



# ***FEMAP***

**Examples**

**Version 7.1**



---

## **FEMAP Version 7.0 Examples**

**Copyright © 1986-2000 by Structural Dynamics Research Corp.**

**Proprietary Data. Unauthorized use, distribution, or duplication is prohibited.**

**All Rights Reserved.**

Portions of this software and related documentation are derived from GHS3D software under license from INRIA, other portions are copyrighted by and are the property of Electronic Data Systems Corporation and Spatial Technology Inc.

The FEMAP Documentation may not be copied, reproduced, disclosed, transferred, or reduced to any form, including electronic medium or machine-readable form, or transmitted or publicly performed by any means, electronic or otherwise, unless Structural Dynamics Research Corp (SDRC) consents in writing in advance.

Use of the software has been provided under a Software License Agreement.

Information described in this document is furnished for information only, is subject to change without notice, and should not be construed as a commitment by SDRC. SDRC assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

FEMAP is a registered trademark of Structural Dynamics Research Corp.

*Dual Engine Geometry Modeling* and *FEMAP Professional* are trademarks of Structural Dynamics Research Corp.

Structural Dynamics Research Corp.

P.O. Box 1172, Exton, PA 19341

Phone: (610) 458-3660

FAX: (610) 458-3665

E-mail: [info@femap.com](mailto:info@femap.com)

Web: <http://www.femap.com>


This manual and software product are both copyrighted and all rights are reserved by SDRC. The distribution and sale of this product are intended for the use of the original purchaser only and for use only on the computer system specified. The software product may be used only under the provisions of the license agreement included with the FEMAP package. Unless otherwise stated, you may only use this software on a single computer, by one person, at one time.

## **Trademark Information**

Throughout this manual, and the software, you will see references to other applications and trademarks which are the property of various companies.

- NASTRAN and Cosmic NASTRAN are registered trademarks of NASA.
- IBM is a registered trademark of International Business Machines Corporation.
- MSC/, MSC/NASTRAN, MSC/pal, MSC/pal 2, and PATRAN are registered trademarks of The MacNeal-Schwendler Corporation.
- CDA/Sprint is a trademark of The CDA Group.
- UAI/NASTRAN is a product of Universal Analytics, Inc.
- CSA/NASTRAN is a product of Computerized Structural Research and Analysis Corp.
- ME/NASTRAN is a product of Macro Engineering, Inc.
- ABAQUS is a registered trademark of Hibbitt, Karlsson, and Sorenson, Inc.
- CAEFEM is a trademark of Concurrent Analysis Corp.
- SDRC, SDRC I-DEAS and I-DEAS are registered trademarks of Structural Dynamics Research Corporation.
- SSS/NASTRAN is a trademark of Schaeffer Software Systems, Inc.
- ANSYS is a registered trademark of ANSYS, Inc.
- STAAD and STARDYNE are products and trademarks of Research Engineers, Inc.
- COSMOS and COSMOS/M are registered trademarks of Structural Research and Analysis Corporation.
- WECAN is a registered trademark of Westinghouse, Inc., marketed by AEGIS Software Corp.
- ALGOR is a registered trademark of Algor Interactive Systems, Inc.
- CFDesign is a trademark of Blue Ridge Numerics, Inc.
- Pro/ENGINEER is a registered trademark of Parametric Technology Corporation
- GENESIS is a registered trademark of Vanderplaats, Miura and Associates, Inc.
- MTAB\*Stress is a trademark of Structural Analysis, Inc.
- AutoCAD and DXF are registered trademarks of Autodesk, Inc.
- Solid Edge is a trademark of Intergraph Corporation.
- MicroStation is a registered trademark of Bentley Systems, Inc.
- SolidWorks is a trademark of SolidWorks Corporation.
- MARC is a trademark of MARC Analysis Research Corporation.
- LS-DYNA is a trademark of Livermore Software Technology Corporation.
- ACIS is a registered trademark of Spatial Technology, Inc.
- Parasolid and Unigraphics are registered trademarks of Electronic Data Systems Corporation.
- Windows, Windows NT, Windows 95, and Windows 98 are registered trademarks of Microsoft Corporation. Portions of the software contained on your FEMAP CD are copyrighted by Microsoft Corporation.

Other brand or product names are trademarks or registered trademarks of their respective holders.

- 
- 
- Portions of this software are copyrighted by Spatial Technology, Inc., Electronic Data Systems, Inc., INRIA, Cypress Software Inc., and Microsoft.

## **Manual Conventions**

This manual uses different fonts to highlight specific features, to identify data that Windows will display, or input that you must type.

<code>Enter, Alt</code>	Shows one or more keys that you should press. In some cases, you will see combinations like Alt+Shift+Back-space. The plus signs show that you should press all keys simultaneously.
<code>a:setup</code>	Shows text that you should type. This is very similar to the keystrokes described above, but is typically used for strings of letters and/or numbers. The keystrokes typically refer to the more specialized, non-alphanumeric keys.
<code>OK, Cancel</code>	Shows text that you will see displayed by FEMAP in a dialog box control, or in the menu.
<b><i>heading</i></b>	Used for headings or titles of sections of the manual. Larger characters of the same style (or italics) are also used depending upon the nature of the section being introduced.
<code>text</code>	Used for all other normal manual text.

Throughout this manual, you will see references to Windows. Windows refers to Microsoft® Windows NT, Windows 95 or Windows 98. You will need one of these operating environment to run FEMAP for the PC. This manual assumes that you are familiar with the general use of the operating environment. If you are not, you can refer to the Windows User's Guide for additional assistance.

Similarly, throughout the manual all references to FEMAP, refer to the latest version of our software.

# Table of Contents

## **1 Introduction**

## **2 Getting Started**

2.1 Hardware Requirements . . . . .	.2-1
2.2 Installation . . . . .	.2-1
2.2.1 Setup Program Execution . . . . .	.2-1
2.2.2 Security Device . . . . .	.2-3
2.3 Errors Starting FEMAP . . . . .	.2-3
2.3.1 Improving Performance (RAM Management) . . . . .	.2-4
2.4 Which examples should I do first? . . . . .	.2-6

## **3 Plate with Hole**

3.1 Problem Description/Objective . . . . .	.3-1
3.2 Creating the Geometry . . . . .	.3-2
3.3 Defining Materials and Properties . . . . .	.3-7
3.4 Generation of Nodes and Elements . . . . .	.3-9
3.5 Loading and Constraining the Model. . . . .	3-11
3.6 Review the Results . . . . .	3-14

## **4 Roof Truss**

4.1 Creating the Geometry . . . . .	.4-1
4.2 Materials, Properties and Meshing . . . . .	.4-4
4.3 Loads and Constraints . . . . .	4-17
4.4 Advanced Loading . . . . .	4-21

## **5 Pressure Vessel**

5.1 Import DXF Geometry . . . . .	.5-1
5.2 Material, Property and Meshing. . . . .	.5-3
5.3 Loads and Constraints . . . . .	.5-6

## **6 Handle**

6.1 Creating the Geometry . . . . .	.6-1
6.2 Materials, Properties and Meshing . . . . .	.6-4
6.3 Loads and Constraints . . . . .	.6-7

## **7 Groups, Layers, Viewing and PostProcessing**

7.1 Working with View Select and View Options . . . . .	.7-2
7.2 Groups and Layers Overview . . . . .	.7-8
7.3 Working with Groups. . . . .	7-10
7.4 Working with Layers . . . . .	7-19
7.5 Combining Grouping, Layers and View Options . . . . .	7-21
7.6 Printing . . . . .	7-21
7.7 Graphical Post-Processing . . . . .	7-23
7.8 XY Style . . . . .	7-33
7.9 Reporting Results . . . . .	7-38
7.10 Getting Your Results to Other Programs . . . . .	7-40

## **8 Simple Solid**

8.1 Create the Geometry . . . . .	8-1
8.2 Loads and Constraints . . . . .	8-5
8.3 Meshing the Solid . . . . .	8-7
8.4 PostProcessing . . . . .	8-8

## **9 Turbine Blade**

9.1 Creating the Geometry . . . . .	9-1
9.2 Loads and Constraints . . . . .	9-4
9.3 Meshing the Solid . . . . .	9-6
9.4 PostProcessing . . . . .	9-9

## **10 Cylindrical Support**

10.1 Creating the Geometry . . . . .	10-1
10.2 Materials, Properties and Elements . . . . .	10-6
10.3 Constraints . . . . .	10-13

## **11 Pipe Intersections**

11.1 Surface Intersection . . . . .	11-1
11.1.1 Geometry . . . . .	11-1
11.1.2 Materials, Properties and Elements . . . . .	11-6
11.2 Solid Intersection . . . . .	11-8
11.2.1 Geometry . . . . .	11-8
11.2.2 Meshing the Solid . . . . .	11-12

## **12 Slotted Guide**

12.1 Creating the Geometry . . . . .	12-1
12.2 Loads and Constraints . . . . .	12-9
12.3 Meshing the Solid . . . . .	12-10

## **13 Connecting Rod**

13.1 Creating the Geometry . . . . .	13-1
13.2 Loads and Constraints . . . . .	13-7
13.3 Meshing the Solid . . . . .	13-10

## **14 Midsurface**

14.1 Introduction . . . . .	14-1
14.2 Creating the Midsurface Model. . . . .	14-2
14.3 Meshing the Model. . . . .	14-6
14.4 Applying Loads and Constraints . . . . .	14-9
14.5 Post-Processing . . . . .	14-10

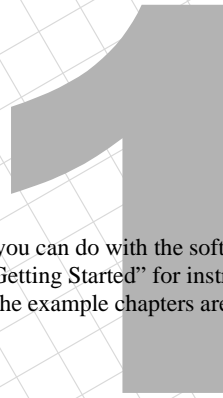
## **15 Hex Meshing Overview**

15.1 Introduction . . . . .	15-1
15.2 Importing the Geometry . . . . .	15-1
15.3 Subdividing the Solid . . . . .	15-2
15.4 Preparing for Meshing . . . . .	15-4
15.5 Meshing . . . . .	15-5

## **16 Hex Meshing**

16.1 Importing the Geometry . . . . .	16-1
16.2 Subdividing the Solid . . . . .	16-2
16.3 Meshing . . . . .	16-3
16.3.1 Free Meshing . . . . .	16-4
16.3.2 Mapped Meshing . . . . .	16-5

# Introduction



This section introduces FEMAP and explains what you can do with the software. When you are finished with this introduction, see Chapter 2, “Getting Started” for instructions on installing FEMAP and for recommendations on which of the example chapters are most appropriate for your interests.

## **About FEMAP**

FEMAP is a finite-element modeling and postprocessing system that allows you to perform engineering analyses both quickly and confidently. FEMAP provides the capability to develop sophisticated analyses of stress, temperature, and dynamic performance directly on the desktop. With easy access to CAD and office automation tools, productivity is dramatically improved compared to traditional approaches.

FEMAP automatically provides the integration that is necessary to link all aspects of your analysis. FEMAP can be used to create geometry, or you can import CAD geometry. FEMAP then provides powerful tools for meshing geometry, as well as applying loads and boundary conditions. You may then use FEMAP to export an input file to over 20 finite element codes. FEMAP can then read the results from the solver program. Once results are obtained in FEMAP, a wide variety of tools are available for visualizing and reporting your results.

## **Geometry**

FEMAP can directly import geometry from your CAD or design system. In fact, FEMAP can directly import a solid model from any ACIS-based or Parasolids-based modeling package. If your modeling package does not use either of these packages, simply use the FEMAP IGES or STEP readers. IGES files can be read and then stitched together to form a solid. This typically requires using just one simple command. STEP AP203 solids can be read directly and automatically converted to Parasolid format.

If you do not have CAD geometry, you can create geometry directly in FEMAP using powerful wire-frame and solid modeling tools. FEMAP Professional contains solid modeling directly in FEMAP with not one but two popular geometry engines (Parasolids and ACIS). You can build solid models in either engine, and then export a model. This is very convenient if you need to export geometry to CAD packages that are either ACIS or Parasolids based.

## **Finite Element Modeling**

Regardless of the origin of your geometry, you can use FEMAP to create a complete finite-element model. Meshes can be created by many methods ranging from manual creation, to mapped meshing between keypoints, to fully-automatic meshing of curves, surfaces and solids. FEMAP can even work with your existing analysis models. You can import and manipulate these models using the interfaces to any of the supported analysis programs.

Appropriate materials and section properties can be created or assigned from FEMAP libraries. Many types of constraint and loading conditions can be applied to represent the design environment. You can apply loads/constraints directly on finite element entities (Nodes and Elements), or you can apply them to geometry. FEMAP will automatically convert geometric conditions to Nodal/Elemental values upon translation to your solver program. You may even convert these loads before translation to convince yourself that the loading conditions are appropriate for your model.

## ***Checking Your Model***

At every step of the modeling process, you receive graphical verification of your progress. You need not worry about making a mistake because FEMAP contains a multi-level undo and redo capability. An On-Line Help system also provides the capability to review information on FEMAP commands without ever opening a manual.

FEMAP also provides extensive tools for checking your model before you analyze it, to give you the confidence that you have properly modeled your part. It constantly examines input to prevent errors in the model, and provides immediate visual feedback. FEMAP also provides a comprehensive set of tools to evaluate your finite element model and identify errors that are often not obvious. For example, FEMAP can check for coincident geometry, find improper connections, estimate mass and inertia, evaluate your constraint conditions, and sum your loading conditions. Each of these methods can be used to identify and eliminate potential errors, saving you considerable time and money.

## ***Analyzing Your Model***

When your model is complete, FEMAP provides interface to over 20 popular programs to perform finite element analysis. You can even import a model from one analysis program and automatically convert it to the format for a different analysis program.

## ***Postprocessing***

After your analysis, FEMAP provides both powerful visualization Tools that enable you to quickly interpret results, and numerical tools to search, report, and perform further calculations using these results. Defamation Plots, Contour Plots, Animations, and X-Y Plots are just some of the postprocessing tools available to the FEMAP user. FEMAP v5.0 now supports OpenGL, which provides even more capability for postprocessing, including dynamic visualization of contours through solid parts. Section Cuts and IsoSurfaces can now be viewed dynamically by simply moving your cursor.

## ***Documenting Results***

Documentation is also a very important factor with any analysis. FEMAP obviously provides direct, high quality printing and plotting of both graphics and text. Frequently, however, graphics or text must be incorporated into a larger report or presentation. FEMAP can export both graphics and text to non-engineering programs with a simply Windows Cut command. You can easily export pictures to such popular programs as MS Word, MS PowerPoint and Adobe Framemaker. You can export to spreadsheets, databases, word processors, desktop publishing software, and presentation graphics, paint and illustration programs. These links enable you to create and publish a complete report or presentation, all electronically, right on your desktop.

With support for AVI files you can even include an animation directly in your PowerPoint Presentation or Word Document. Creating illustrations for reports and presentations has never been this easy.

FEMAP can also write out analysis models as VRML files, deformed and contoured just as you see on your screen. This enables you to share analysis results across networks or the web with any standard VRML viewer.

## ***FEMAP Documentation***

In addition to the On-Line Help in FEMAP, FEMAP also comes with a complete set of documentation. Four manuals are provided with FEMAP: (1) Examples (2) Release Notes, (3) Users' Guide, and (4) Command Reference.



**Examples...**

... provides an introduction to FEMAP, basics on installing the software, and a number of detailed examples of building a finite element models from start to finish using FEMAP. This manual provides the new user a quick tour of just some of the capabilities of FEMAP, while familiarizing them with the program.

**Release Notes...**

... contains a brief overview of new features in the latest release of FEMAP. This document is especially useful to users who are upgrading from a previous version of the software.

**Users Guide...**

... includes general information about FEMAP. Included in this manual are descriptions of FEMAP's Graphical User Interface, including the command toolbars, common dialog boxes in FEMAP, and a general overview of the finite element process. This manual also contains information on shortcut keys, as well as the use of function keys in FEMAP.

This manual provides an excellent overview if you are not certain how to do something in FEMAP as well as information on getting started with FEMAP.

**Command Reference...**

... contains explanations of every menu command in FEMAP. If you are unclear about a certain command in FEMAP, you can use this manual to find its description. In general, the commands are separated into chapters which correspond to the Main FEMAP menu. The few exceptions to this are the List, Modify, and Delete Menu commands, which are explained under the Chapter which contains the type of entities (i.e. if you are Deleting Geometry, look under the Geometry chapter for Delete). In addition, the Tools and List menus are combined under the Checking Your Model chapter, and the View and Group menus are combined under the Viewing Your Model chapter.

There is also a special Postprocessing chapter describing commands specifically used for post-processing, and there is a brief description of using some of the View commands to check your Model under the Checking Your Model chapter.

**Overview of this Manual**

This manual is designed to get you using FEMAP quickly and with proficiency. It contains a number of examples that take you step by step through the main processes for building and using an FEA model.

As there are many different types of real analysis problems, there are different types of example problems shown here. There are examples using just beams, plates, or solids, as well as combinations. They are based on real world problems but are not intended to represent any actual part or structure. Again they are meant to be used only as an aid in learning to use FEMAP. You may find that some of the examples do not resemble anything you will be doing and may want to skip them, however, it is recommended that you work through all the problems because they may contain some commands you will find useful.

As mentioned before, the example problems are spelled out step by step. All necessary dialog boxes are shown and buttons/icons pressed are indicated. Simple commands that are repeated within the same problem may not be shown completely. If you have trouble, refer to the previous instance of that command for complete instruction. As you complete more and more examples you should become familiar with many of the FEMAP commands, particularly the view orientation commands, and be able to execute them on your own.

The general purpose of this manual is to familiarize you with using FEMAP to complete the following FEA Modeling tasks.

Creating 2-D and 3-D wire-frame and solid geometry.

Importing geometry from popular CAD packages.

Midsurfacing techniques

Creating basic materials

Creating Line, Plane and Solid Element Property types

Using various types of meshing in your models

Applying loads and boundary conditions.

Manipulating views using the view commands and groups and layers

Viewing, postprocessing and reporting analysis results for your models

The examples in this manual should help you learn the basic FEA modeling process and some general FEMAP commands and the FEMAP command structure. For a more complete description of the FEMAP interface and modeling procedures refer to the FEMAP User Manual. For an in depth description of all the commands in FEMAP refer to the FEMAP Command Reference

# Getting Started

Welcome to FEMAP! This section will help you install and start using the software.

To help you quickly become familiar with FEMAP there are recommendations at the end of this chapter on which examples to do first, depending on your interests.

This section contains information specific to getting started on a PC, which includes Windows NT (INTEL and DEC Alpha), Windows 95, and Windows 98. Information related to FEMAP installation and operation on UNIX platforms can be found in the FEMAP Unix Guide. Supported UNIX platforms include HP/ux, IBM RS6000/aix, SUN/Solaris, and SGI/IRIX.

## 2.1 Hardware Requirements

There are no special hardware requirements for FEMAP beyond those imposed by Windows operating systems. There are many types of hardware that will allow you to use FEMAP. Proper choice of hardware however, can often make the difference between frustration and productivity. Here are two suggestions concerning hardware:

### **Memory, RAM:**

You will need at least 16 Mbytes of RAM to run FEMAP. Furthermore, if you are going to use the solid modeling engines, 32 Mbytes of memory is required. Obviously, the more amount of RAM the better. Adding RAM can be one of the most cost effective means of increasing performance.

### **Graphics Boards:**

Standard graphics adapters work very well with FEMAP. Specialized boards which contain support for OpenGL will provide increased graphical performance when dynamically rotating large, complex models. They also usually provide higher resolution and more colors which makes graphics easier to see and more realistic.

## 2.2 Installation

### 2.2.1 Setup Program Execution

#### **Windows NT**

1. Log into your computer as **Administrator**.
2. Insert the FEMAP CD. Setup should automatically start within a few seconds.

#### **Windows 95/98**

1. Log on to your computer as any user.
2. Insert the FEMAP CD. Setup should automatically start within a few seconds.

If the setup program does not start automatically use the Windows Explorer to view the CD-ROM's contents. You can manually start the setup by double clicking on the file **Setup.bat**

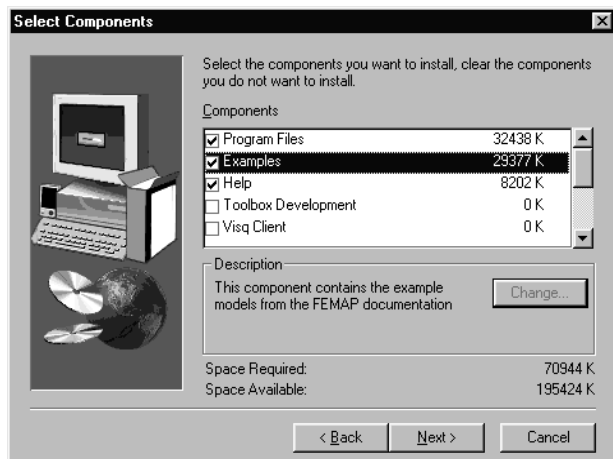
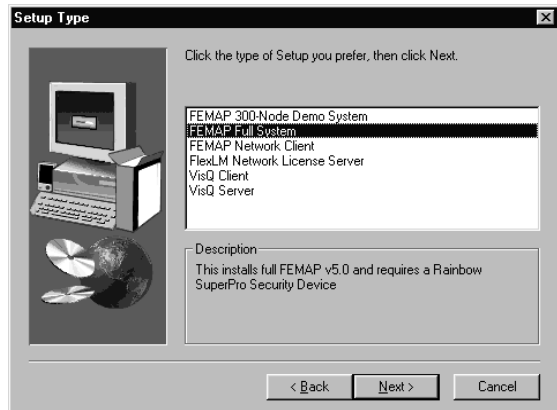
Once setup is running you will see a license Agreement. Assuming that you agree with the license agreement, press Yes to continue and select the directory where you would like to have the FEMAP program files installed.

You will now be asked which type of installation to perform. There are four options related to FEMAP, Demo System, Full System, Network Client, or FLEXlm License Manager. The Network Client and FLEXLM License Manager are required only for network installations, and will not be discussed here. Please see Chapter 2 of the FEMAP Users Guide for information on network installations.

1. If you have purchased a stand alone license of FEMAP or have a 30-day evaluation system, choose **FEMAP Full System..**
2. If you have requested a Free Demonstration License of FEMAP install **FEMAP 300-Node Demo System.**

Stand alone and evaluation licenses come with a security device that attaches to your parallel port. If you have not received this device, you will need to install the demo version.

You will now be prompted for which components of FEMAP to install. Choose *Program Files*, *Examples*, *Help* for the typical installation.



## 2.2.2 Security Device

### Note:

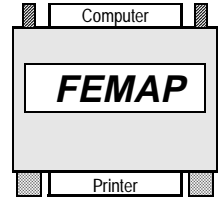
If you have the 300-Node Demo version or network version, this section should be skipped.

First locate the security device. It should look like the following.

