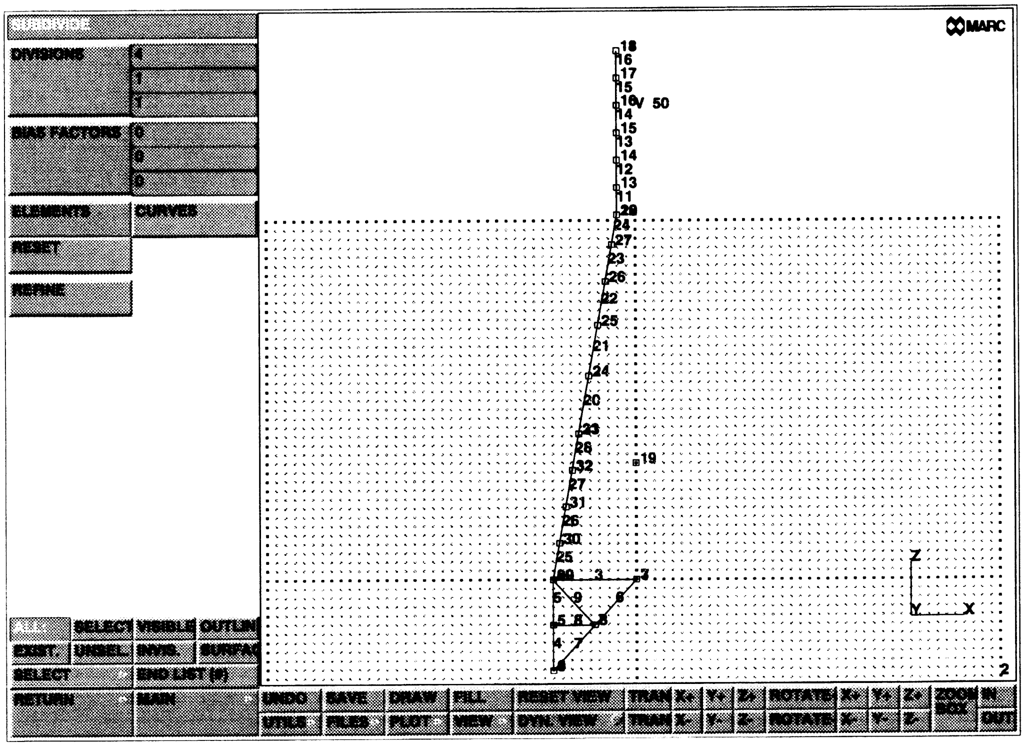


Figure 11.9 Subdivided Elements in the Mid-Section

As a next step, the lower-mid-section is completed and cross members are added in this part.

MAIN
    MESH GENERATION
        elems ADD
            `33` *(pick middle node of mid-section)*
            `19` *(pick auxiliary node at u= 0)*
            `19` *(pick auxiliary node at u= 0)*
            `29` *(pick upper left node of base)*
        SUBDIVIDE
            DIVISIONS
                `3 1 1`
            ELEMENTS
                `30`
                END LIST (#)

MAIN
MESH GENERATION
elems ADD
34 *(pick auxiliary node at u= 0)*
32 *(almost horizontal to the left)*
32 *(work the diagonals)*
35
35
31
31
36
36
30
PLOT
label ELEMENTS *(off)*
REGENERATE
RETURN

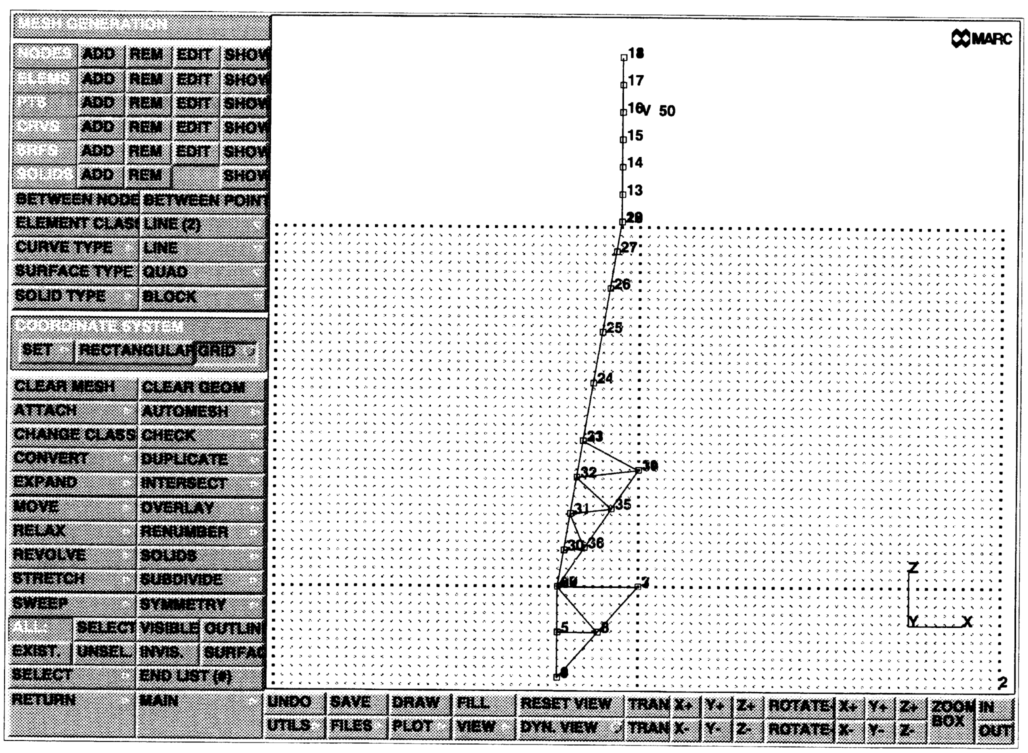


Figure 11.10 Completed Mid-Section

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 II.

MAIN
    MESH GENERATION
        SYMMETRY
            ELEMENTS
                all: EXIST.

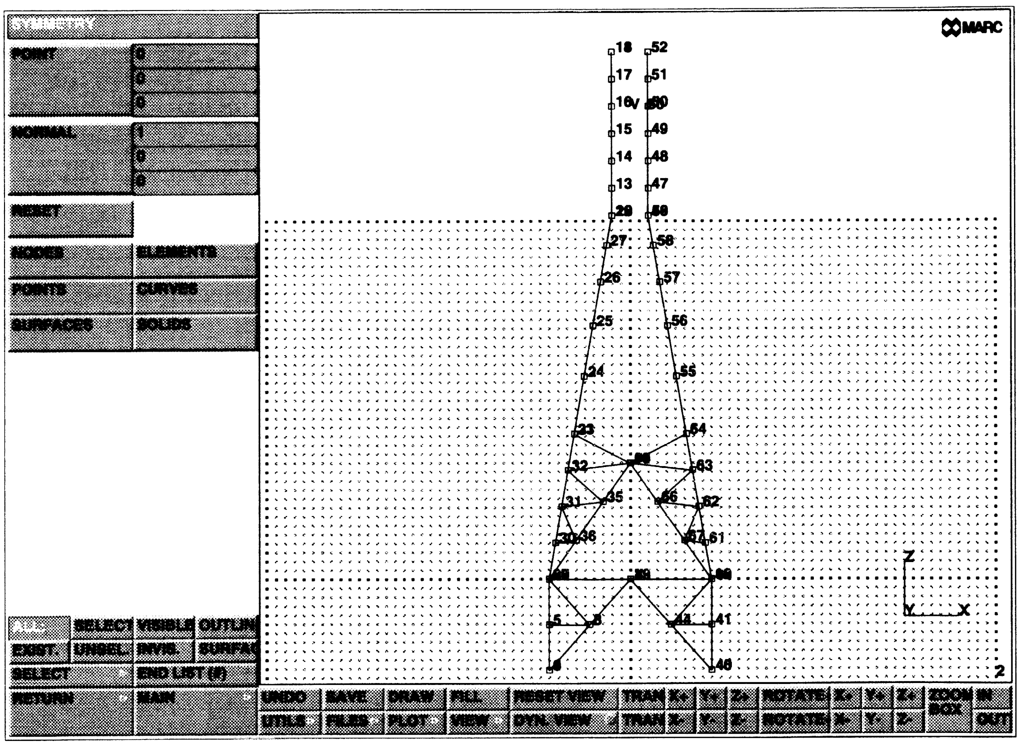


Figure 11.11 Cross Section of Tower Member

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

(first generate the horizontal cross member between nodes 33 and 64, then work the diagonals in the high-mid-section)

PLOT
label NODES *(off)*
REGENERATE
RETURN

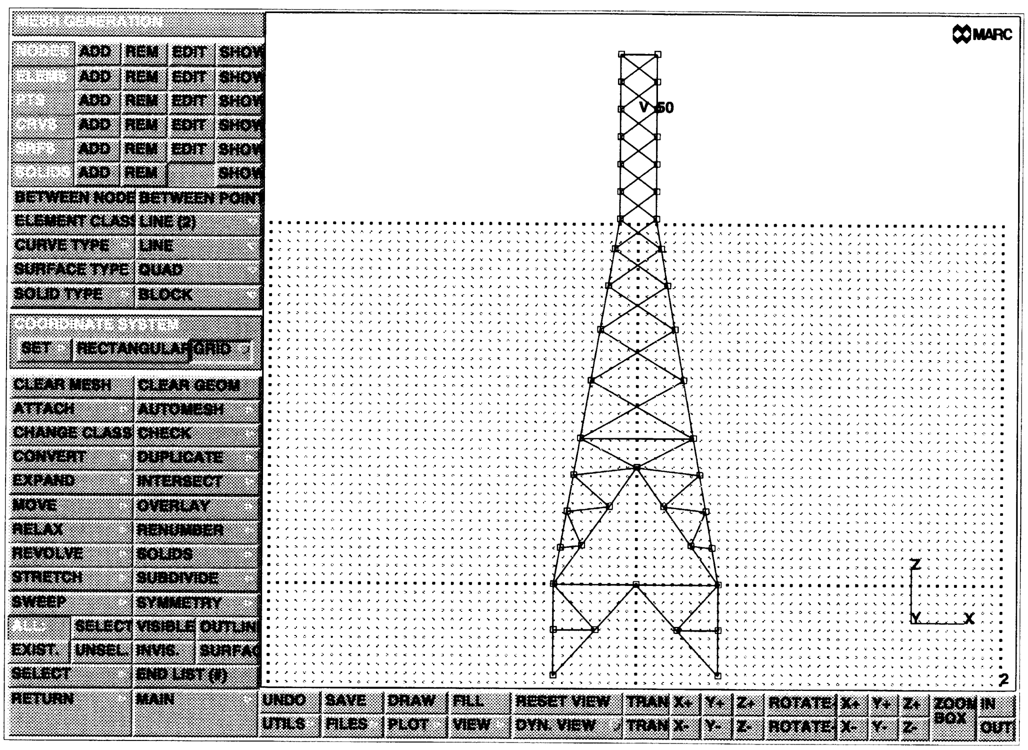


Figure 11.12 Tower Face Completed

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                                  (to switch off grid)
            RETURN
        DUPLICATE
            ROTATIONS
                0  0  90
            REPETITIONS
                3
            ELEMENTS
                all: EXIST.
            VIEW
                SHOW ALL VIEWS
                FILL
                RETURN
```

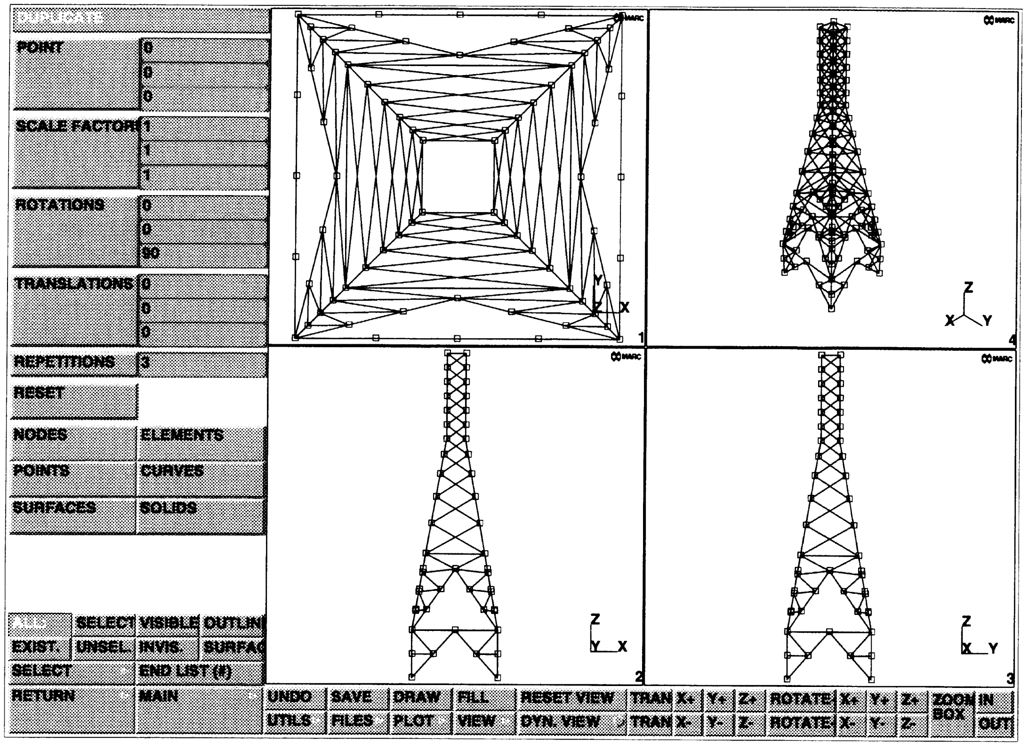


Figure 11.13 Four Views of Tower Face Duplication

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.

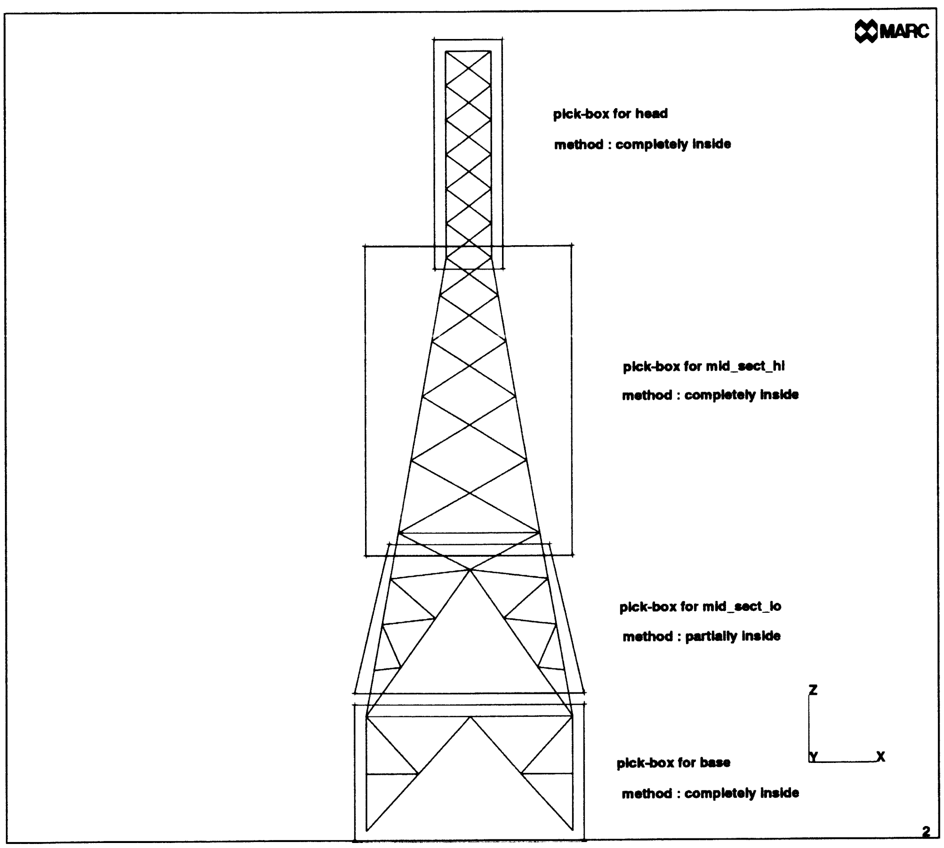


Figure 11.14 Boxes Used for Element Set Selection

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 +∞ and -∞ in the direction perpendicular to the screen. Refer to Section 2.5 for more details on list specification.

```
MAIN
    MESH GENERATION
        VIEW
            show 2
            RETURN
        SELECT
            elements STORE
                base
                                    (Box Pick the elements in the base of
                                         the tower, see Figure 11.14)
                END LIST (#)
MAIN
    DEVICE
        picking PARTIAL
        SELECT
            elements STORE
                mid_sect_lo
                                    (Box Pick the elements, realizing
                                    that all elements partially within
                                       the box will also be included)
                END LIST (#)
            RETURN
    DEVICE
        picking COMPLETE
        SELECT
            elements STORE
                mid_sect_hi
                                    (Box Pick the elements)
                END LIST (#)
            elements STORE
                head
                                    (Box Pick the elements)
                END LIST (#)
```

MAIN
PLOT
MORE
IDENTIFY
SETS
REGENERATE
NONE
REGENERATE

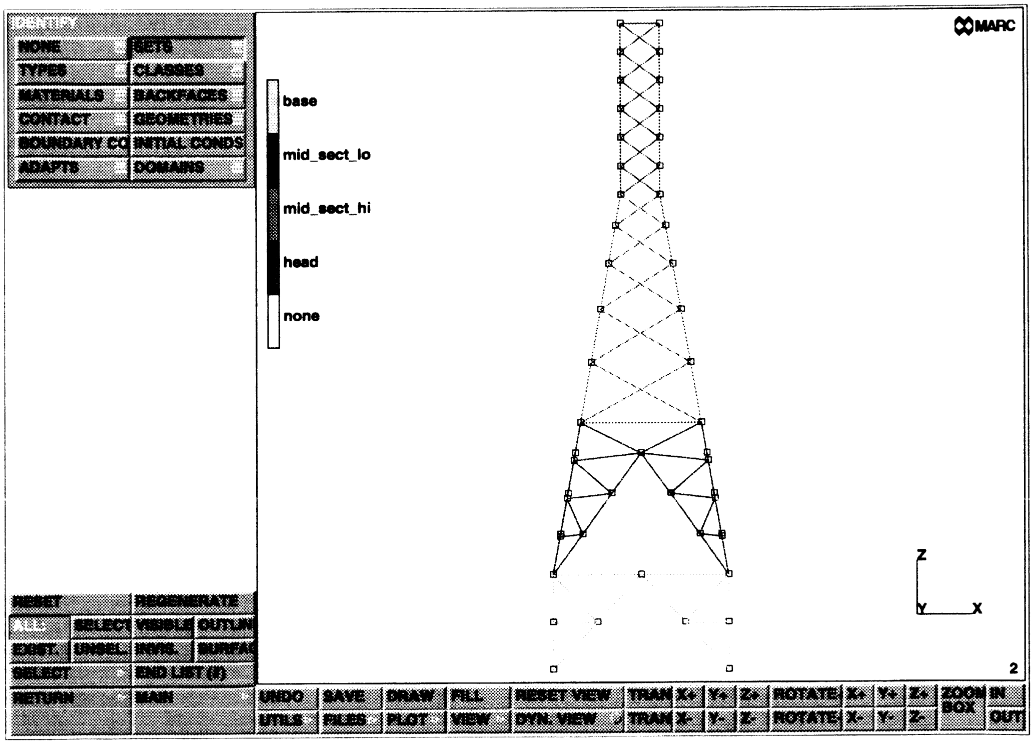


Figure 11.15 Element Sets Created

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.

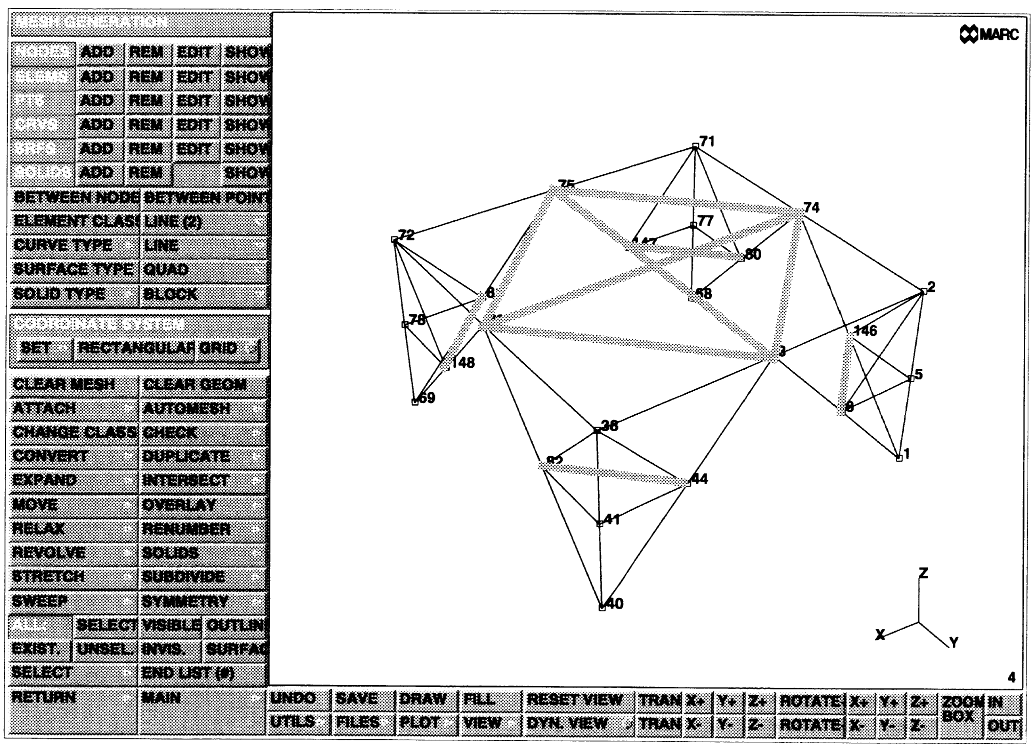


Figure 11.16 Elements To Be Added in the Base of the Tower