Follow these steps to install the device:

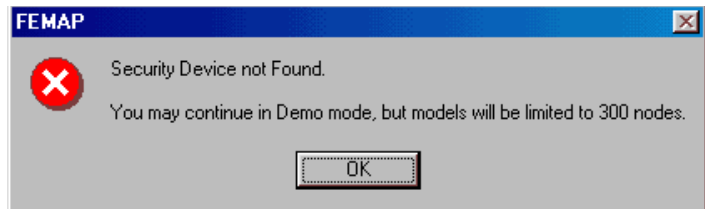
1. Turn off your computer. You should always turn off your computer when installing any peripheral device or cable.
2. Attach the security device to any standard, IBM-compatible (Centronics) parallel printer port (or into another security device, if present). Make certain you tighten the screws. This ensures proper electrical connection, and allows proper installation of the printer cable in the next step.
3. Attach your printer cable, if any, into the other end of the security device.
4. Turn on your computer and proceed with the installation of the FEMAP software.

The security device (hardware key) may be destroyed if you connect any devices other than a printer in-line with the security device. If you need to use the parallel port to connect an external device such as an external hard drive, you must remove the hardware key from the series during the operation.



## 2.3 Errors Starting FEMAP

If you receive an error indicating that the security device cannot be found, there are two possible problems:



1. You have installed the Full Version without a security device.
2. The security device is not properly installed.

If you have the 300-Node Demonstration version, you will need to re-install FEMAP as the 300-Node Demo version. If you have a security device attached to the parallel port, turn the power off, check the connections, and then re-boot. If you still have problems, contact ESP.

If you receive an *Unable to allocate Scroll Buffer File* error or have any other difficulty starting FEMAP where abnormal termination occurs, you either do not have enough disk space, or your Windows TEMP is not set to a valid, accessible directory. You may either change your Windows TEMP directory environment variable, or specify a path for the FEMAP scratch files (which default to the Windows TEMP directory set by the environment variable) to a valid directory.

This and all other FEMAP Preferences are stored in a file called femap.ini typically located in the FEMAP executable directory. You will have to create this file or modify it to include the appropriate lines as shown below:

```
DiskModelScr=c:\femap700
```

```
DiskUndo=c:\femap700
```

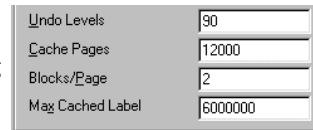
```
DiskListBuff=c:\femap700
```

where c:\femap700 can be any valid path. The DiskModelScr, DiskUndo, and DiskListBuff parameters are case sensitive, and must be defined exactly as above. Once you make these changes and FEMAP starts, you can modify this path under File Preferences Database.

### 2.3.1 Improving Performance (RAM Management)

To improve FEMAP performance on Windows 95 and Windows NT personal workstations, you should modify the default settings that FEMAP uses to manage RAM. This procedure is explained in more detail in the Command Reference Manual under File Preferences, Startup Preferences (File Manipulation Chapter). To access the internal FEMAP memory management system, follow the procedure below:

1. Choose the File Preferences command and hit the Database button.
2. Change Cache Pages and Blocks/Page in the dialog box.
3. Max Cached Label should be set to a number that is higher than any entity you will create in your model file. This sets aside a small portion of memory that stores all of the ID's in FEMAP.
4. Select OK and say Yes to the Permanent question. (Remember to say Yes or your selection will not be saved).



Undo Levels	90
Cache Pages	12000
Blocks/Page	2
Max Cached Label	6000000

#### Note:

You should never allow FEMAP to allocate more than the physical memory of the machine. The internal memory management (swapping) in FEMAP will allow the program to run much faster than Windows memory swapping. Therefore, you should set the Cache Pages and Blocks/Page at a level which is comfortably below the physical memory of the machine. Also, to optimize performance, you should always increase Cache Pages (max 15000) to its limit before increasing Blocks/Page.

### 2.3.1.1 Setting Guidelines

Starting with FEMAP version 7.0 FEMAP handles memory differently and requires different settings from earlier versions.

#### **.FEMAP Version 7.0**

The following figures are provided as a starting point to improve performance.

Operating System	Installed RAM (Mb)	Cache Pages	Blocks/Page
Windows 95/98, NT	64	10500	1
	128	12000	2
	256	14000	4
	512	15000	8
	1000	15000	15

Actual performance will vary depending upon other concurrent applications and model specifics. Once again, it is best to increase Cache pages to 15000 before increasing Blocks/Page

#### **Note:**

For best performance you should have enough physical Ram to load the entire model file into memory. i.e. if you expect your model files to be a maximum of 100 Mb then you would want FEMAP to allocate at least 100 Mb of memory. If you had 128 Mb of physical RAM this would leave 28 Mb for windows and other programs that may be running at the same time as FEMAP.

#### **FEMAP Version 5 & 6**

Memory is handled differently starting with FEMAP version 7.0. For earlier versions of FEMAP use the table below.

Operating System	Installed RAM (Mb)	Cache Pages - Max	Blocks/Page
Windows 95/98	32	480	20
	64	1024	24
	96	1024	38
	128	1024	52
Windows NT	32	390	20
	64	1024	22
	96	1024	36
	128	1024	50

## ***2.4 Which examples should I do first?***

**All new users** should start with Chapter 3, a basic “plate with a hole” example.

Users that will be **creating FE models from CAD geometry** will want to do Chapters 8, 9, 14, 15 and 16. These illustrate geometry based loads and ways to idealize geometry.

Users interested in **beam and shell modeling** will want to review Chapters 4 and 10.

**Building geometry directly in FEMAP** is covered in Chapters 6, 8, 11, 12 and 13.

Once you start to feel comfortable with FEMAP, **all new users** will definitely want to spend some time with Chapter 7 on Groups, Layers and Postprocessing. **This is a very important chapter** and will help unlock the power of FEMAP for many users.

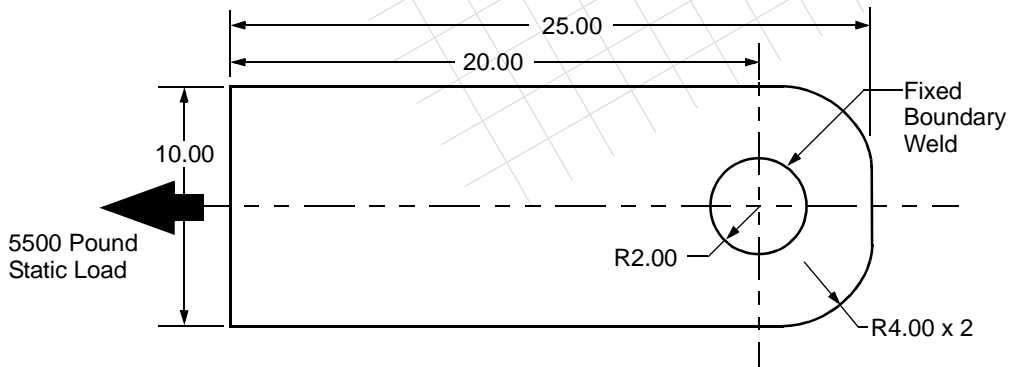
**Current FEMAP users** will find features of Version 6 & 7 in Chapters 4, 10, 14, 15 and 16.



# Plate with Hole

## 3.1 Problem Description/Objective

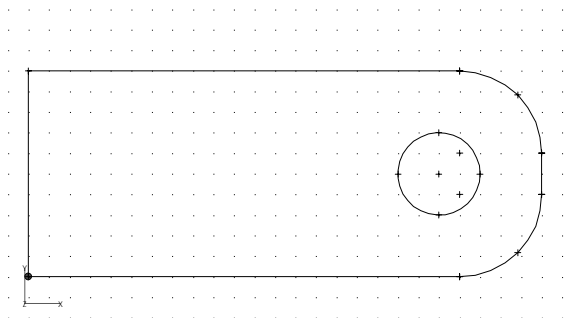
In this example we need to review the physical performance of a proposed mounting bracket configuration. We will define the geometry and material properties, prepare a finite element mesh and define boundary conditions and loads. The bracket will be constructed of AISI 4130 Steel. We will completely fix the plate as if the hole were welded to a very rigid support, and then apply a static tensile load of 5500 pounds. Finally, we will review how to prepare the model for solution and then look at the results of an analysis.



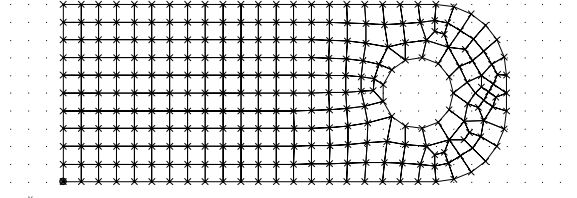
### Process Overview

Before we begin the detailed step-by-step instructions for completing the sample model, an overview of the process is in order. There are three basic steps to be taken to turn this sample model into a model suitable for finite element analysis.

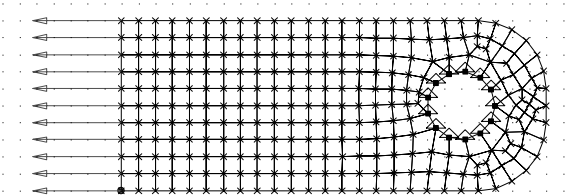
- 1 Create or Import the underlying geometry.



- 2 Mesh the Part using the FEMAP Surface Mesh Command. Material properties and any element physical properties (such as thickness) are also defined in this step.



- 3 Apply various sets of Loads and Boundary Conditions

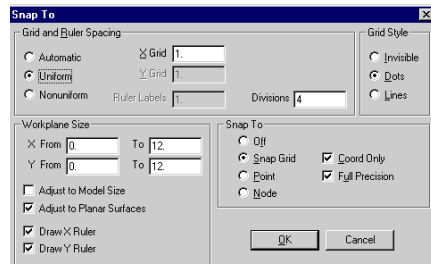
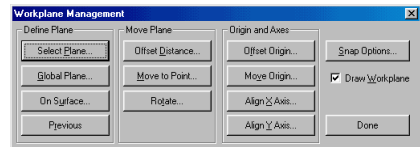


## 3.2 Creating the Geometry

If you have not already done so, start FEMAP. Select New Model to begin a new modeling session. Throughout this example, all commands that you will select from the FEMAP menu will be shown in the following style - **FILE - NEW**. Which means, first select File from the menu, and then move to the New command.

### Set Snap Mode

- 1 Choose **TOOLS - WORKPLANE**. Press the Snap Options Button.
- 2 Set the Grid and Ruler Spacing to Uniform with an X grid of 1.
- 3 Set the Workplane Size in both directions at 0 to 25 and turn off Adjust to Model Size.
- 4 Set the Snap To to Snap Grid. This will snap the coordinates that you select from the screen to predefined grid locations.
- 5 Set the Grid Style to Dots. This displays the snap grid locations as dots.
- 6 Press OK to accept.

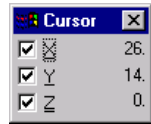


- 7 The Grid may be displayed too coarsely. Use the Zoom-Out Icon to adjust the view of the default grid.



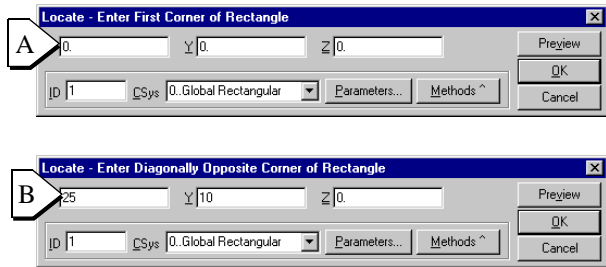
## Cursor Position

- 1 Turn on the Cursor Position Dialog Box by choosing **TOOLS - CURSOR POSITION**. The Cursor Position Dialog Box appears in the top right corner of the screen. This guide is very useful when creating your own geometry.
- 2 Use the A.) Zoom Out icon and the B.) Zoom Window icons on the toolbar to view the general area between two X-Y Positions, such as 0,0 and 30,20. Moving the cursor around in the graphics windows and watch the values change in the Cursor Position Dialog Box.



There are many ways to create the required geometry. Here we will demonstrate one

- 1 Choose **GEOMETRY - CURVE LINE - RECT-ANGLE** and enter two corner points of the base rectangle that forms this part, A=0,0,0 and B=25,10,0. Press OK after each Corner.



The direct entry technique was simplest for this example. As an alternate, you may graphically pick the locations of corner points on screen.

### Graphically Select the Two Corners:

Since the Snap Mode is set to Grid, and the Grid is spaced at every 1.0 unit, it is very easy to use the Cursor Position Dialog Box, and move the cursor in the graphics window to the precise location required for each point, press the mouse, and have the dialog box filled in for you.

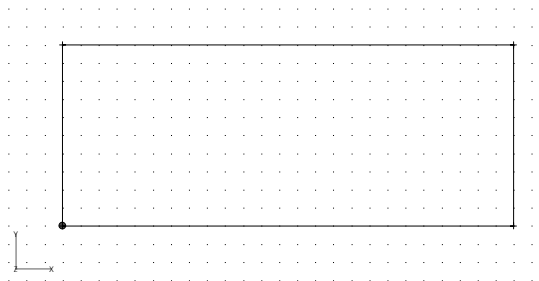
### Hint:

When selecting from the graphics window, instead of moving the mouse all the way down the OK button and pressing it when you are done, you can double click the mouse on the last selection and FEMAP will know that you are done and effectively press OK for you.

### Program Short-Cut Keys

FEMAP has a number of pre-defined short-cut keys to speed up the work you do. Some use the function keys, e.g. F3 (Print), and some use the control key (Ctrl) in combination with another letter key.

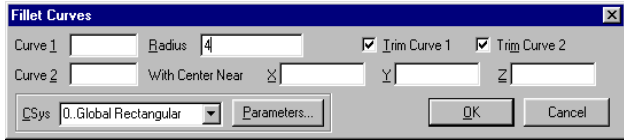
These are denoted in the menu structure and detailed in the full FEMAP manual and in the on-line



help (search for Command Keys). For instance, Press Ctrl-A (**VIEW - AUTOSCALE** command), and FEMAP will autoscale the view. Your model should look similar to the one above.

We will now fillet the right side of the part.

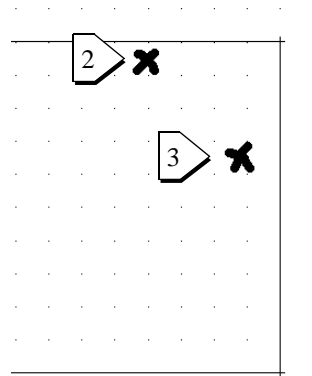
- 1 Select **MODIFY - FILLET** from the FEMAP Menu.



- 2 Set the select snap to screen by using the mouse to select the Snap Off Tool Bar Icon at A.)

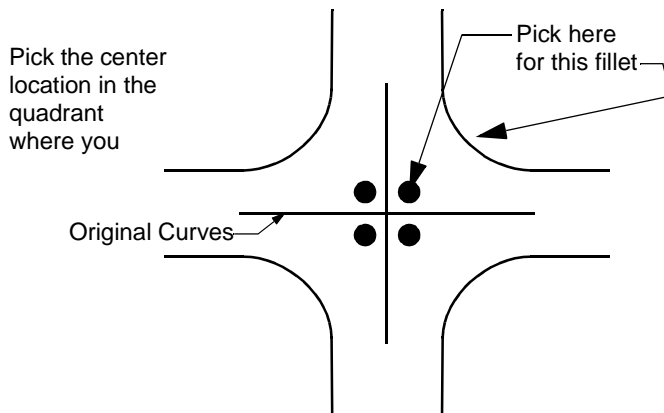


- 3 The Fillet Curve Dialog Box starts off waiting for you to enter the first curve of the fillet. FEMAP also uses the location on screen that you select the curve at to determine which of the four possible fillets between two curves should be used. When picking the top curve, move the mouse to a location slightly inside the rectangle, towards the right side of the line. You will notice the line highlighting, giving you a preview of exactly which curve will be picked. When you have the mouse in the position indicated, press the left mouse button to pick the curve.

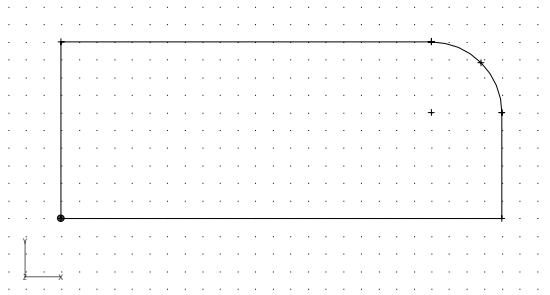
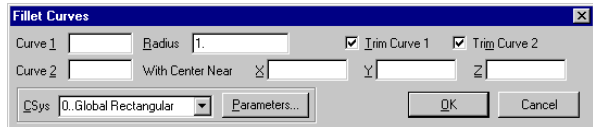


### Note:

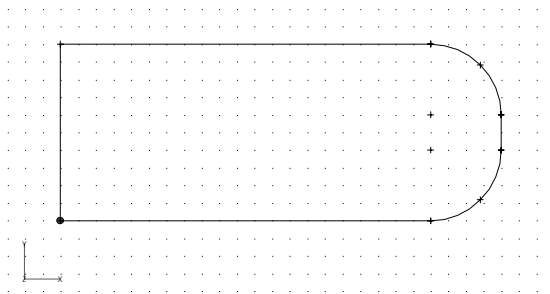
By picking inside the two lines you are specifying a fillet radius whose center will be toward the sides of the picks. The effect of picking on the other sides is illustrated below:



- 3 Pick the right side in a similar manner with mouse positioned slightly to the inside, and towards the top of the line.
  - 4 Now that the curves have been selected, FEMAP highlights the fillet radius field in the dialog box. Simply type a radius of 4.0. Press OK or Enter to create the fillet.
- 1 Select **MODIFY - FILLET** from the FEMAP Menu.

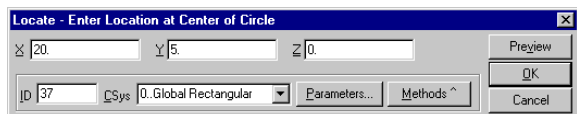


Fillet the bottom right corner following the same procedure. When you are done, Press the ESC key, or Press the Cancel button to dismiss the Fillet Curves dialog box. The model should now look like this:

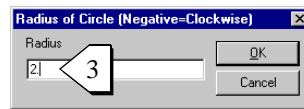
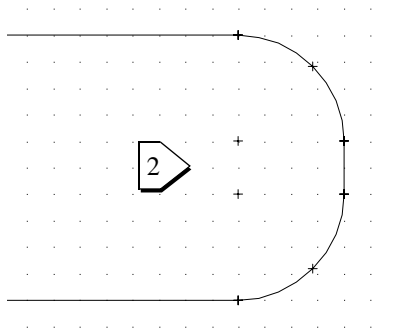


We will now create the circular hole at the right side of this part.

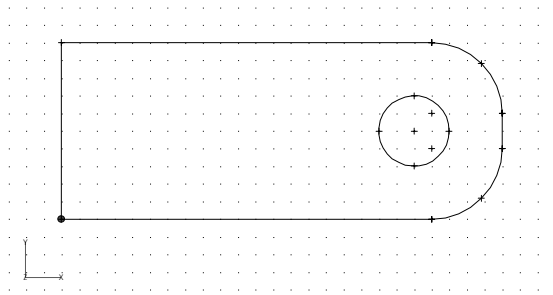
- 1 Select **GEOMETRY - CURVE CIRCLE - CENTER** from the FEMAP Menu.



- 2 You can enter the center point manually, at 20,5,0 or pick it from the screen. To pick from the screen, move the mouse directly on top of the screen grid that is between the two center points of the fillet arcs and one to the left, and press the left mouse button. You can watch the location numbers in the Cursor Position Dialog Box to ensure that you are at 20,5,0. Press OK when finished.

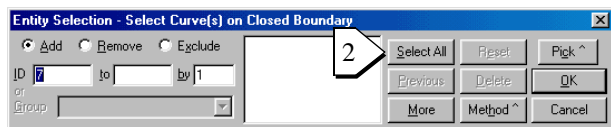


- 3 You are now prompted for the radius of the hole, type in 2.0. Press OK to create the circle. Press ESC or Cancel to end Circle Creation.



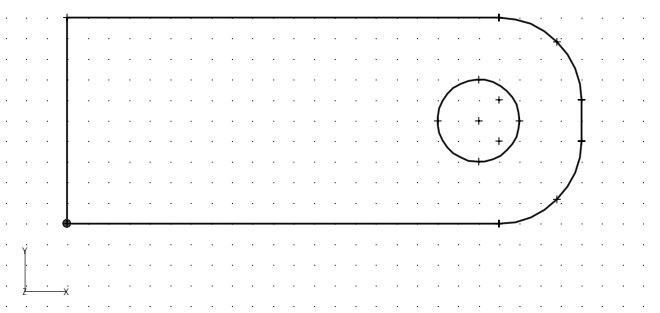
This part will be meshed by using the FEMAP Surface Mesher. The Surface Mesher is designed to take any enclosed boundary, including internal voids, and fill that boundary with planar finite elements.

- 1 Select **GEOMETRY - BOUNDARY SURFACE** from the FEMAP Menu. FEMAP displays the Standard Entity Selection Dialog Box.



- 2 Move the cursor around in the graphics window. You will notice the individual curves highlight as you pass over them. You can pick the curves individually, or, since these are the only curves in the model, press Select All to select them. Press OK to move on.
- 3 FEMAP automatically detects the hole/holes in the boundary and includes them

The boundary is displayed as a highlighted entity on top of its underlying geometry.

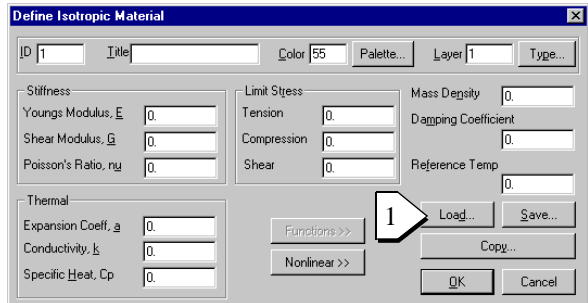


The geometry creation phase of this example is now complete. From now on we will be creating entities directly related to Finite Element Analysis. First Material and Element Properties will be defined. Then Nodes and Elements will be generated automatically using the underlying geometry. Finally Loads and Boundary Conditions will be defined on the finite element mesh.

### 3.3 Defining Materials and Properties

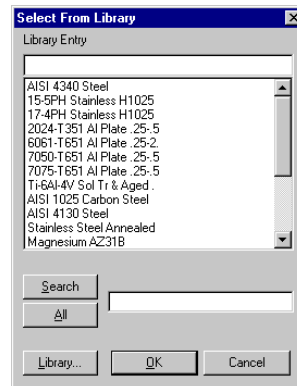
Before any finite elements can be created, we must first define a material.

- 1 Select **MODEL - MATERIAL** from the FEMAP Menu. FEMAP displays the Define Isotropic Material Dialog Box. You could enter all the physical properties of a material in this box, or to use the library of materials included with FEMAP, Press the Load Button.