```
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
```

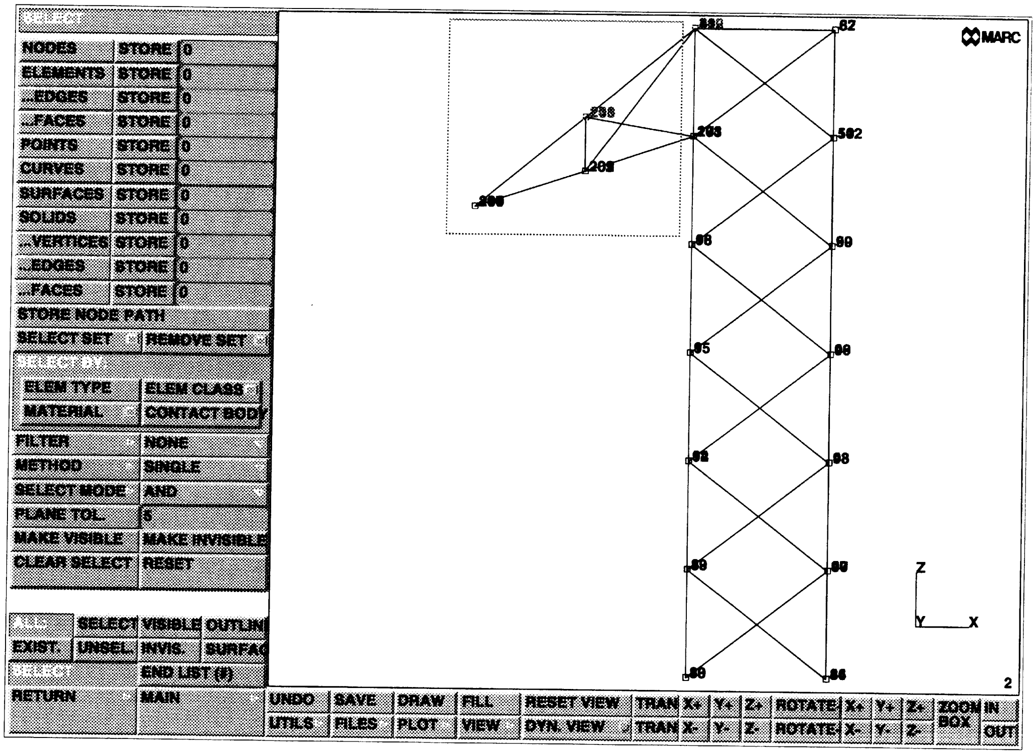


Figure 11.20 Box Pick of Transmission Tower Arm

Store the elements of the arm in an element set called *arm* for later reference.

MAIN
DEVICE
picking PARTIAL
SELECT
elements STORE
`arm`

(Box Pick the elements in the arm)

END LIST (#)
RETURN
picking COMPLETE

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 *(off)*
      - MORE
        - IDENTIFY
          - SETS
          - REGENERATE
          - NONE

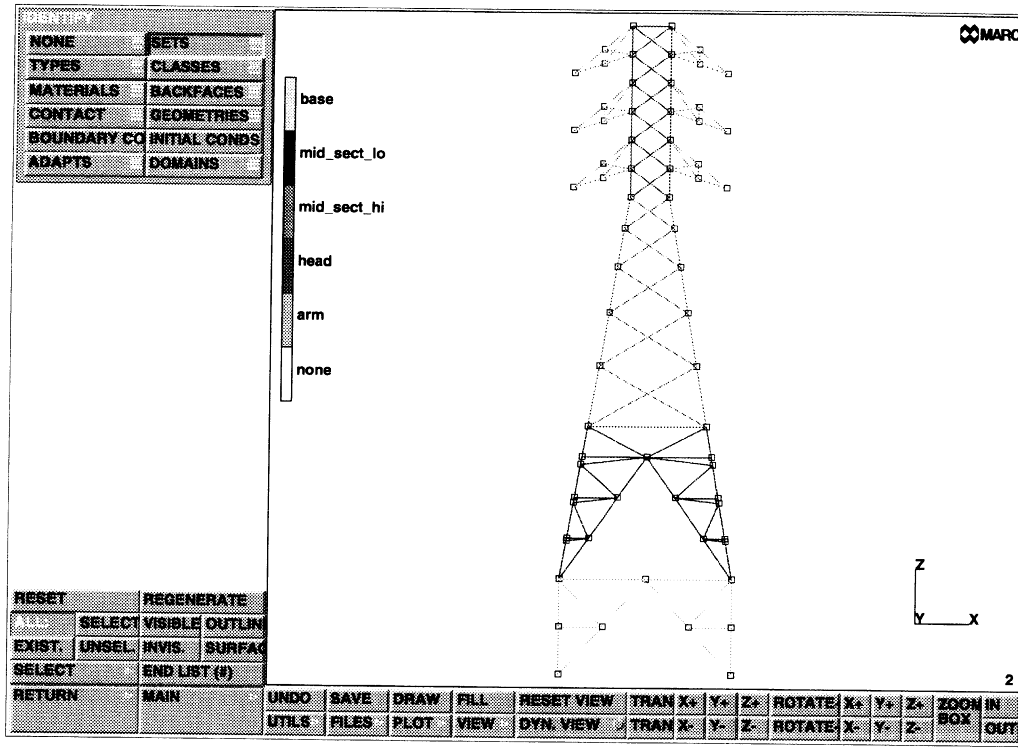


Figure 11.21 Completed Transmission Tower Model

You have now completed Steps 1 through 4 outlined in Section 11.1.5. 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.

This section outlined the advantage of using the symmetry of a structure in two-dimensional 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; correct is emphasized because a small mistake here can have catastrophic consequences. For example, if you apply incorrect boundary conditions, you may end up analyzing a completely different problem.

Step 5

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 load. 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 *(on)*
        ON y displace *(on)*
        ON z displace *(on)*
        OK
      nodes ADD
        (pick 4 nodes at base)
        END LIST (#)

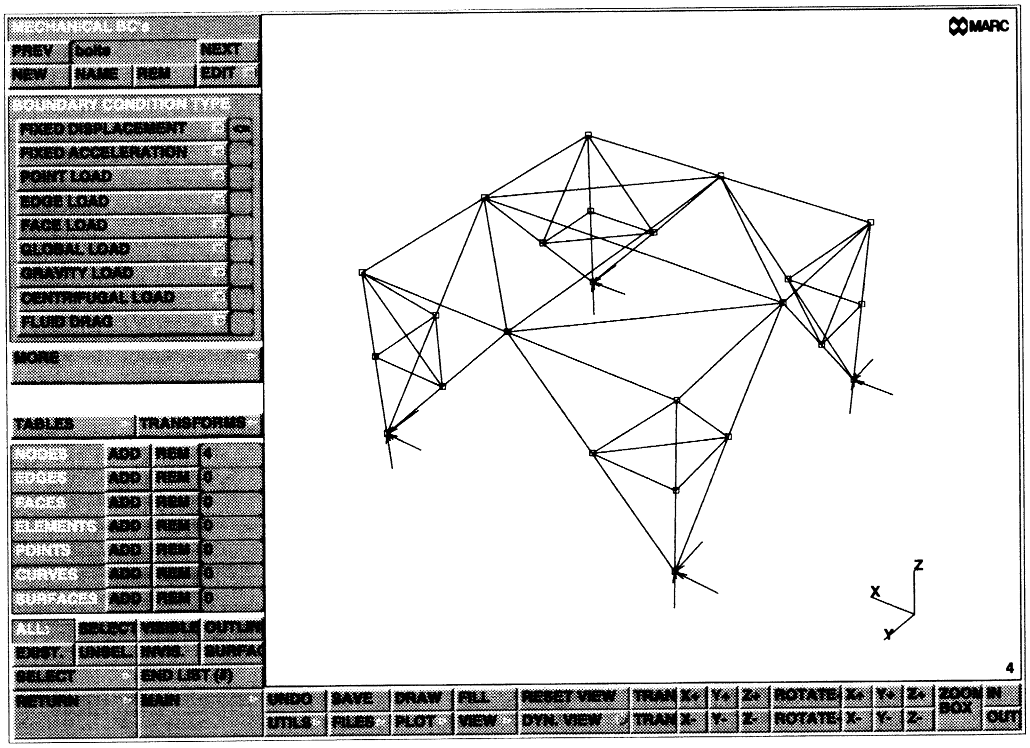


Figure 11.22 Boundary Conditions

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.

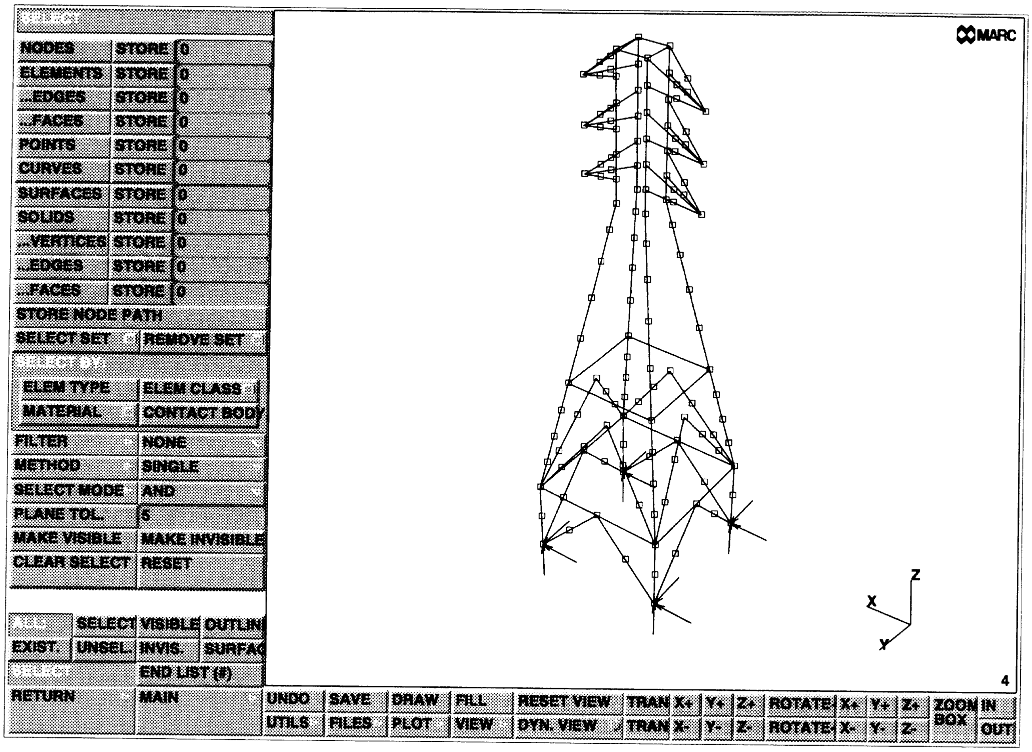


Figure 11.23 Elements To Be Contained in Set L3x3

MAIN
    MESH GENERATION
        SELECT
            elements STORE
                `L3x3`

(pick elements from base)

                END LIST (#)
            SELECT SET
                mid_sect_lo
                OK
            MAKE VISIBLE
            FILL
                elements STORE
                `L3x3`

(pick elements from mid_sect_lo)

                END LIST (#)

MAIN
  MESH GENERATION
    SELECT SET
      mid_sect_hi
      OK
    MAKE VISIBLE
    FILL
    elements STORE
      L3x3
      (pick elements from mid_sect_hi)
      END LIST (#)
    SELECT SET
      head
      OK
    MAKE VISIBLE
    FILL
    elements STORE
      L3x3
      (pick elements from head)
      END LIST (#)
    SELECT SET
      arm
      OK
    MAKE VISIBLE
    FILL
    elements STORE
      L3x3
      (pick elements from arm)
      END LIST (#)
    ELEMENTS
      all: EXIST.
    MAKE VISIBLE
    FILL
    ELEMENTS
      all: EXIST.
    select mode AND *(to switch to EXCEPT)*
      L3x3
    elements STORE
      L2x2
      all: SELECT.

- MAIN
  - MESH GENERATION
    - SELECT
      - CLEAR SELECT
      - RESET
      - PLOT
        - MORE
        - IDENTIFY
          - SETS
          - REGENERATE
          - NONE
          - REGENERATE

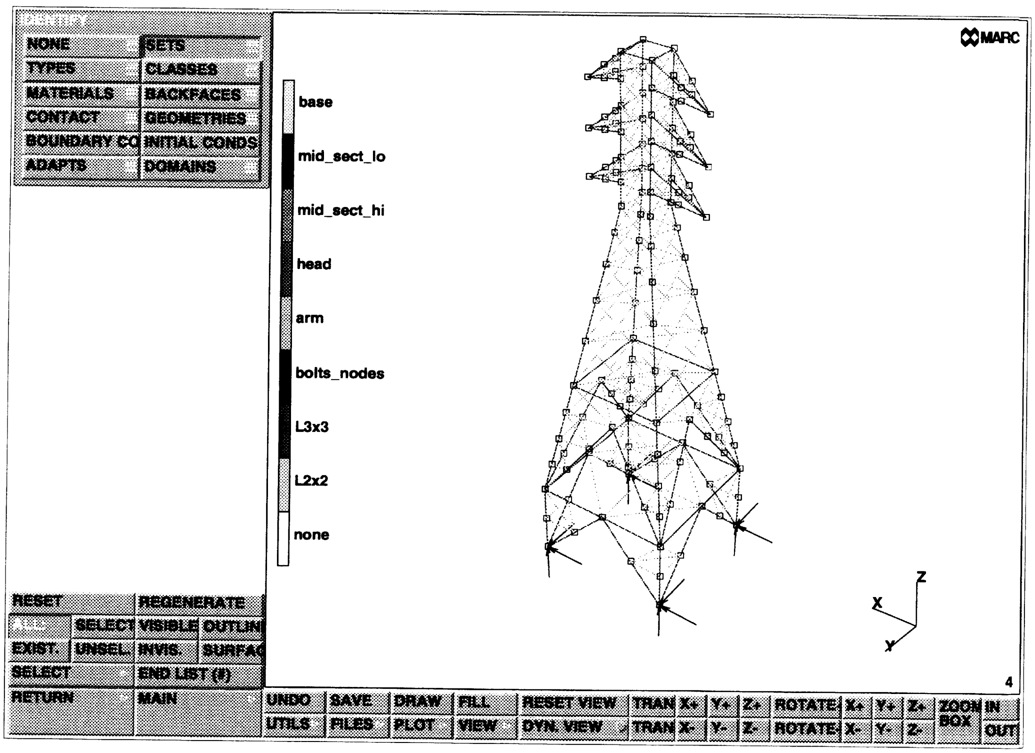


Figure 11.24 Identify Sets

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.

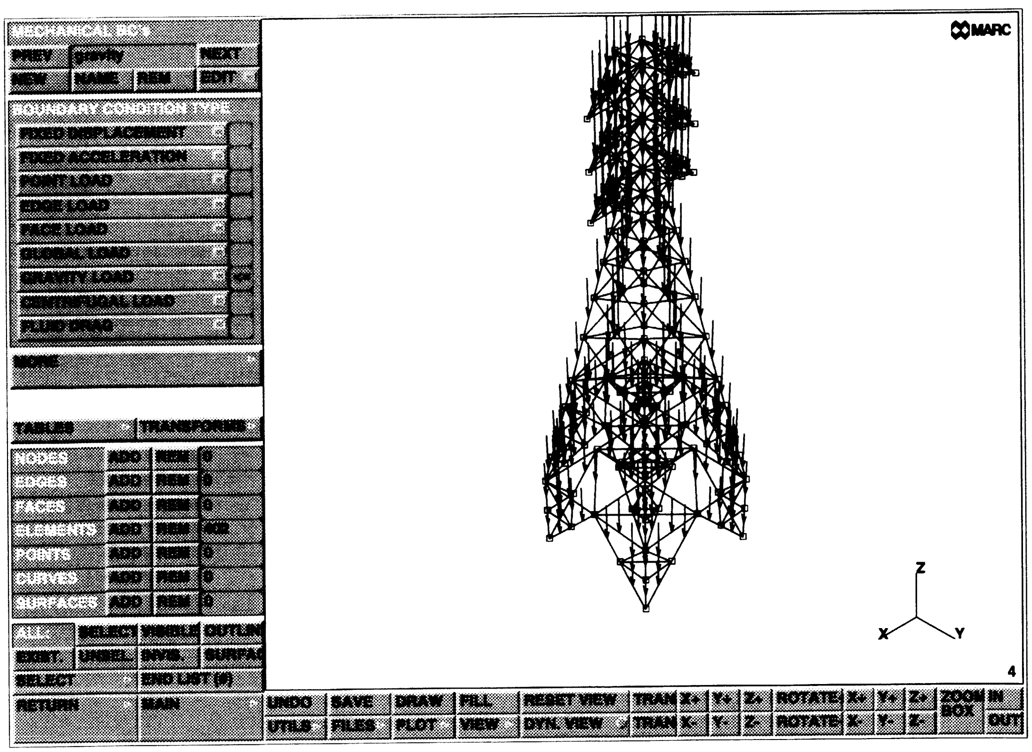


Figure 11.25 Gravity Load for the Structure

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.

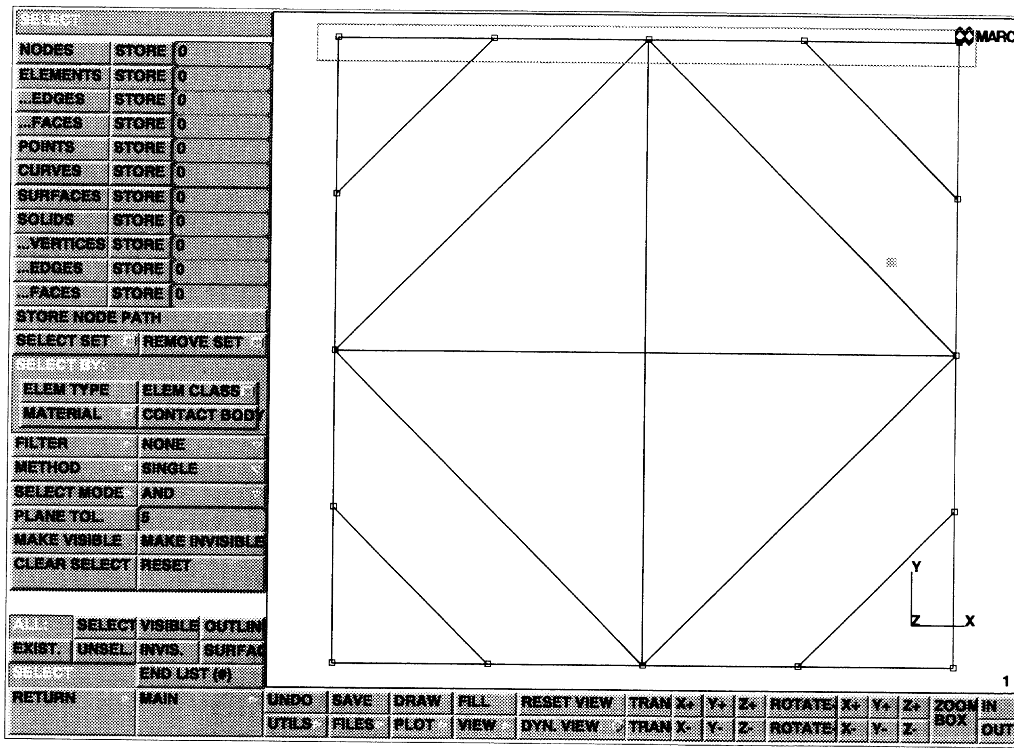


Figure 11.26 Box Pick from base for lo_front

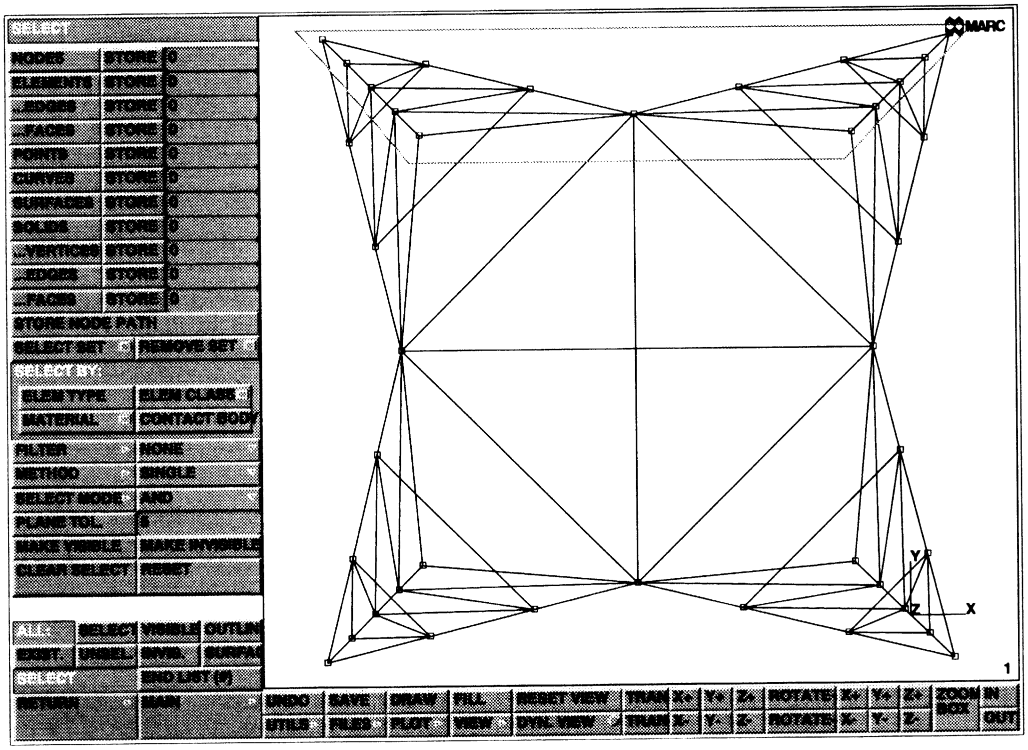


Figure 11.27 Polygon Pick from mid_sect_lo for lo_front

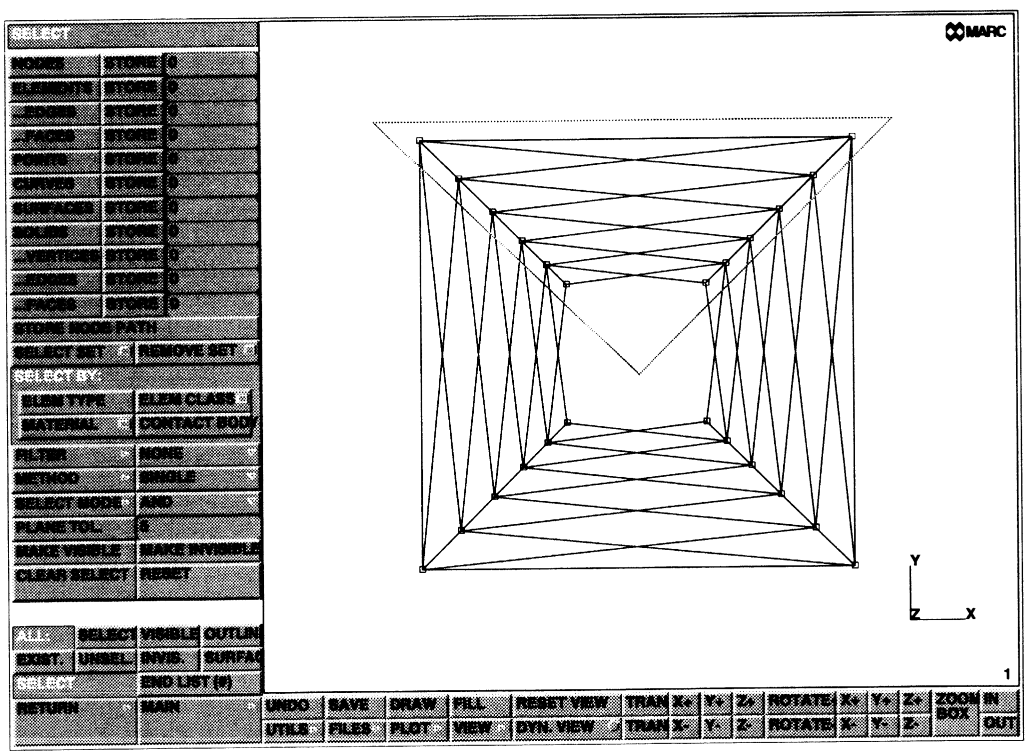


Figure 11.28 Polygon Pick from mid_sect_hi for hi_front

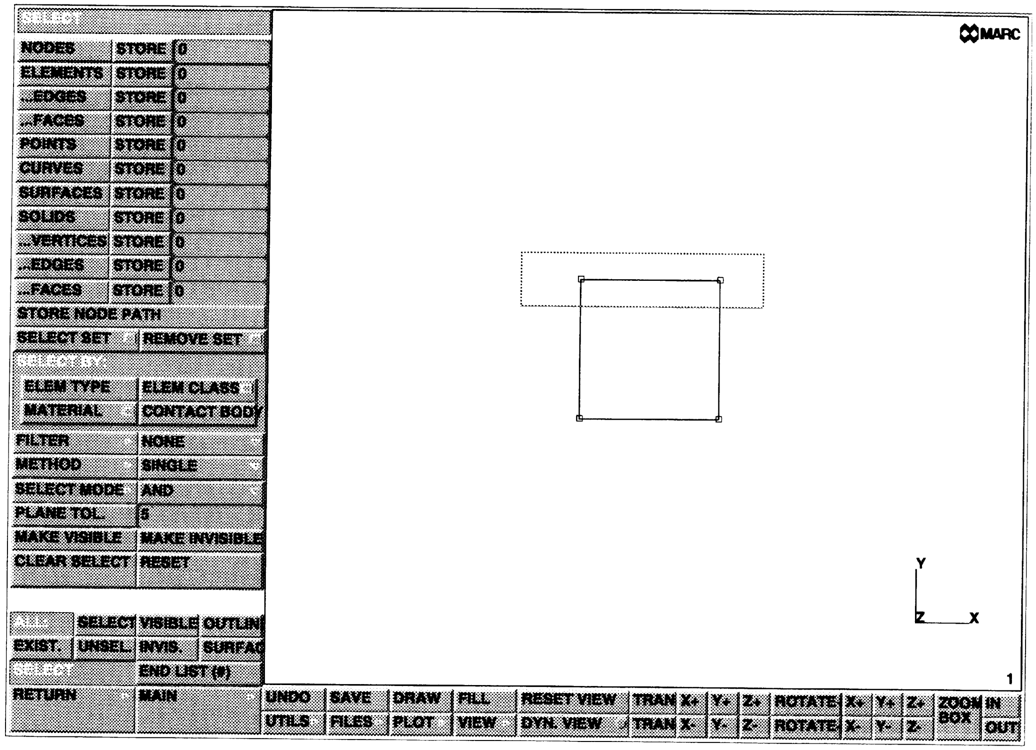


Figure 11.29 Box Pick from head for hi_front

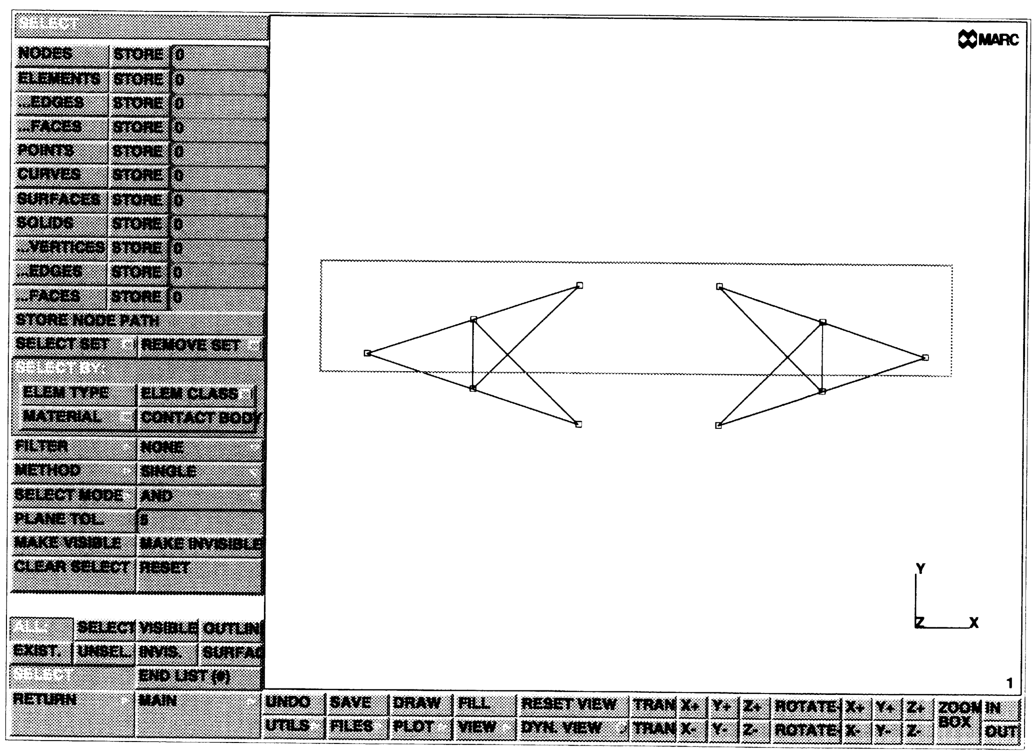


Figure 11.30 Box Pick from arm for hi_front

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:

MAIN
BOUNDARY CONDITIONS
MECHANICAL
NEW
NAME
`hi_wind`
GLOBAL LOAD
Y FORCE
`-120`
OK
elements ADD
`hi_front`

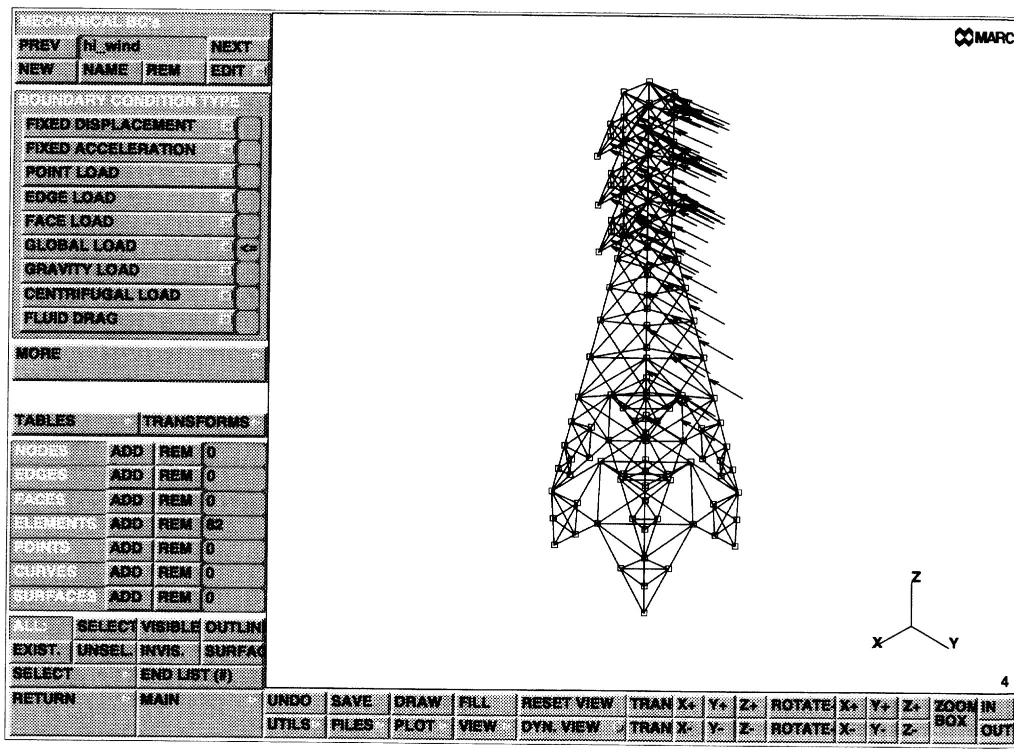


Figure 11.31 Strong Wind Load

In an identical way a Y FORCE of -80 can be applied to all elements contained in lo_front.

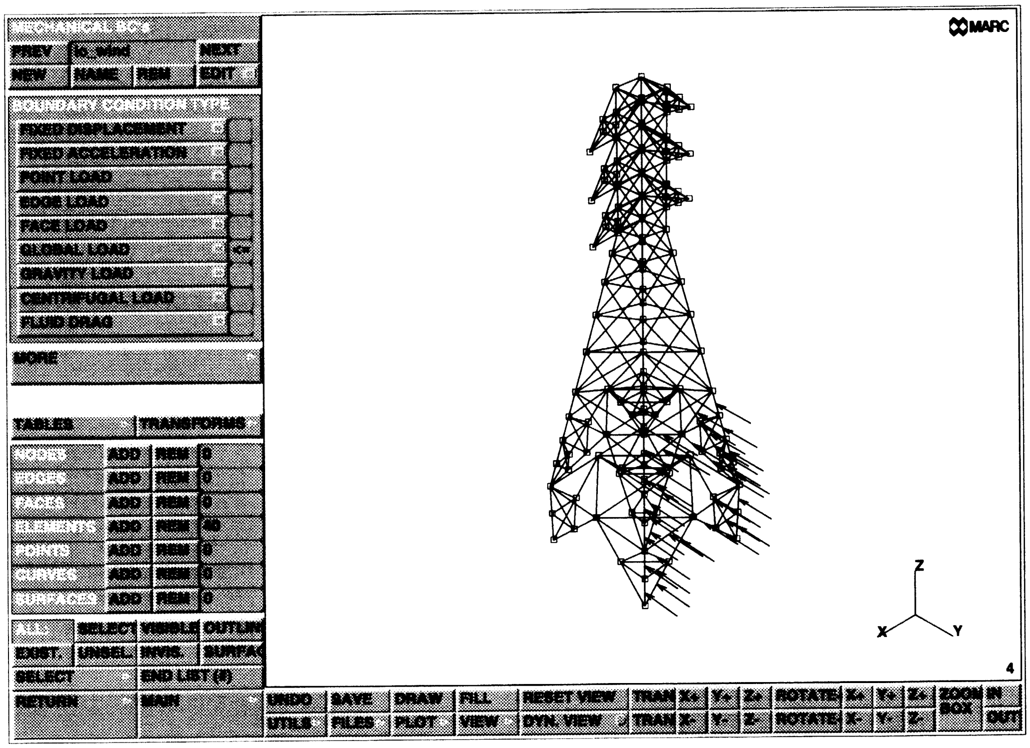


Figure 11.32 Weak Wind Load

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
(pick the 6 nodes on the tip of the arms)
END LIST (#)

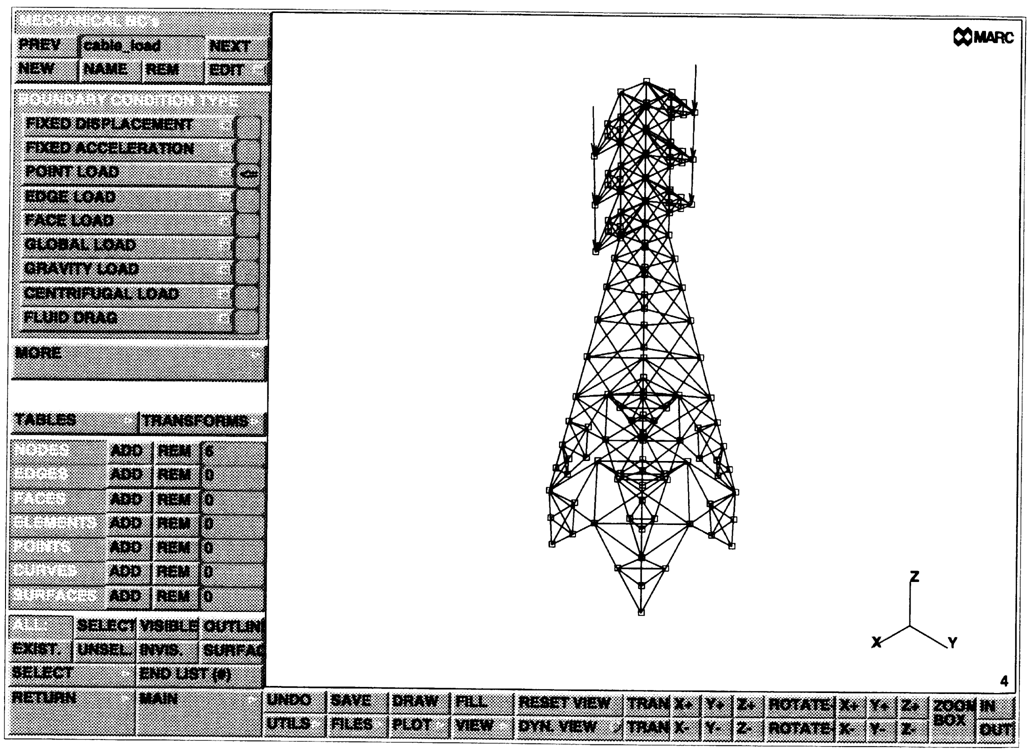


Figure 11.33 Cable Loads

Step 6

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 in section 1.4 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 one 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                          (for Ixx)
                    6.0e-05                          (for Iyy)
                    0                                (for direction)
                    0                                (for direction)
                    1                                (for direction)
                OK
            SELECT
                SELECT SET
                    L3x3
                    OK
                select mode AND                      (to switch to EXCEPT)
                SELECT SET
                    upright
                    OK
                RETURN
            elements ADD
                all: SELECT.
            ID GEOMETRIES                            (on)
```

MAIN
    GEOMETRIC PROPERTIES
        3-D
            NEW
            NAME
                `L3x3_x`
            ELASTIC BEAM
                AREA
                    `0.01`
                    `6.0e-05` *(for Ixx)*
                    `6.0e-05` *(for Iyy)*
                    `1` *(for direction)*
                    `0` *(for direction)*
                    `0` *(for direction)*
                OK
            SELECT
                CLEAR SELECT
                RESET
                SELECT SET
                    upright
                    OK
                select mode AND *(to switch to EXCEPT)*
                SELECT SET
                    L2x2
                    OK
                RETURN
            elements ADD
                all: SELECT.

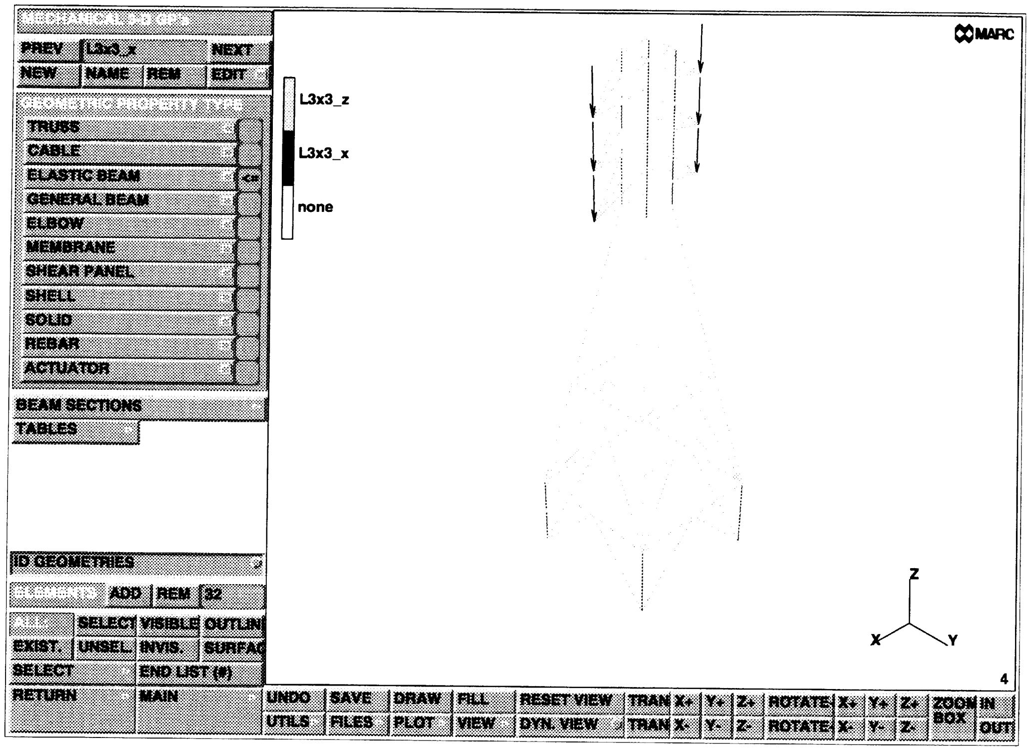


Figure 11.34 Geometric Properties Assignment for L3x3 Angles

MAIN
  GEOMETRIC PROPERTIES
    3-D
      NEW
      NAME
        `L2x2_z`
      ELASTIC BEAM
        AREA
          `0.00651`
          `2.0e-05` *(for Ixx)*
          `2.0e-05` *(for Iyy)*
          `0` *(for direction)*
          `0` *(for direction)*
          `1` *(for direction)*
        OK
      SELECT
        CLEAR SELECT
        RESET
        SELECT SET
          L2x2
          OK
        select mode AND *(to switch to EXCEPT)*
        SELECT SET
          upright
          OK
        RETURN
      elements ADD
        all: SELECT.