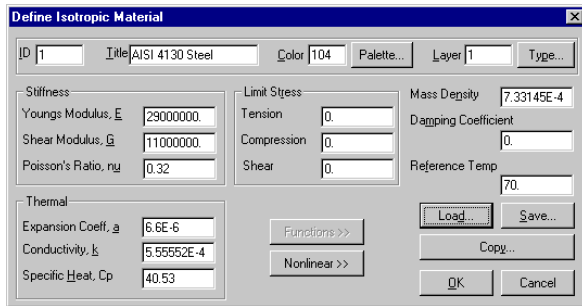


- 2 FEMAP Material Libraries are easy to extend with your own materials. The library that ships with FEMAP contains several common material in U.S. (in. lb. sec.) units. In this example select AISI 4130 Steel using the arrow keys or the mouse, and Press OK when finished.

2



- 3 The physical properties corresponding with AISI 4130 Steel in traditional U.S. units from the library will be placed in the Define Isotropic Material Dialog Box. Press OK to Create the Material, and then Cancel to end creation of materials.



The 'Define Isotropic Material' dialog box is shown. It contains the following fields and values:

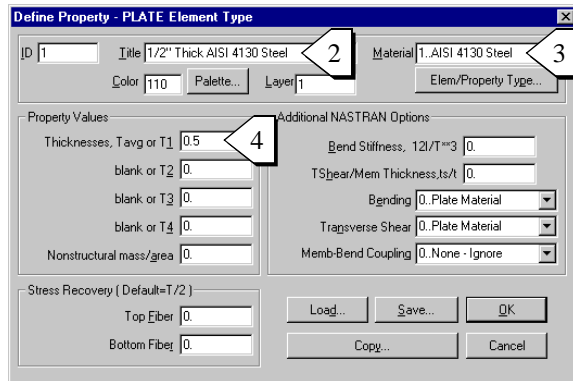
- ID: 1
- Title: AISI 4130 Steel
- Color: T104
- Palette: ...
- Layer: 1
- Type: ...
- Stiffness:
  - Young's Modulus, E: 29000000
  - Shear Modulus, G: 11000000
  - Poisson's Ratio, nu: 0.32
- Limit Stress:
  - Tension: 0
  - Compression: 0
  - Shear: 0
- Mass Density: 7.33145E-4
- Damping Coefficient: 0
- Reference Temp: 70
- Thermal:
  - Expansion Coeff, alpha: 6.6E-6
  - Conductivity, k: 5.55552E-4
  - Specific Heat, Cp: 40.53
- Buttons: Functions >>, Nonlinear >>, Load..., Save..., Copy..., OK, Cancel.

### Note:

FEMAP automatically repeats any creation commands assuming that during the creation of most finite element models that each command will be required more than once. This behavior is designed to minimize the number of times you will need to go into the menus or use the toolbox to access commands. This feature can be turned off in the File - Preferences command.

Element properties define the physical characteristics of the actual finite elements in your model. For example, plate thickness, or area and moments of inertia for beams, etc. Element properties also reference a material to define their stiffness properties. We will now create a property for the plate elements that will be used to build this model.

- 1 Choose **MODEL - PROPERTY** from the FEMAP menu.
- 2 Add a title by using the TAB key to move to the title field, or by clicking the mouse in the title field.
- 3 Identify the Material that Property is associated with by pressing the down arrow on the Material, and select AISI 4130 Steel we just created.



The 'Define Property - PLATE Element Type' dialog box is shown. It contains the following fields and values:

- ID: 1
- Title: 1/2" Thick AISI 4130 Steel
- Color: T10
- Palette: ...
- Layer: 1
- Material: 1\_AISI 4130 Steel
- Elem/Property Type: ...
- Property Values:
  - Thicknesses, Tavg or T1: 0.5
  - blank or T2: 0
  - blank or T3: 0
  - blank or T4: 0
  - Nonstructural mass/area: 0
- Additional NASTRAN Options:
  - Bend Stiffness, 12I/T\*\*3: 0
  - TShear/Mem Thickness, ts/t: 0
  - Bending: 0. Plate Material
  - Transverse Shear: 0. Plate Material
  - Mem-Bend Coupling: 0. None - Ignore
- Stress Recovery (Default=T/2):
  - Top Fiber: 0
  - Bottom Fiber: 0
- Buttons: Load..., Save..., OK, Copy..., Cancel.

- 4 A constant thickness for the plate is specified in the Thickness, Tavg, T1 field. Use the TAB key or mouse to move to this field and enter 0.5.

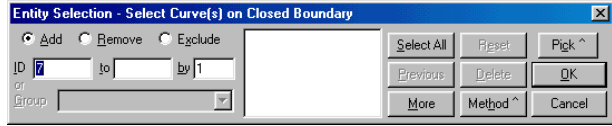
The remainder of the fields can be used for more specialized capabilities of FEMAP and some of the FEA solvers. For this example, we have entered all of the required information. Press OK to create the Material, and then ESC or Cancel to end creation of materials.



### 3.4 Generation of Nodes and Elements

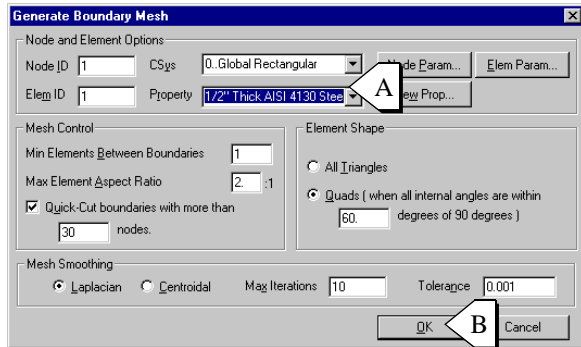
We will now create the actual finite elements by using the FEMAP Surface Mesher.

- 1 Select **MESH - GEOMETRY-SURFACE** from the FEMAP Menu. FEMAP will display the Standard Entity Selection Dialog Box, and prompt you for the Boundary to mesh.



Select the Boundary from the screen, or use Select All. Press OK to continue.

- 2 FEMAP will now display the Generate Boundary Mesh Dialog Box.



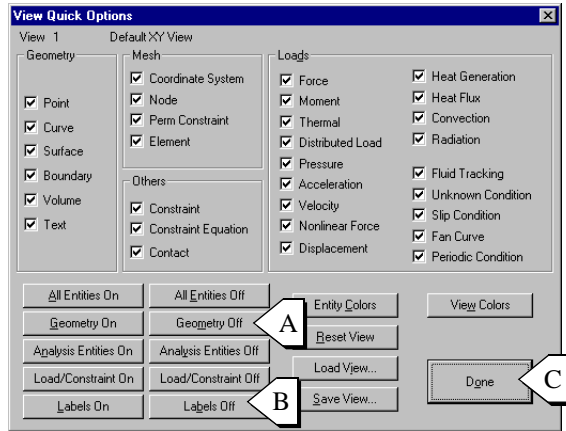
A.) Select the Property to be used for the elements generated, and B.) Press OK to generate them. There are many options in the Boundary Mesh Dialog Box that provide extensive control over the automatic mesh.

For this example we will use the default settings. Defaults

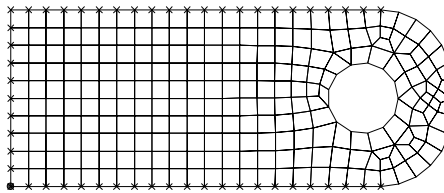
have been carefully designed to generate a quality mesh for a wide variety of geometries.

The part will be meshed totally automatically. The size of the individual elements was determined by the default global mesh size, that you can control. In addition, the number of element generated along each geometric entity can be specified with the Mesh - Mesh Control - Size Along Curve command overriding the global mesh size. In this example, the standard default mesh size of 1.0 was sufficient and did not need to be modified. At this point the display is fairly cluttered with the node and element numbers. We will now use some of the View Options in FEMAP to modify the display and remove unnecessary information.

- 1 Press Ctrl-Q on the keyboard. This will bring up the FEMAP Quick View Options Dialog Box. Since we are finished with the geometry side of FEA Pre- and Post-Processing, A.) Press the Geometry Off Button. To clear the entity labeling, and clean up the display, B.) Press the Labels Off Button. C.) Press Done.

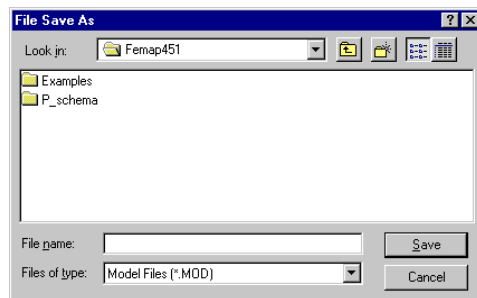


Your model should be displayed as follows:



Now would be a good time to save your work up to this point. To save a model to a new name.  
:

Select **FILE - SAVE AS** from the FEMAP Menu. The Windows Common File Save Dialog box will be displayed. The dialog box shown here is from Windows 95. The dialog boxes from Windows 3.1x and Windows NT are similar. Using this box, you can navigate around your local computer, and across a network. Move to or create a directory where you wish to store your sample FEMAP models, type in the name CH2MESH, and press Save.

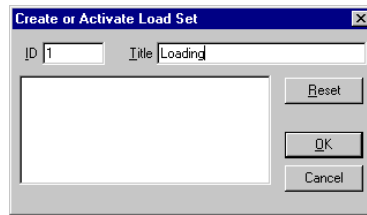


### 3.5 Loading and Constraining the Model

Recall that we want to weld and fix the inside of the hole, then pull on the left side of the bracket with a total force of 5,500 pounds.

We will now add the loads and constraints required to perform a static finite element analysis, first we must first create an empty Load Set. FEMAP supports multiple load and constraint sets to allow you to define more than one loading condition or constraint condition for the same finite element model.

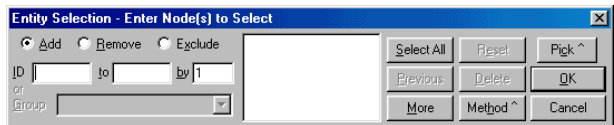
To create the empty load set, choose **MODEL - LOAD - SET** from the menu and add a descriptive name in the Title box. Press OK to create the Set.



Similar to Load Sets, there are also Constraint Sets for storing different boundary conditions for your model. To create the first empty Constraint Set, select **MODEL - CONSTRAINT - SET** from the menu, type a descriptive name, such as "Welded Hole" and create the empty set.

Let's first constrain the part. Constraints define how a part is held in reaction to the applied loads. In this case we will completely fix all the nodes around the hole as if the plate were welded to a significantly stiff underlying structure.

- 1 Choose **MODEL - CONSTRAINT - NODAL** from the menu. Again FEMAP will display the Standard Entity Selection Dialog Box.



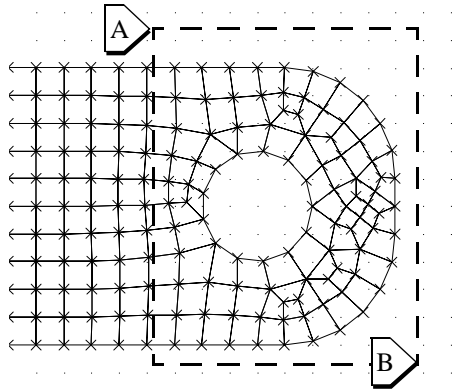
Instead of box picking this time, we will circle pick the nodes around the hole.

- 2 Zoom in on the area around the circle. To do so, press the Zoom Icon on the toolbar.

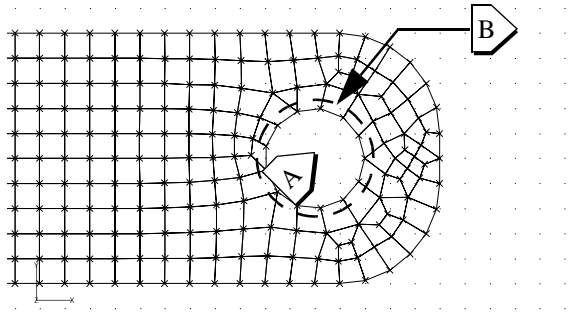


## 3-12 Plate with Hole

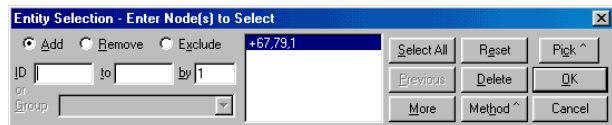
- 3 Now pick a box around the hole with picks at A and B.



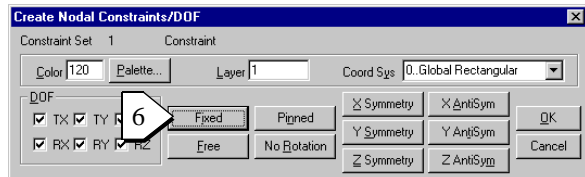
- 4 Circle picking is similar to box picking, but instead of pressing and holding the shift key, you press and hold the control key. The trick in circle picking is to make the first pick A.) at the center of the circle and then move out to the outside radius B.)



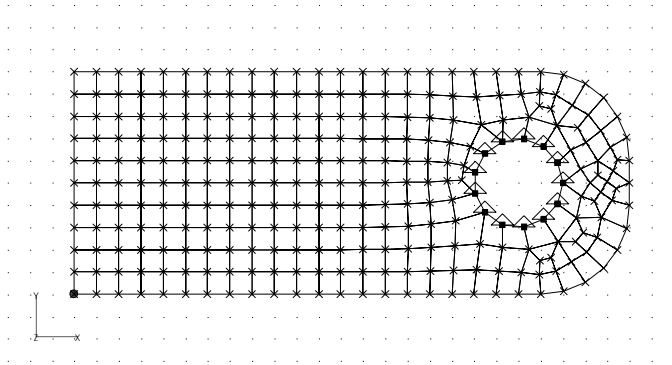
- 5 FEMAP will now fill in the nodes selected within the circle. Press OK to select these nodes.



- 6 You will now see the Create Nodal Constraints/DOF Dialog Box. Press the Fixed Button to constrain all six degrees of freedom at this location. Press OK to create the constraints. Press the ESC Key or Press Cancel to end creation of constraints.

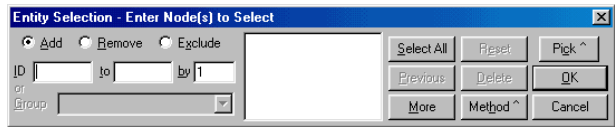


The model should look like the one below. If you do not see all of your model, press Ctrl-A to Autoscale the view.

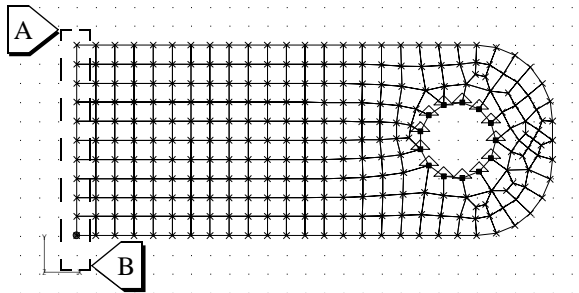


We will now apply a load to the model along the left hand side, in the negative x direction totaling 5,500 pounds.

- 1 Select **MODEL - LOAD - NODAL** from the FEMAP menu. FEMAP will now display the Standard Entity Selection Box requesting nodes.



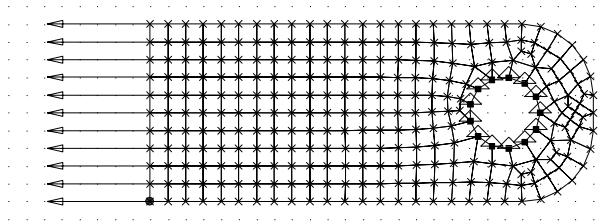
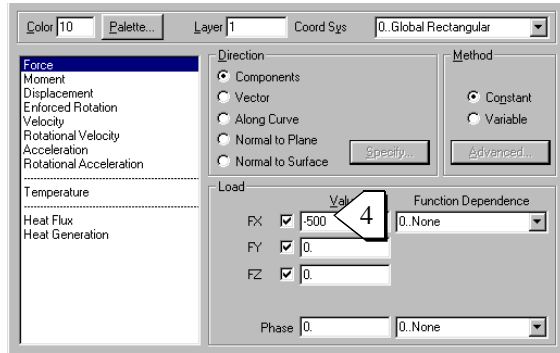
- 2 You could pick the 11 nodes at the end of this part one by one, or instead, use the box-picking capability of FEMAP. To box pick, press and hold down the shift key, then, A.) move the mouse to one corner of the rectangle you wish to select inside of, and press the left mouse button and drag the mouse to B.) the other corner of the region. You will see a dashed rectangle on screen indicating the region that will be selected.



- 3 FEMAP will fill in the Entity Selection Dialog box with the nodes inside the rectangle. Press OK to select these nodes.



- 4 FEMAP now displays the Create Nodal Loads Dialog Box. Move to the TX (Translation Degree of Freedom, X-Direction) and enter a value of negative 500 (-500.0). Press OK to create the loads. Press ESC or Cancel to end creating nodal loads. You will now see the loads on your model:



### Running the Analysis

The model building stage is now complete. The model is ready for analysis by any one of a number of finite element codes that FEMAP supports. Refer to the translation section of the FEMAP User Guide and your analysis program documentation to analyze this model. Typically, select **FILE - EXPORT - ANALYSIS MODEL** from the FEMAP menu. Choose the analysis program that you have access to, set the Analysis Type to Static, and press OK. FEMAP will then lead you through the creation of an input deck with dialog boxes custom designed to properly set up the analysis for your particular solver.

Once you have created an input file for your FEA program, you need to run that program.

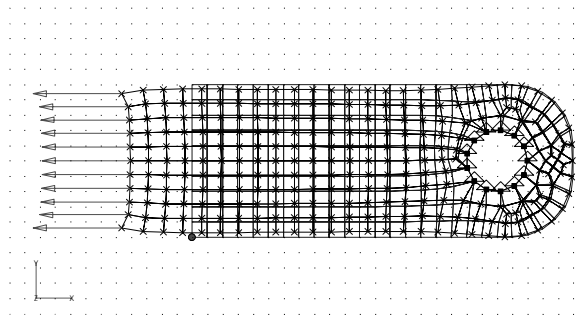
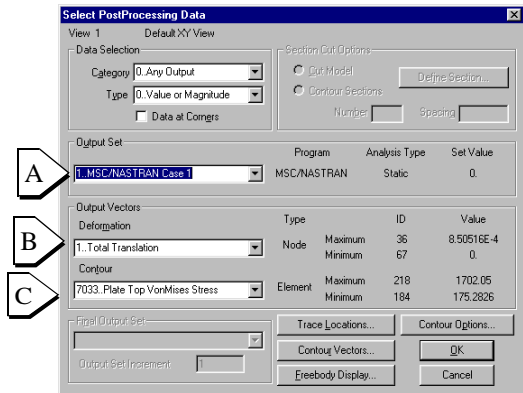
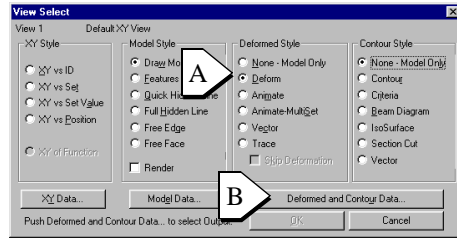
## 3.6 Review the Results

After solution, FEMAP can read the output results for post-processing. Reading output results is similar to writing the model for analysis, select **FILE - IMPORT - ANALYSIS RESULTS** from the FEMAP menu, set the translator to your FEA program, and then press OK. FEMAP will lead you through the rest of the process based on requirement of your particular analysis code. Once read in, the results are available for a wide array of graphical and numerical post-processing.

If you do not have access to a finite element analysis program, the results of analyzing this sample model are available on the diskette included with this manual. Select File - Open from the FEMAP menu, navigate to the directory where you installed the example problems for this manual. In the /examples subdirectory, open CH3POST.MOD.

## Viewing the Deformed Model

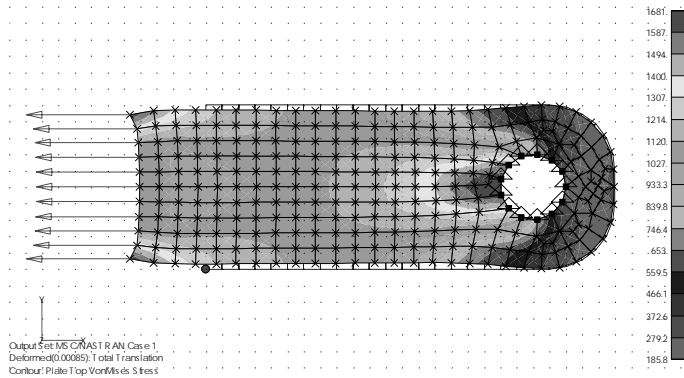
- 1 Choose **VIEW - SELECT** from the menu. The View - Select command determines how your model will appear on screen. To view the model deformed by the results of your analysis, A.) change the Deformed Style to Deform. To specify exactly what data to use for Post-Processing, B.) Press the Deformed and Contour Data Button.
- 2 The Select Post-Processing Dialog Box is now displayed. Here, you specify what data is used for on-screen Post-Processing. Data from multiple analysis runs or multiple Load Set/Constraint Set combinations will be stored in multiple Output Sets. The combo box at A.) is used to specify which Output Set the results will be obtained from, and the B.) Deformation and C.) Contour combo boxes are used to choose which particular pieces of output data are used for the Deformed Style and Contour Style options in the previous View - Select Dialog Box. Press OK to accept these choices, and then Press OK in the View Select Dialog box to activate the Deformed view.



## Viewing the Stress Distribution

Similar to turning on the deformed plot, you can return to View - Select and change the Contour Style option from None - Model Only to Contour and pressing OK. FEMAP displays the

model with a color representation of the stress levels superimposed on top of the deformed shape.

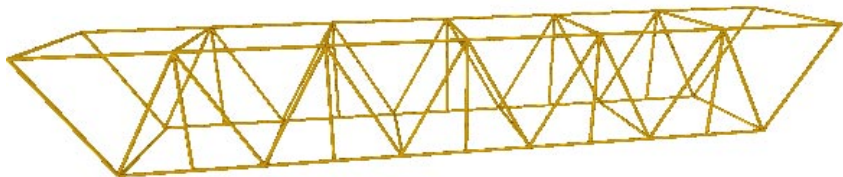


This chapter was designed to provide you with an overview of working with FEMAP, and to let you work through one sample problem from start to finish. The following chapters contain different types of examples to give you a broad range of experience with FEMAP.



# Roof Truss

# 4

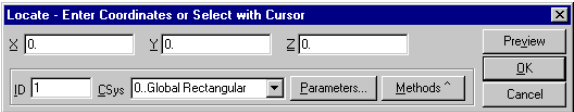


The above roof truss will be a simple example of modeling using both rod and L Beam elements. The symmetry of the truss will be used to reduce the problem to half the size. If you have not already done so, start FEMAP. Select New Model when prompted, or if you are already running FEMAP, select **FILE - NEW** from the menu.

## 4.1 Creating the Geometry

### Create Points

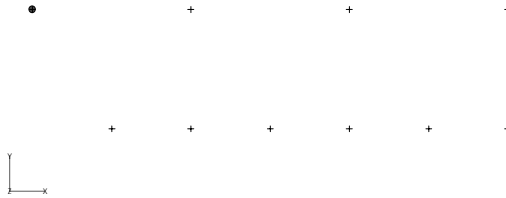
- 1 Select **GEOMETRY - POINT** from the FEMAP Menu. Enter the following data points:



Point	X	Y	Z
1	0	0	0
2	40	0	0
3	80	0	0
4	120	0	0
5	20	-30	0
6	40	-30	0
7	60	-30	0
8	80	-30	0
9	100	-30	0
10	120	-30	0

After the last point is entered select cancel. **After all screen picks you must select OK or double click on the last selection to complete the command or move to the next dialog box. Press cancel to exit a command.**

Press Ctrl-A to Autoscale the view.



### Create Lines Between the Points

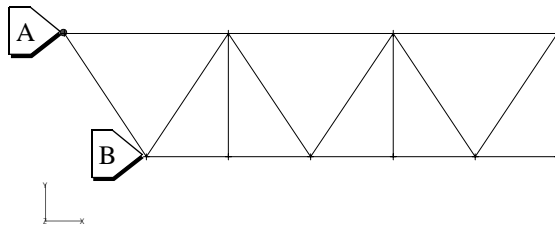
- 1 Choose **GEOMETRY - CURVE LINE - POINTS** from the FEMAP Menu.
- 2 Now create 17 lines connecting the points in this pattern.



#### Hint:

To speed creation of the lines, select the first point by A.) moving the mouse on top of the first point in

a line (you will see the point highlight) click once, and then move to B.) the point at the other end of the line and once it highlights, double-click. By double-clicking on the second point, it will be selected, and the OK button will be pressed for you.



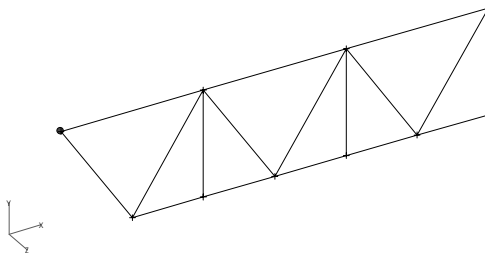
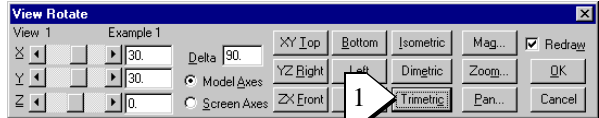
#### Note:

Be sure to create all 17 line segments. You do not want to have any lines crossing other lines. During mesh generation, you are guaranteed of having nodes at the end-points of geometric entities, therefore you are guaranteed that the finite element model will be connected at each joint.

These lines will be copied in the Z direction to create the other side of the truss. Since we will be working in 3-D for the first time, let's rotate the view so we can see the new lines that will be generated behind the ones you just created.

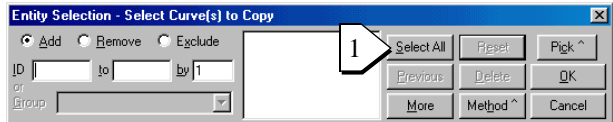
### Rotate the View

- 1 Choose **VIEW - ROTATE** from the FEMAP Menu (or use the Ctrl-R or F8 short-cut keys) and you will see the View Rotate Dialog Box. There are several pre-defined 3-D views that you can select from, you may want to experiment and press some of them. Before leaving View Rotate, press Trimetric and then OK to dismiss the View Rotate Dialog Box.



### Copy the Curves

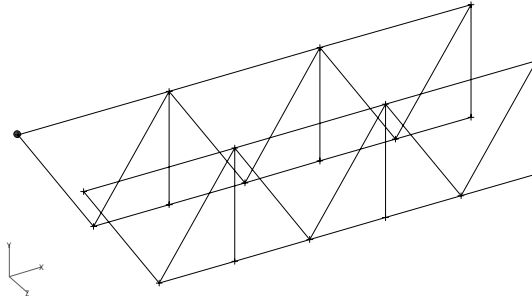
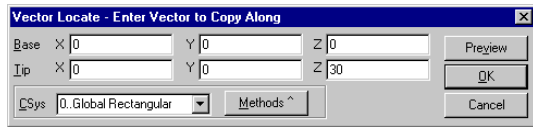
- 1 Select **GEOMETRY - COPY - CURVE** from the FEMAP menu. Press Select All to pick all the curves in this model. Press OK to continue



- 2 FEMAP now displays the Generation Options Dialog Box. This box allows you to control certain features about the new entities that will be created by this generation command, i.e. will the new entities match the original ones regarding layer, color, etc. or will they be created using the current settings defined in the FEMAP Tools - Parameters command. Choose OK to accept the defaults and continue.

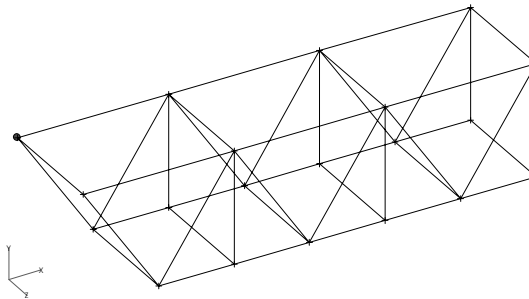


- 3 FEMAP now displays the Standard Define Vector Dialog Box. Fill in the box as shown, and Press OK to define the vector and complete the copy.



### ***Connect the Two Sides***

We will now connect the two side of the truss. Again, use the **GEOMETRY - CURVE LINE - POINTS** command, and create new lines that connect the corresponding points from the two sides of the truss. You should create ten 10 lines.



You have completed the geometry section of this example.

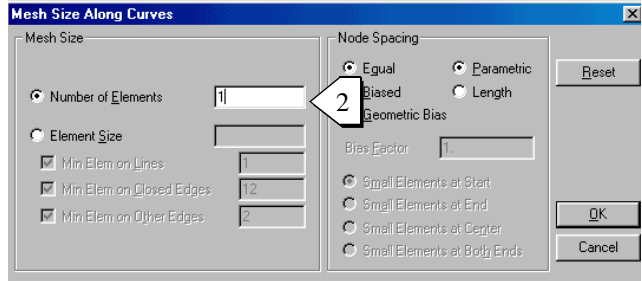
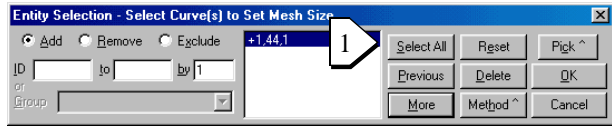
## ***4.2 Materials, Properties and Meshing***

We will now create the elements for the roof truss.

The first thing we need to do is set the mesh sizes for the curves (actually lines in this example) to an appropriate value; appropriate in that it will keep the total number of nodes generated small enough to work in the demo version of FEMAP.

## Setting the Element Mesh Sizes

- 1 Select **MESH - MESH CONTROL - SIZE ALONG CURVE** from the FEMAP Menu. We will press **Select All** and **OK** to define the number of line elements per curve.
- 2 In the **Number of Elements** field, enter 1 and Press **OK** to continue. FEMAP prompts you for more curves, press **Cancel** or **ESC** to exit the command.



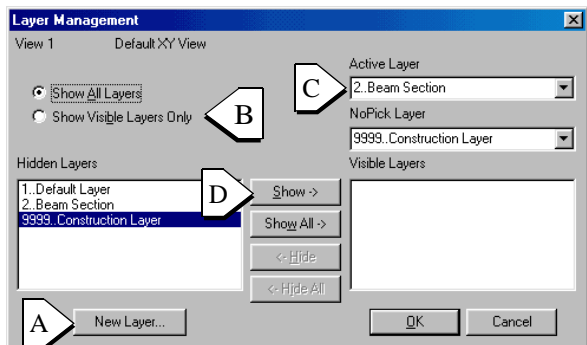
## Defining Beam Property for Arbitrary Cross Section

You may not always want to use a simple Beam cross section in your model. You can define any arbitrary cross section by either creating the corresponding surface inside of FEMAP or importing external geometry and then selecting a surface to calculate the Section Properties.

First create a new layer for our beam cross section so we can easily separate it from the rest of the model.

5. Select **VIEW - ROTATE XY TOP** and say **OK** to orient our view.
6. Select **VIEW - LAYERS** and the Layer Management dialog box will appear.

- A.) Press **New Layer** and create a new layer called **Beam Section**.
- B.) Select **SHOW VISIBLE LAYERS ONLY**.
- C.) Scroll Down and change the **ACTIVE LAYER** to **2..Beam Section**
- D.) Use the **HIDE** and **SHOW** keys to orient the high-lighted layer so that the only visible layer is **2..Beam Section**.



Geometry has been created on the Default and Construction Layers but we are viewing the empty Layer 2..Beam Section.

**Note:**

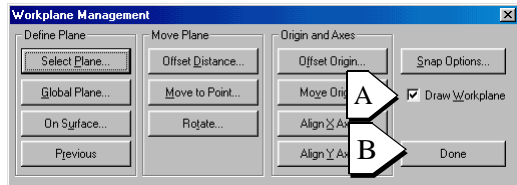
Up to this point all of the existing geomerty has been created on the Default and Construction Layers. Since we just switched to viewing only Layer 2..Beam Section which has no entities on it yet we will see only a blank screen.

Next we will turn on the workplane so that we have a reference when drawing the beam cross section.

Select **TOOLS - WORKPLANE** and click On (A) **DRAW WORKPLANE** to activate it and press DONE (B).

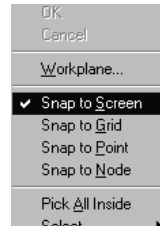
Use CTRL-G to regenerate the screen and draw the workplane. Next Zoom in so that the extents of the screen are about:

**X axis = +7    Y axis = -7**

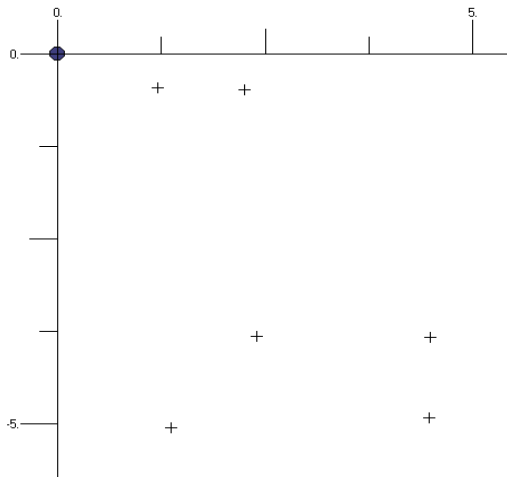
**Creating Geomerty for Cross Section**

Now we can use the Geomerty creation commands to draw any surface.

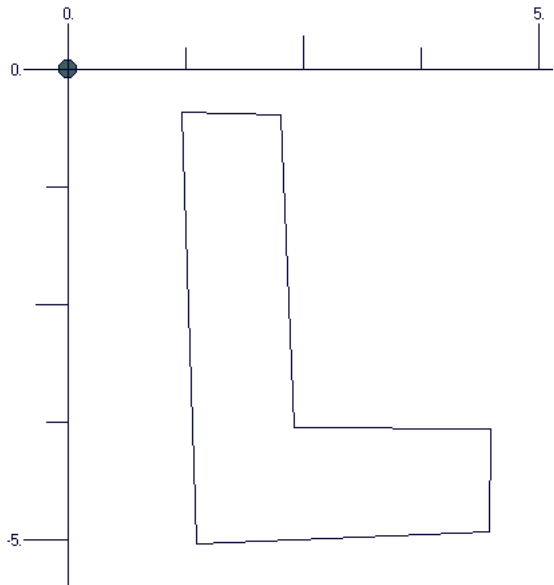
- 1 S Right mouse Click anywhere in the Default XY view and set your **SNAP to SCREEN**.



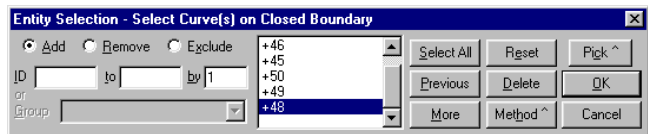
- 2 Select **GEOMERTY-POINT** and create 6 points by left mouse clicking on the screen to define the outline of a L beam approximately where the 6 points above are located to the right.



- 3 Once all of the points have been created we will connect them with line segments. Select **GEOMETRY - CURVE - POINTS** to connect the points and create the geometric cross section

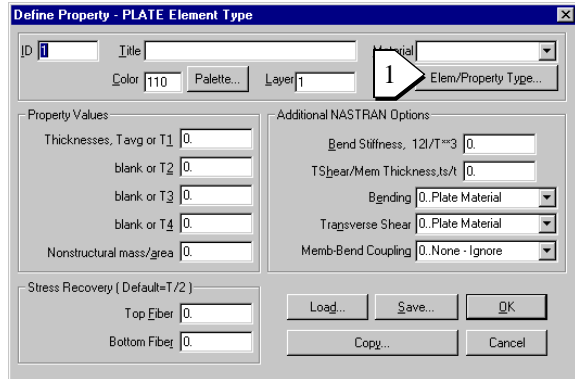


- 4 Now we have line segments that define our beam section but we need to create a surface. Select **GEOMETRY - BOUNDARY SURFACE** and pick each of the curve you have just created. Once you choose all the curves press OK and a boundary surface will be created.

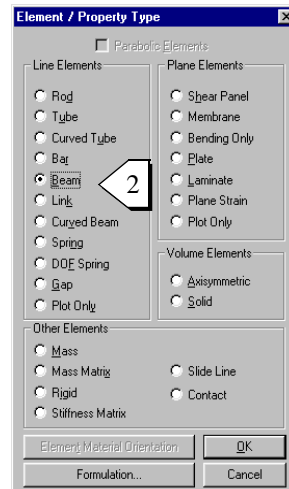


### Creating a New Property

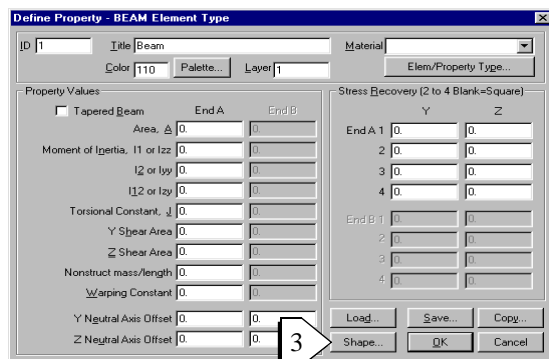
- 1 Select **MODEL - PROPERTY**. The default property type is Plate. To change to the Beam property we want to use in this example, and to change to a different element/property type in general, press the Elem/Property Type button.



- 2 In the Element/Property Type Dialog Box, change to a Line Element/Beam and press OK to continue.



- 3 FEMAP now returns you to the Define Beam Element Dialog Box. Press Shape to define the shape of the General Section.





- 4 Now we can select the surfaces you have chosen for the cross section.

A) Scroll down the Shape Drop-down list and select the type of beam as **GENERAL SECTION**

B.) Press **Surface** to select the surface to use as the cross section.

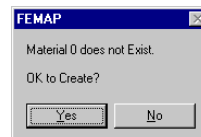
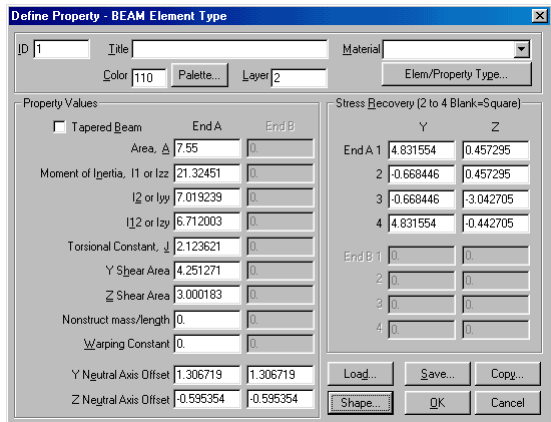
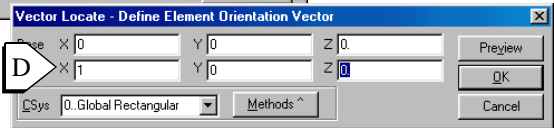
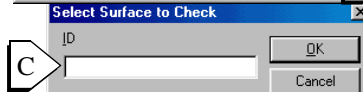
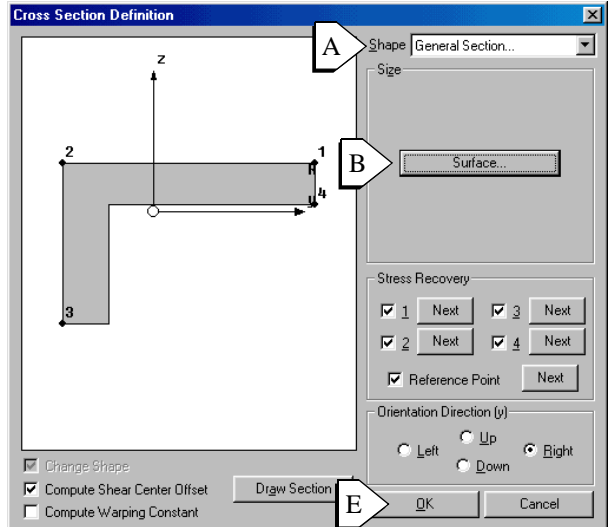
C.) Using the mouse select the Cross Section Boundary surface we have just created.

D.) Define the Orientation Vector for the surface. This vector orients the Y-Axis of the cross-section. Enter a vector coincident with the Y-Axis.

E.) Press OK and FEMAP will calculate the section properties

- 5 FEMAP now returns you to the Define Property Dialog Box which now includes the calculated values. Since we have yet to define a Material, there is nothing to select in the Material field. Press OK to continue.

- 6 FEMAP determines that we have not specified a Material for this Property and asks if you would like to create one. Press Yes to create a Material.



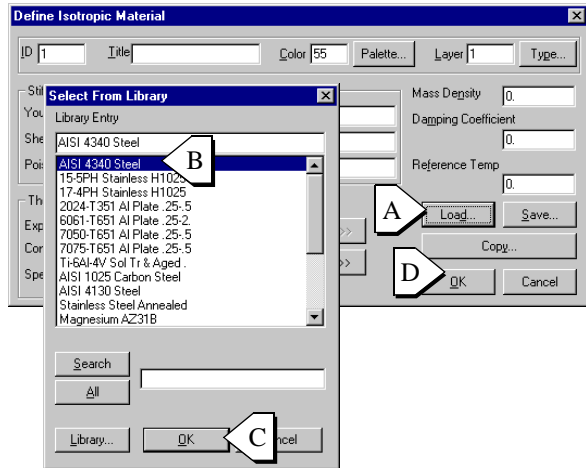
- 7 FEMAP displays the Define Isotropic Material Dialog Box.

A.) Press Load to access the material library. When the Select From Library box appears,

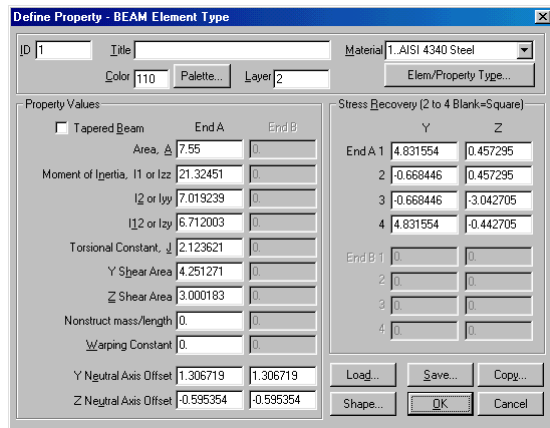
B.) select AISI 4340 Steel,

C.) press OK in the Select From Library Box, and then

D.) press OK in the Define Isotropic Material Box to create the material.

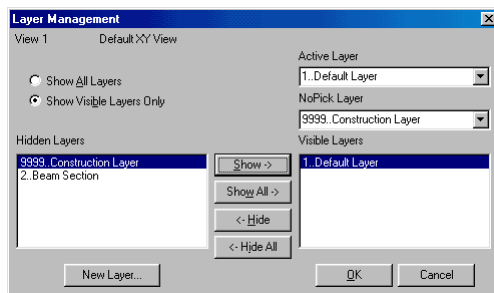


- 8 FEMAP returns to the Define Property Dialog Box with the Material reference filled in. Press **OK** to create the property.



- 9 Select **VIEW - LAYERS** and reset the active layer to 1.. Default View. Once again use the **SHOW** and **HIDE** buttons so that we can view our original model.

Use **VIEW-REGENERATE** to rebuild the screen and **CTRL - A** to auto scale the part.

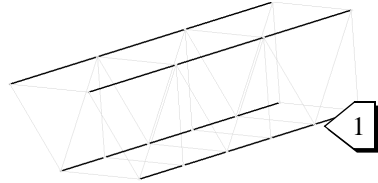


At this point we have defined the property to use during meshing and the size of the elements to generate, now we can mesh the part.

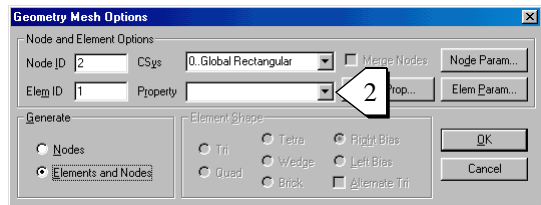
## Creating the Mesh

We will first mesh the longitudinal curves with beams whose properties were defined in the previous section. The second meshing section will then demonstrate meshing the cross-braces with rod elements.

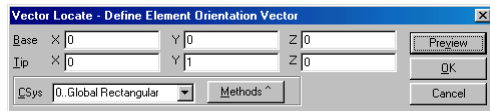
- 1 To mesh the curves containing beams select **MESH - GEOMETRY - CURVE**. Once the selection dialog box appears, select the darkened curves shown to the right by using your mouse.



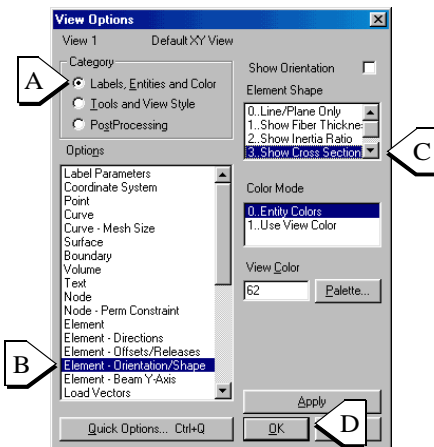
- 2 The Geometry Mesh Options dialog box then appears. Choose the Beam Property 1 just created under Property to mesh with this property. You can accept all other defaults and press OK.



- 3 FEMAP then prompts you to define the orientation vector that defines the Y axis of the beam. It is important that this vector be identical to the one used to define the cross-section properties, otherwise analysis results may be incorrect. Use an orientation vector along the Y-Axis and press OK.



- 4 To view the L Beam elements with their cross section select **VIEW - OPTIONS** and set
  - A.) Category to Labels Entities and Colors
  - B.) Options to Element Orientation/Shape
  - C.) Element Shape to Show Cross Section
  - D.) Press OK.