MAIN
    GEOMETRIC PROPERTIES
        3-D
            NEW
            NAME
                `L2x2_x`
            ELASTIC BEAM
                AREA
                    `0.00651`
                    `2.0e-05` *(for Ixx)*
                    `2.0e-05` *(for Iyy)*
                    `1` *(for direction)*
                    `0` *(for direction)*
                    `0` *(for direction)*
                OK
            SELECT
                CLEAR SELECT
                RESET
                SELECT SET
                    upright
                    OK
                select mode AND *(to switch to EXCEPT)*
                SELECT SET
                    L3x3
                    OK
                RETURN
            elements ADD
                all: SELECT.
            SELECT
                CLEAR SELECT
                RESET

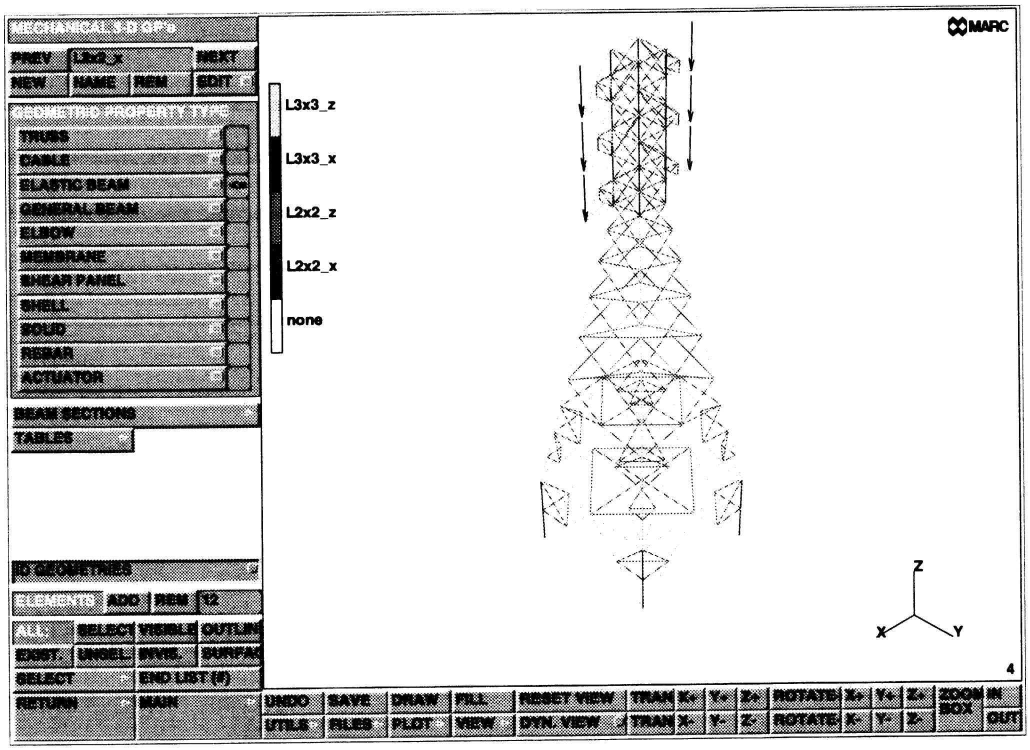


Figure 11.35 Geometric Properties Assignment for L2x2 Angles

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` *(for Poisson's Ratio)*

`15.217` *(for mass density)*

OK

elements ADD

all: EXIST.

ID MATERIALS *(on)*

ID MATERIALS *(off)*

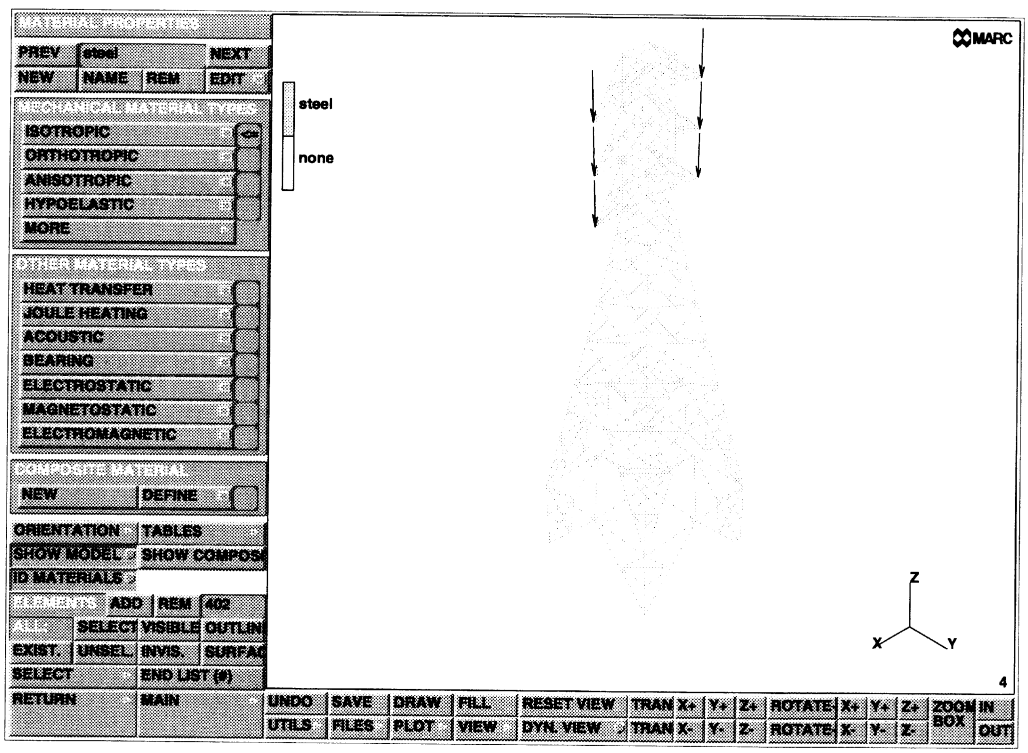


Figure 11.36 Material Properties Assignment

Step 7

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

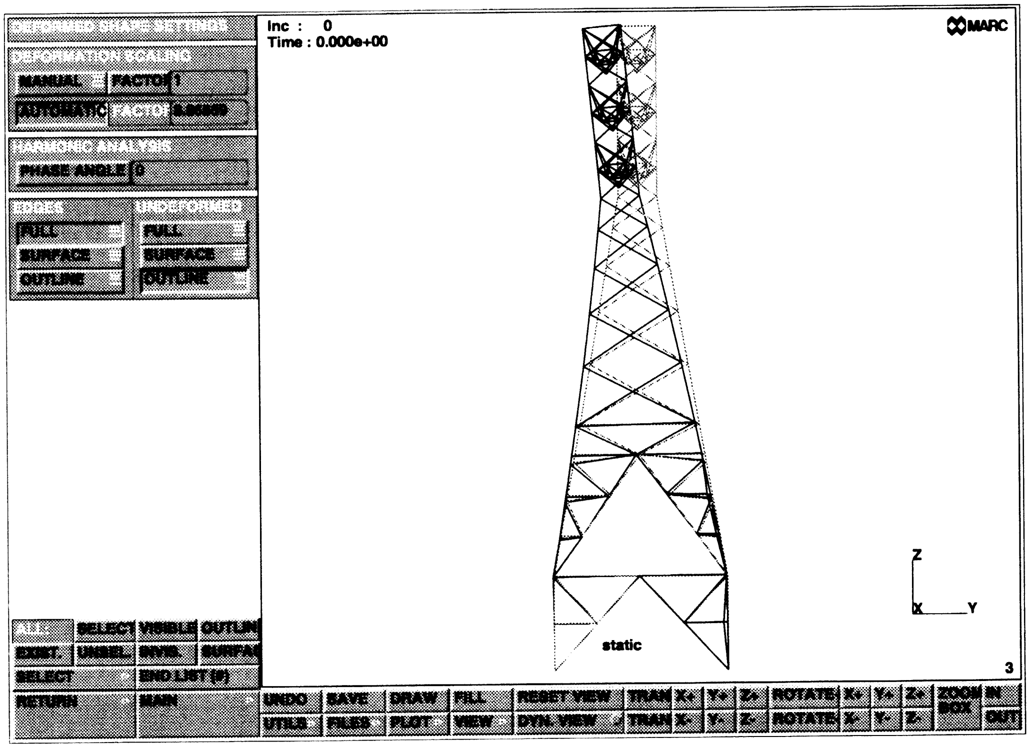


Figure 11.37 Deformation of Tower under Combined Load

Step 9

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
```

Step 10

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 mode 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
                REPEAT
                PLAY
                STOP
                RETURN
            PREVIOUS
        SKIP TO INC
            0:15
        DEF & ORIG
        FILL
```

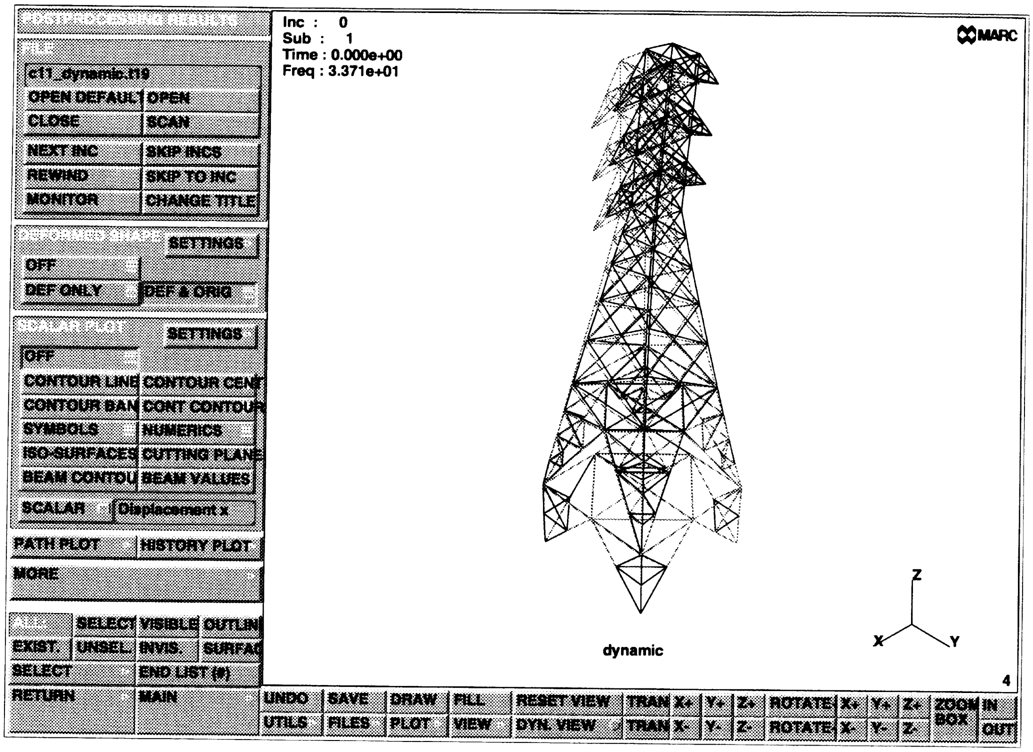


Figure 11.38 Eigenmode of Tower, f = 33.71 radians/sec

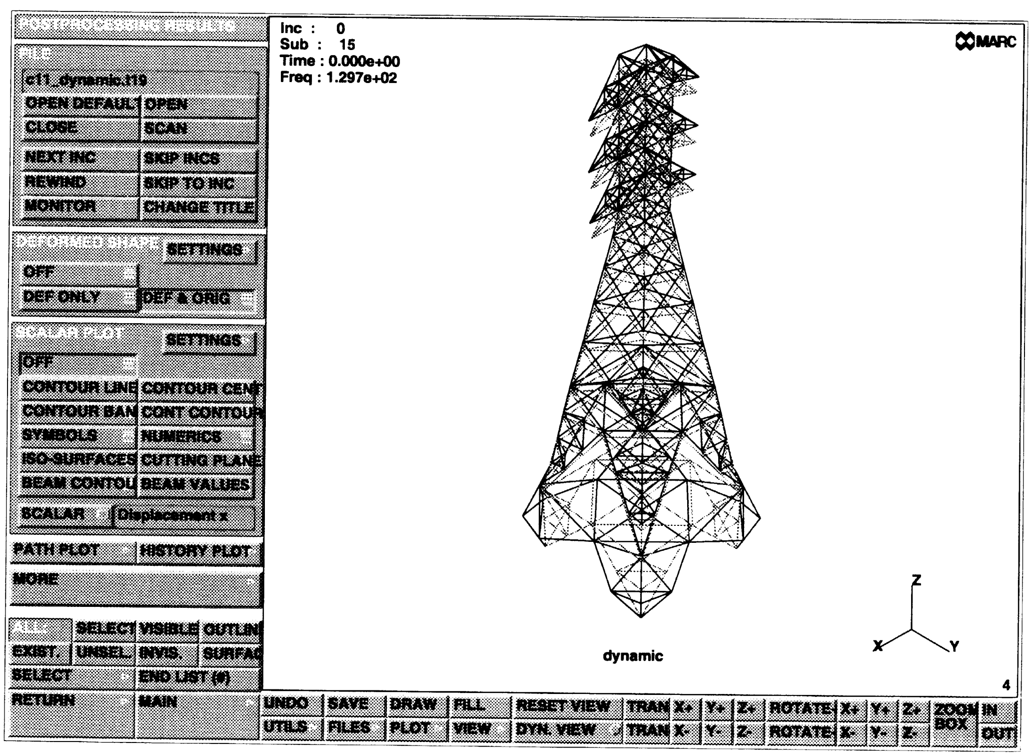


Figure 11.39 Eigenmode of Tower, f = 129.7 radians/sec

11.3 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 II, 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.

11.4 Procedure file

```
| Version : MENTAT II 2.3
|
| This problem describes the modeling and analysis of a tower structure.
| The mesh will consist of linear beam-elements. The following mesh generation
| features will be used:
|      - user defined local coordinate systems
|      - node and element creation
|      - element subdivision and duplication
|
| A static and modal analysis of this model will be performed.
|
|
| Step 1
|
| Create a grid and position it in space such that the first elements
| for the base of the tower can be created
|
*set_grid_spacing
1 1
*set_grid_size
10 10
*set_grid on
*system_rotate
90 0 0
*system_translate
0 9 0
*activate_view 2
*show_view 2
*fill_view
*set_node_labels on
*set_element_labels on
|
| add the first line elements
|
*set_element_class line2
*add_elements
node(-9,0,0)
node(-9,10,0)
node(0,10,0)
 1
 2
 3
| Subdivide and add additional stiffeners
|
*subdivide_elements
 1
 2
```

```
# | End of List
*add_elements
 5
 8
 8
 6
|
| Reposition the grid so the basic structure of the head of the
| tower can be generated
|
*set_grid_spacing
2 2
*set_grid_size
20 20
*system_translate
0 -7 50
 *deactivate_view 1
 *deactivate_view 2
*activate_view 4
*view_perspective
*activate_all_views
*show_all_views
*fill_view
|
| Add the primary elements for the head and subdivide
|
*show_view 2
*add_elements
node(-2,0,0)
node(-2,18,0)
*sub_divisions
6 2 2
*subdivide_elements
 10
# | End of List
|
| Again, reposition the grid, now such that the base and the head
| can be connected
|
*system_align
0 9 10
-9 9 10
-2 2 50
*set_grid_spacing
1 1
*set_grid_size
50 50
*show_view 3
|
| Connect base and head, subdivide, split, and add
```

```
 | additional stiffeners
 |
 *show_view 2
 *add_nodes
 0 13 0
 *add_elements
  6
  12
 *sub_bias_factors
 -.2 0 0
 *sub_divisions
 2 1 1
 *subdivide_elements
  17
 # | End of List
 *sub_divisions
 5 1 1
 *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
 *add_elements
 33 19
 19 29
 *sub_divisions
 3 1 1
 *subdivide_elements
  30
 # | End of List
 *add_elements
  34
  32
  32
  35
  35
  31
  31
  36
  36
  30
 *set_element_labels off
 *regen
```

```
|
| Step 2
|
| Using the symmetry option, the first face of the tower will be completed.
| Next, cross-members will be added. Some useful element sets will be generated
| and the arms will be added to the tower.
|
*symmetry_elements
all_existing
*add_elements
 33
 64
 64
 24
 24
 56
 56
 26
 26
 58
 58
 28
 28
 47
 47
 14
 14
 49
 49
 16
 16
 51
 51
 18
 18
 52
 52
 17
 17
 50
 50
 15
 15
 48
 48
 13
 13
 59
 59
 27
```

```
  27
  57
  57
  25
  25
  55
  55
  33
  *set_node_labels off
 *sweep_nodes
 all_existing
 *system_reset
  *set_grid off
 *set_duplicate_rotations
 0 0 90
 *set_duplicate_repetitions
 3
 *duplicate_elements
 all_existing
 *show_all_views
 *fill_view
 *sweep_nodes
 all_existing
 *sweep_elements
 all_existing
 |
 | 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
*identify_none
*regen
|
| Bring only 'base' into view and add stiffeners
|
*set_node_labels on
*select_sets
base
*visible_selected
*show_view 4
*fill_view
 *deactivate_view 1
 *deactivate_view 2
 *deactivate_view 3
*rot_model_mspace_z_rev
*add_elements
 146
 8
 80
 147
 81
 148
 82
 44
 74
 3
 3
 76
 76
 75
 75
```

```
 74
 3
 75
 74
 76
|
| Include the newly generated elements into 'base'
|
*store_elements
base
all_visible
|
| Bring only 'mid_sect_lo' into view and add stiffeners
|
*select_sets
mid_sect_lo
*visible_selected
*fill_view
*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
|
| Include the newly generated elements into 'mid_sect_lo'
|
```

```
*store_elements
mid_sect_lo
all_visible
|
| Step 3
|
|
| Bring only 'head' into view and add the extreme outward node
| for the arms. Then start to create the arms
|
*select_sets
head
*visible_selected
*fill_view
*zoom_box
*zoom_box(4,0.303409,0.497462,0.740909,0.095178)
*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
*add_nodes
-8 0 63
*add_elements
 200
 101
 200
 17
 200
 86
 200
 11
*sub_divisions
2 1 1
*subdivide_elements
 378
 379
 377
 376
# | End of List
*add_elements
202 208
208 211
211 205
205 202
202 209
209 211
211 206
```

```
206 202
202 211
|
| Store these elements as the set arm
|
*activate_all_views
*fill_view
*show_view 2
*pick_inside_partial
*store_elements
arm
 380 381 382 383 384 385 386 387 388 389 390 391 392 393 394 395 396
# | End of List
*pick_inside_complete
|
| Step 4
|
| Duplicate the set arm once, rotating it over 180 degrees
|
*duplicate_reset
*set_duplicate_rotations
0 0 180
*set_duplicate_repetitions
1
*duplicate_elements
arm
*fill_view
|
| Duplicate it twice translating it downwards
|
*duplicate_reset
*set_duplicate_translations
0 0 -6
*set_duplicate_repetitions
2
*duplicate_elements
arm
# | End of List
*sweep_nodes
all_existing
*select_elements
all_existing
*visible_selected
*fill_view
*set_node_labels off
*identify_sets
*regen
*identify_none
|
| Step 5
```

```
|
|
| Add boundary conditions
|
| First fix to earth
|
*apply_name
bolts
*select_sets
base
*visible_selected
*show_view 4
*fill_view
*apply_type fixed_displacement
*apply_dof x
*apply_dof y
*apply_dof z
*add_apply_nodes
 69
 40
 1
 68
# | End of List
*fill_view
|
| Select all elements that will be heavy L3x3 profiles
| first select from set base then mid_sect_lo, mid_sect_hi,
| head and arms.
|
*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
# | End of List
*select_sets
mid_sect_lo
*visible_selected
*fill_view
*store_elements
L3x3
 264
 261
 258
 165
 168
 171
 64
 63
 62
 31
 32
 33
 262
 259
 256
 163
 166
 169
 263
 260
 257
 164
 167
 170
 149
 152
 155
 158
 148
 151
```

```
 154
 157
 60
 59
 58
 57
 25
 26
 27
 28
# | End of List
*select_sets
mid_sect_hi
*visible_selected
*fill_view
*store_elements
L3x3
 134
 137
 140
 143
 146
 24
 23
 22
 21
 20
 70
 282
 281
 280
 52 53 54 55 56 133 136 139 142 145
# | End of List
*select_sets
head
*visible_selected
*fill_view
*store_elements
L3x3
 116 119 122 125 128 131
 46 47 48 49 50 51 115 118 121 124 127 130
 11 12 13 14 15 16
# | End of List
*select_sets
arm
*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
 426
 422
 424
 416
 428
 383
 381
 387
 385
 384
 386
 380
 382
  82
 316
 317
 318
```

```
# | End of List
*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
L2x2
all_selected
*select_clear
*select_reset
*identify_sets
*regen
*identify_none
*regen
*reset_view
*fill_view
|
| Apply gravity loading to all elements
|
*new_apply
*apply_name
gravity
*apply_type gravity_load
*apply_value z
-32.2
*add_apply_elements
all_existing
*show_view 1
*fill_view
|
| Select all elements in the lower part of a frontal face.
|
*select_sets
base
*visible_selected
*store_elements
lo_front
 3 4 5 6 7 8 9 39 40 41 42 43 44 45
# | End of List
*select_sets
mid_sect_lo
*visible_selected
*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
# | End of List
|
| Select all elements in the higher part of a frontal face
|
*select_sets
mid_sect_hi
*visible_selected
*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
# | End of List
*select_sets
head
*visible_selected
*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
arm
*visible_selected
*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
*show_view 4
|
| Apply a wind load to the hi_front elements
|
*new_apply
*apply_name
hi_wind
*apply_type global_load
*apply_value y
-120
*add_apply_elements
hi_front
|
| Apply a wind load to the lo_front elements
|
*new_apply
*apply_name
```

```
lo_wind
*apply_type global_load
*apply_value y
-80
*add_apply_elements
lo_front
|
| Apply the cable load
|
*new_apply
*apply_name
cable_load
*apply_type point_load
*apply_value z
-500
*add_apply_nodes
 250
 249
 213
 200
 225
 226
# | End of List
|
| Step 6
|
| Assign the geometrical and physical properties
|
| Now create a set of all elements pointing in z-direction
| and store as set 'upright'
|
*set_nodes off
*select_sets
base
*visible_selected
*fill_view
*store_elements
upright
 97 100
 98 101
 40 41
 4 5
# | End of List
*select_sets
head
*visible_selected
*fill_view
*store_elements
upright
 116 119 122 125 128 131
```

```
 46 47 48 49 50 51 115 118 121 124 127 130
 11 12 13 14 15 16
# | End of List
*select_sets
arm
*visible_selected
*store_elements
upright
 407
 405
 468
 464
 469
 465
 431
 430
 388
 390
 434
 435
# | End of List
*select_clear
*select_reset
*select_elements
all_existing
*visible_selected
*fill_view
|
| Apply geometric properties to all elements of type L3x3,
| not pointing in z-direction
|
*new_geometry
*geometry_name
L3x3_z
*geometry_type mech_three_beam_ela
*geometry_value area
0.01
6.0e-5
6.0e-5
0
0
1
*select_sets
L3x3
*select_mode_except
*select_sets
upright
*add_geometry_elements
all_selected
*identify_geometries
```

```
*regen
|
| Apply geometric properties to all elements of type L3x3,
| pointing in z-direction
|
*new_geometry
*geometry_name
L3x3_x
*geometry_type mech_three_beam_ela
*geometry_value area
0.01
6.0e-5
6.0e-5
1
0
0
*select_clear
*select_reset
*select_sets
upright
*select_mode_except
*select_sets
L2x2
*add_geometry_elements
all_selected
|
| Apply geometric properties to all elements of type L2x2,
| not pointing in z-direction
|
*new_geometry
*geometry_name
L2x2_z
*geometry_type mech_three_beam_ela
*geometry_value area
0.00651
2.0e-5
2.0e-5
0
0
1
*select_clear
*select_reset
*select_sets
L2x2
*select_mode_except
*select_sets
upright
*add_geometry_elements
all_selected
|
```

```
| Apply geometric properties to all elements of type L2x2,
| pointing in z-direction
|
*new_geometry
*geometry_name
L2x2_x
*geometry_type mech_three_beam_ela
*geometry_value area
0.00651
2.0e-5
2.0e-5
1
0
0
*select_clear
*select_reset
*select_sets
upright
*select_mode_except
*select_sets
L3x3
*add_geometry_elements
all_selected
*select_clear
*select_reset
|
| Assign steel properties to all elements
|
*new_material
*material_name
steel
*material_type mechanical:isotropic
*material_value isotropic:youngs_modulus
4.176e9
*material_value isotropic:poissons_ratio
0.3
*material_value isotropic:mass_density
15.217
*add_material_elements
all_existing
*identify_materials *regen
*identify_none *regen
|
| Step 7
|
| Assemble a job, analyzing all loads and B.C.'s as initial
| conditions. Save model, submit and run.
|
*new_job
*job_name
```

```
static
*job_class mechanical
*element_type 52
all_existing
*check_job
*save_as_model
c11
y
*update_job
*submit_job 1
*monitor_job
|
| Step 8
|
|
| Postprocess the static analysis
|
*post_open_default
*show_view 3
*set_deformed both
*post_next
*set_automag on
*fill_view
|
| Step 9
|
|
| Create a new, modal_dynamic job
|
*post_close
*set_deformed off
*restore_model
*new_loadcase
*loadcase_name
dynamic
*loadcase_type dynamic_modal
*loadcase_value nmodes
15
*new_job
*job_name
dynamic
*job_class mechanical
*add_job_loadcases
dynamic
*check_job
*save_model
*update_job
*submit_job 1
*monitor_job
|
```

```
| Step 10
|
| Postprocess the dynamic analysis
|
*post_open_default
*show_view 4
*set_deformed both
*post_next
*post_next
*set_automag on
*fill_view
*set_deformed on
*post_animate_mode
15
*animation_repeat on
*animation_play
*animation_terminate
*post_skip_to
0:15
*set_deformed both
*fill_view
```

Chapter 12: Importing a Model

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.

12.1 Background Information

12.1.1 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 II installation directory, in the subdirectory *examples / userguide* and is named *c12.igs*.

12.2 Detailed Session Description

Step 1

Assume you are already in Mentat II 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
      `c12.igs`
    FILL

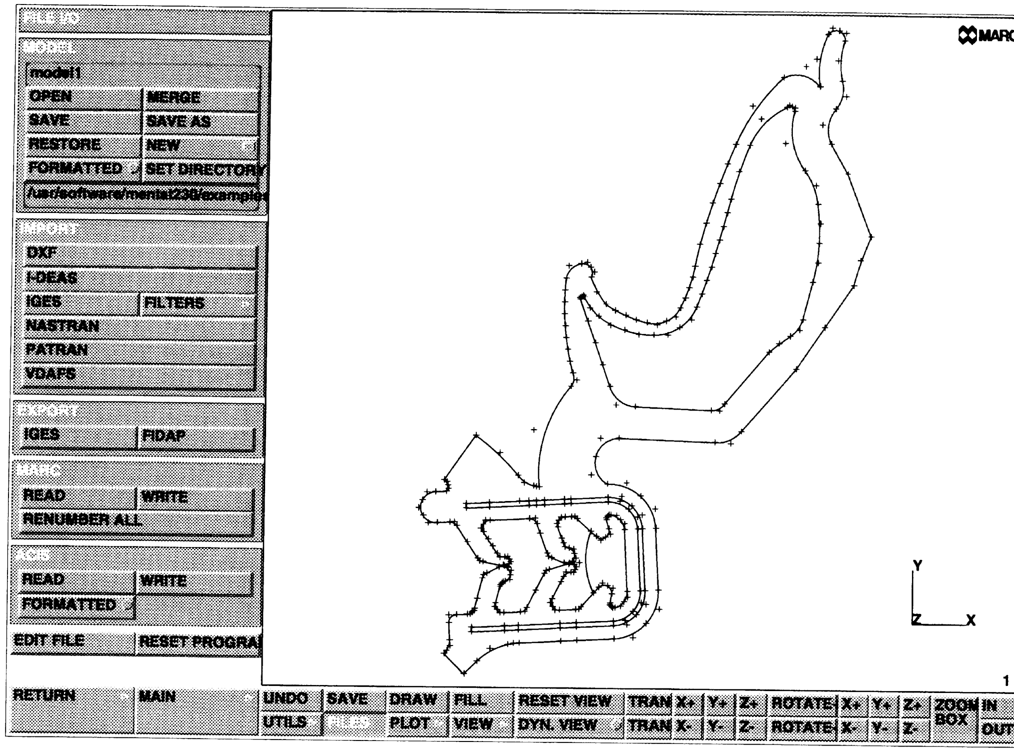


Figure 12.1 Scaled Model of Imported IGES File

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 II responds by sweeping all duplicate points and curves, respectively.

To improve the quality of the display, it is useful to increase the number of divisions used to display the curves.

```
PLOT
    MORE
    DIVISIONS
        CURVES
            20
        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 II 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 BOX

(Zoom in on the base of the neck)

crvs ADD *(Use the default curve type)*

235 *(Pick point)*

188 *(Pick point)*

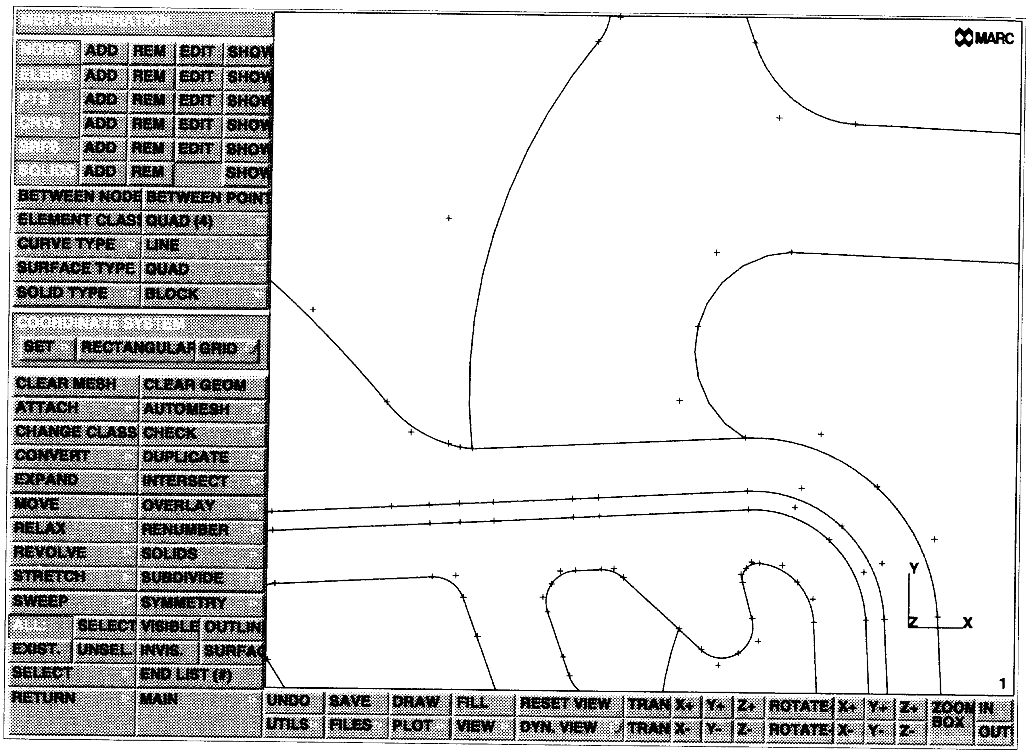


Figure 12.2 Curved Added at Base of Neck

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 Section 2.5 of Chapter 2 to select the members for each set.

MAIN
  PLOT
    draw POINTS *(off)*
    RETURN
  FILL
  SELECT
    crvs STORE
      `upperpart` *(The curve set name)*
      (Use the Polygon Pick Method to select the curves)
      END LIST (#)

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.

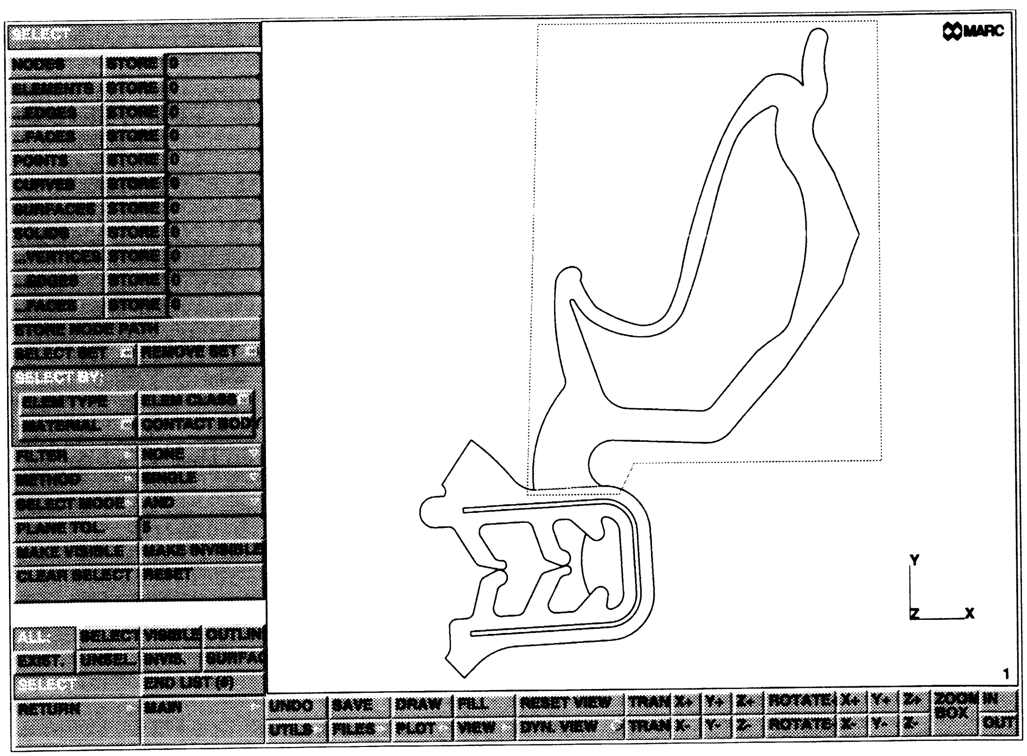


Figure 12.3 Polygon Pick Contour for Upper Part

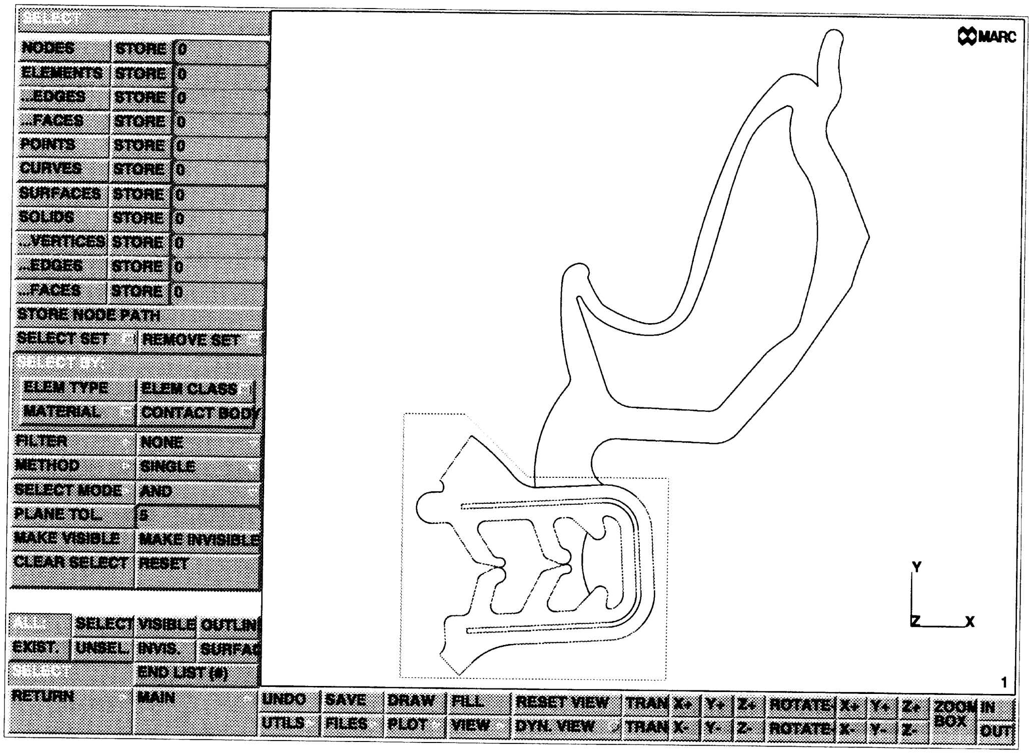


Figure 12.4 Polygon Pick Contour for Lower Part

To verify that you have created two sets, click on the sets SELECT 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.

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

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.

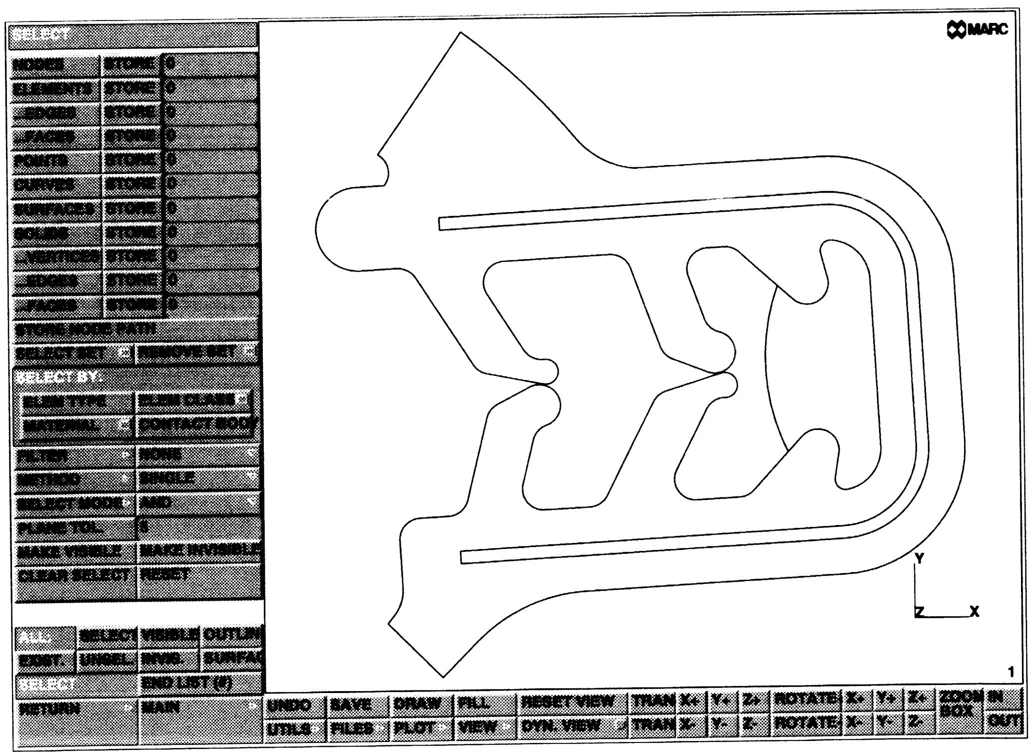


Figure 12.5 Lower Part of Model Scaled to Fill the Graphics Area

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` *(Pick curve)*
END LIST (#)

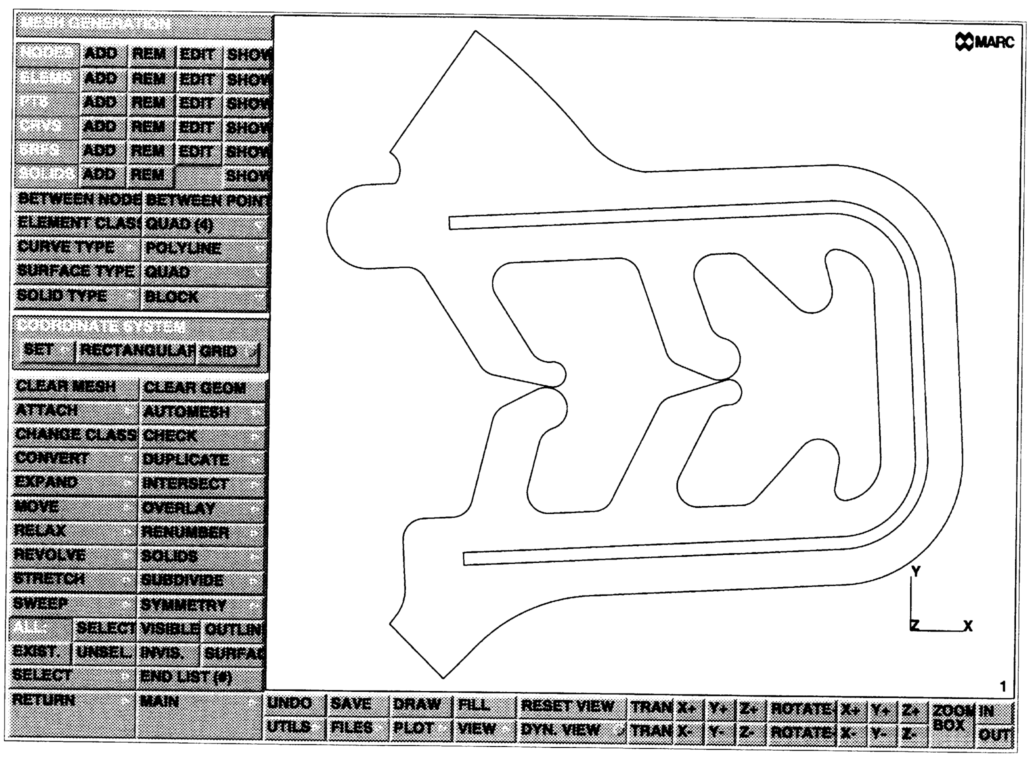


Figure 12.6 Lower Part of Model with Curve Removed

The model is ready to be meshed using the **OVERLAY** processor from the mesh generation menu. 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
        OVERLAY
            DIVISIONS
                70  70
            CURVE MESH
                all: VISIBLE
```

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
        SAVE
```

Figure 12.7 shows the resulting mesh that should appear in the graphics area.

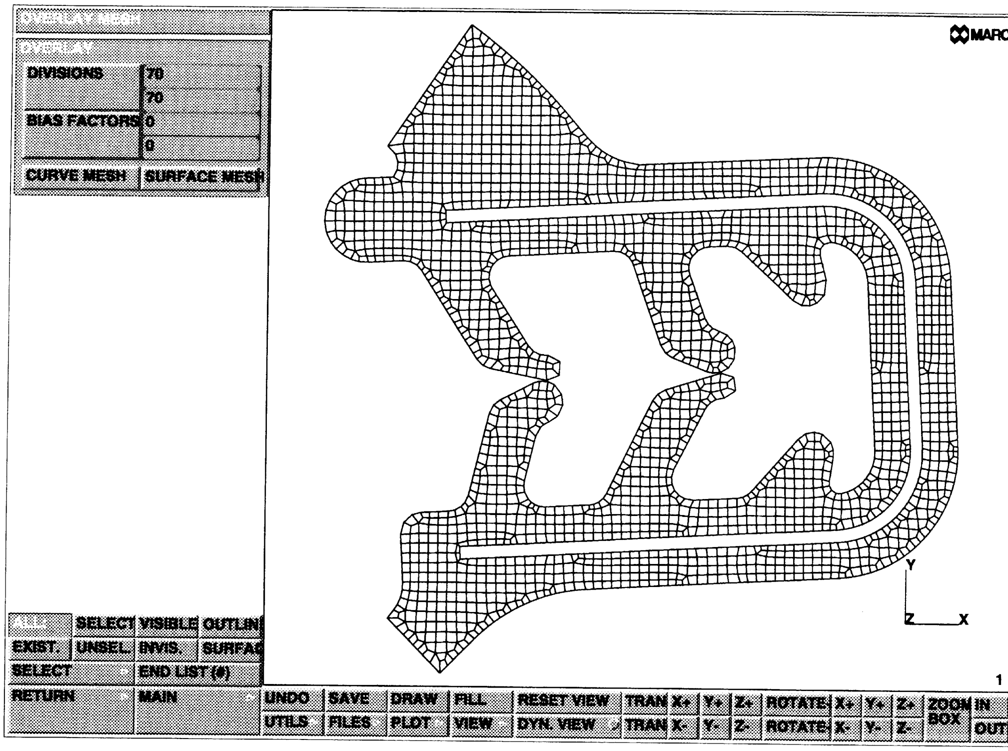


Figure 12.7 Mesh Generated with a Density of 70 in x and y Directions

12.3 Procedure File

```
| Version : MENTAT II 2.3
*import iges
c12.igs
*fill_view
*sweep_points
all_existing
*sweep_curves
all_existing
*zoom_box
*zoom_box(1,0.311364,0.772843,0.523864,0.562183)
*add_curves
 235
 188
 *set_points off
*fill_view
*store_curves
upperpart
 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 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
 135 136 137 138 139 140 141 142 213
# | End of List
*store_curves
lowerpart
 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 90 143 144 145 146 147 148 149 150 151 152 153 154 155 156 157 158 159 160
 161 162 163 164 165 166 167 168 169 170 171 172 173 174 175 176 177 178 179 180
 181 182 183 184 185 186 187 188 189 190 191 192 193 194 195 196 197 198 199 200
 201 202 203 204 205 206 207 208 209 210 211 212 213
# | End of List
*select_sets
lowerpart
*visible_selected
*fill_view
*remove_curves
 180
# | End of List
*set_overlay_divisions
70
70
*overlay_mesh
all_visible
 *set_nodes off
 *set_faces off
 *set_curves off
```

```
*regen
*save_as_model
c12
y
```

Chapter 13: Punch

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.

13.1 Background Information

13.1.1 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.

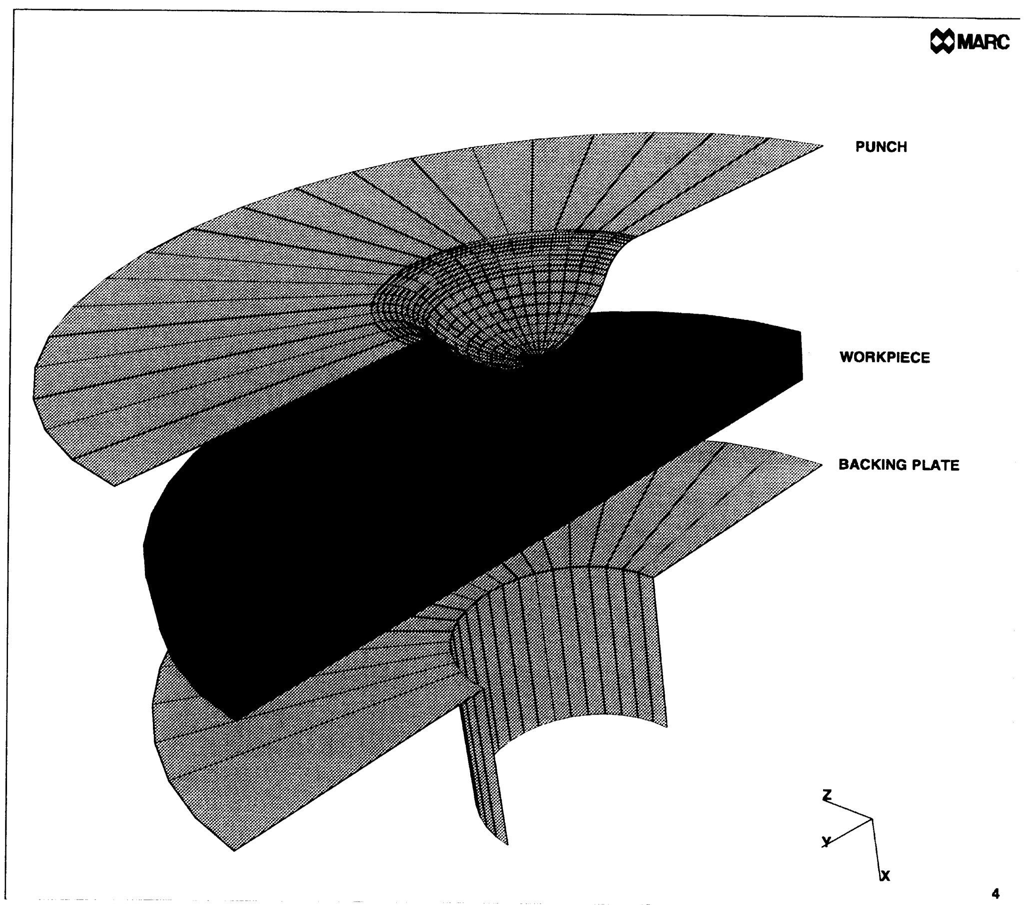


Figure 13.1 Punch, Workpiece, and Backing Plate

13.1.2 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.

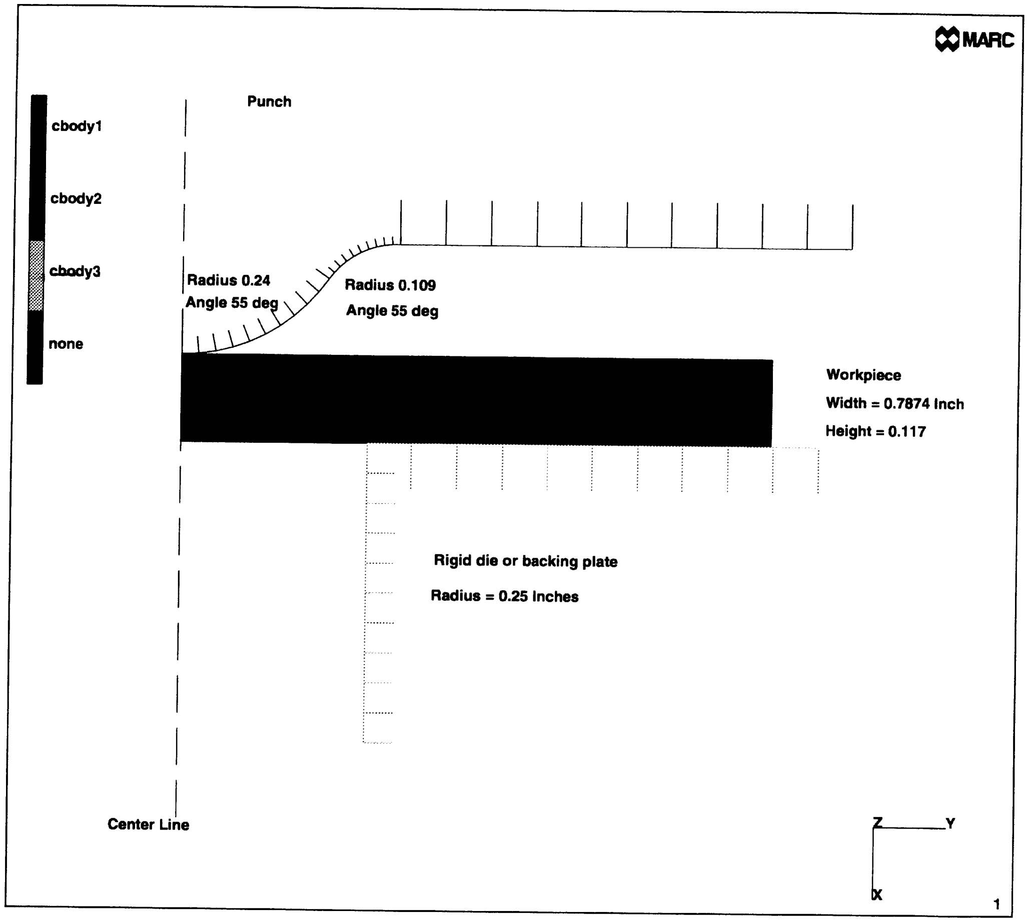


Figure 13.2 Dimensions of Punch, Workpiece, and Backing Plate

13.1.3 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.

13.1.4 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.

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.

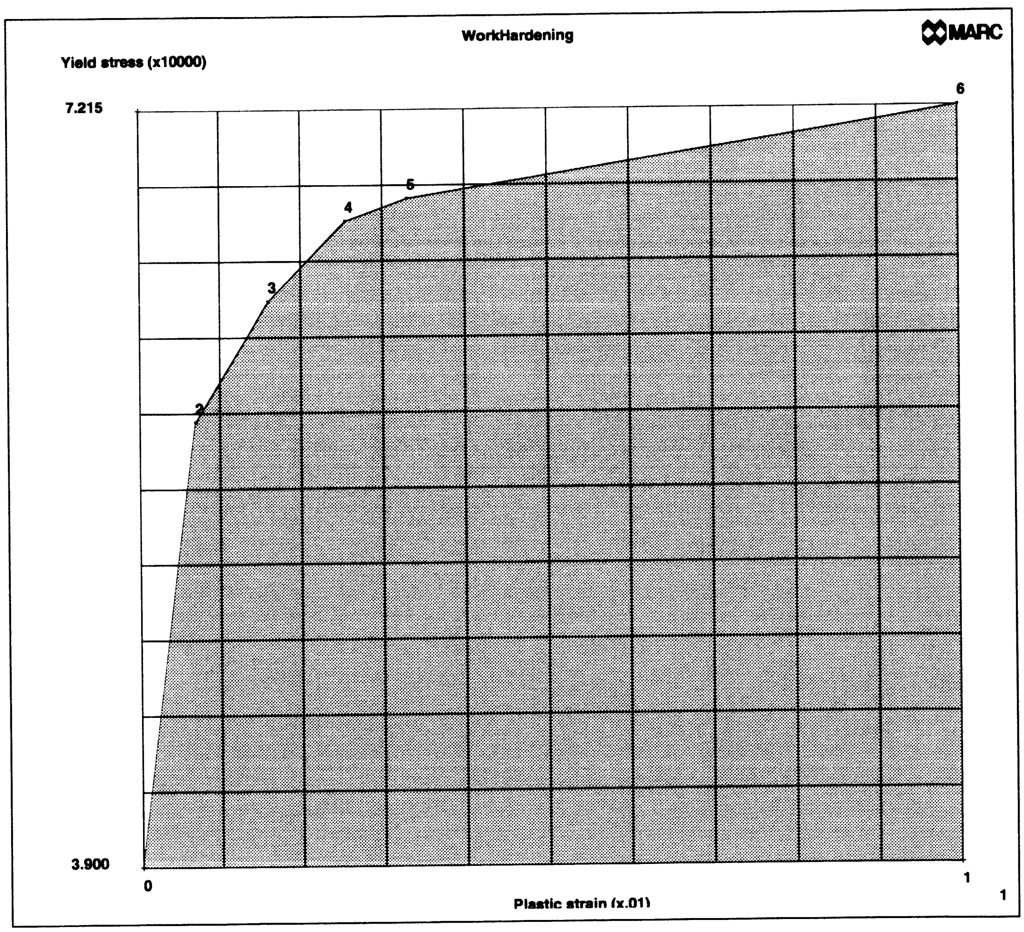


Figure 13.3 The Workhardening Curve for the Workpiece Material

13.1.5 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 Post process the results by displaying the deformed structure and the residual stresses and strains.

13.2 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 *(on)*
        REGENERATE
        RETURN

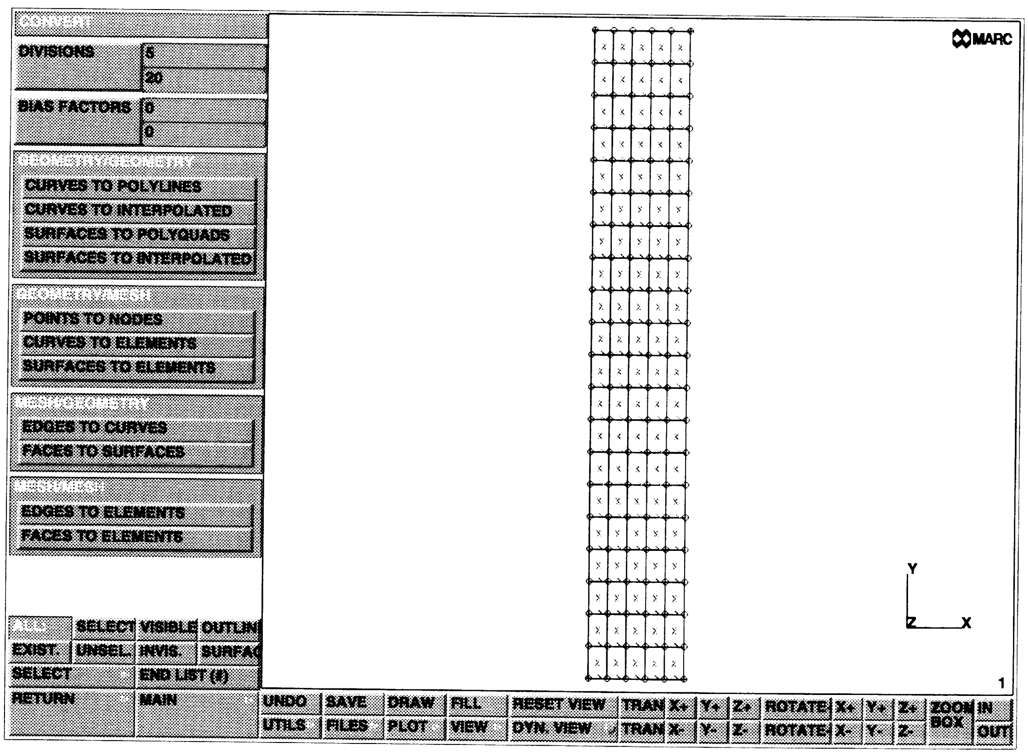


Figure 13.4 Result of the Convert Command

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
                `x`
                `0.25`
                `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 *(Box pick the 7th row of elements)*
                `26 27 28 29 30`
                END LIST (#)

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
                1                              (Pick the lower left point)
                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)
```

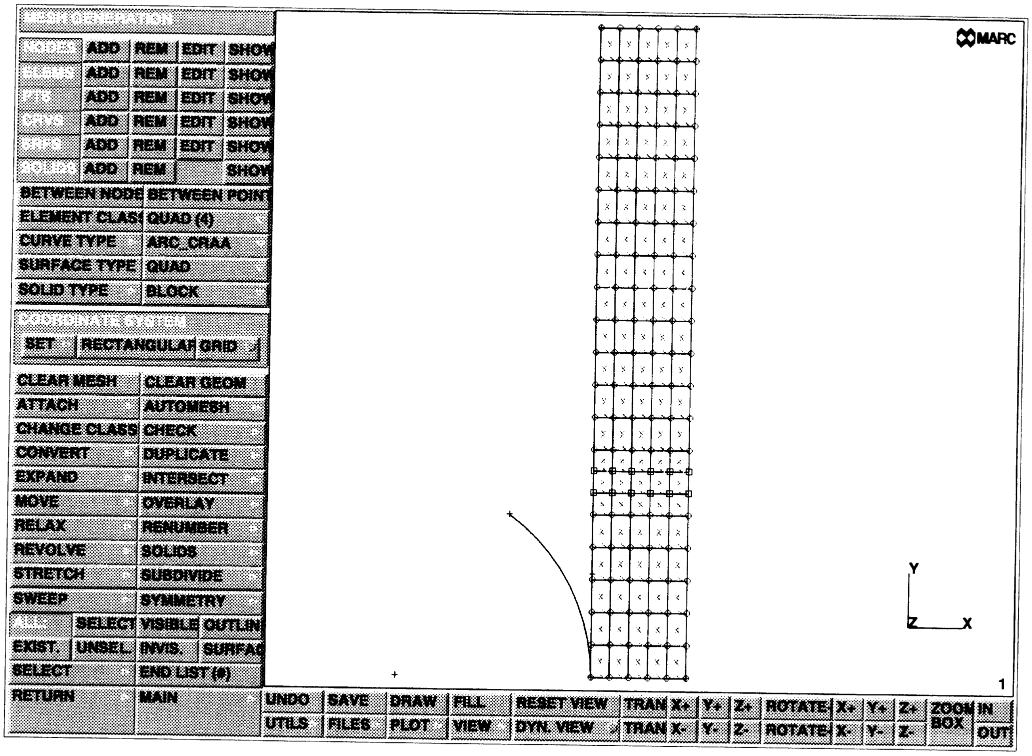


Figure 13.5 The Spherical Part for the Punch

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.