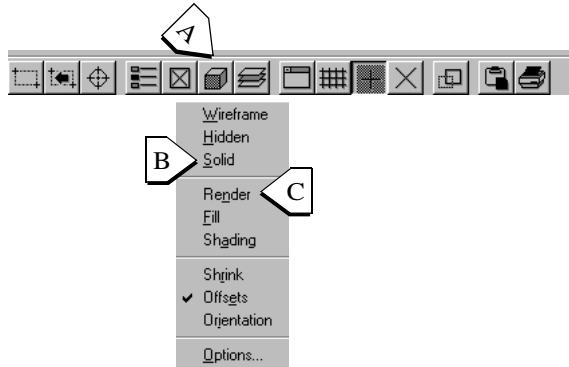
- 5 Use the tool bar across the top of the screen to chose  
A.) **VIEW Style** and

B.) Check on **SOLID**

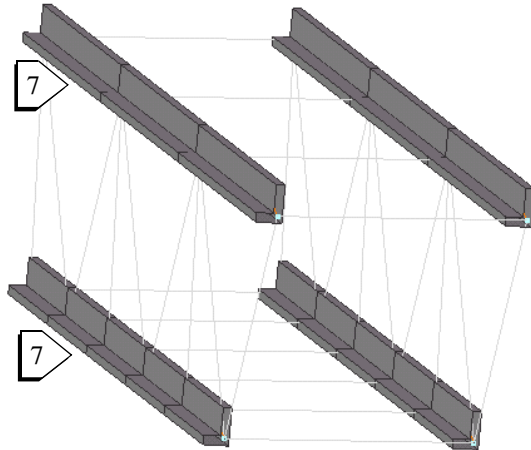
and

C.) Select **RENDER**

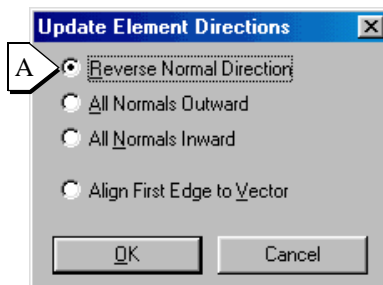
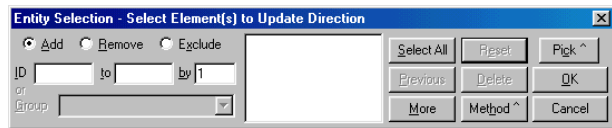
The display show look similar to (7) below.

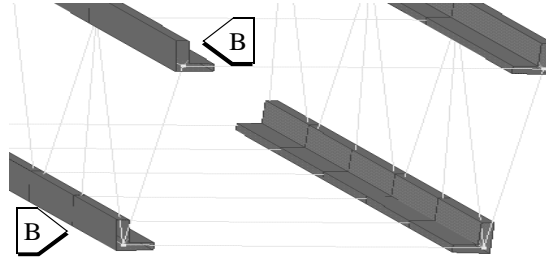


- 6 Since we are now in render mode, dynamically rotate the model simply by left clicking in the graphics window and dragging the mouse. This will rotate the part around XY. If you want to rotate around Z, Pan, or Zoom, simply hold the Alt, Ctrl, or Shift key down, respectively, when you first press the left mouse button.



- 7 With the cross-sections of the beams drawn, you can tell if there is an incorrect orientation. Some of the beams might be oriented with the L section facing outward rather than inward (as in 7 above). To modify the orientation Select **MODIFY - UPDATE ELEMENTS - REVERSE**, select the elements and then choose A. Reverse Normal Direction. If you select the appropriate elements, your display should look similar to the one below (B), with all L beams pointing inward.





### Note:

The order in which you picked the points to create each of the curves will determine which direction the cross section will face. Therefore, you may need to use the reverse command on elements different that the one shown in the previous picture.

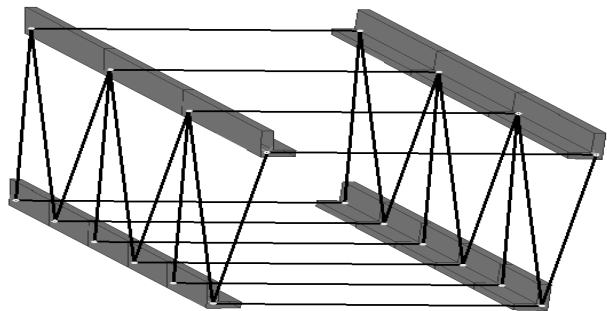
### Hint:

The incorrect orientation could have been avoided by defining the mesh attributes originally along the curve. The **Mesh - Mesh Control - Attributes Along Curve** command allows you to define the property, orientation of the cross-section along the curve, and any offsets before you mesh the curve. This command, including offsets, is used in Chapter 9, Cylindrical Support. Thus, you could have avoided the **Modify - Update Elements - Reverse** command above.

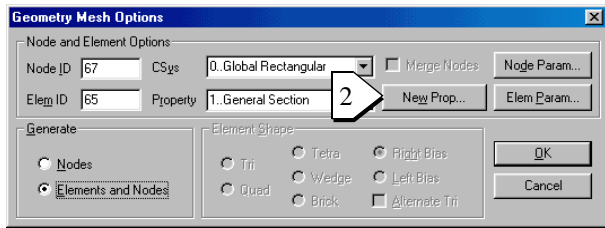
### Rod Mesh

The above section meshed the longitudinal curves with beam elements. The remaining curves connecting the longitudinal curves as well as the cross-braces will now be meshed with rod elements (all curves except those that were just meshed with beams).

- 1 Select **MESH - GEOMETRY - CURVE** from the FEMAP menu. Use the cursor to pick all of the curves shown in bold in the diagram (the cross-braces, as well as the connection between the two rows of beams).

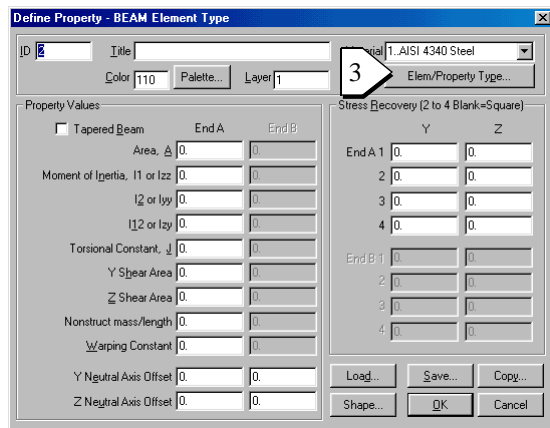


- 2 FEMAP now displays the Generate Mesh Options Dialog Box. Since we only created the General Section property and do not have a rod property available, we will need to create a new property for the upcoming elements. Press the New Prop. Button.

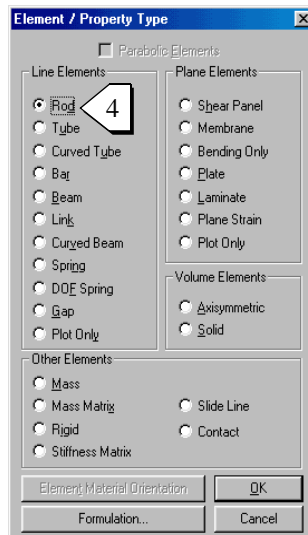


- 3 FEMAP now displays the Define Property Dialog Box. The previous property we created was a Beam type.

To change to the Rod property we want to use in this example, and to change to a different element/property type in general, press the Elem/Property Type button.

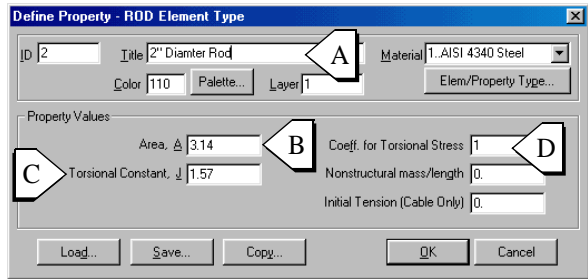


- 4 In the Element/Property Type Dialog Box, change to a Line Element/Rod type and press OK to continue.



- 5 FEMAP now returns to the Define Property Dialog Box, however the input options now represent those required for a Rod element/property. Fill in the box as shown, entering
- A.) "2 in. Dia. Rod" for a title,  
 B.) 3.14 for the Area,  
 C.) 1.57 for the Torsional Constant,  
 D.) 1.0 for the Coef. for Torsional Stress.

Since we have already defined a Material we can chose it in the Material field. Press OK to continue.



- 8 You are now back at the original Geometry Mesh Options Dialog Box with the Property reference filled in. Press OK to generate the rod elements.

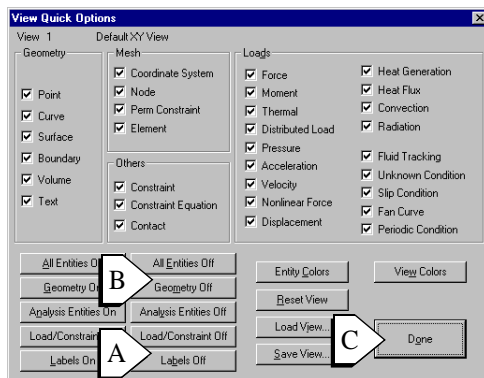


- 9 To reduce the amount of information displayed, use the FEMAP Quick View Options, Ctrl-Q. Once the View Quick Options box appears:

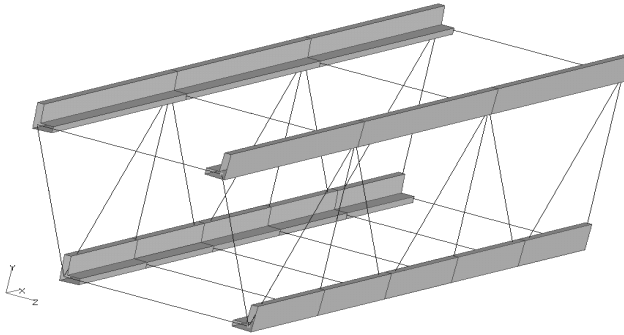
A.) Press Labels Off to turn off all entity labels

B.) Geometry Off to turn off display of the underlying geometry since it will no longer be needed.

C.) Press Done to execute the new view options.



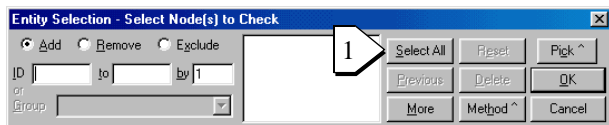
The display should now look like the figure below.



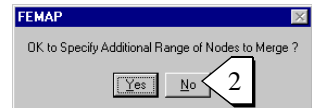
Each line in this model now has either rod or beam elements along its lengths. Since they were generated independently, there will be multiple nodes at the end of every line coincident with the nodes generated for the other lines that also end in the same point. In the world of FEA, these structural members will not be connected. To effectively “sew” the model together we will use FEMAP’s ability to check and optionally merge coincident nodes of a model.

### Merging Coincident Nodes:

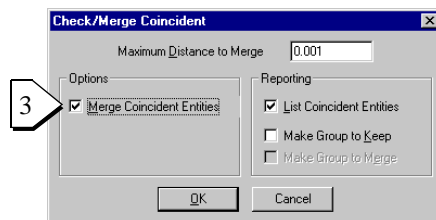
- 1 Select **TOOLS - CHECK - COINCIDENT NODES** from the FEMAP menu. FEMAP prompts you for a list of nodes to check, press **Select All** and then **OK** to continue.



- 2 FEMAP now asks if you would like to select another range of nodes to merge. If we answered yes to this question, FEMAP would require us to select another batch of nodes. The significance here is that the second range of nodes would be merged into the first. In a finite element model where you are trying to keep a certain node numbering arrangement, this feature allows you to control which node numbers are kept (the first set selected), and which could be potentially merged away. Since this example does not have any specific node numbering requirements, press **No** and nodes will be merged without any consideration to numbering.



- 3 The Check/Merge Coincident Dialog Box is now displayed. Since this command is titled “Check” Coincident Nodes, it is just that a check. If you took the defaults, it would just check and list to the Message and Lists Windows which nodes are coincident. To actually merge the node, check the Merge Coincident Entities box. Press **OK** to continue.



Coincident nodes will be merged, connecting all rod and beam elements of this example together.

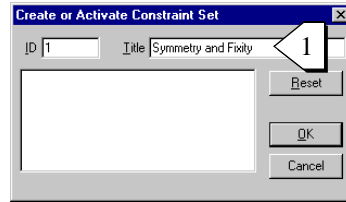


## 4.3 Loads and Constraints

Since Loads and Constraints are Set based, you must create an empty Constraint Set before defining any actual constraints.

### Create Empty Constraint Set

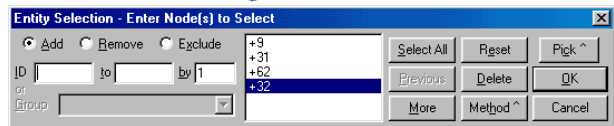
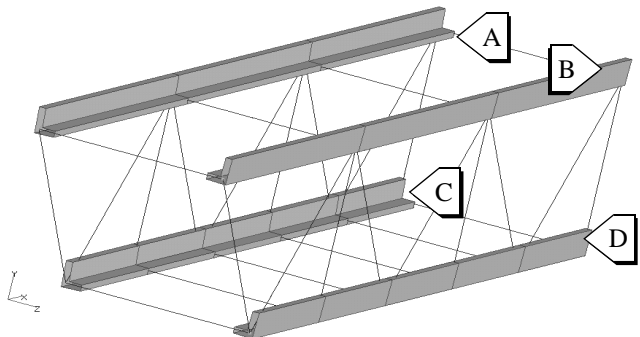
- 1 Select **MODEL - CONSTRAINT - SET** from the FEMAP menu. Fill in the Title and press OK to create the empty set.



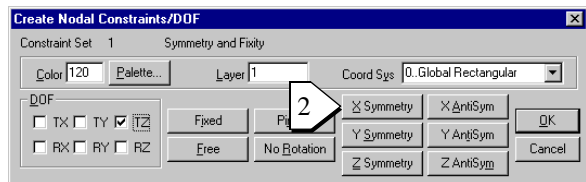
### Create the Boundary Conditions - Symmetry and Fixed End

To simulate the symmetry of this part, we will constrain the four nodes that are at the halfway point of our structure. In this model, we are defining symmetry across the X-Plane through these four points. By imposing this type of constraint condition, we are actually introducing a stiffness exactly equal to the structure modeled, just mirrored about the X-Plane.

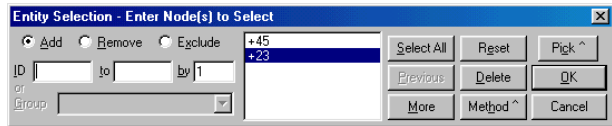
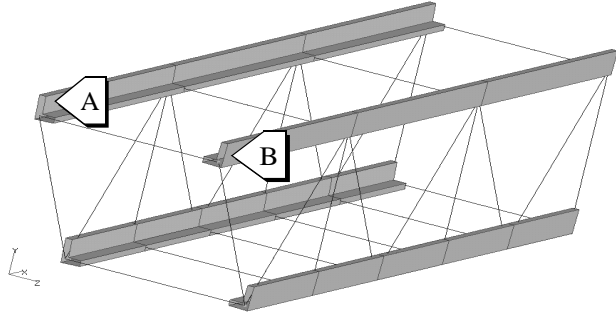
- 1 Select **MODEL - CONSTRAINT - NODAL** from the FEMAP menu. FEMAP prompts you for the nodes to constrain, select the four nodes at A, B, C and D. Press OK.



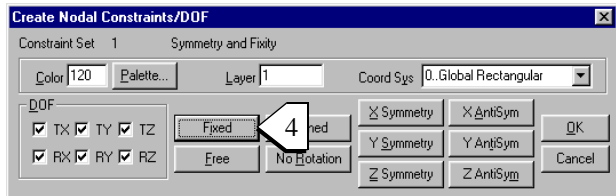
- 2 FEMAP now prompts you for the constraint conditions to apply to the selected nodes. You could control the constraint of each degree of freedom individually in the DOF box, or use the built in quick keys to apply common constraint conditions. For this example press the X Symmetry button to apply constraints that impose symmetry about the X-plane. Press OK to create the constraints.



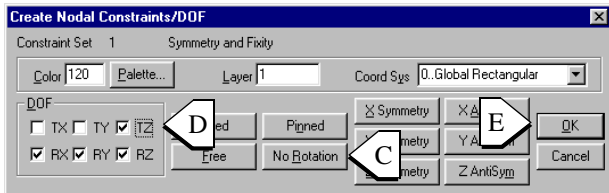
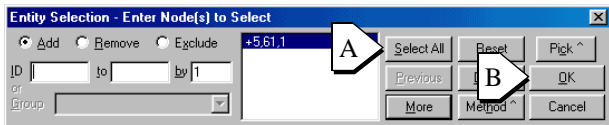
- 3 FEMAP automatically prompts you for more nodes to constrain. Select the two nodes A and B at the left end of the truss. Press OK to continue.



- 4 Again, the Create Nodal Constraints Dialog Box is displayed. This time, we will fix these nodes. Press the Fixed button, which will toggle all six degrees of freedom to constrained, and then press OK to continue.

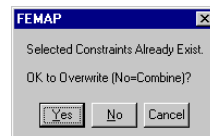


- 5 Since we will be loading this part in the negative Y direction, this part will also require constraint against movement in the global Z direction and against all rotations. To achieve this, we will constrain all nodes against translation in the Z direction.

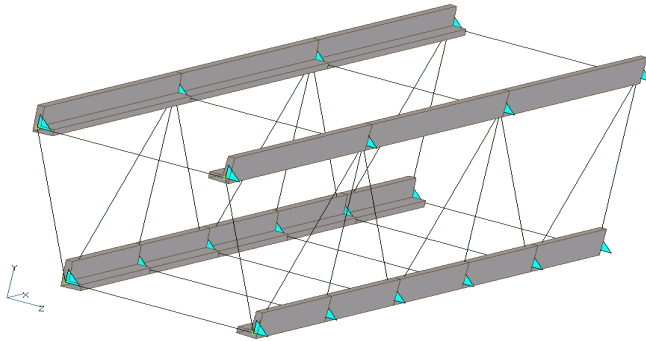


- A.) Press Select All in the Entity Select Box and then
- B.) Press OK. FEMAP now displays the Create Nodal Constraint Dialog Box,
- C.) Press the No Rotation button,
- D.) Toggle the TZ Degree of Freedom, and
- E.) Press OK to continue.

- 5 Since some of the nodes already have constraints, FEMAP asks you whether to overwrite or combine, press No to combine. Again FEMAP will prompt you for more nodes to constrain. Press Cancel or the ESC key to dismiss the dialog box.



The constraints are displayed as small triangles.



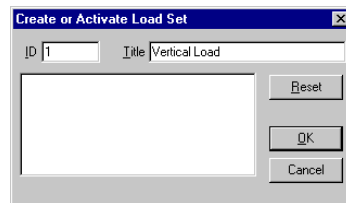
Since all labels were turned off back in the geometry creation phase, you will not see any labels on the constraints themselves. If you were to turn the constraint labels on, you would see the actual degree of freedom numbers displayed for each constraint.

### **Loading the Model**

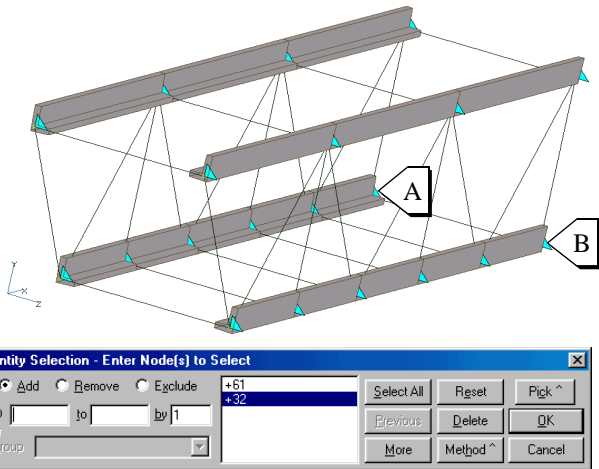
We will now apply a load in the negative Y direction to simulate something hanging from this truss. Again, like constraints, loads are grouped in sets. Before creating any loads we must create a set to hold them.

Loads will be placed along the bottom nodes at the symmetry plane of the truss. To create the load

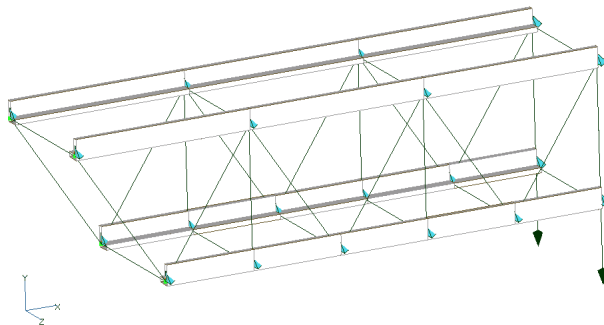
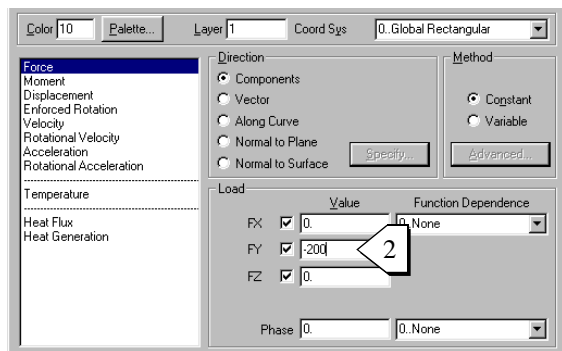
Select **MODEL - LOAD - SET** from the FEMAP menu. Similar to the Constraint Set previously created, enter a Title and press OK to create the new empty Load Set.



- 1 Select **MODEL - LOAD - NODAL** from the FEMAP menu. FEMAP prompts you for the nodes. Select two nodes, at A and B.



- 2 FEMAP now displays the Create Nodal Load Dialog Box. By default the load Type is set to Force. Enter -200 in the TY field to create a 200 pound load in the negative Y direction. Press OK to create the loads. Again, FEMAP automatically prompts you for more nodes for load application. Press the ESC key or the Cancel button to terminate creation of loads.



This model is now complete and ready for analysis. Again, you may save the model now, complete with the loads and boundary conditions applied for later analysis.

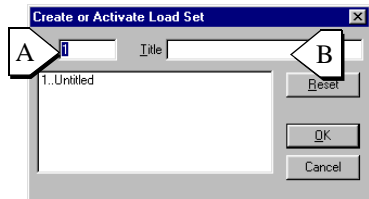
## 4.4 Advanced Loading

Previously we applied a static load to our roof truss. In this section we will apply a load that varies with time in preparation for a NASTRAN Transient Dynamic Analysis. We will now create a time varying load to obtain a response history. The load will be a sinusoidal load with a frequency close to the natural frequency. We have already performed a modal analysis and determined that the first mode natural frequency is close to 51 Hz. If necessary, open CH4DONE.MOD in the /examples subdirectory.

### Create a New Load Set

In the current model, Load Set 1 contains the static loading. We must first create a new load set to define our time dependent loading and Transient Dynamic Analysis setup information.