MAIN
    MESH GENERATION
        CURVE TYPE
            TANGENT/RADIUS/ANGLE
            RETURN
        crvs ADD
            `11` *(Pick the end point of the arc just created (Tangent point))*
            `0.109` *(Radius)*
            `-55` *(Angle)*

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.

```
MAIN
    MESH GENERATION
        EXPAND
            TRANSLATIONS
                0 0.6 0
            POINTS
                14                  (Pick the end point of the last arc)
                END LIST (#)
            FILL
```

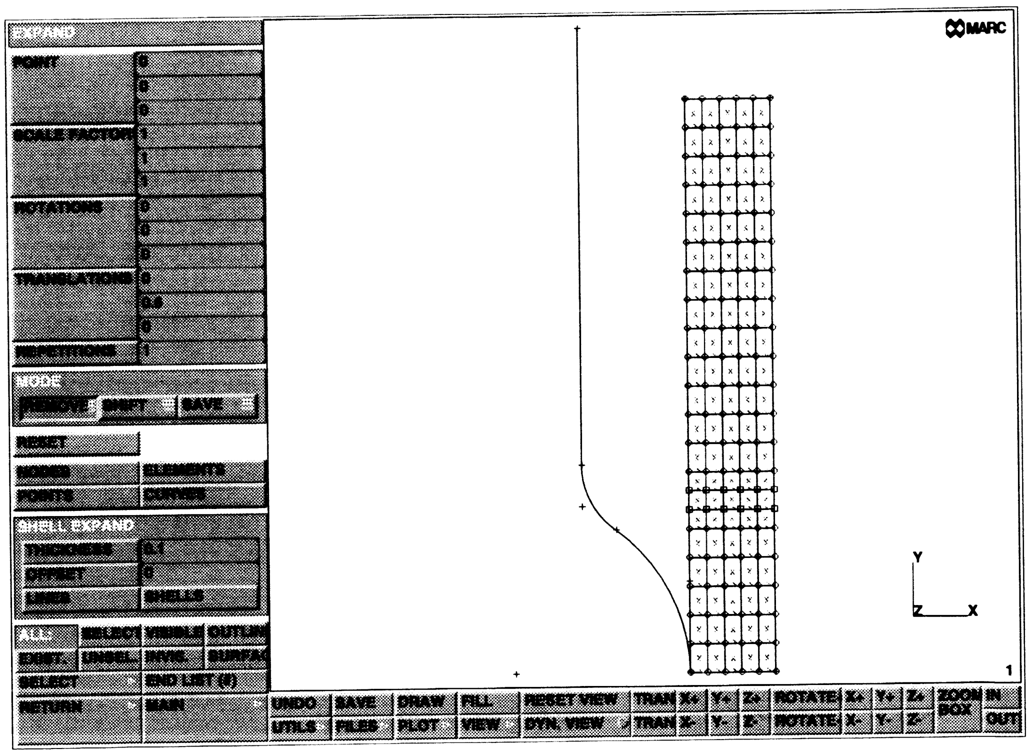


Figure 13.6 The Geometry of the Punch

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.

MAIN
  MESH GENERATION
    pts ADD
      `0.357 25 0`
    EXPAND
      POINTS
        `16` *(Pick the point just created)*
        END LIST (#)
      TRANSLATIONS
        `0.4 0 0`
      POINTS
        `16` *(Pick the corner point)*
        END LIST (#)
      FILL

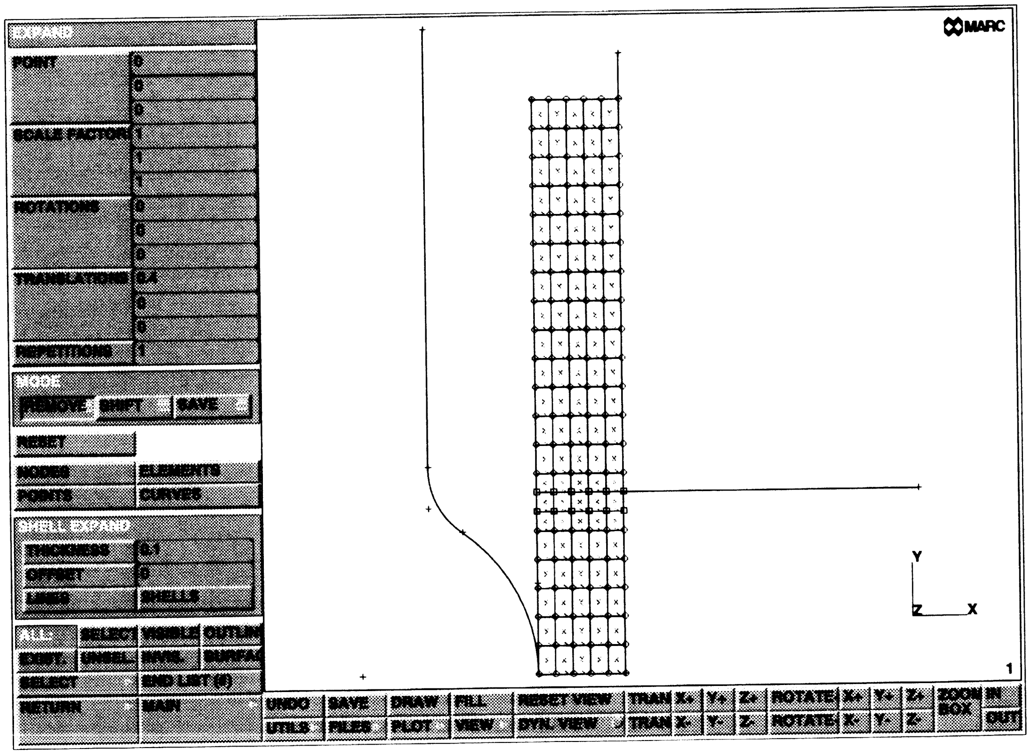


Figure 13.7 Punch, Workpiece, and Backing Plate

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
            label X-AXIS
                plastic strain
            label Y-AXIS
                yield stress
            SHOW MODEL
```

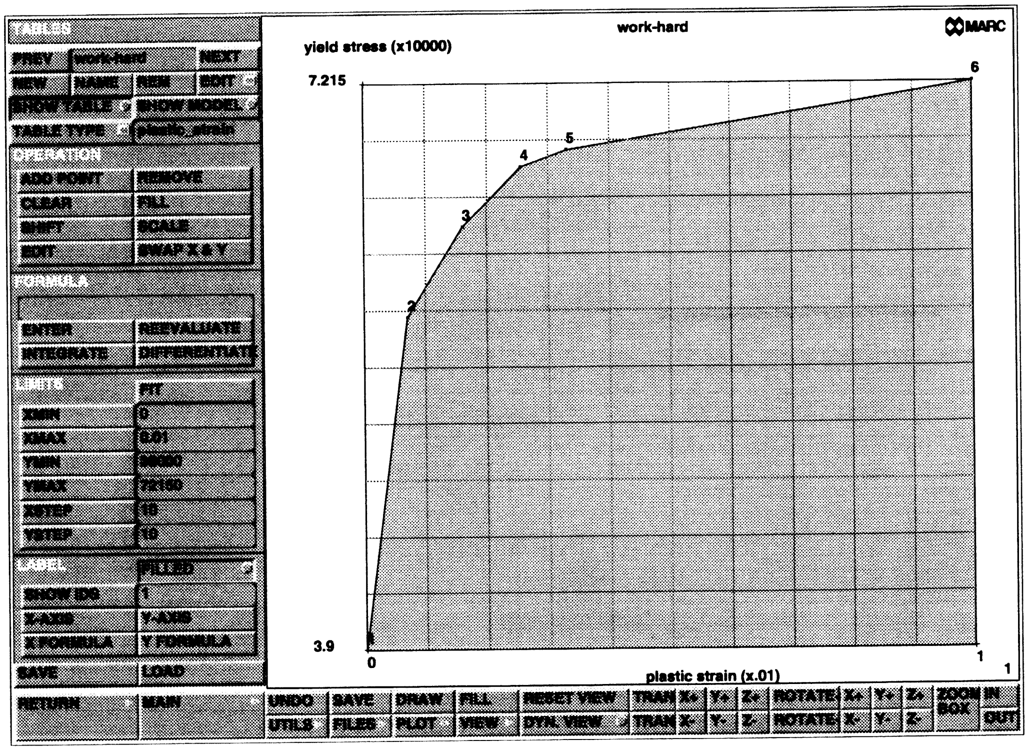


Figure 13.8 Workhardening Curve

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.

```
MAIN
    MATERIAL PROPERTIES
        ISOTROPIC
            YOUNG'S MODULUS
                30.0e6
            POISSON'S RATIO
                0.3
            PLASTICITY
                INITIAL YIELD STRESS
                    1
                initial yield stress TABLE 1
                    work-hard
                    OK
                OK
            OK
        elements ADD
            all: EXIST.
```

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 *(on)*
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)*

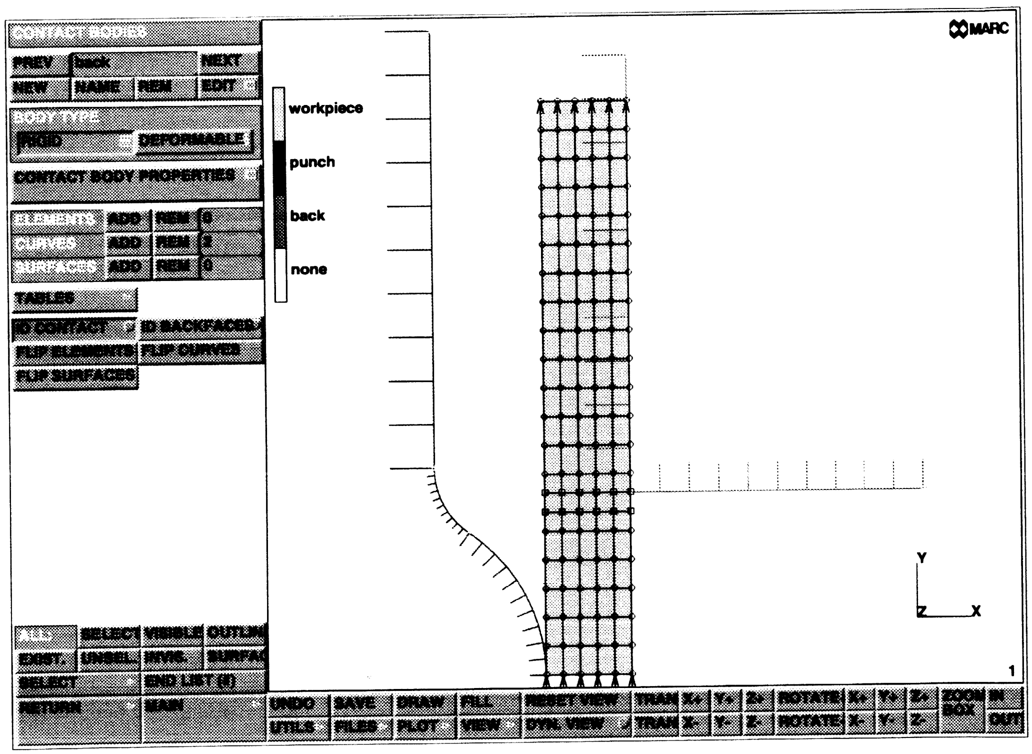


Figure 13.9 Incorrect Definition Direction of Curves in Back-Plate

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.

MAIN
CONTACT
CONTACT BODIES
FLIP CURVES
5 *(Pick curve)*
END LIST (#)
ID CONTACT *(off)*

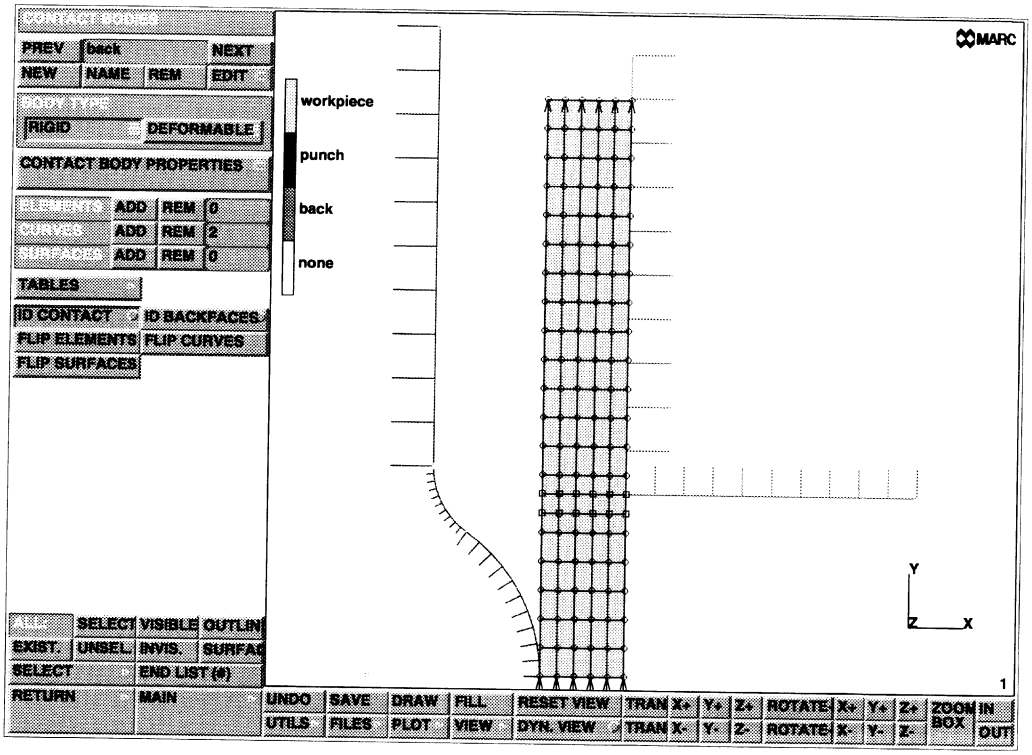


Figure 13.10 Corrected Definition

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            (0.1488/0.4 = velocity)
                    0.4  0.1488/0.4
                    0.4  -10*0.1488/0.4           (-10*velocity)
                    0.5  -10*0.1488/0.4
                FIT
                SHOW MODEL
```

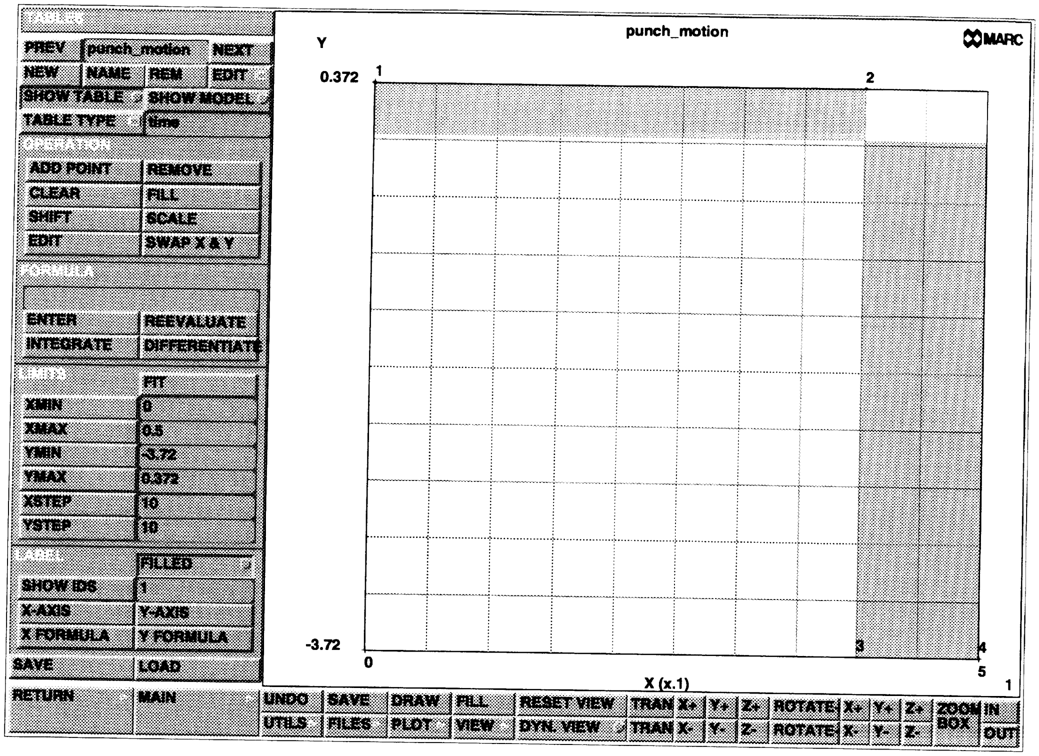


Figure 13.11 Velocity as a Function of Time

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 *(to activate body 2)*
    CONTACT BODY PROPERTIES
      velocity X
        `1`
      velocity x TABLE
        punch_motion
        OK
      OK

Step 5

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
```

Step 6

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.

```
MAIN
    JOBS
        MECHANICAL
            loadcases SELECT
                indent
                release
            ANALYSIS OPTIONS
                LARGE DISPLACEMENT                    (on)
                UPDATED LAGRANGE PROCEDURE            (on)
                FINITE STRAIN PLASTICITY              (on)
                OK
            JOB RESULTS
                SELECT TENSORS
                    stress
                SELECT VARIABLES
                    von_mises
                    epl_strain
                OK
            AXISYMMETRIC          (This makes sure that the
                                  default element type 10 is used)
            OK
        CHECK
        SAVE
        RUN
            SUBMIT 1
            MONITOR
```

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.

Step 7

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                                  (off)
            MORE
                edges OUTLINE
                PREVIOUS
            RETURN
        DEF & ORIG
        SCALAR
            Equivalent Von Mises Stress
            OK
        CONTOUR BANDS
        MONITOR
```

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.

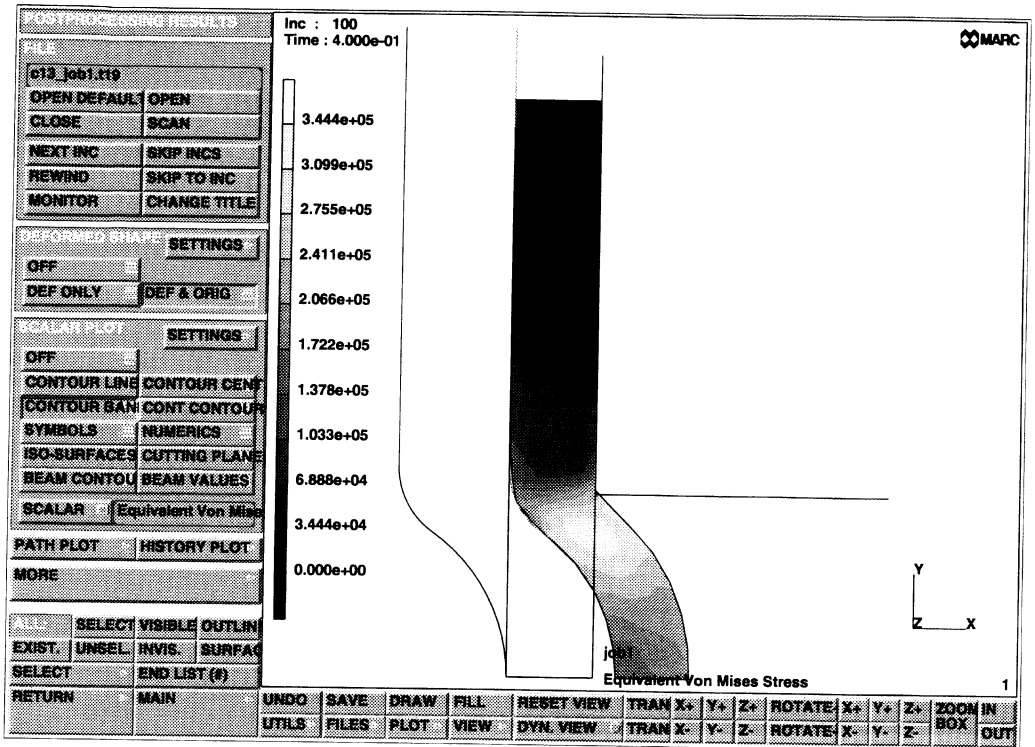


Figure 13.12 Deformations and Stresses at Maximum Indentation

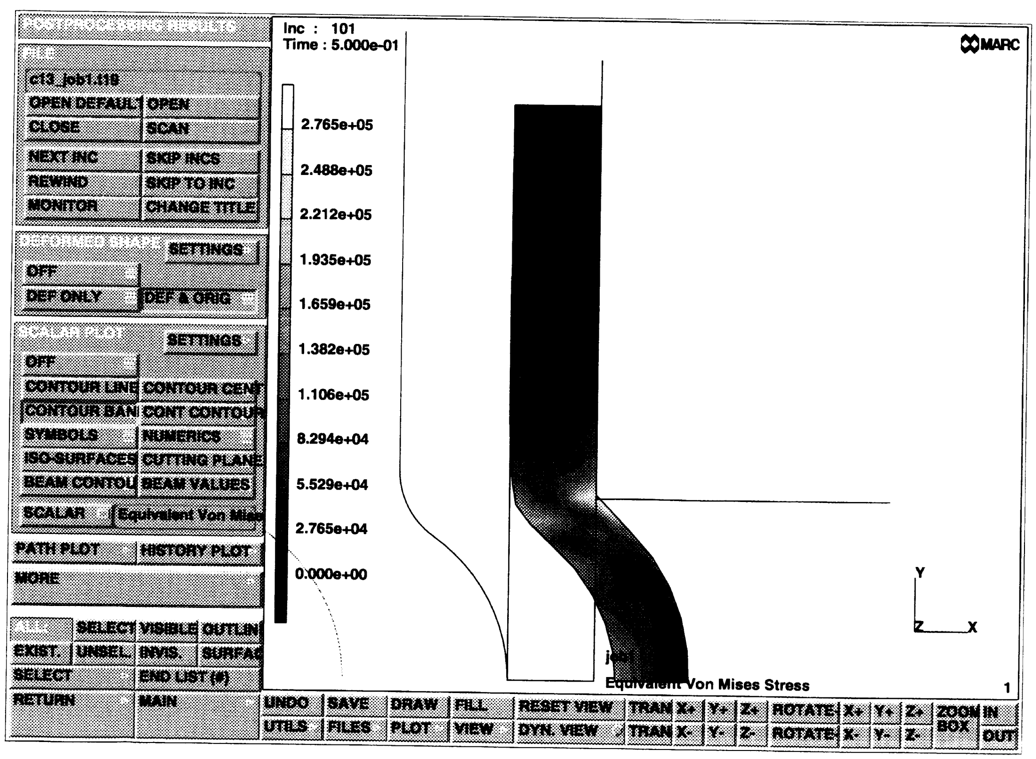


Figure 13.13 Deformations and Stresses after Spring-back

13.3 Procedure file

```
| Version : MENTAT II 2.3
|
| This problem demonstrates 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 a spherical indentation.
| The goal of the static analysis described here is to determine the residual
| stresses and plastic strains in the workpiece after the operation.
|
|
| Step 1
|
|
| Generate a quad surface and convert to elements.
|
| Create a point and expand it in x-direction creating a line curve
|
*add_points
0.24 0 0
*set_expand_translations
0.117 0 0
*expand_points
all_existing
# | End of List
|
| Expand the curve in y-direction creating a surface
|
*set_expand_translations
0 0.7874 0
*expand_curves
all_existing
# | End of List
*fill_view
|
| Generate a 5 x 20 mesh on the surface
|
*set_convert_divisions
5 20
*convert_surfaces
all_existing
*set_surfaces off
*regen
|
| Find the row of nodes closest to R = 0.25
| and shift those nodes to the exact position
|
*show_nodes
42
```

```
*set_move_formulas
x
0.25
z
*move_nodes
 37 38 39 40 41 42
# | End of List
|
| Subdivide the row of elements with a too high
| aspect ratio
|
*sub_divisions
1 2 1
*subdivide_elements
 26 27 28 29 30
# | End of List
*sweep_nodes
all_existing
*renumber_all
|
| Step 2
|
| Create the curves for the two rigid bodies
| (the punch and the backing-plate)
| Start with the punch by creating a 55 degrees
| circular arc just touching the mesh
|
*set_duplicate_translations
-0.24 0 0
*duplicate_points
 1
# | End of List
*set_curve_type arc_craa
*add_curves
0 0 0
0.24
0 55
|
| Add a curve that bends back to vertical again
|
*set_curve_type arc_tra
*add_curves
 11
0.109
-55
|
| Finish with a straight line segment
|
*set_expand_translations
0 0.6 0
```

```
*expand_points
 14
# | End of List
*fill_view
|
| Make the backing plate, starting with the corner point
|
*add_points
0.357 0.25 0
*expand_points
 16
# | End of List
*set_expand_translations
0.4 0 0
*expand_points
 16
# | End of List
*fill_view
|
| Step 3
|
| Apply material data and boundary conditions
|
| Define the work-hardening curve
|
*set_table_type
plastic_strain
*show_table
*table_add
0 39000
0.7e-3 58500
1.6e-3 63765
2.55e-3 67265
3.3e-3 68250
10e-3 72150
*table_fit
*table_name
work-hard
*set_table_xname
plastic strain
*set_table_yname
yield stress
*show_model
|
| Apply Young's modulus, Poisson's ratio and
| work hardening to all elements
|
*material_type mechanical:isotropic
*material_value isotropic:youngs_modulus
30e6
```

```
.3
*material_type plasticity
*material_value plasticity:yield_stress
1
*material_table plasticity:yield_stress0
work-hard
*add_material_elements
all_existing
|
| Fix the upper-right corner node
|
*apply_type fixed_displacement
*apply_dof x
*apply_dof y
*add_apply_nodes
 126
# | End of List
|
| Fix the radial displacements
| at the symmetry axis and at the largest radius
|
*new_apply
*apply_type fixed_displacement
*apply_dof y
*add_apply_nodes
 1 2 3 4 5 6
 121 122 123 124 125 126
# | End of List
|
| Step 4
|
| Identify the contact bodies and define their motion
|
*contact_deformable
*contact_body_name
workpiece
*add_contact_body_elements
all_existing
*new_contact_body
*contact_body_name
punch
*add_contact_body_curves
 2
 3
 4
# | End of List
*new_contact_body
*contact_body_name
back
*add_contact_body_curves
```

```
 5
 6
# | End of List
*elements_solid
*identify_contact *regen
*flip_curves
 5
# | End of List
 *identify_none *regen
|
| define the punch velocity as a function of time
|
*new_table
*table_name
punch_motion
*set_table_type
time
*table_add
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
*table_fit
*show_model
*previous_contact_body
*contact_value vx
1
*cbody_table vx0
punch_motion
|
| Step 5
|
|
| Define two loadcases, one for the indentation
| and one for the release
|
*loadcase_type static
*loadcase_name
indent
*loadcase_value time
.4
*loadcase_value nsteps
100
*loadcase_value maxrec
20
*new_loadcase
*loadcase_name
release
*loadcase_type static
*loadcase_value time
```

```
.1
*loadcase_value nsteps
1
*loadcase_value maxrec
20
*add_loadcase_cbodies
punch
|
| Step 6
|
|
| Activate the nonlinear job options and
| submit a job
|
*job_class mechanical
*add_job_loadcases
indent
release
*job_option large:on
*job_option update:on
*job_option finite:on
*add_post_tensor
stress
*add_post_var
von_mises
epl_strain
*job_option dimen:axisym
*check_job
*save_as_model
c13
y
*update_job
*submit_job 1
*monitor_job
|
| Step 7
|
|
| Evaluate the results
|
*post_open_default
*fill_view
*set_nodes off
*edges_outline
*set_deformed both
*post_value
Equivalent Von Mises Stress
*post_contour_bands
*post_monitor
```

Chapter 14: Bracket

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. The 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.

14.1 Background Information

14.1.1 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 during earthquake loads.

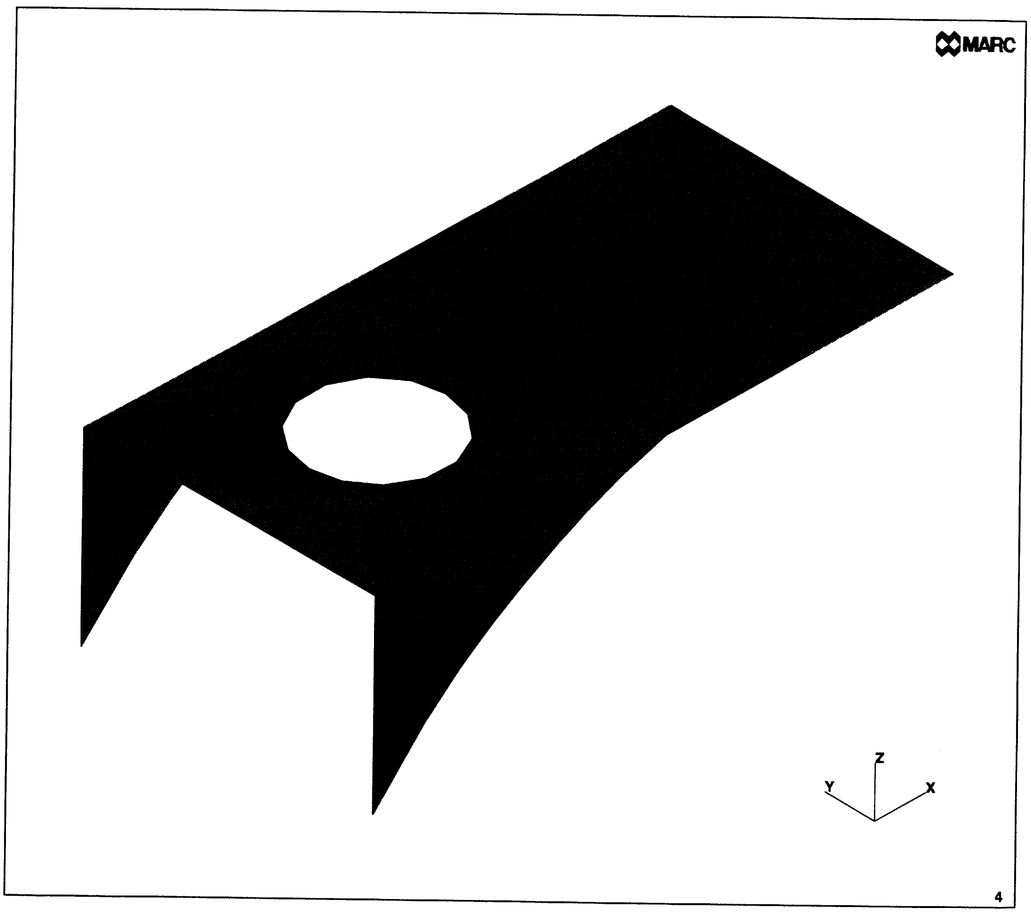


Figure 14.1 Bracket

14.1.2 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.

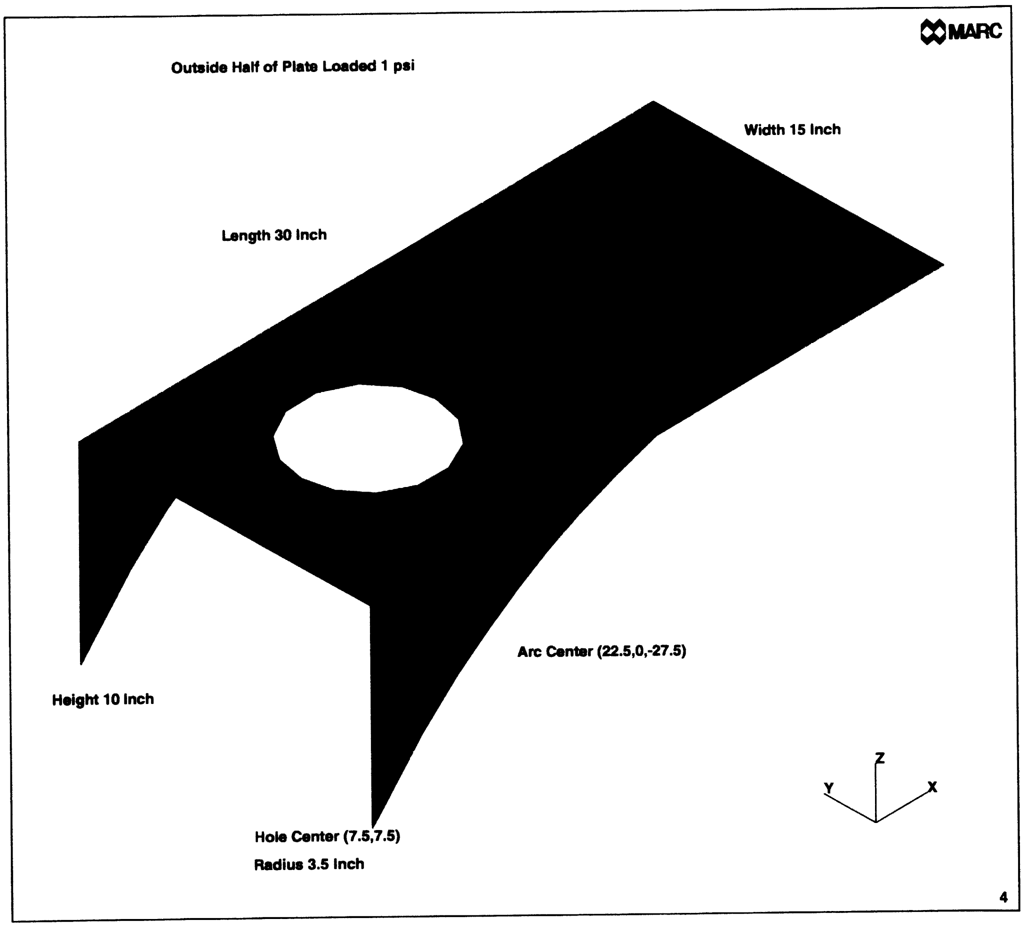


Figure 14.2 Dimensions and Loads for the Bracket

14.1.3 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 an earthquake event.

14.1.4 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.

14.1.5 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.

14.2 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
            SPACING
                5   5
            SIZE
                30  30
            grid ON                                        (on)
            RETURN
```

The next step will be to create the 3 vertical lines of the model. The right side of the grid is all that will be used, so the first thing is to fill the screen and then zoom in on the right side. Then the end points of the 3 lines will be easier to pick. The following button sequence will create the lines.

MAIN
    MESH GENERATION
        FILL
        ZOOM BOX

(Zoom in on right side of grid)

crvs ADD *(Pick points from grid)*

```
point(0,-10,0)
point(0,25,0)
point(15,0,0)
point(15,15,0)
point(30,0,0)
point(30,15,0)
```

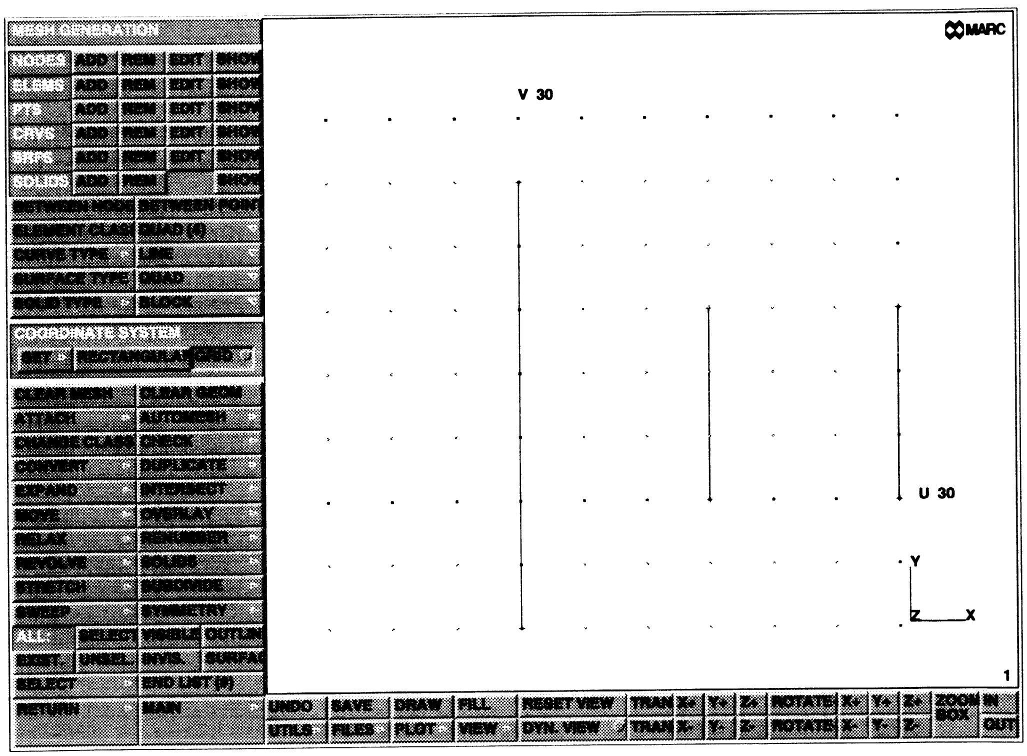


Figure 14.3 Grid and Straight Line Segments

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.

MAIN
    MESH GENERATION
        CURVE TYPE
            CENTER/POINT/POINT
            RETURN
        crvs ADD
            `22.5  27.5  0` *(Center point)*
            `15 0 0` *(Pick lower end point of the second line)*
            `0 -10 0` *(Pick lower end point of the first line)*
            `22.5  42.5  0` *(Center point)*
            `0  25  0` *(Pick upper end point of the first line)*
            `15  15  0` *(Pick upper end point of the second line)*

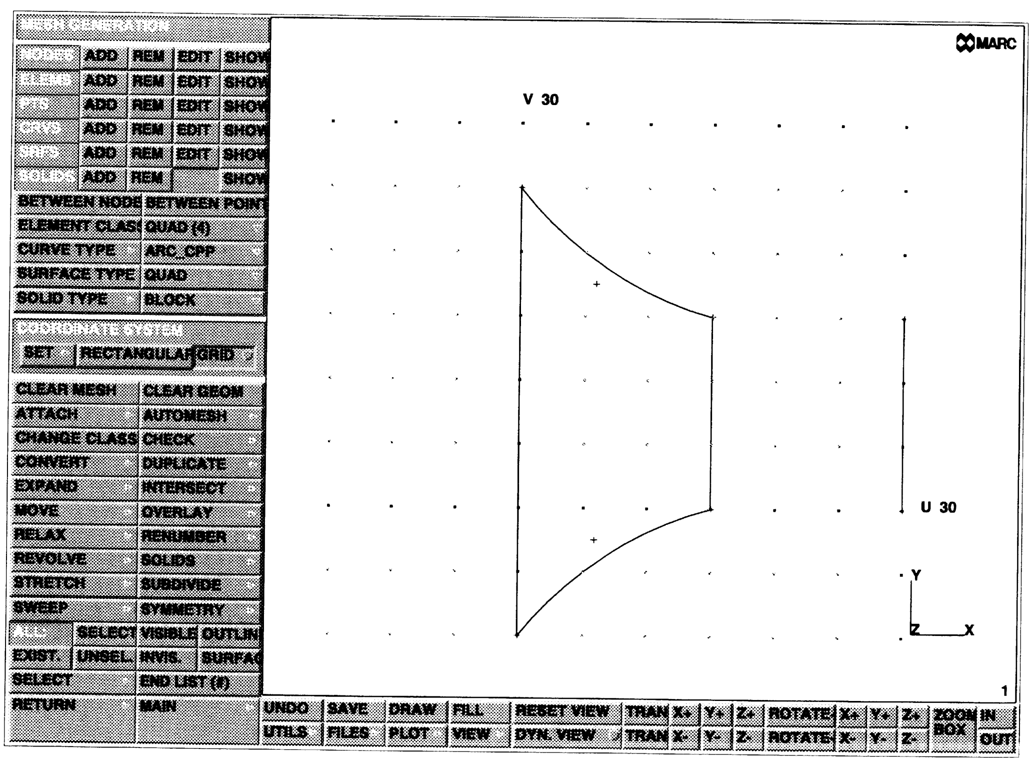


Figure 14.4 Line Segments and Fillets

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 XY
                7.5  7.5
            SIZE
                10  10
            SPACING
                0.5  0.5
            RETURN
        ZOOM BOX
                                        (Zoom in on the center of the grid)
        CURVE TYPE
            CENTER/POINT
            RETURN
        crvs ADD
            0  0  0                     (Pick the center point)
            3.5  0  0                   (Pick a point on the circle)
        FILL
```

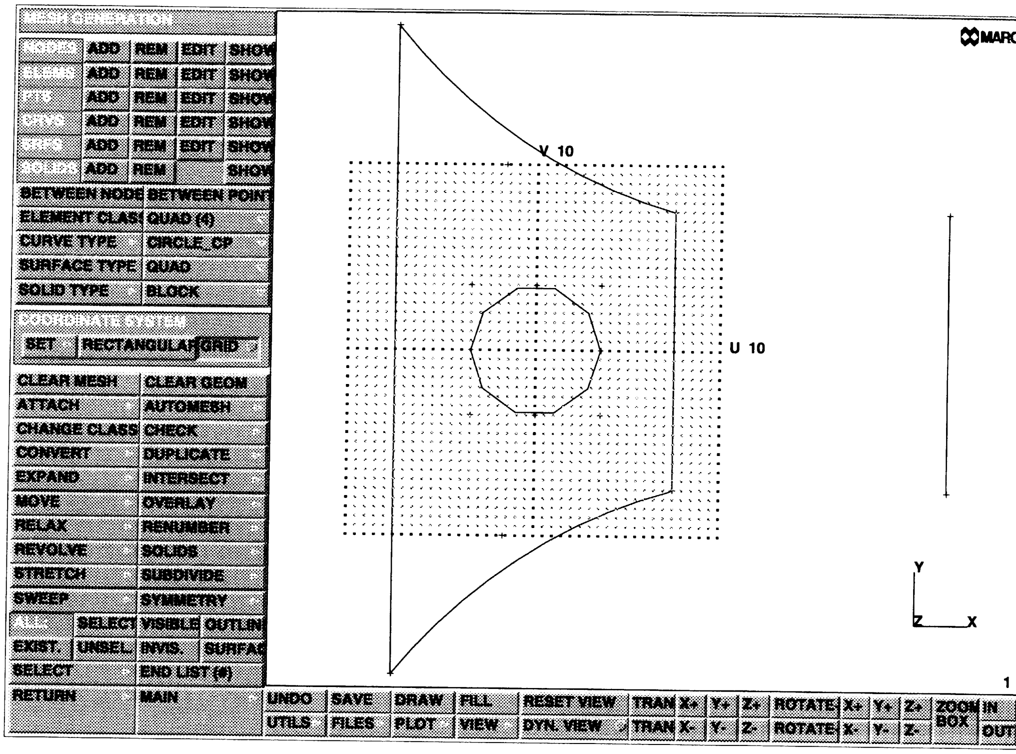


Figure 14.5 Generation of the Circular Hole

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.

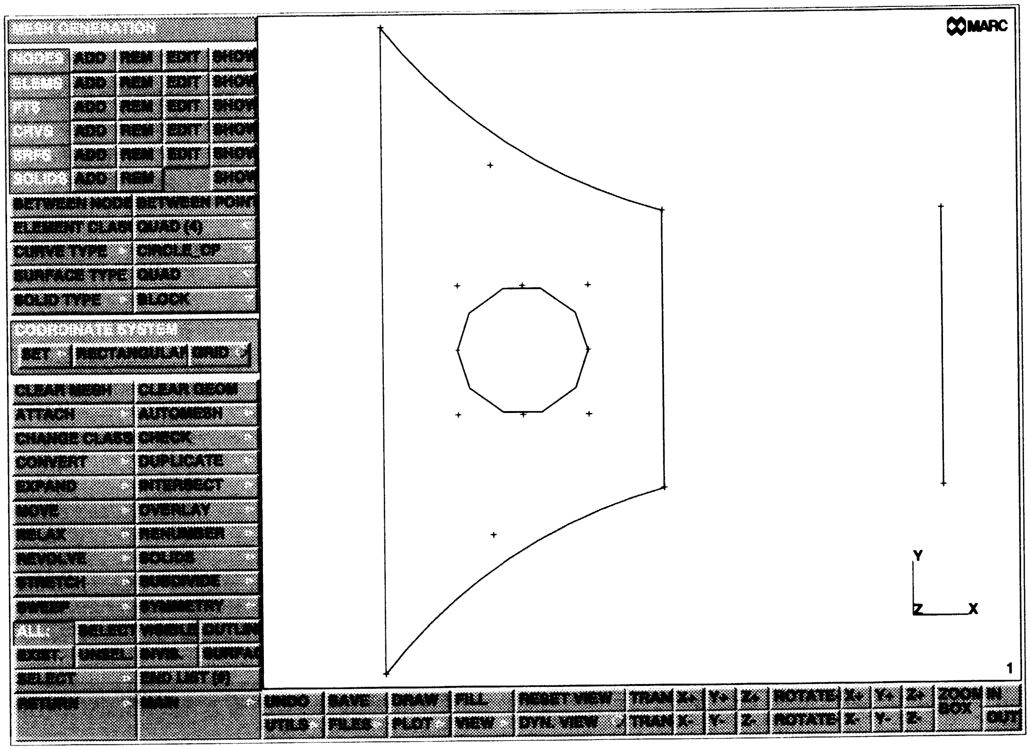


Figure 14.6 The Closed Contour for the Overlay Command

MAIN
  MESH GENERATION
    GRID *(off)*
    OVERLAY
      DIVISIONS
        `15  20`
      CURVE MESH *(Use the Box Pick Method)*
        `1  2  4  5  6`
        END LIST (#)

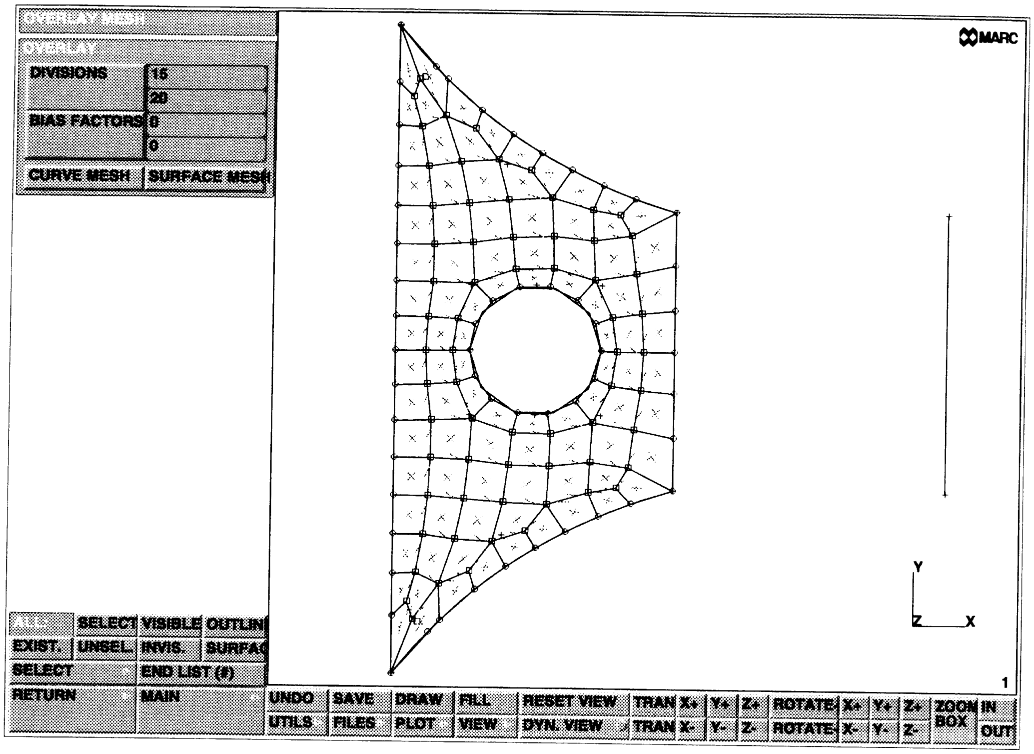


Figure 14.7 The Automeshed Part

Step 2

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)*
            `7`
            `5`
            `6`
            `12`
        CONVERT
            DIVISIONS
                `6 6`
            SURFACES TO ELEMENTS
                `1`   *(Pick surface)*
                END LIST (#)

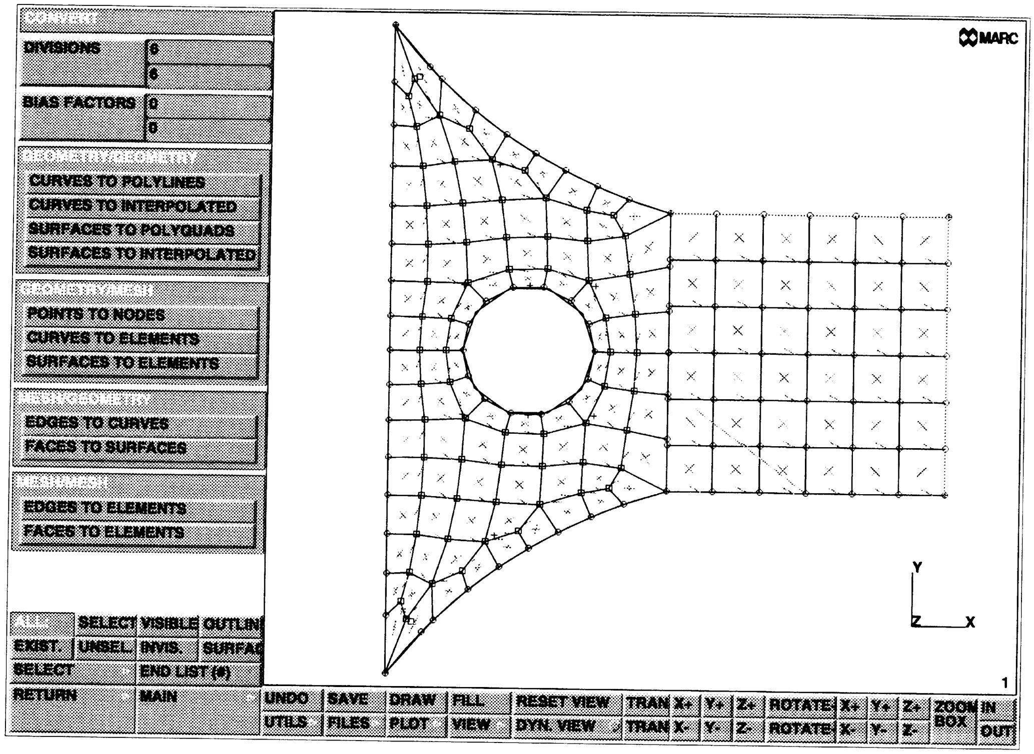


Figure 14.8 Elements on the Square Surface

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 will merge the nodes. The sweep tolerance should be changed back to the default when the merge operation is finished.

MAIN
    MESH GENERATION
        SWEEP
            REMOVE UNUSED NODES
            TOLERANCE
                `0.5`
            SWEEP NODES

(Box Pick the nodes on the interface)

                END LIST (#)
            TOLERANCE
                `0.0001`

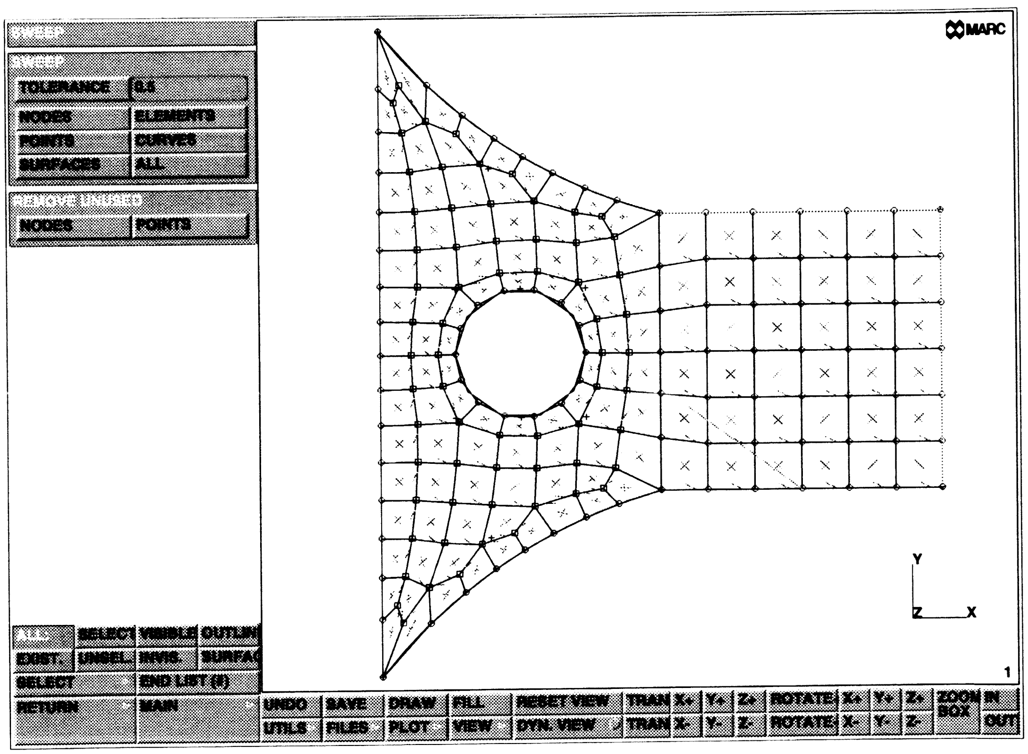


Figure 14.9 Correctly Connected Mesh

Step 3

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
            SPACING
                5  5
            SIZE
                15  15
            grid ON                                   (on)
            RESET
            RETURN
        CURVE TYPE
            LINE
            RETURN
        crvs ADD
            point(0, 15, 0)                           (Pick grid point)
            12                 (Pick upper left point of the surface)
            point(0, 0, 0)                            (Pick grid point)
            7                  (Pick lower left point of the surface)
        GRID                                          (off)
        ATTACH
            attach nodes CURVE
                7                                     (Pick lower line)
                                           (Pick nodes near lower line)
                END LIST (#)
                8                                     (Pick upper line)
                                           (Pick nodes near upper line)
                END LIST (#)
```

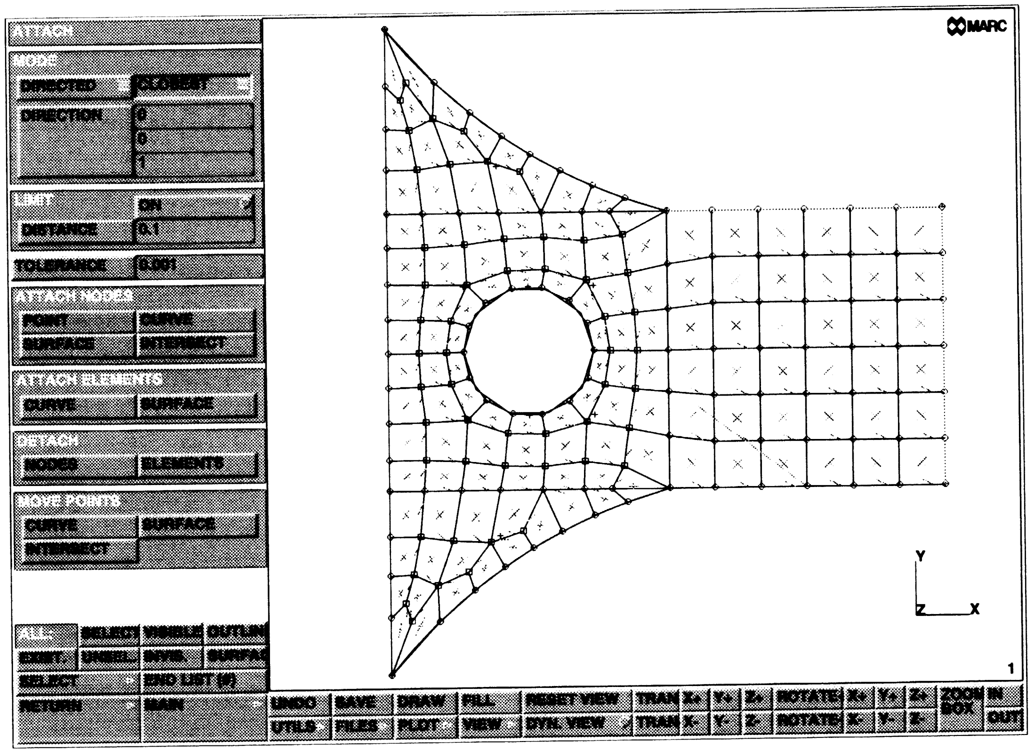


Figure 14.10 Nodes at the Top Half Attached to the Line

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

82 53 *(Pick elements at the triangular corners)*

END LIST (#)

elems ADD *(Pick nodes)*

113

58

109 *(First click on this node)*

109 *(Second click on this node)*

Add three more triangular elements in the same way.

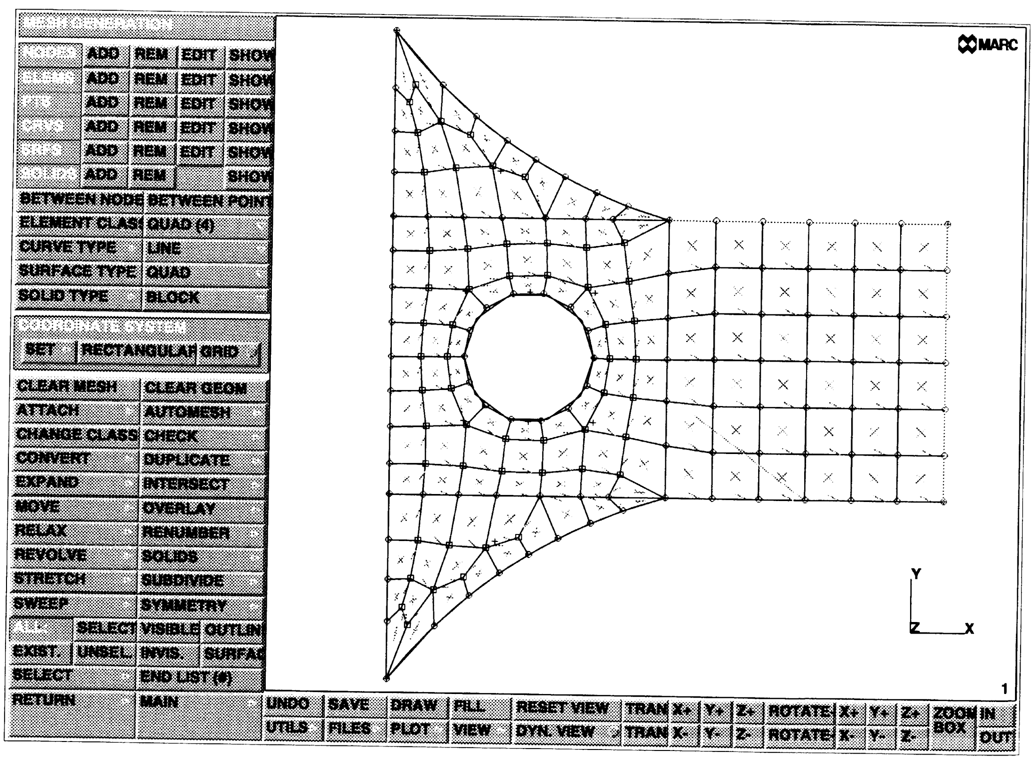


Figure 14.11 Corner Elements Replaced by Triangular Elements

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.

MAIN
 VIEW
  SHOW ALL VIEWS
  ACTIVATE ALL
  PLOT
   draw POINTS *(off)*
   draw CURVES *(off)*
   REGENERATE
   RETURN
  FILL

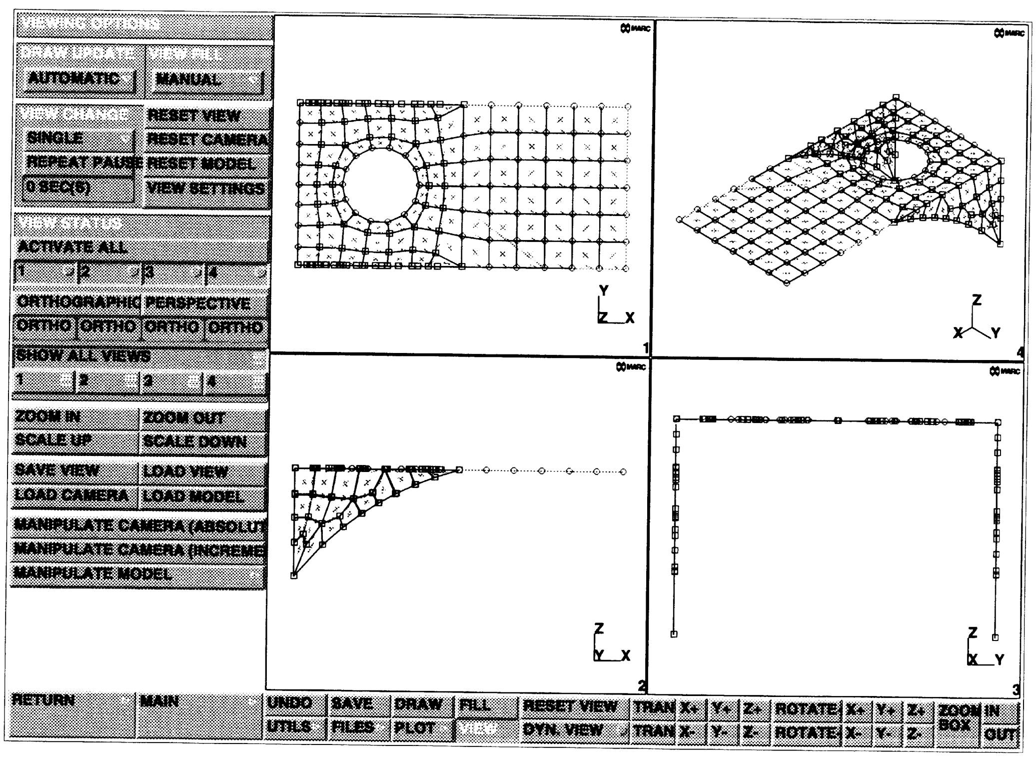


Figure 14.12 The Complete FE Model

Step 4

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 *(on)*
                ON y displace *(on)*
                ON z displace *(on)*
                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` *(Pick the surface)*
    END LIST (#)

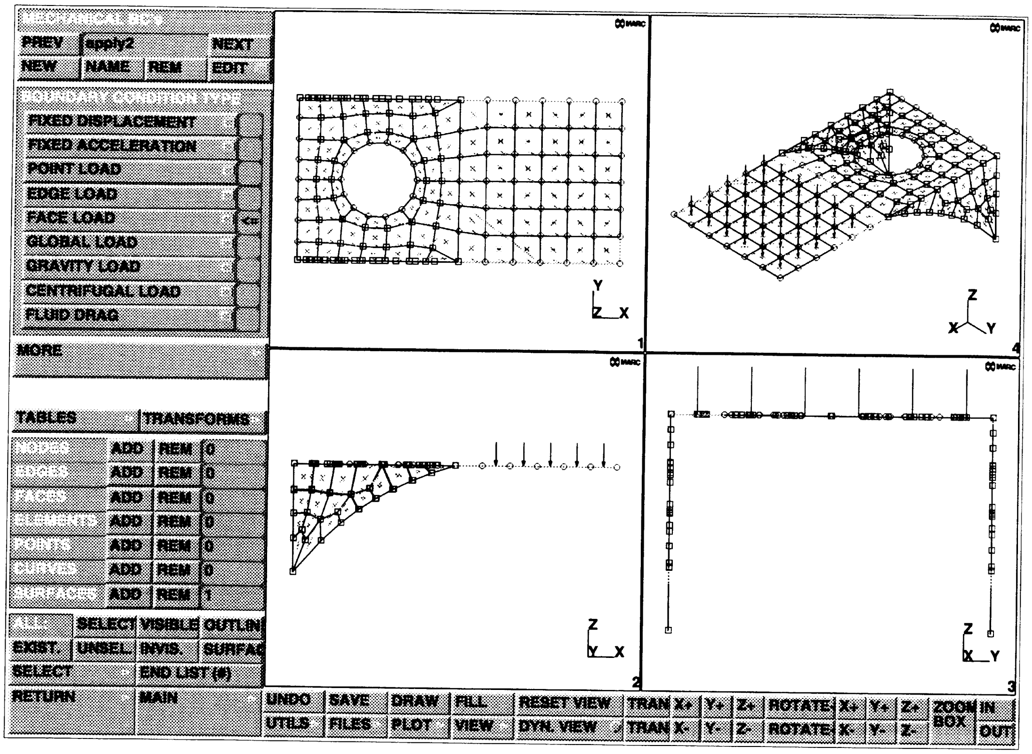


Figure 14.13 Loads Applied to All Elements Attached to the Surface

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
1 *(Pick surface)*
(Box Pick the Elements which are detached from the surface)

END LIST (#)

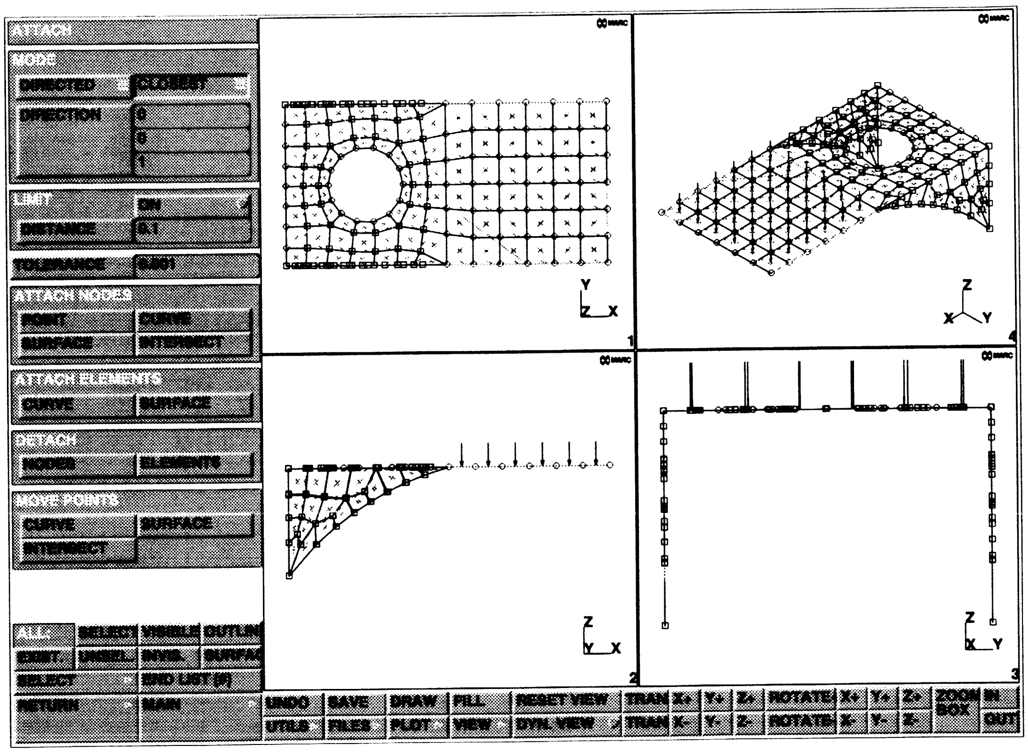


Figure 14.14 Added Surface Loads Due to Renewed Attach

Step 5

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                          (Poisson's ratio)
                0.283/386.4                  (Mass density)
            OK
        elements ADD
            all: EXIST
```