- 1 Select **MODEL - LOAD - SET** from the FEMAP menu. You will see the first Load Set that was previously defined. To add another Load Set, A.) Type a 2 in the ID Field, and B.) Type a new title in the Title field. Press OK to create the empty set.

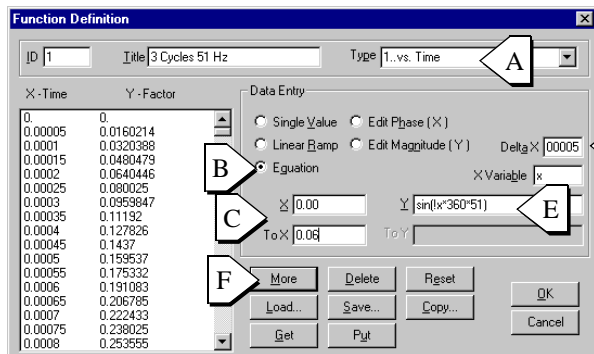


### Create the Magnitude vs. Time Function

Before we can define a time varying load, we must first create a FEMAP Function that defines the multiplier at each time step. We will use direct transient analysis, 1200 steps, a time per step of 0.00005 and output interval of 40. The number of steps and time per step gives a total time of 0.06 seconds which corresponds to about 3 cycles. To create a time-dependent function

- 1 Select **MODEL - FUNCTION** from the FEMAP menu.  
A.) Set the Type of Function to vs. Time.  
B.) Selection Equation under Data Entry.

We will now define an equation for  $X=0.0$  to  $X=0.06$ , first,



C.) set the X and To X values as to 0.00 and 0.06, respectively.

D.) Next, since we want 1200 steps between 0.00 and 0.06, enter a Delta X of 0.00005.

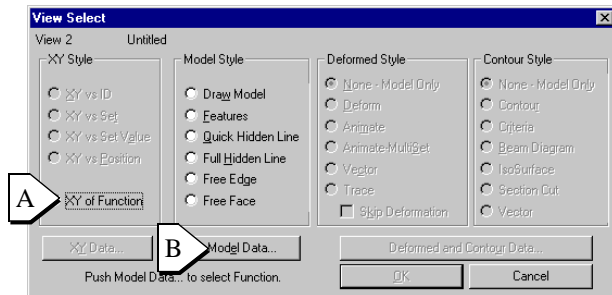
We now need to enter the equation. You will notice that the X variable is defined as x, therefore in our equation, we will use the !x FEMAP convention for accessing a variable.

E.) Type in the equation:  $\sin(1x*360*51)$ .

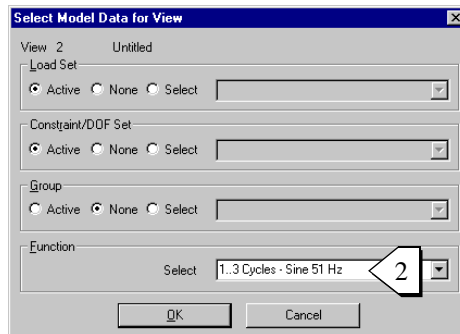
F.) Press More to see the function values calculated. Press OK to create the function and continue. Press Cancel or ESC to exit the command.

To view this function, and make certain that it represents the sinusoidal time history that we want, do the following:

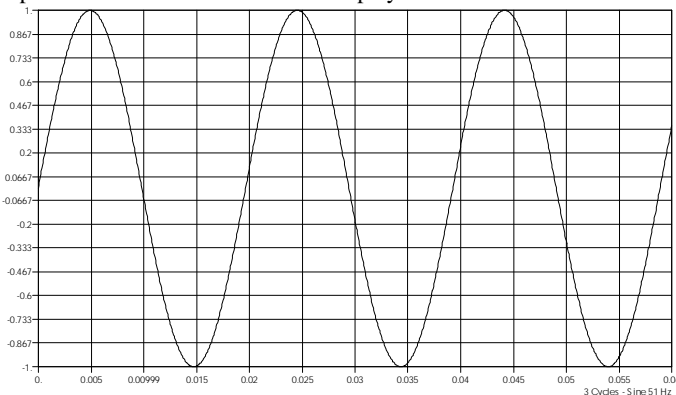
- 1 Choose **VIEW - SELECT** (F5 or Ctrl-S) from the FEMAP menu. A.) Change the XY Style to XY of Function. You will notice the Push Model Data...to select Function text appear at the bottom. B.) Push the Model Data button.



- 2 In the Select Model Data for View Dialog Box, select the Function that we just defined and then press OK to return to the View Select Dialog Box. Press OK in the View Select Dialog Box to view the XY Function plotted on screen.



An XY plot of the function will now be displayed:



To return to the regular model view,

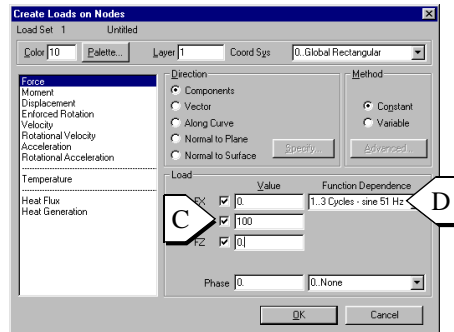
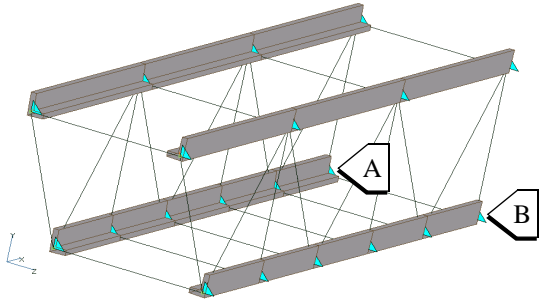
1. Access **VIEW - SELECT** either from the menu or press F5 or Ctrl-S.
2. Change the Model Style back to Draw Model.
3. Press OK to redraw the model.

## Create the Time Varying Load

Creating a time varying load is almost as easy as creating a regular static load, with the additional step of attaching a function to it. To create our time varying load:

- 1 Select **MODEL - LOAD - NODAL** from the FEMAP menu. Just like we did in the static loading example of Chapter 4, pick the bottom two nodes A.) and B.), and press OK to continue.

FEMAP will now display the Create Nodal Loads Dialog Box. Again, similar to applying a static load, C.) enter a value of -100.0 in the TY field. To attach this load to a function, move to the Temp/Time/Freq. Dependence field and D.) select the function that we just created. Press OK to create the loads. Press Cancel to end creation of nodal loads.



## Set up the Transient Dynamic Analysis

In addition to a time varying load, a transient dynamic analysis also requires some additional input that defines the analysis. We are going to specify 1200 steps, a time per step of 0.00005 and an output interval of 40. The number of steps and time per step gives us a total time of 0.06 seconds which corresponds to about 3 cycles. The output interval of 40 will give us 30 data sets. Set the overall structural damping coefficient to 5% of critical or 0.05.

Select **MODEL - LOAD - DYNAMIC ANALYSIS** from the FEMAP menu.

A.) Choose Direct Transient as the solution method and Enter:

B.) 0.05 as the damping coefficient.

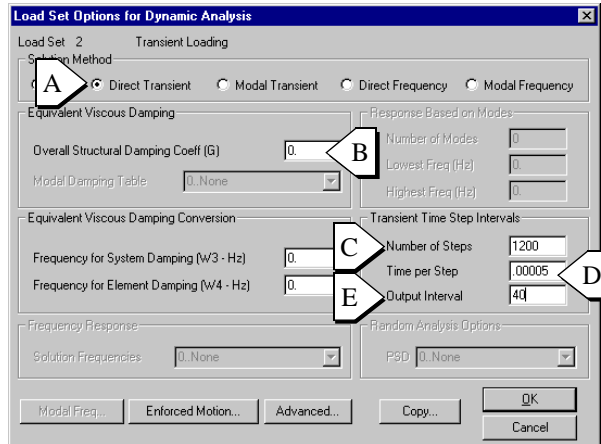
C.) 1200 for the Number of Steps,

D.) .00005 for the Time Step

E.) 40 for the output interval.

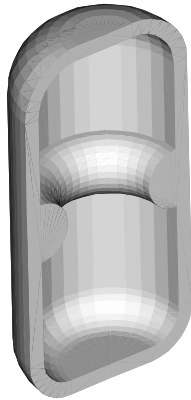
Press OK to set the Dynamic Analysis options.

The model is now ready for transient analysis. Refer to the translator reference for your particular analysis program. Save this model if you wish. A completed version is available in the / function subdirectory titled CH4FUNC.MOD.





# Pressure Vessel



# 5

In this example we will take advantage of the axisymmetric nature of this and most pressure vessels and analyze the part using axisymmetric finite elements.

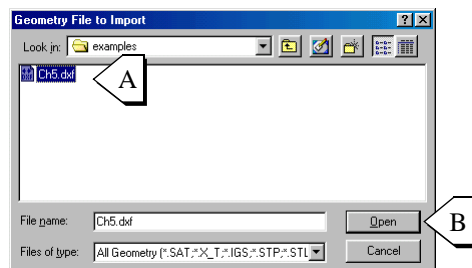
In the previous examples we created geometry completely inside of FEMAP. Many times, geometry for your part may already exist in a CAD system. In this example the geometry required for building the finite element model will be imported.

Again, first start FEMAP and create a new model, or if FEMAP is already running, select **FILE - NEW** from the menu.

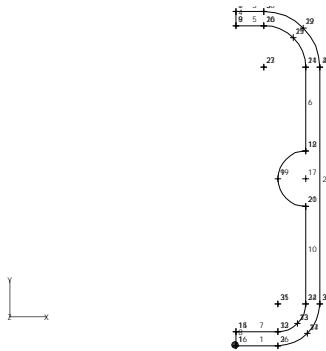
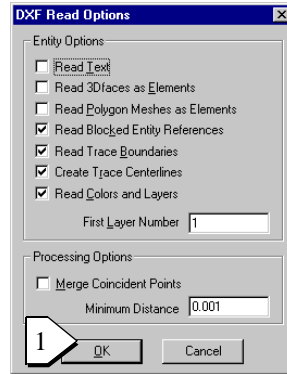
## 5.1 Import DXF Geometry

The geometry for this example was created in AutoCAD. From AutoCAD, the geometry was exported as an AutoCAD DXF file. DXF files are a popular means for transferring geometry between CAD programs and desktop publishing packages. To import the geometry:

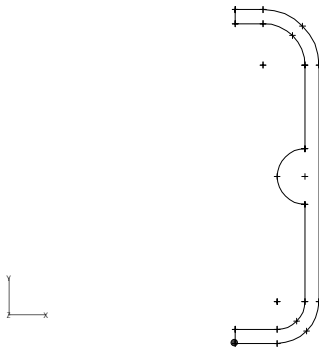
- 1 Select **FILE - IMPORT - GEOMETRY** from the FEMAP menu.
- 2 FEMAP will now display the Windows File Open Common Dialog Box, maneuver to the directory where the examples for this manual are stored, go to the /example subdirectory, and A.) select CH5.DXF, and then B.) Press Open to begin importing the geometric information.



- 3 FEMAP now displays DXF Read Options Dialog Box. This box contains various options to control how the information in the DXF is read in and stored in FEMAP. Once again, the defaults have been designed to work best in the general situation of importing geometry, therefore, just press OK to continue.



If the labels for the points and curves that make up the geometry of this example are displayed, use the FEMAP Quick View Options (Ctrl-Q), and turn off all labels.



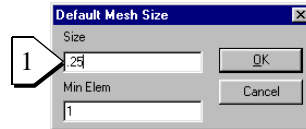
The geometry section of this example is complete, representing the simple import of existing geometry that will later form the basis of your finite element.

## 5.2 Material, Property and Meshing

We will first set the global mesh size, then view it to verify that the mesh spacing is appropriate.

### Set the Global Mesh Size

Select **MESH - MESH CONTROL - DEFAULT SIZE** from the FEMAP menu. Enter a Size of 0.25 as shown, press OK to continue.



The global mesh size is not displayed on your model by default. To turn on the mesh size preview:

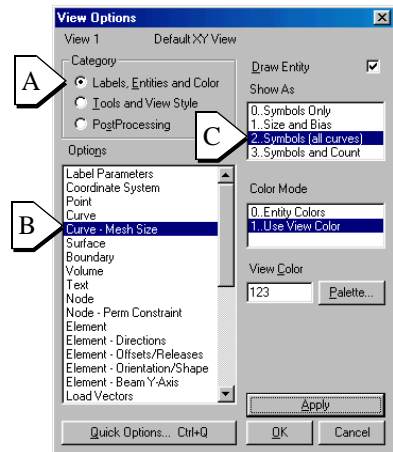
Select **VIEW - OPTIONS** from the FEMAP menu (shortcut keys are F6 or Ctrl-O). The View Options Dialog Box is displayed. View options are divided into three specific categories as shown in the figure to the right. Make sure that

A.) The Category is set to Labels, Entities and Color.

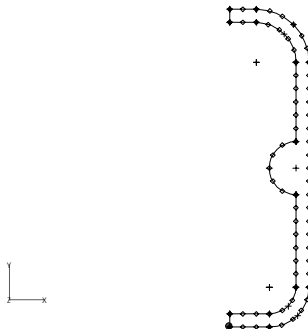
B.) Switch the Option to Curve - Mesh Size, and then,

C.) Change the Show As option to “2..Symbols (all curves)”.

Press OK when completed to update the view.

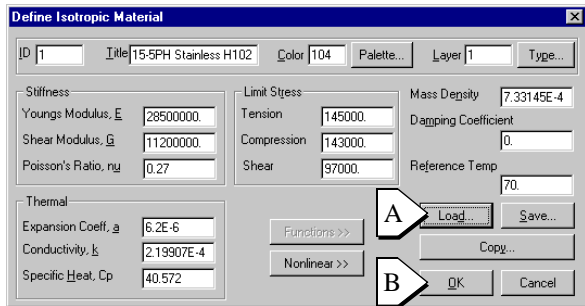


You should see small diamonds along each of the geometric entities in FEMAP. These diamonds indicate where nodes will be created during any automatic meshing commands that use this geometry.



### Create a Material

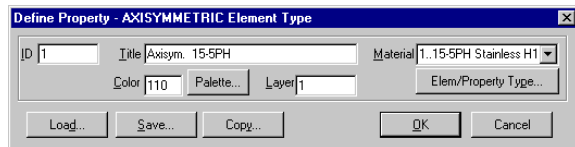
Select **MODEL - MATERIAL** from the FEMAP menu. A.) Press the **LOAD** button and choose 15-5PH Stainless Steel from the FEMAP material library. Press OK to return to the material creation dialog box. The physical properties will now be filled in, B.) Press OK to continue, and then ESC or Cancel since this is the only material required in this model.



### Create the Axisymmetric Property

We will now create an Axisymmetric Property that references the 15-5 Stainless material that we just created.

Select **MODEL - PROPERTY** from the FEMAP menu. Initially, the default Property Type is Plate, A.) press the **ELEM/PROPERTY TYPE** button, and select Axisymmetric for the element type.

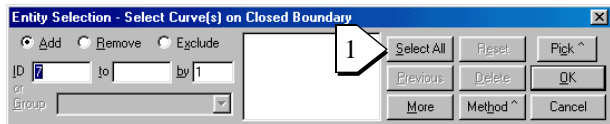


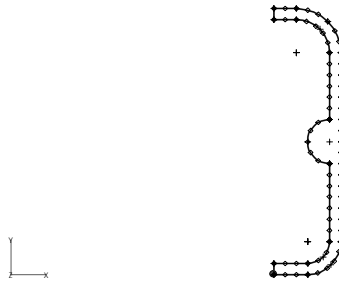
Press OK to return to the property creation dialog box. Fill in the Title as shown, select the 15-5PH Stainless that we just created as the material, and press OK to continue. FEMAP now prompts you for the creation of another property, press Cancel or hit the ESC key to end creation of properties.

### Create a Boundary

This shape could be broken down into simple geometric regions for mapped meshing, but it is much easier to create a FEMAP Boundary Surface and automatically mesh the whole part.

- 1 Select **GEOMETRY - BOUNDARY SURFACE** from the FEMAP Menu. FEMAP prompts you to select the curves for the boundary, Press Select All, and then OK to continue.

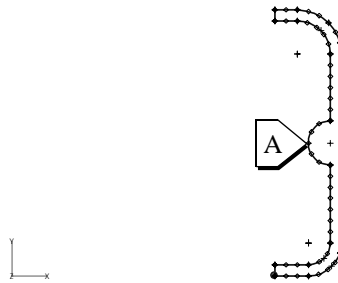




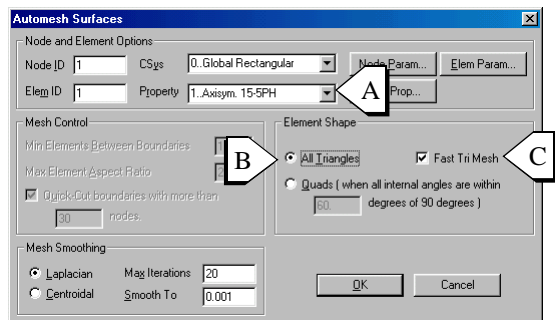
### Boundary Mesh

We are now ready to mesh the part.

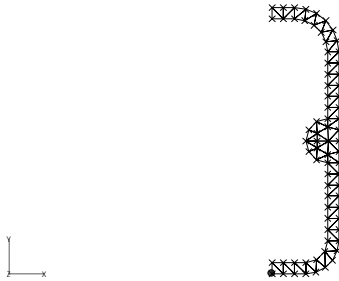
- 1 Select **MESH - GEOMETRY - SURFACE** from the FEMAP menu, A.) select the Boundary from the graphics window and B.) press OK to continue. As an alternate, recall that you could simply double-click on the boundary, which selects the boundary and accepts the selection immediately.



- 2 The Automesh Surfaces Dialog Box now appears. A.) Select the Axisymmetric Property we just created in the Property Box, and B.) change the Element Shape option to All Triangles. C.) Check the box for Fast Tri Mesh and Press OK to generate the mesh.



Use the FEMAP Quick View Options (Ctrl-Q) and turn off labels and geometry. The mesh will look like:



The meshing portion of this example is now complete.

### 5.3 Loads and Constraints

In the previous examples every step of creating the model geometry, finite element data, loads and boundary conditions was detailed. We will now assume that you have acquired sufficient proficiency in FEMAP that you will require less step-by-step instructions.

We will now constrain the vessel and apply an internal pressure load.

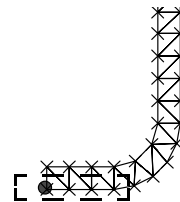
#### Create Empty Load and Constraint Sets

Similar to the last example, create empty load and constraint sets.

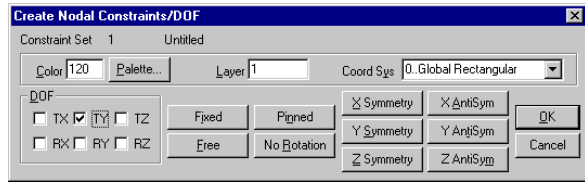
#### Constrain the Model

We will constrain this model at its base. Since this model is axisymmetric, there is no need to apply any loads and constraints out of plane. We will constrain the bottom nodes along the flat edge against vertical movement.

- 1 Select **MODEL - CONSTRAINT - NODAL** from the FEMAP menu. When the Create Nodal Constraint Dialog Box appears, select the four nodes along the bottom of the part. You can select these nodes individually, or use the method of box picking previously described in this manual.



- 2 Constrain these nodes in the vertical direction by toggling the TY DOF box. Press OK to continue, and then Cancel to end the Create - Constraint - Nodal command.

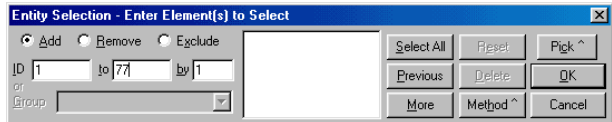


## Applying the Internal Pressure Load

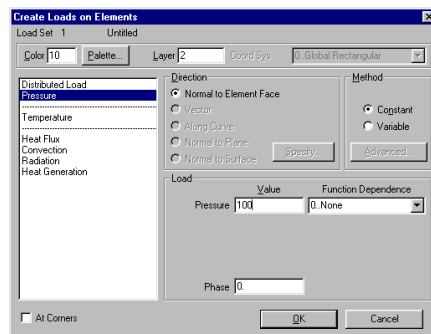
In an axisymmetric model, a pressure load is applied along the edges of the axisymmetric elements. Be careful when applying an internal pressure to an axisymmetric model, some analysis codes require a pressure load per radian, while others want a pressure load per degree. Be sure to check your analysis code's documentation and verify which convention is appropriate.

Since the load is applied to the edges of the elements, in addition to the element number itself, FEMAP requires you to specify which face of the element gets the load. Here we will use a FEMAP feature that makes applying loads to the outside of an axisymmetric or solid model very easy - the Adjacent Faces method.

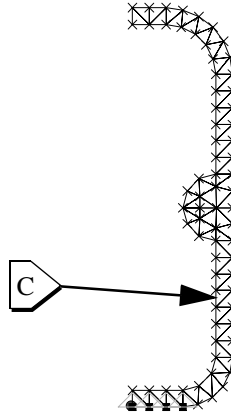
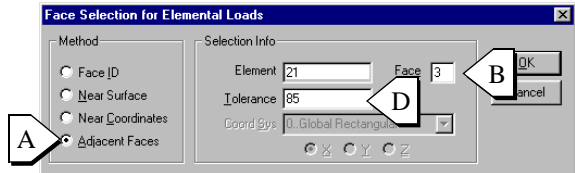
- 1 Select **MODEL - LOAD - ELEMENTAL** from FEMAP menu. Even though we only want to apply a load to the elements that border the interior of this part, we will first select all elements, and then narrow this selection down with the Adjacent Faces Method. Select all the elements and then press OK to continue.



- 2 In the Create Elemental Loads box, enter a pressure of 100.00 and press OK to continue.

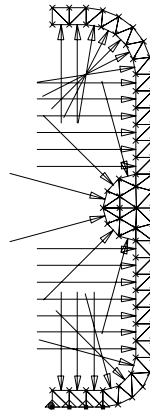


- 3 A.) Change the Method to Adjacent Faces. B.) Now click in the Face Box, this will move the focus (flashing cursor) to the face box. Since the focus is in the Face box, when you now select from the graphics window, FEMAP will know that you are trying to pick a particular face. Move to the graphics window and C.) pick one face on the inside of the part. You will see that face that you choose highlighted. Make certain that it is on the inside of the vessel. Now move to the tolerance field and enter a value of 85.00. Press OK to generate the loads. Press Cancel to end the creation of elemental loads.



The Adjacent Faces Method simply walks connected free faces (or edges in this case) until it gets to a connected free face that exceeds the angle tolerance. In this example, by entering a very large angle tolerance of 85.0 degrees, all the internal faces are selected right up to the corners along the Y Axis. To go around this corner, an angle of 90.0 degrees or greater would be required as a tolerance.