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                           (on)
            PLOT
                elements SOLID
                REGENERATE
                RETURN
            ID GEOMETRIES                           (off)
```

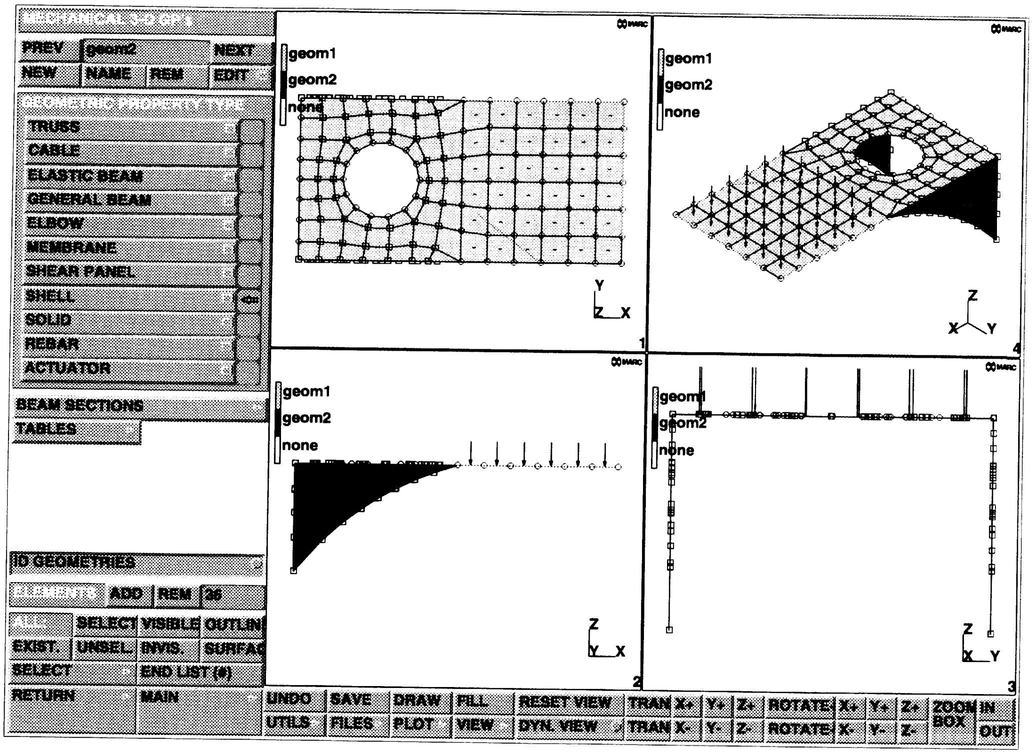


Figure 14.15 Graphical Confirmation of Applied Geometries

Step 6

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.

```
MAIN
    JOBS
        MECHANICAL
            loadcases SELECT
                lcase1
            JOB RESULTS
                SELECT VARIABLES
                    von_mises
                OUTER & MIDPLANE
                    von_mises
                SELECT TENSORS
                    stress
                OUTER & MIDPLANE
                    stress
                OK
            JOB PARAMETERS
                # SHELL/BEAM LAYERS
                    3
                OK
            OK
        SAVE
        RUN
            SUBMIT 1
            MONITOR
```

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
        SCALAR
            DOWN
            Equivalent Von Mises Stress Layer 2
            OK
        CONTOUR BANDS
        NEXT INC
        deformed shape SETTINGS
            deformation scaling AUTOMATIC
```

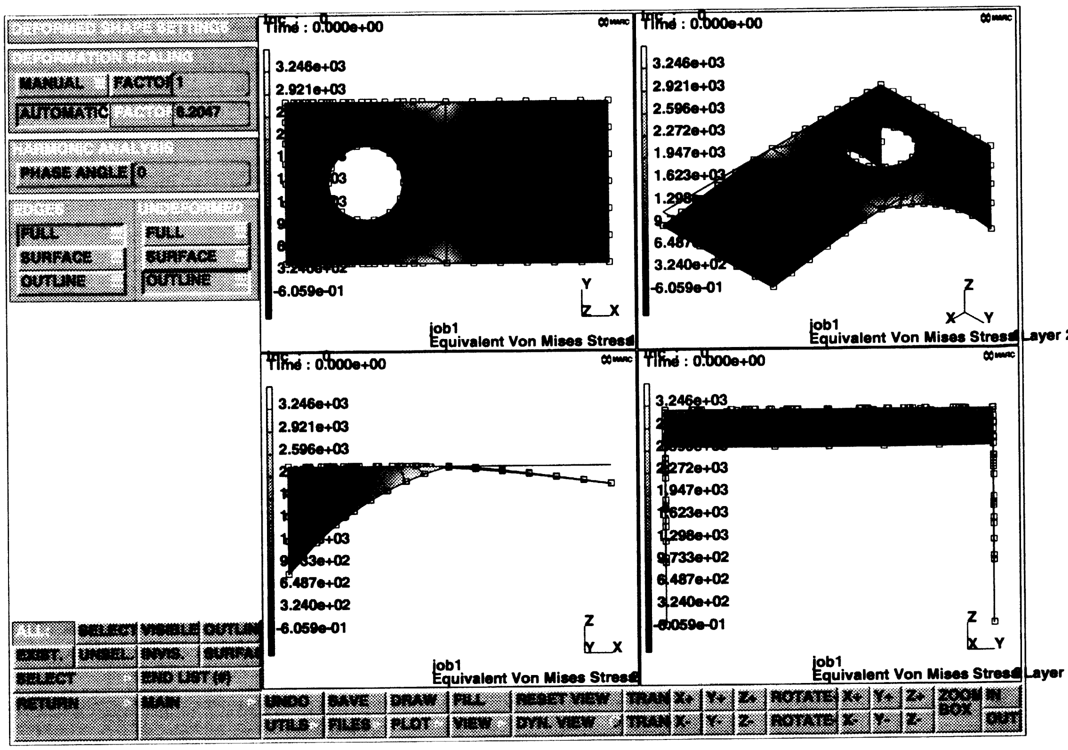


Figure 14.16 Deformed Structure and Contours of von Mises Stress

14.3 Conclusion

The bracket will sustain the required static loads.

14.3.1 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.

14.4 Detailed Session Description of the Mode Shape Analysis

Step 1

For the dynamic analysis, the same geometry will be used. The first step is to restore the database. The following button sequence will restore the database.

MAIN
- RESULTS
  - CLOSE
  - scalar plot OFF
- FILES
  - RESTORE
  - RESET PROGRAM
  - RETURN
- VIEW
  - show 4
- PLOT
  - draw POINTS *(off)*
  - draw CURVES *(off)*
  - draw SURFACES *(off)*
  - RETURN
- FILL

Step 2

The next step is to create a modal dynamic loadcase. The default is 10 modes which is acceptable. 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
            loadcases SELECT
                lcase2
            ANALYSIS OPTIONS
                LANCZOS
                OK
            OK
        RUN
        SAVE
            SUBMIT 1
            MONITOR
```

Step 3

The next step is to postprocess the results. For modal analyses, the shape of the deflections or mode shapes is of most 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                                    (off)
            MORE
                edges OUTLINE
                RETURN
        CONTOUR BANDS
        SCALAR
            Displacement z
            OK
        deformed shape SETTINGS
            deformation scaling AUTOMATIC
            RETURN
        NEXT INC            (Repeat until all modes have been viewed)
```

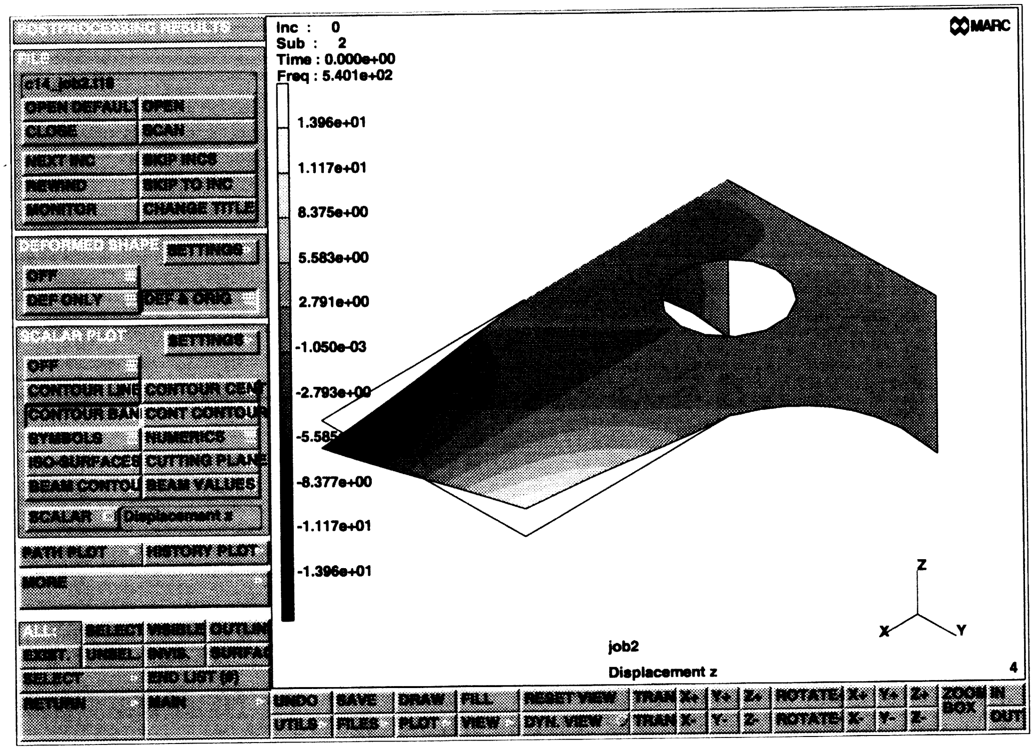


Figure 14.17 The Second Eigenmode

Finally generate an animation sequence of one mode shape:

MAIN
RESULTS
MORE
animate MODE
9 *(Number of frames for animation)*
ANIMATION
PLAY

14.4.1 Overview of steps

Step 1 Create an earthquake table for the loading. Then apply it as a gravity loadcase.

Step 2 Create and submit a transient analysis.

Step 3 Postprocess the results.

14.5 Detailed Session Description of Dynamic Transient Analysis

Step 1

Restore the database before continuing the analysis. The next step is to include the earthquake loading using the table option in the boundary condition menu. Because there are 85 entries, the table is included in the file *c14.tbl* on the Mentat II installation tape. To load the table, the user will need the full path name to the Mentat II 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
        scalar plot OFF
        RETURN
    FILES
        RESTORE
        RESET PROGRAM
        RETURN
    PLOT
        draw CURVES                                   (off)
        draw POINTS                                   (off)
        draw SURFACES                                 (off)
        FILL
```

MAIN
BOUNDARY CONDITIONS
MECHANICAL
TABLE
LOAD
`c14.tbl`
FIT
SHOW MODEL
RETURN
NEW
GRAVITY LOAD
z accel. TABLE
table1
OK
OK
elements ADD
(Box Pick the cantilevered elements)
END LIST (#)

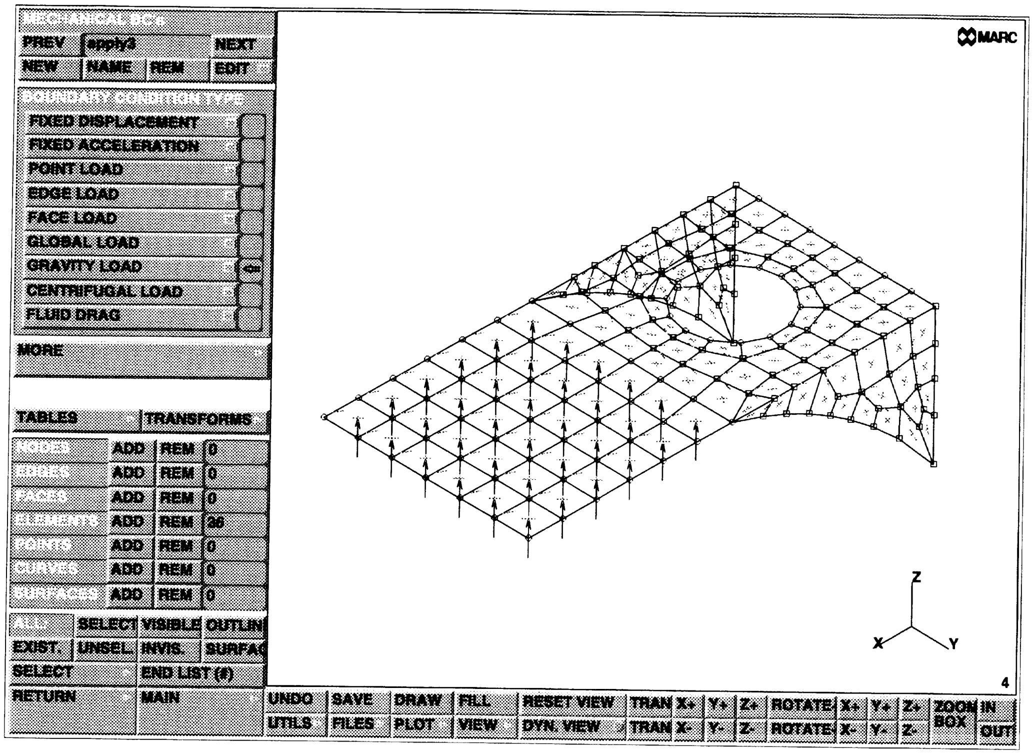


Figure 14.18 Transient Gravity Loading

Step 2

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
                DESELECT
                    apply2
                    OK
                TOTAL LOADCASE TIME
                    9
                fixed PARAMETERS
                    # STEPS
                        90
                    OK
                OK
            OK
```

Step 3

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
            loadcases SELECT
                lcase3
            JOB RESULTS
                SELECT VARIABLES
                    von_mises
                OUTER & MIDPLANE
                    von_mises
                OK
            JOB PARAMETERS
                # SHELL/BEAM LAYERS
                    3
                OK
            OK
        SAVE
        RUN
            SUBMIT 1
            MONITOR
```

Step 4

The final step in any analysis is to post process 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
        FILL
        DEF & ORIG
        SCALAR
            Equivalent Von Mises Stress Layer 2
            RETURN
        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 (#)
FILLED *(off)*
SHOW IDS
`5`
VARIABLES
ADD CURVE
Arc Length
DOWN
DOWN
Equivalent Von Mises Stress Layer 1
Arc Length
DOWN
DOWN
Equivalent Von Mises Stress Layer 2
Arc Length
DOWN
DOWN
Equivalent Von Mises Stress Layer 3
RETURN
FIT
YMAX
`36000`
MONITOR

The stresses for the nodes in the node path do not exceed the yield stress of 36000 psi. As a last step, a history plot is made. The resulting plot is shown in Figure 14.19.

```
MAIN
    RESULTS
        REWIND
        HISTORY PLOT
        SET NODES
            166
            109
            113
            END LIST (#)
        COLLECT DATA
            0  90  1
        FILLED                                        (off)
        SHOW IDS
            0
        NODES/VARIABLES
            ADD VARIABLE
                Time
                DOWN
                DOWN
                DOWN
                DOWN
                DOWN
                Equivalent Von Mises Stress Layer 3
            FIT
```

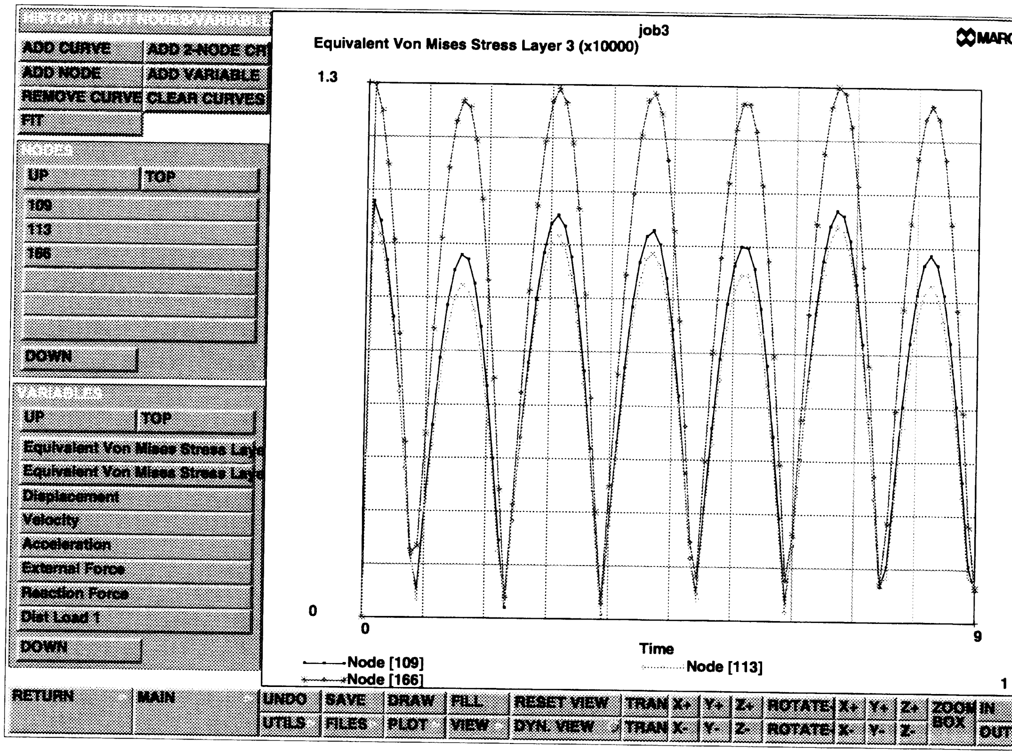


Figure 14.19 History Plot of von Mises Stress

14.6 Conclusion

The bracket is shown to withstand the dynamic loads. The stresses never exceed yield.

14.7 Earthquake Table

Earthquake table file c14.tbl:

```
# Title
table1
# X-axis Label
X
# Y-axis Label
Y
# Type
                   1
# Steps in X and Y
                  10                    20
# X-min, X-max, Y-min, Ymax
  0.000000000000e+00  9.000000000000e+00 -1.000000000000e+00  1.000000000000e+00
                  85
  0.000000000000e+00  0.000000000000e+00                   1
  1.000000000000e-01  5.000000000000e-02                   2
  2.000000000000e-01  2.500000000000e-01                   3
  3.000000000000e-01 -5.000000000000e-02                   4
  4.000000000000e-01  3.500000000000e-01                   5
  5.000000000000e-01 -1.000000000000e-01                   6
  6.000000000000e-01  5.000000000000e-02                   7
  7.000000000000e-01  4.000000000000e-01                   8
  8.000000000000e-01  1.500000000000e-01                   9
  9.000000000000e-01 -5.000000000000e-02                  10
  1.000000000000e+00  4.000000000000e-01                  11
  1.100000000000e+00 -1.500000000000e-01                  12
  1.200000000000e+00  5.000000000000e-02                  13
  1.300000000000e+00 -5.000000000000e-02                  14
  1.400000000000e+00  1.500000000000e-01                  15
  1.500000000000e+00 -1.500000000000e-01                  16
  1.600000000000e+00 -5.000000000000e-02                  17
  1.700000000000e+00 -1.000000000000e-01                  18
  1.800000000000e+00  3.000000000000e-01                  19
  1.900000000000e+00  1.500000000000e-01                  20
  2.000000000000e+00  2.500000000000e-01                  21
  2.100000000000e+00  2.000000000000e-01                  22
  2.200000000000e+00  4.500000000000e-01                  23
  2.300000000000e+00 -1.000000000000e-01                  24
  2.400000000000e+00  1.000000000000e-01                  25
  2.500000000000e+00 -5.000000000000e-02                  26
  2.600000000000e+00  2.500000000000e-01                  27
  2.700000000000e+00  3.500000000000e-01                  28
  2.800000000000e+00  2.500000000000e-01                  29
```

2.900000000000e+00	5.500000000000e-01	30
3.000000000000e+00	3.500000000000e-01	31
3.100000000000e+00	6.000000000000e-01	32
3.200000000000e+00	3.000000000000e-01	33
3.300000000000e+00	1.500000000000e-01	34
3.400000000000e+00	2.500000000000e-01	35
3.500000000000e+00	1.000000000000e-01	36
3.600000000000e+00	-1.000000000000e-01	37
3.700000000000e+00	-2.000000000000e-01	38
3.800000000000e+00	-1.000000000000e-01	39
3.900000000000e+00	-2.000000000000e-01	40
4.000000000000e+00	-5.000000000000e-02	41
4.100000000000e+00	-1.500000000000e-01	42
4.200000000000e+00	4.500000000000e-01	43
4.300000000000e+00	3.000000000000e-01	44
4.400000000000e+00	5.500000000000e-01	45
4.500000000000e+00	4.000000000000e-01	46
4.600000000000e+00	6.500000000000e-01	47
4.700000000000e+00	3.500000000000e-01	48
4.800000000000e+00	8.500000000000e-01	49
4.900000000000e+00	5.000000000000e-01	50
5.000000000000e+00	3.000000000000e-01	51
5.100000000000e+00	4.500000000000e-01	52
5.200000000000e+00	2.000000000000e-01	53
5.300000000000e+00	5.000000000000e-02	54
5.400000000000e+00	2.000000000000e-01	55
5.500000000000e+00	1.000000000000e-01	56
5.600000000000e+00	-5.000000000000e-02	57
5.700000000000e+00	-2.000000000000e-01	58
5.800000000000e+00	-1.000000000000e-01	59
5.900000000000e+00	-5.000000000000e-02	60
6.000000000000e+00	-3.000000000000e-01	61
6.100000000000e+00	-1.500000000000e-01	62
6.200000000000e+00	-4.500000000000e-01	63
6.300000000000e+00	-3.500000000000e-01	64
6.400000000000e+00	-4.000000000000e-01	65
6.500000000000e+00	-1.000000000000e-01	66
6.600000000000e+00	-2.000000000000e-01	67
6.700000000000e+00	1.500000000000e-01	68
6.800000000000e+00	5.000000000000e-02	69
6.900000000000e+00	2.000000000000e-01	70
7.000000000000e+00	1.000000000000e-01	71
7.100000000000e+00	-5.000000000000e-02	72
7.200000000000e+00	5.000000000000e-02	73
7.300000000000e+00	-1.000000000000e-01	74
7.400000000000e+00	1.500000000000e-01	75
7.500000000000e+00	5.000000000000e-02	76
7.700000000000e+00	-5.000000000000e-02	77
7.800000000000e+00	0.000000000000e+00	78
8.000000000000e+00	5.000000000000e-02	79

```
8.200000000000e+00 -5.000000000000e-02        80
8.300000000000e+00 -5.000000000000e-02        81
8.400000000000e+00  5.000000000000e-02        82
8.600000000000e+00  5.000000000000e-02        83
8.800000000000e+00 -5.000000000000e-02        84
9.000000000000e+00  0.000000000000e+00        85
```

14.8 Procedure File

```
| Version : MENTAT II 2.3
|
| This example 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. The 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.
|
|
| THE LINEAR STATIC CASE
|
| Step 1: Create the boundary of a flat area representing
|         the half of the plate with the hole in it.
|         Convert it to finite elements.
|
*set_grid_spacing
5 5
*set_grid_size
30 30
*set_grid on
*fill_view
*zoom_box
*zoom_box(1,0.268182,0.686548,0.979545,-0.025381)
*add_curves
point(0,-10,0)
point(0,25,0)
point(15,0,0)
point(15,15,0)
point(30,0,0)
point(30,15,0)
*set_curve_type arc_cpp
*add_curves
22.5 -27.5 0
15 0 0
0 -10 0
22.5 42.5 0
0 25 0
15 15 0
*origin_xy
7.5 7.5
*set_grid_size
10 10
*set_grid_spacing
0.5 0.5
*zoom_box
```

```
*zoom_box(1,0.275000,0.602792,0.510227,0.389594)
*set_curve_type circle_cp
*add_curves
0 0 0
3.5 0 0
*fill_view
 *set_grid off
*set_overlay_divisions
15 20
*overlay_mesh
 1 2 4 5 6
# | End of List
|
| Step 2: Create the cantilevered section of the plate
|         Convert it to finite elements. Merge the two
|         parts together.
|
*add_surfaces
 7
 5
 6
 12
*set_convert_divisions
6 6
*convert_surfaces
 1
# | End of List
*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
*set_sweep_tolerance
0.001
|
| Step 3: Fold the vertical sections and modify the
|         elements in the triangular region.
|
*set_grid_spacing
5 5
*set_grid_size
15 15
*set_grid on
*system_reset
*set_curve_type line
*add_curves
point(0,15,0)
 12
point(0,0,0)
```

```
  7
  *set_grid off
 *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
 *remove_elements
  82
  53
 # | End of List
 *add_elements
  113
  58
  109
  109
  58
  52
  109
  109
  22
  16
  83
  83
  16
  81
  83
  83
*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
# | End of List
*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
*show_all_views
*activate_all_views
 *set_points off
 *set_curves off
*regen
```

```
*fill_view
|
| Step 4: Apply boundary conditions and loads.
|
*apply_type fixed_displacement
*apply_dof x
*apply_dof y
*apply_dof z
*add_apply_nodes
 70 71 72 75 78 82 84 90 94 98 102 108 110 114 117 120 121
# | End of List
*new_apply
*apply_type face_load
*apply_value p
1
*add_apply_surfaces
 1
# | End of List
*attach_elements_surface
 1
 93 99 105 111 117 123
# | End of List
|
| Step 5: Assign material and geometric properties.
|
*material_type mechanical:isotropic
*material_value isotropic:youngs_modulus
30e6
0.3
0.283/386.4
*add_material_elements
all_existing
*pick_inside_partial
*geometry_type mech_three_shell
*geometry_value thick
0.25
*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
*geometry_value thick
.5
*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
*elements_solid
*regen
 *identify_none *regen
|
| Step 6: Create the loadcase and submit it.
|
*loadcase_type static
*job_class mechanical
*add_job_loadcases
lcase1
*add_post_var
von_mises
*post_var_outer_layers
von_mises
*add_post_tensor
stress
*post_tensor_outer_layers
stress
*job_param layers
3
*save_as_model
c14
y
*update_job
*submit_job 1
*monitor_job
|
| Step 7: Postprocess the results.
|
*post_open_default
*set_deformed both
*post_value
*pick_list_next(post_value,14)
Equivalent von Mises Stress Layer 2
*post_contour_bands
*post_next
*set_automag on
|
| MODE SHAPE ANALYSIS
|
| Step 1: Restore the database from the static analysis.
|
*post_close
*post_off
*restore_model
*reset
*show_view 4
 *set_points off
 *set_curves off
```

```
 *set_surfaces off
*fill_view
|
| Step 2: Create a modal dynamic loadcase and submit it.
|
*new_loadcase
*loadcase_type dynamic_modal
*new_job
*job_class mechanical
*add_job_loadcases
lcase2
*job_option modal:lanczos
*save_model
*update_job
*submit_job 1
*monitor_job
|
| Step 3: Post process the results.
|
*post_open_default
*set_deformed both
 *set_nodes off
*edges_outline
*post_contour_bands
*post_value
Displacement z
*set_automag on
*post_next
*post_next
*post_next
*post_next
*post_next
*post_next
*post_next
*post_next
*post_next
*post_next
*post_next
*post_animate_mode
9
*animation_play
|
| DYNAMIC TRANSIENT ANALYSIS
|
| Step 1: Create an earthquake table for loading.
|         Then apply it as a gravity loadcase (transient).
|
*post_close
*post_off
*restore_model
```

```
*reset
 *set_points off
 *set_curves off
 *set_surfaces off
*fill_view
*new_apply
*table_read
c14.tbl
*table_fit
*show_model
*apply_type gravity_load
*apply_table z0
table1
*add_apply_elements
 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
# | End of List
|
| Step 2: Create and submit the transient analysis.
|
*new_loadcase
*loadcase_type dynamic_transient
*remove_loadcase_loads
apply2
*loadcase_value time
9
*loadcase_option stepping:fixed
*loadcase_option stepping:fixed
*loadcase_value nsteps
90
*new_job
*job_class mechanical
*add_job_loadcases
lcase3
*add_post_var
von_mises
*post_var_outer_layers
von_mises
*job_param layers
3
*save_model
*update_job
*submit_job 1
*monitor_job
|
| Step 3: Postprocess the results.
|
*post_open_default
*set_deformed both
```

```
*fill_view
*post_value
*pick_list_next(post_value,14)
*pick_list_next(post_value,14)
Equivalent Von Mises Stress Layer 2
*post_contour_bands
*post_monitor
*post_rewind
*set_pathplot_path
 171
 109
 121
# | End of List
 *pathplot_unfilled
*set_pathplot_node_id
5
*pathplot_add
Arc Length
*pick_list_next(pathplot_variable,14)
*pick_list_next(pathplot_variable,14)
Equivalent Von Mises Stress Layer 1
*pathplot_add
Arc Length
*pick_list_next(pathplot_variable,14)
*pick_list_next(pathplot_variable,14)
Equivalent Von Mises Stress Layer 2
*pathplot_add
Arc Length
*pick_list_next(pathplot_variable,14)
*pick_list_next(pathplot_variable,14)
Equivalent Von Mises Stress Layer 3
*pathplot_fit
*set_pathplot_ymax
36000
*post_monitor
*post_rewind
*set_history_nodes
 166
 109
 113
# | End of List
*history_collect
0 90 1
 *history_unfilled
*set_history_increment_id
0
*history_add_var
Time
*pick_list_next(history_variable,7)
*pick_list_next(history_variable,7)
```

```
*pick_list_next(history_variable,7)
*pick_list_next(history_variable,7)
*pick_list_next(history_variable,7)
Equivalent Von Mises Stress Layer 3
*history_fit
```

Chapter 15: Cooling Fin

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 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.

15.1 Background Information

15.1.1 Description

This problem demonstrates the preparation of a heat transfer model including convection boundary conditions.

15.1.2 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.

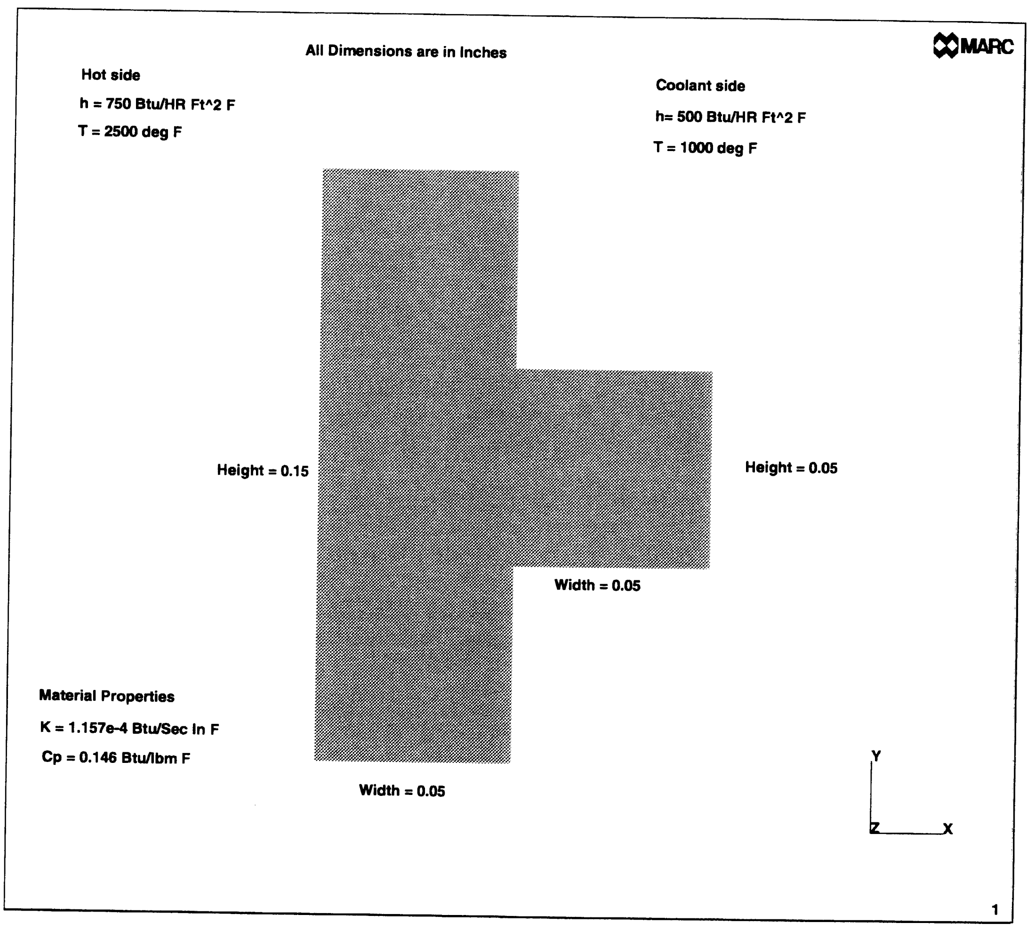


Figure 15.1 Cross-Section of Cooling Fin

15.1.3 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.

15.1.4 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

$$h = 750\ Btu/HR\ Ft^2\ {}^oF$$

$$= 750 \times \frac{1}{3600(12)^2}\ Btu/\sec\ in^2\ {}^oF$$

$$T_{\infty} = 2500\ {}^oF$$

The right side has a convection:

$$q = h(T - T_{\infty})$$

with

$$h = 500\ Btu/HR\ Ft^2\ {}^oF$$

$$= 500 \times \frac{1}{3600(12)^2}\ Btu/\sec\ in^2\ {}^oF$$

$$T_{\infty} = 1000\ {}^oF$$

The material has a coefficient of thermal conduction $\lambda = 1.157\ 10^{-4}\ Btu/\sec\ in^2\ {}^oF$

For a steady state analysis it is not required to enter the mass density and the heat capacity.

15.1.5 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.

15.2 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
            SPACING
                0.05  0.05
            SIZE
                0.2  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
                1                   (Pick the first surface)
                END LIST (#)
            DIVISIONS
                3  2
            SURFACES TO ELEMENTS
                2                  (Pick the second surface)
                END LIST (#)
```

The next button sequence will merge the duplicate nodes on the interface of the two surfaces.

MAIN
- MESH GENERATION
  - SWEEP
    - sweep NODES
      - all: EXIST.
    - PLOT
      - draw POINTS *(off)*
      - draw SURFACES *(off)*
      - REGENERATE
      - RETURN
    - FILL

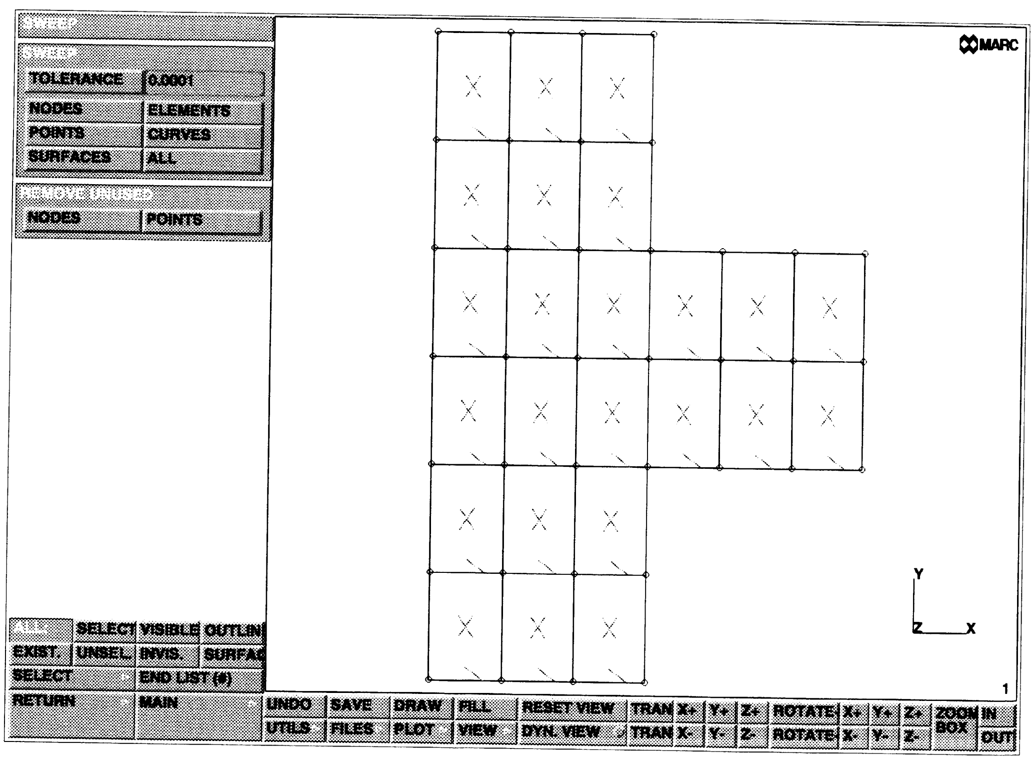


Figure 15.2 The Mesh Generated Using the Convert Option

Step 2

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.

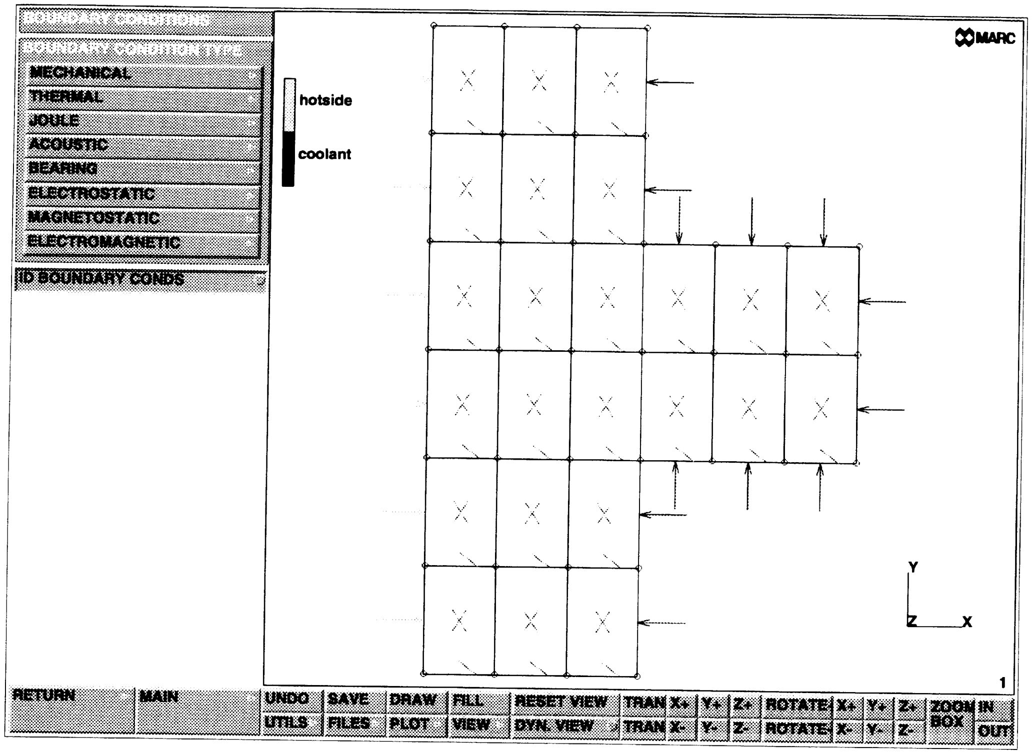


Figure 15.3 The Film Conditions on the Hot and Coolant Side

Step 3

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.
```

Step 4

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
                lcase1
            analysis dimension PLANAR
            OK
        ELEMENT TYPES
            PLANAR
                39
                OK
            all: EXIST.
            RETURN
        CHECK
        SAVE
        RUN
            SUBMIT 1
            MONITOR
```

Step 6

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

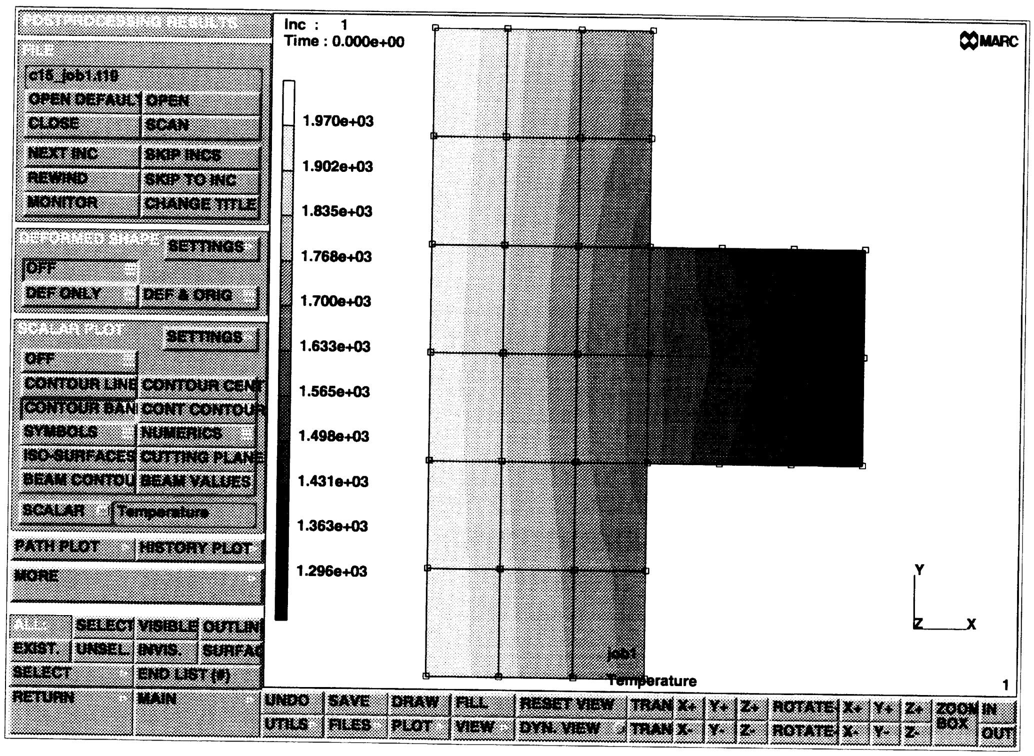


Figure 15.4 Contours of Temperature for Structure with Fin

Step 7

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

(Box Pick the fin elements)

END LIST (#)
SWEEP
remove unused NODES

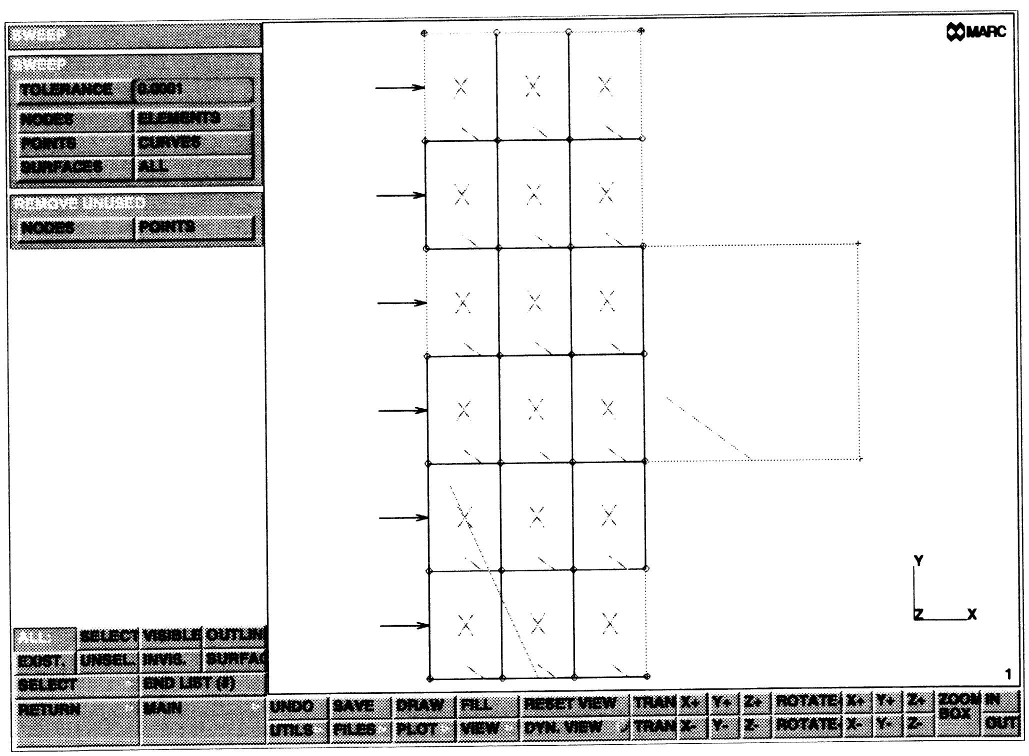


Figure 15.5 Mesh without Fin

Step 8

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.

MAIN
    BOUNDARY CONDITIONS
        ID BOUNDARY CONDS *(on)*
        ID BOUNDARY CONDS *(off)*
        THERMAL
            NEXT *(to edit the second boundary condition)*
            edges ADD
                `12:1  9:1` *(Box Pick the right edges)*
                END LIST (#)
            RETURN
        ID BOUNDARY CONDS *(on)*
        ID BOUNDARY CONDS *(off)*

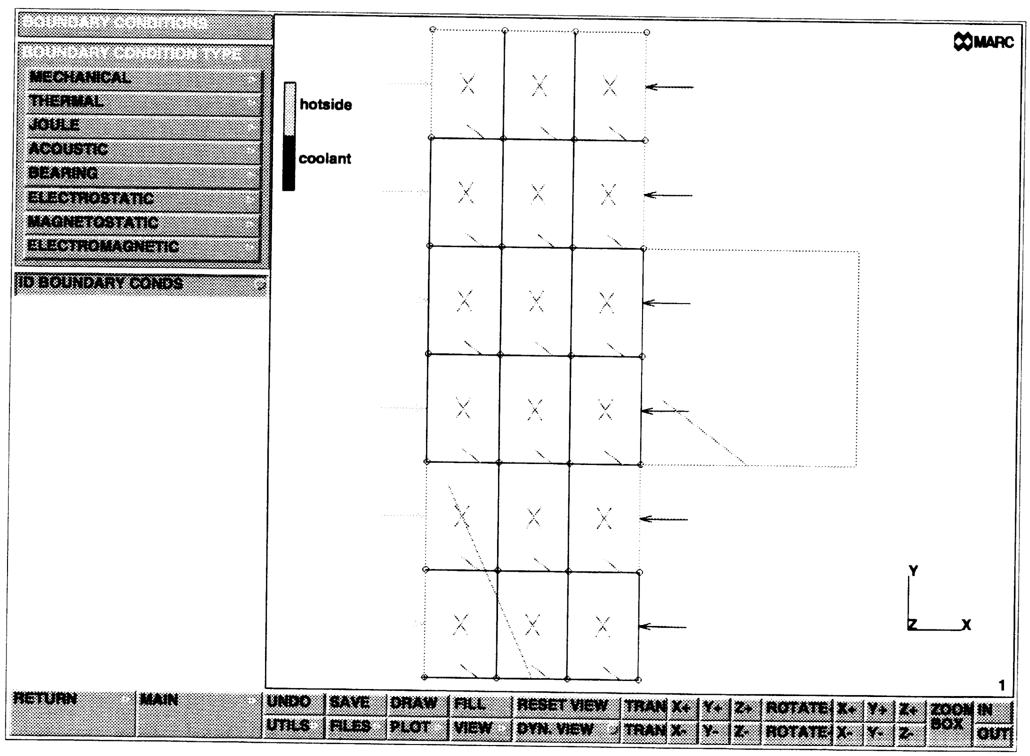


Figure 15.6 New Thermal Boundary Conditions

Step 9

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
```

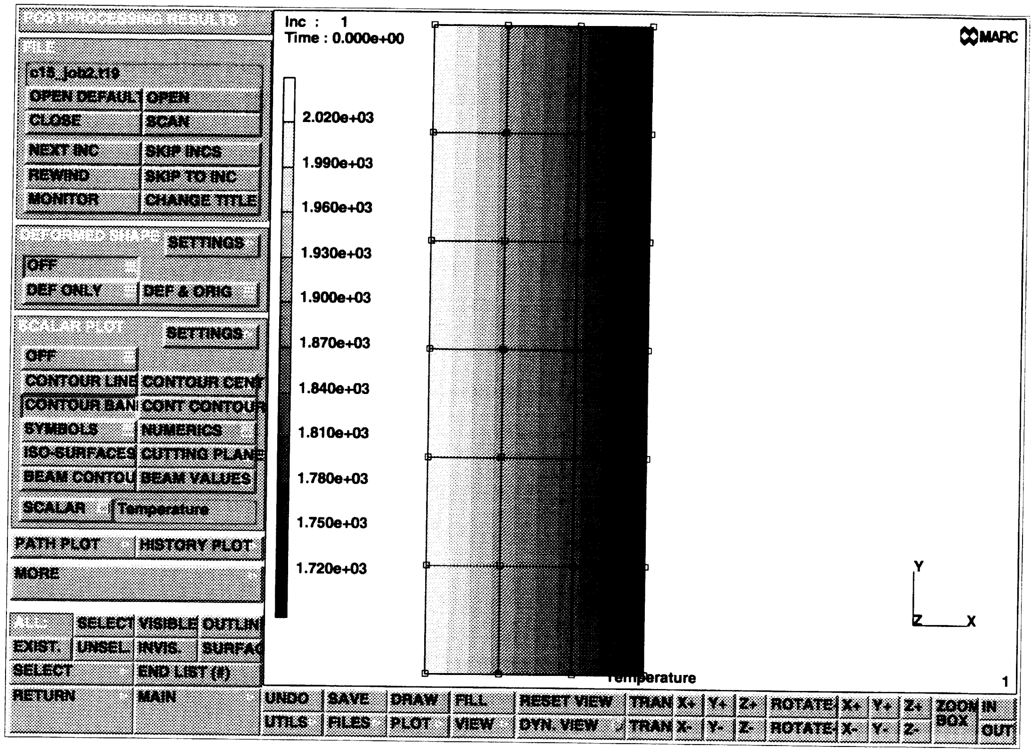


Figure 15.7 Results for Structure without Fin

15.3 Procedure File

```
| Version : MENTAT II 2.3
|
| This problem demonstrates the preparation of a heat transfer model including
| convection boundary conditions.
| 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 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.
|
|
| Step 1
|
| Set grid dimensions and activate
|
*set_grid_spacing
0.05 0.05
*set_grid_size
0.2 0.2
*set_grid on
*fill_view
*zoom_box
*zoom_box(1,0.462500,0.560914,0.928409,0.024112)
|
| Add two surfaces
|
*add_surfaces
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)
*set_grid off
*set_convert_divisions
3 6
*convert_surfaces
 1
# | End of List
*set_convert_divisions
3 2
*convert_surfaces
 2
# | End of List
```

```
*sweep_nodes
all_existing
*set_points off
*set_surfaces off
*regen
*fill_view
|
| Step 2
|
| Add two sets of convective Boundary Conditions
|
*apply_name
hotside
*apply_type edge_film
*apply_value h
750/(3600*144)
*apply_value tinf
2500
*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
*apply_value h
500/(3600*144)
*apply_value tinf
1000
*add_apply_edges
 15:1 18:1 22:2 23:2 24:2 21:1 24:1 19:0 20:0 21:0 3:1 6:1
# | End of List
*identify_applys *regen
*identify_none *regen
|
| Step 3
|
| Specify the material properties
|
*material_type heat
*material_value heat:conductivity
1.157e-4
*add_material_elements
all_existing
|
| Step 4
|
| Create a steady-state loadcase
|
*loadcase_type steady_state
```

```
|
| Step 5
|
| Define a job and submit
|
*job_class heat
*add_job_loadcases
lcase1
*job_option dimen:two
*element_type 39
all_existing
*check_job
*save_as_model
c15
y
*update_job
*submit_job 1
*monitor_job
|
| Step 6
|
| Postprocess results
|
*post_open_default
*post_contour_bands
*post_next
*post_next
|
| Step 7
|
| Recapture the database
|
*post_close
*post_off
*restore_model
*reset
|
| Delete fin elements
|
*remove_elements
 19 20 21 22 23 24
# | End of List
*remove_unused_nodes
|
| Step 8
|
| Repair the convective B.C.'s
|
*identify_applys *regen
*identify_none *regen
```

```
*next_apply
*add_apply_edges
 12:1 9:1
# | End of List
*identify_applys *regen
 *identify_none *regen
|
| Step 9
|
| Define a new job and submit
|
*new_job
*job_class heat
*add_job_loadcases
lcase1
*job_option dimen:two
*check_job
*save_model
*update_job
*submit_job 1
*monitor_job
|
| Step 10
|
| Postprocess the results
|
*post_open_default
*post_contour_bands
*post_next
*post_next
```

MARC
Analysis
Research
Corporation

CALIFORNIA - Palo Alto
MARC Corporate Headquarters
260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306, USA
Tel: 1 415 329 6800
Fax: 1 415 323 5892
Email: support@marc.com

CALIFORNIA - San Diego
MARC Analysis Research Corp.
4350 LaJolla Village Drive, Suite 300
San Diego, CA 92122
Tel: 1 619 546 4414
Fax: 1 619 587 8710

CONNECTICUT - Manchester
MARC Analysis Research Corp.
Prestige Office Center
150 North Main Street, Suite 5
Manchester, CT 06040
Tel: 1 203 647 4841
Fax: 1 203 647 4870

MICHIGAN - Ann Arbor
MARC Analysis Research Corp.
25 Frank Lloyd Wright Drive
Ann Arbor, MI 48106-0523
Tel: 1 313 998 0540
Fax: 1 313 998 0542

CZECH REPUBLIC
MARC Overseas, Inc.
Podolska 50
147 00 Praha 4
Czech Republic
Tel: 011 42 2 6121 4123
011 42 2 6121 4111 x252
Fax: 011 42 2 6121 4123

GERMANY - Hannover
MARC Software Deutschland Gmbh
Alte Dohrener Str. 66
D 30173 Hannover, Germany
Tel: 011 49 511 980 5182
Fax: 011 49 511 980 5187

GERMANY - Munich
MARC Software Deutschland GmbH
Ismaninger Str. 9
85609 Aschheim (bie Munich)
Germany
Tel: 011 49 89 904 50 33
Fax: 011 49 89 903 06 76
Email: system@marc.co.de

HOLLAND
MARC Analysis Research Corporation
Dublinstraat 32
2713 HS Zoetermeer
The Netherlands
Tel: 011 31 79 3510 411
Fax: 011 31 79 3517 560
Email: support@marc.nl

ITALY
Espri-MARC s.r.l.
16129 Viale Brigata Bisagno 2/10
I-16129 Genova
Italy
Tel: 011 39 10 585 949
Fax: 011 39 10 585 949

UNITED KINGDOM
MARC UK Ltd.
35, Shenley Pavilions
Chalkdell Drive
Shenley Wood
Milton Keynes, MK56LB
Tel: 011 44 1908 506 505
Fax: 011 44 1908 506 522

CHINA
MARC Overseas, Inc.
Beijing Yanshan Apartment #1703/04
138 Haidian Road
Beijing 100 086
Tel: 001 86 10 256 3388 x1703/04
001 86 10 256 4375
Fax: 001 86 10 261 5714

JAPAN - Osaka
Nippon MARC Co., Ltd.
4F 2nd Kimi Building
2-11 Toyotucho Suita-shi
Osaka 564
Tel: 011 81 6 385 1101
Fax: 011 81 3 385-4343

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
Tel: 011 81 3 3345 0181
Fax: 011 81 3 3345 1529
Email: system@marc.co.jp

KOREA
Dong Kyung Bldg. 7FL
824-19 Yuksam-Dong
Kngnam-Ku
Seoul, Korea
Tel: 011 82 2 561 7543
Fax: 011 82 2 561 7767

Document Title: **Mentat II User's Guide**
Part Number: UG-3009-2.3
Revision Date: June, 1996