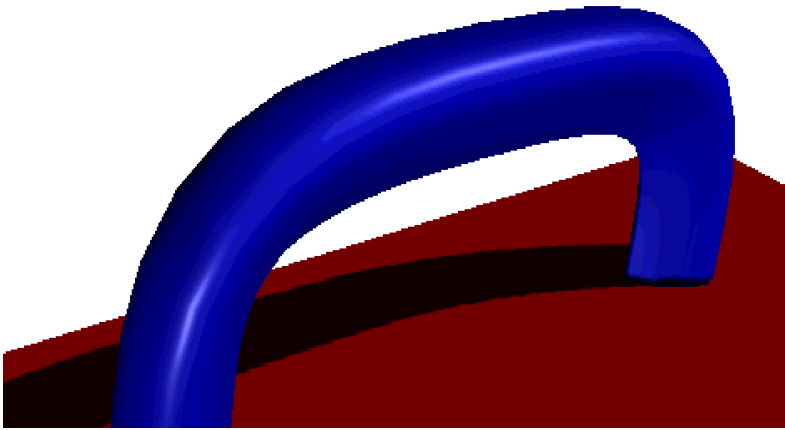
The model is now complete and ready for analysis. You can save it now for later analysis, or simply use the pre-made version CH5DONE.MOD in the /examples subdirectory when the time comes.





# *Handle*

# 6



The above handle represents a good model for solid elements. The curved shape and rounded profile together with the variable loading conditions would not be possible to achieve with other types of elements. Because we are limited to 300 nodes in the demo version we will only model half of the handle and with a lower mesh density than we would like for normal analysis.

**You must be licensed for the PARASOLID Geometry Module and have it active (under Tools - Advanced Geometry) to complete this example.**

Again, start FEMAP or start a new model from an existing FEMAP session.

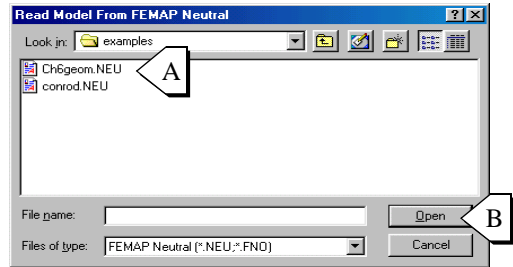
## **6.1 Creating the Geometry**

### **Importing the Curve Geometry**

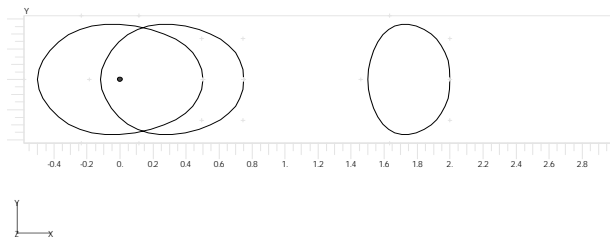
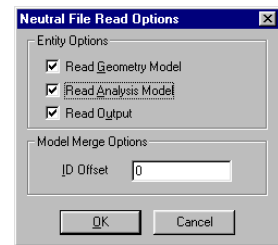
Once again the geometry for this example has been created for you. It is a simple series of curves that were created in FEMAP and written out as a neutral file. To import the base geometric data:

- 1 Select **FILE - IMPORT - FEMAP NEUTRAL** from the FEMAP menu.

- FEMAP displays the standard Windows File Open Dialog Box. Maneuver to the /examples subdirectory and A.) select the CH6GEOM.NEU file, and B.) Press Open.



- The FEMAP Neutral Read Options Dialog Box is displayed, providing several options for how to treat the incoming data. Accept the defaults and press OK to continue.



To modify the view to view the three dimensional nature of the imported geometry, use the Dynamic Rotate button on the FEMAP toolbar.

## Rotate the View

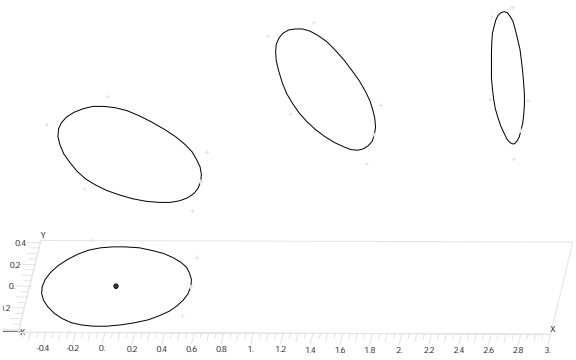
- 1 Press the Dynamic Display button on the toolbar.



- 2 FEMAP displays the Dynamic Display dialog box at the bottom of the screen.



- 3 To dynamically rotate your model, move the cursor inside the graphics window, and then press and drag it left to right and up and down. This will dynamically rotate the model. By pressing and holding the Shift key, and pressing and dragging the mouse up and down, you can scale the view dynamically. Using the Ctrl key in combination with pressing the left mouse button and dragging, you can dynamically pan the view. When you get the model in an orientation similar to the one shown, press OK or the Return to key to leave Dynamic Display.

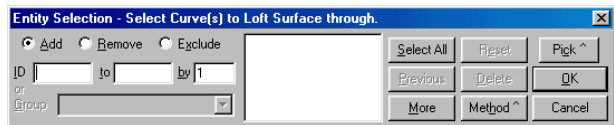


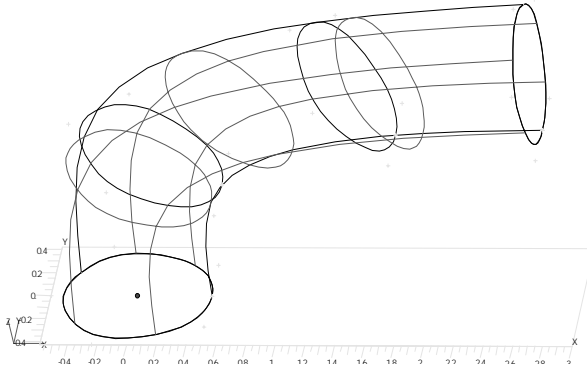
We will now create the surfaces that represent the outside of this part.

## Create Surfaces

The surfaces that represent the outside of this handle will be created with the Geometry - Surface - Aligned Curves function. Geometry - Surface - Aligned Curves takes curves aligned in the same parametric direction and construct a lofted surface between them.

- 1 Select **GEOMETRY - SURFACE - ALIGNED CURVES** from the FEMAP Menu. FEMAP prompts you for the curves. Select All and press OK to loft the surface.

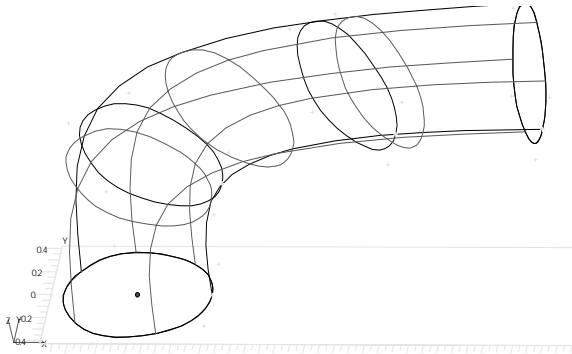




The surfaces are now defined. It is also necessary to mesh the end of the part so that a complete enclosing triangular surface mesh can be generated. We will do this with two FEMAP Boundaries. For each of the closed curve ends, select **GEOMETRY - BOUNDARY SURFACE** from the FEMAP menu, select the curves at one end, repeat the command and select the other.

The geometry portion of this example is now complete.

## 6.2 Materials, Properties and Meshing



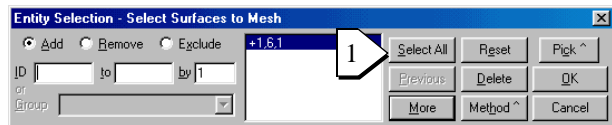
We can now mesh the surfaces and boundaries just created. This part will be solid meshed with the tetrahedral automatic mesher.

The first step in a solid tetrahedral mesh is to create a triangular surface mesh that completely encloses the solid. This triangular surface mesh is used as the starting point for the volume mesher which fills the interior of the

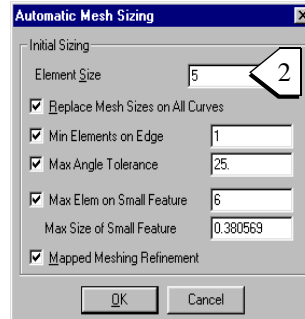
triangular surface mesh with elements. In this example the solid mesh command automatically creates the surface mesh needed for the solid mesh and deletes it when the solid mesh is complete.

### Mesh the Surfaces

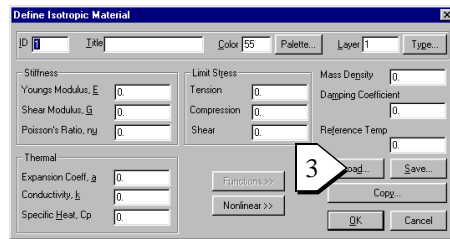
- 1 Choose **MESH - GEOMETRY - SOLIDS FROM SURFACE** from the FEMAP menu. Press Select All to mesh all the surfaces in this model. Press OK to continue.



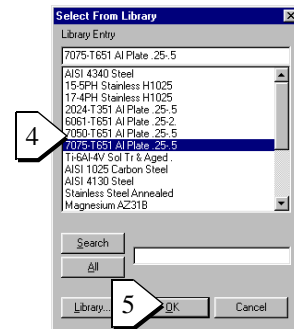
- 2 FEMAP now displays the Automatic Mesh Sizing Dialog Box. A.) Change the Element Size to 0.4 and press OK.



- 3 Since no material has been created FEMAP prompts you to make one. You can enter in values or press the Load button to bring up the material library.



- 4 The material library shipped with FEMAP contains material properties using English units (lb, ft, sec). You can create your own materials and store them in this library or create your own library. For this example select a material from this library and press OK.

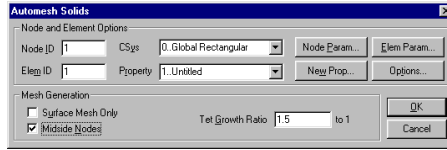


### Note:

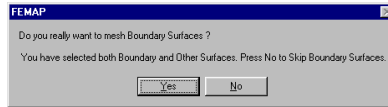
Remember, there are no units in FEMAP. All dimensions must be kept consistent with the unit system you use to define your material properties. Always make certain your units are consistent from the beginning of model creation because it is extremely difficult to fix inconsistencies in units once the model is built.

- 5 Press OK in the define material dialog box when the properties have been loaded.

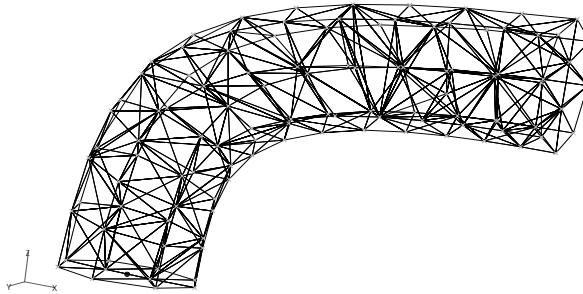
- The automesh solids dialog box appears. Leave the values as the defaults and press OK.



- FEMAP asks you if it is OK to mesh boundary surfaces. In this case they are part of the solid so press Yes.



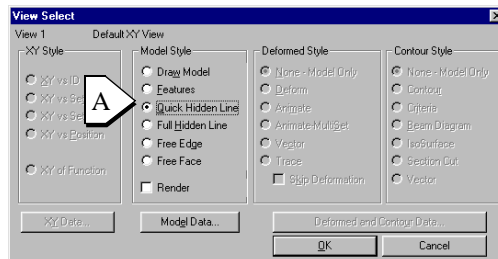
Press Ctrl-Q to bring up the FEMAP Quick View Options Dialog Box. Select Geometry Off, Labels Off and press Done.



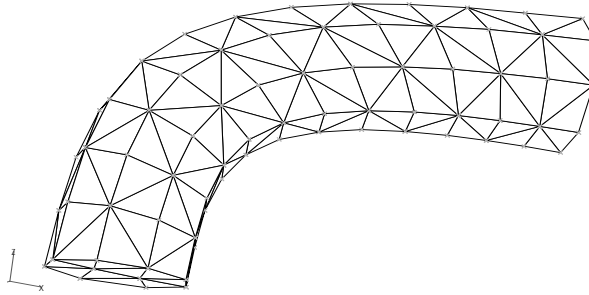
The model is shown in wire-frame mode. As your models become more complicated, it is often necessary to view them with their hidden lines removed. To do so:

## Switch to Hidden Line Mode

- Choose View - Select from the FEMAP menu (shortcut keys Ctrl-S or F5). Press OK to continue.

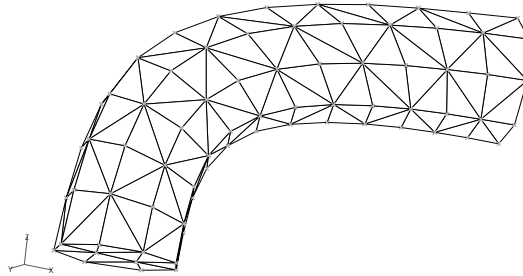


FEMAP now displays your model sorted back to front, effectively hiding elements that are behind other elements.

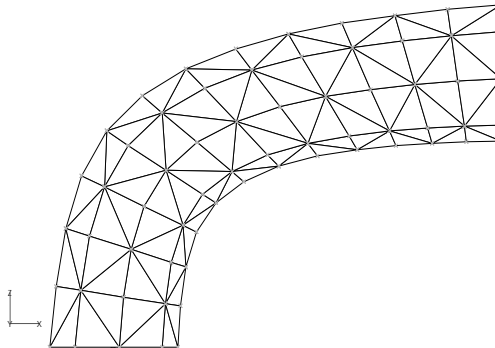


## 6.3 Loads and Constraints

Use **VIEW - SELECT** (F5 or Ctrl-S) and change the Model Style to Draw Model.



Use **VIEW - ROTATE** (F8 or Ctrl-R) and rotate the view to ZX Front.



### ***Constraining the Model***

Since this model uses tetrahedral elements, the individual nodes only use translational degrees of freedom. We will first fix the base by constraining all three translational degrees of freedom, and then constrain the right side with an X-Symmetry boundary condition.

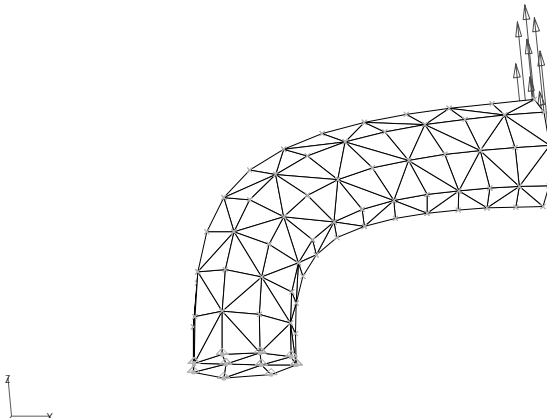
1. Select **MODEL - CONSTRAINT - NODAL** from the FEMAP menu.
2. Box pick the nodes along the bottom side of the handle. Press OK.
3. Constrain the TX, TY and TZ degrees of freedom. Press OK.
4. Box pick the nodes along the right side of the handle. Press OK.
5. Press the X-Symmetry button, or just constrain the TX degree of freedom. Press OK
6. Press Cancel or use the ESC key to end the Create - Constraint - Nodal command.

### ***Loading the Model***

A load will be applied to the center of this handle in the plane of symmetry. Since we are applying a load in the plane of symmetry, we are effectively applying twice the load that is input. To apply the load:

1. Select **MODEL - LOAD - ON SURFACE** from the FEMAP menu.
2. Pick the edge surface the right side of the handle (the plane of symmetry). Press OK.
3. We want to apply a total load of 5000 pounds. Select a load type of Force. Enter a value of 5000 in the Z direction. This will apply a total force of 5000 pounds on the surface. The load will be expanded to the nodes during translation. Press OK to create the loads. Press Cancel to end input of nodal loads.

This model is now ready for analysis. As usual, you can save it, or use the model provided.





# Groups, Layers, Viewing and PostProcessing

This chapter is a break in the model creation format of the manual and instead deals with many of the model manipulation commands. These commands are useful in any FEMAP model so review them carefully.

In addition to the numerous Pre- and Post-Processing options provided by FEMAP to make the generation and interpretation of FEA easier, FEMAP also provides a wide array of viewing options that play a key role in increasing your FEA productivity.

The options and methods for controlling how your model is displayed on screen can be divided into two broad categories:

## **View Select and View Options:**

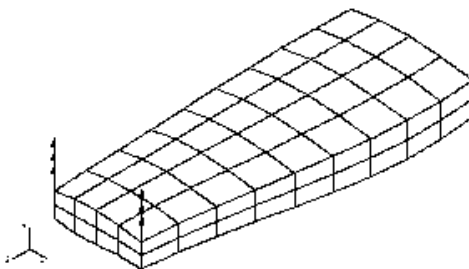
View Select controls the top level display options. With View - Select you can control whether your model is displayed in hidden line or plain wire-frame mode, turn on and off stress contours, animations and deformed plots, etc. View - Options provides the detailed control over how entities are displayed, i.e. what color elements are drawn with, whether or not labels for nodes are displayed, whether or not perspective is turned on, etc. View - Options also provides extensive control over the display of post-processing options that is more fully described later in this chapter.

## **Groups and Layers:**

By using groups and layers, you can segment your model into smaller, more manageable, discrete pieces. These pieces can then be used to minimize the amount of information presented in the graphics windows or in printed reports by specifying which group will be seen or which group will be used to create a report. Groups and layers also make it easier to manipulate, update, and apply loads to your model.

Throughout this chapter the following model will be used to demonstrate the grouping, layer, and viewing capabilities of FEMAP.

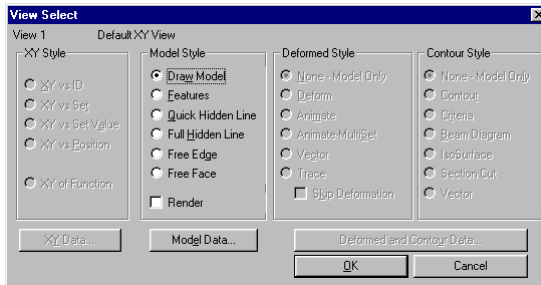
If you wish to follow the upcoming examples, load the model now. Start FEMAP, move to the /examples subdirectory in the directory where the examples for this manual were installed, and open CH7WING.MOD.



The model represents an extremely simplified version of an aircraft wing. The model is constrained at its base, and loaded vertically at the front and back spars with nodal loads. It contains multiple materials, properties, and element types. We will use the various grouping, layering and view options available in FEMAP to better understand the model and its analysis results.

## 7.1 Working with View Select and View Options

### View Select



View - Select provides top level control of how your model is displayed.

We will concentrate on the Model Style and Model Data options in this section. The XY Style, Deformed Style, and Contour Style options will be discussed later in the Post-Processing section.

FEMAP provides numerous styles in which you can display your model. Each style provides certain benefits. Choice of the best style depends upon what you need to accomplish. The following table describes all of the styles, their advantages and disadvantages:

**Table 1:**

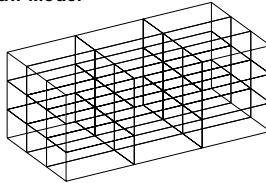
Style	Description	Advantages	Disadvantages
Draw Model	Simply displays all entities.	Fast. Everything visible. Usually best "working mode". Good for screen selection.	Complex 3D models can be hard to visualize. Entities drawn on top of each other may make it difficult to locate a particular detail.
Features	Draws all entities. Lines of the same color, which overlap, alternately draw and erase themselves.	Fast. Results in a plot which only shows color boundaries. With proper color assignments can show property or material boundaries.	Not usually appropriate for screen selection. Resulting display depends on your color choices.
Quick Hidden Line	Sorts all elements, then displays from the back of view. Only shows entities which are visible - hidden lines are removed.	Good for final display and visualization of complex 3D models. Can be helpful for screen selection in complicated models.	Not usually best for picking - many entities are not visible. Does not properly remove hidden lines for some elements (see Full Hidden Line).
Full Hidden Line	Same as Quick Hidden Line, but does additional checking to properly remove all hidden lines.	Same as Quick Hidden Line.	Same as Quick Hidden Line. Slower than quick hidden line due to the additional checking required.

Table 1:

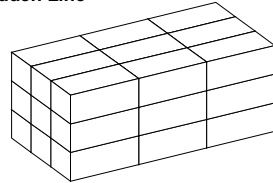
Style	Description	Advantages	Disadvantages
Free Edge	Finds and displays all element edges which do not join to another element.	Can quickly point out holes or disconnections in your model.	Not appropriate as a working mode. Really just intended for checking your model.
Free Face	Finds and displays all element faces which do not join to another element.	Can quickly point out disconnections between solid elements. Reduces complexity of solid model plots. Can help to find duplicate plate elements.	Usually not used for a working mode. Intended for checking model.
Render	Turns on OpenGL drawing mode. It is either on or off and another view style must still be selected.	Faster graphics. Dynamic rotations of shaded solid models. Extra postprocessing features for solids.	Not all view options/ styles available in this mode.

The pictures, below, show examples of the various model styles.

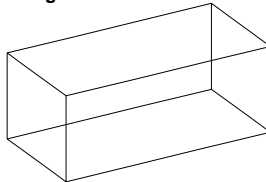
Draw Model



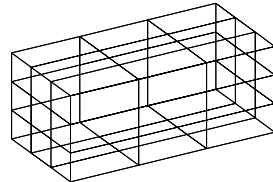
Hidden Line



Free Edge



Free Face



Although the hidden line removal options do require substantial calculations, and are therefore somewhat slower, they can often be the best approach to understanding a complex model. This is especially true for 3D models. After you make the first hidden line display, FEMAP retains a display list of the sorted information. This dramatically speeds up redrawing hidden line views. Refer to the View Redraw and View Regenerate commands for more information.



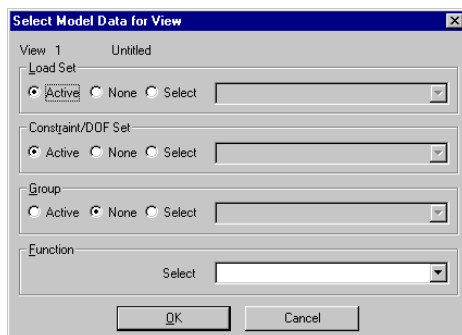
For Solid Element Models, you can also use the Free Face option to simulate a hidden line view. In fact, you can even use this mode to show hidden lines in a different line style (like dashed), instead of removing them. To remove backfaces, use the Fill, Backfaces and Hidden Option, in the View Options command, and chose one of the “Skip” methods. Choose the “Show All Faces” method to show hidden lines as a different color/style,

then go to the Free Edge and Face option and set the Free Edge Color to “Use View Color”. Finally, choose the color and linestyle that you want to use.

### Selecting Data for a Model Style

You can control what portions of your model are displayed by any of the model styles by pressing the Model Data command button in the View - Select Dialog Box. The Select Model Data for View dialog box will then be displayed.

Here you can choose the Load Set, Constraint Set and Group which will be displayed in the view. By default, whatever load and constraint set you activate will be displayed. You can however eliminate loads and/or constraints by choosing the None options, or you can Select a particular set for display whether or not it is active. If you choose the Select option, you must specify an existing set in the appropriate drop-down list.



By default, your entire model will always be displayed. Since the Group option is set to None, activating a Group will not change the display. This enables you to activate a Group and then graphically select entities, from your entire model, into the Group. If you want to display only a portion of your model, switch this option to either Active or Select. Then only the entities which are in the appropriate Group will be displayed.

The final section of this dialog box, Function, is used to select the function that will be displayed when you choose the XY of Function display style. Even though this is obviously an XY style of plot, you must choose the function to be displayed from this dialog because it is a display of model information - not postprocessing information like all other XY plotting styles.

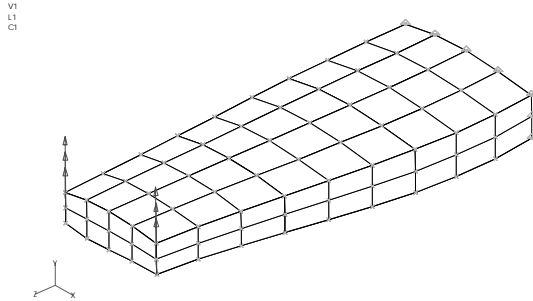
### Example Using View - Select

Instead of walking you through View - Select option by option, we encourage you to use CH7WING.MOD and experiment with how the various option affect the model display. To do so, go to View - Select in the FEMAP menu (shortcut keys Ctrl-S or F5, and change the model between the various Model Style and Model Data options.

### View Options

The View - Options command in FEMAP provides detailed control of the display of all entities in the FEMAP graphics window(s). Each view in FEMAP is independent, and the View - Select and View - Options changes will affect only that view, unless you select the All Views option. The quickest way to assess how View - Options can help you tailor the display of a finite element model is to experiment with the various settings on our sample wing model.

If you have not already done so, or if you have and already significantly changed its display, open CH7WING.MOD in the /examples subdirectory of the example problems for this manual. Use **VIEW - SELECT**, and change the Model Style to Quick Hidden Line:



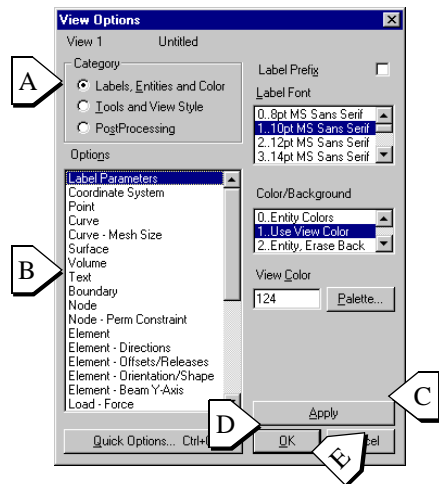
This model will now be utilized to demonstrate some of the available choices under View Options.

### Using View - Options

In the following walk-through we will adjust several of the options that control the details of how a finite element model is displayed on screen.

Select **VIEW - OPTIONS** from the FEMAP menu. The View - Options dialog box is displayed. Some highlights:

A.) Category - View - Options is divided into three distinct categories. In this portion of the manual we will look at the first two: (1) Labels, Entities and Colors which controls how the individual geometric and analysis entities will be displayed, and (2) Tools and View Style which provides control over some broader aspects of how a model is displayed. As you change between the three categories, you will see the list of available options change:



B.) The individual options themselves are displayed in the Options Box. Once you have selected a broad category in A, you then move to B to choose the particular view option that you wish to adjust.

As you move from option to option you will notice the controls on the right side of the dialog box will change to reflect what aspects of the current view option are available for modification.

C.) The Apply button allows you to preview the changes you make to the View - Options before returning to FEMAP. As you adjust various options in the View - Options dialog box you can press Apply at any time to see what effect they have.

D.) The OK button will update the display based on your changes to the various options and return you to FEMAP.

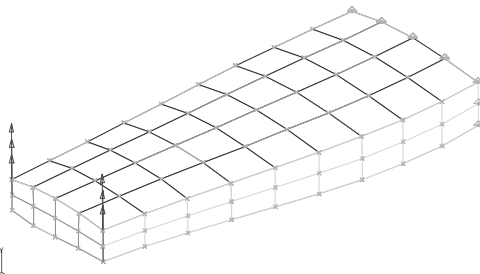
E.) Cancel will return you to FEMAP without any changes you have made to the View - Options.

- 2 We will first change the View - Options to display the elements based on their property color. Make sure that the Category is set to Labels, Entities and Colors.
  - A.) Change the Option to Element.
  - B.) Change the Color Mode to 3..Property Colors.
  - C.) Press Apply to preview the changes.

**Note:**

Initially, elements were displayed with the color model set to "Entity Color", therefore, each element is displayed based on its own color which is held inside FEMAP with the element itself. By changing to "Property Colors", FEMAP draws each element based on the color of that element's property. This capability is very useful for visualizing how the various properties in your model are distributed, and to guarantee that elements reference the appropriate properties.

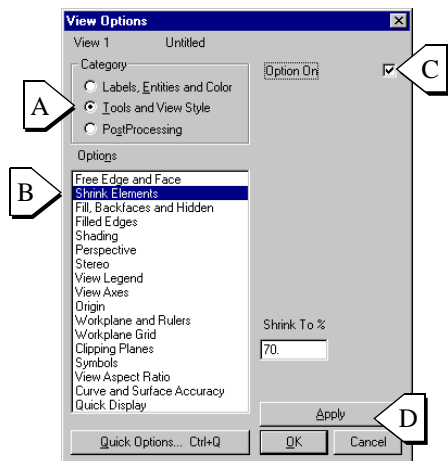
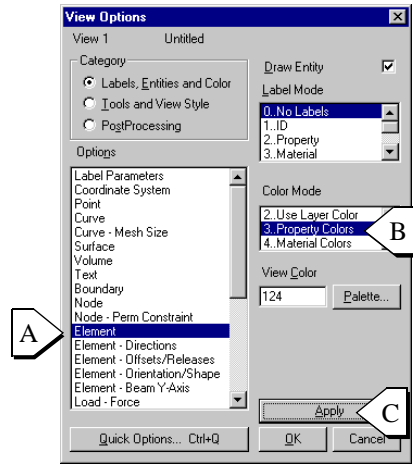
Now that multiple colors are being used to display the elements, areas where element edges overlap will be the color of the last element drawn. A View - Option that can make the elements clearer is Shrink Elements.



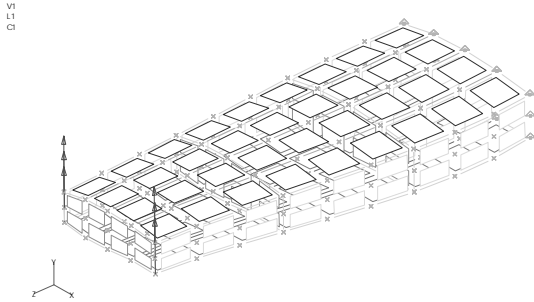
- 3 A.) Change the Category to Tools and View Style.
  - B.) Change the Option to Shrink Elements.
  - C.) In the upper right corner you will notice a "Option On". Toggle it to the checked position to turn on Element Shrink.
  - D.) Press Apply to see the effect.

**Note:**

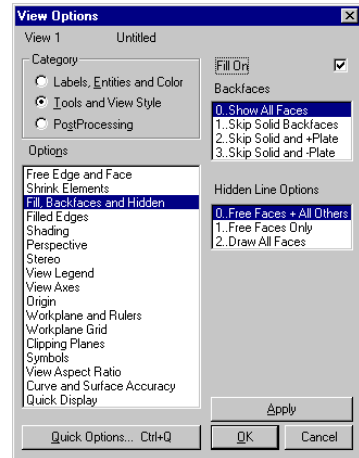
This toggle control will be found on almost every option in view options. A shortcut for toggling is to double-click on the Option itself in B. If you double-click on any "option" in the Options list, it will select it and reverse the state of this toggle.



The model is now displayed with the element shrink turned on. Shrinking the elements makes it easier to see the now colored property boundaries since elements no longer overlap. In addition, the line elements representing the caps of the spars and ribs are now visible between adjacent plates. Elements are drawn by default by their outline only, to view the elements filled in, we will turn the fill on.



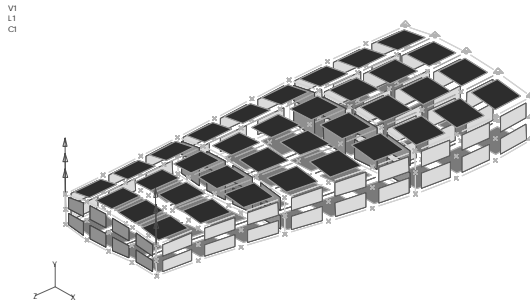
- 4 To turn the fill on, again in the Tools and View Style category, change:
  - A.) the Option to Fill, Backfaces and Hidden.
  - B.) Toggle the Fill On in the upper right hand corner, and
  - C.) Press Apply to preview the changes.



### Note:

The View Option Fill, Backfaces and Hidden actually combines three separate view options. In this command, the “Fill” part of the title refers to the toggling fill on or off for elements and surfaces, the “Backfaces” part is controlled by the second box, and controls which faces of a model are considered in a hidden line plot, and the “Hidden” part refers to the third box which provides some additional Hidden Line Options. Several of the individual view options are grouped this way to reduce the overall complexity of this dialog box.

The model is now displayed with the elements “filled” with their property color.



As you can see from the View - Options Dialog Box there is an enormous amount of control over how your model is displayed. Describing how each option affects the display of your model is beyond the scope of this manual. Detailed descriptions of each option can be found in the FEMAP Command Reference, and in the on-line help. At this point we encourage you to try the various options and see how they affect this sample model.

## 7.2 Groups and Layers Overview

Some main points about groups and layers that will help you understand them better:

- Each entity in FEMAP can have only one layer reference.
- An entity can be in more than one group.
- Only one group can be displayed, by itself at one time.
- Any combination of layers from none to all can be displayed at any time.
- A model can have only one “active” group at a time.
- FEMAP graphics windows can use the entities in a group to display one of the following:
  - Entities from the active group.
  - Entities from a specified group.
  - All Entities, i.e. no group.

Groups are designed to mimic how FEA models were numbered and arranged when there were built by hand. For example, in the aircraft industry, a model of a complete aircraft would be very carefully numbered. All the nodes and elements at a frame at a particular location along the fuselage would be numbered in such a way to clearly identify them as belonging to that frame. FEMAP grouping makes it very easy to isolate portions of a finite element model that are numbered in such a manner.

Layers are designed similar to layering in most CAD systems. The name layer comes from the clear sheet of paper analogy for CAD layering, where all the entities associated with a given layer would be drawn on a clear sheet of paper, and only the “active” clear sheets being overlaid to produce a visual image.

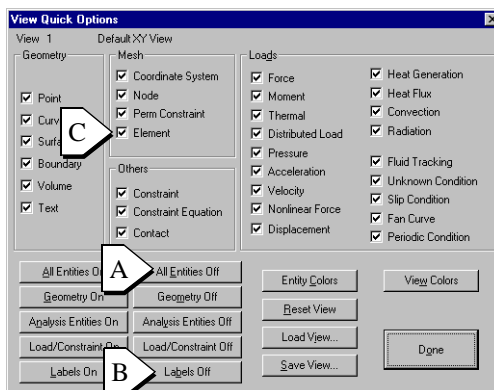
### Clean Up the View

Again, open CH7WING.MOD in the /examples subdirectory where the examples for this manual were installed.

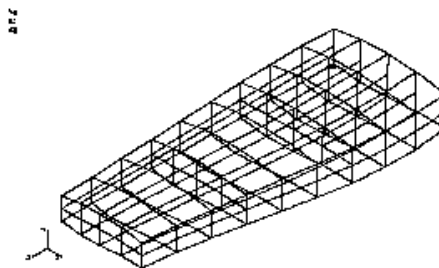


The wing model will come up with all entities on, and all labels on, in wire-frame mode. To clean up the display:

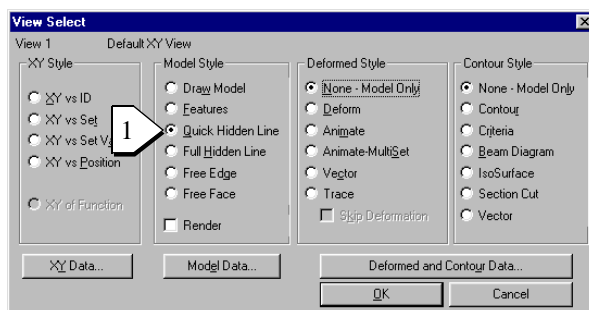
- 1 Press **CTRL-Q** to access the FEMAP Quick View Options. First, turn off everything by  
A.) pressing All Entities Off.  
B.) Press Labels Off to turn off all labeling.  
C.) Toggle the Element selection box to turn only elements on.  
Press Done to return to FEMAP.



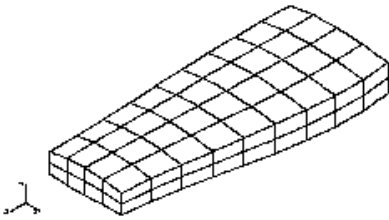
The model will now look like the picture shown here, a plain wire-frame representation of all the elements that comprise this model. The elements are drawn along their edges only, with no fill in the middle, which is why you can see through the closer elements to the ones behind. To achieve a hidden line view, a hidden line being a view that would show only those elements that you would see if this were a real combination of plates and line in space, we will use the FEMAP View - Select command.



- 1 Choose **VIEW - SELECT** from the FEMAP menu. Instead of using the mouse to pick this command, you could also use the F5 or Ctrl-S shortcut keys to access the command. To change to a hidden line view, change the Model Style option from Draw Model (wireframe) to Quick Hidden Line. Press OK to return to FEMAP.



The model will now appear as follows:



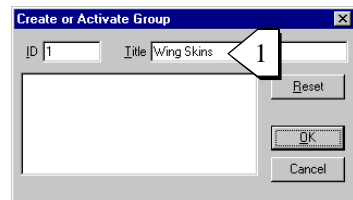
### Note:

You may have noticed both the “Quick” Hidden Line and “Full” Hidden Line options in View - Select. A Quick Hidden Line is a depth sort, where the center of the elements are sorted, back to front, based on their distance along an axis perpendicular to the screen. For most models, this will produce a true hidden line plot. The Full Hidden Line plot will first do the same depth sort of all the element being displayed, and then go through some additional checks to remedy situations where the depth sort could not produce the true hidden line plot. During model generation and construction, it is best to use the Quick Hidden Line option since it is faster, and use the Full option for final prints. In the OpenGL accelerated Render mode there is no difference between quick and full hidden line since OpenGL contains its own depth buffer.

## 7.3 Working with Groups

To jump start you into the world of grouping in FEMAP, we will use our wing test model. In this first example, we will use FEMAP’s group creation capability to create a group that represents just the upper and lower wing skins.

- 1 To create a new, empty group, select **GROUP - SET** from the FEMAP Menu. Enter a title of “Wing Skins” in the title field, and press OK to continue.



- 2 A new group is completely empty. To add all the elements that represent the top and bottom wing skins, select **GROUP - ELEMENT**

- **PROPERTY**. This com-

mand will modify the elements in the current group based on their property reference.

FEMAP displays the standard entity selection dialog box asking for properties. Just like picking elements in the graphics window, you can also pick properties. Move to the graphic windows and select one element from the top wing skin (Property 2) and one element from the bottom wing skin (Property 3). Press OK to continue.

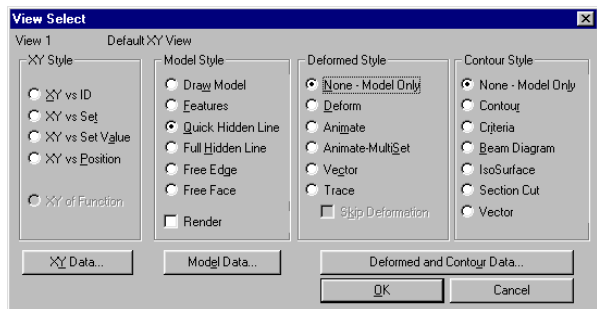


### Note:

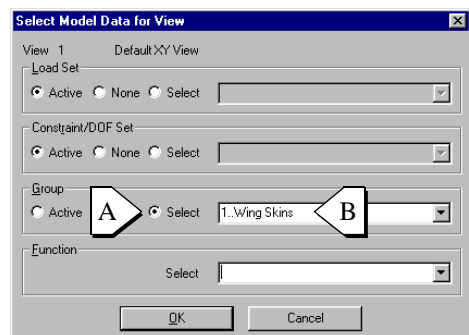
Even though we were selecting elements on screen, FEMAP extracts the property that the element selected references and fills in the selection box accordingly.

At this point we do not see anything different on screen. A group has been created, elements added to that group, but the display will still show the entire model. To cause FEMAP to display only the entities in a certain group, we will once again use the View - Select command.

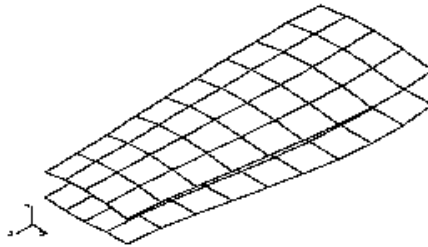
- 1 Select **VIEW - SELECT** from the FEMAP menu (or press F5 or Ctrl-S). Press the Model Data button to access control of group display.



- 2 FEMAP now displays the Select Model Data for View. In addition to controlling what group is displayed in the current view, this dialog box can also be used to set which Load Set and Constraint Set are seen. To view the group we just created, A.) toggle Group Option to Select, and then B.) select the group that we just created. Press OK to return to the View - Select dialog box. Press OK in the View - Select Box to make the changes to the view.



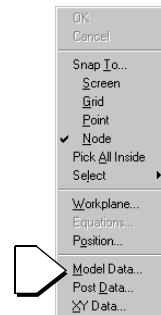
Only those elements referenced by Property 2 and Property 3 are shown.



### Hint:

To quickly get in and out of the Select Model Data for View dialog box, you can use the shortcut menu that is attached to the right mouse button. First left-click in the graphics window you want to change to make that view current, then right-click anywhere in that graphics window. The short-cut menu will appear. You can then move down and select the Model Data option and go directly to the model data dialog box.

At this point you may be asking “What is grouping good for, and why do I want to use it?” With a model as simple as this, grouping for visualization and mesh construction is not that important, but as your models grow in size (they will), grouping makes it possible to work on discrete areas of your model more efficiently. Even in a small model, grouping can be very useful, consider:



### Using a Group to Trim Down a Report

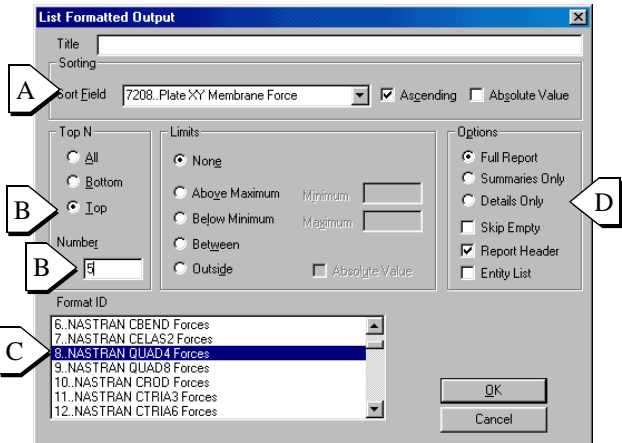
FEMAP provides numerous capabilities for creating printed reports that will be covered in a later section. We will print a quick report here to demonstrate how groups can be useful.

Say you are interested in finding the element in either the upper and lower wing skins with the maximum shear flow (FXY Force). Now that we have a group defined that includes all the wing skin elements, this will be easy.

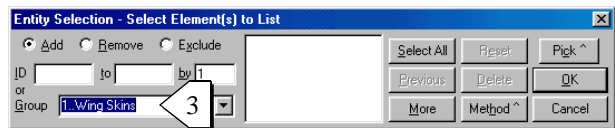
- 1 Select **LIST - OUTPUT - STANDARD** from the FEMAP menu. FEMAP first asks you which output set to use. Since this model contains only one set of output data from a single static stress run, just press OK to select Output Set 1 and continue.



- 2 Next, FEMAP displays the List Formatted Output dialog box. Fill the box in as shown, with:
- A.) the sort field specified as data vector 7208.Plate XY Membrane Force,
  - B.) that we want the Top 5 elements,
  - C.) that the report format we want to use is number 8..NASTRAN CQUAD4 Forces, and
  - D.) Details Only in the Options section to limit the information to details. Press OK to continue.



- 3 FEMAP now prompts for which elements to use. Instead of box picking or individually picking, we now have a new option that uses the group we just created. Move to the Group field, and select the Wing Skins group. Press OK to generate the report.

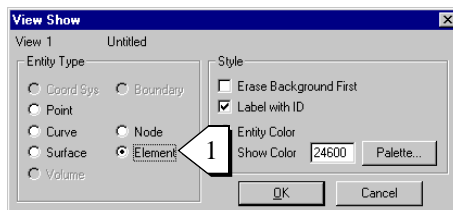


The report now appears in the FEMAP Messages and Lists Window. Press **CTRL-U** to temporarily maximize the Messages and Lists Window. You will see a report that list the plate element forces corresponding to the top five shear flow plates in the upper and lower wing skins. The report is sorted in ascending order as specified in the List Formatted Output dialog box. We can now easily see that the maximum shear flow is 62.29 #/in. and can be found in element number 120. Press **CTRL-U** again to return to the regular view style.

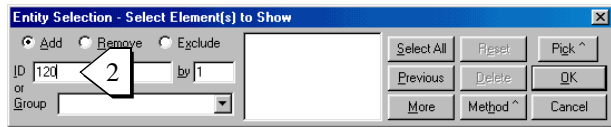
### Hint:

To quickly find element number 120, do the following:

- 1 Select **VIEW - SHOW** from the FEMAP menu. Set the Entity Type to Element and press OK to continue.



- FEMAP now displays the standard entity selection dialog box. Type 120 in the ID box. Press OK and element 120 will be temporarily highlighted on screen so you can easily identify it. View - Show can be very useful for finding entities in complicated FEA models.



### Note:

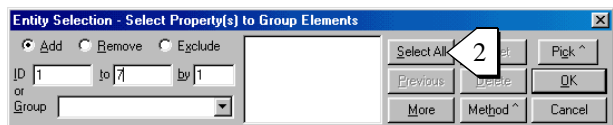
The default ID (usually the highest numbered entity) is already highlighted when you enter the entity selection dialog box. If you want to select a different entity, simply type in the number of your selection. Since the text in the box was highlighted, it will be replaced just like any highlighted text in any Windows program.

As you have just seen, a group can be used in the standard entity selection box. Any time FEMAP is prompting you to select entities, you can specify a group. FEMAP will then use the group information, and bring all of the requested entities from the group into the current selection. This is extremely useful when applying loads to a portion of your model, or when you wish to update sections of your model.

## Automatic Generation of Groups

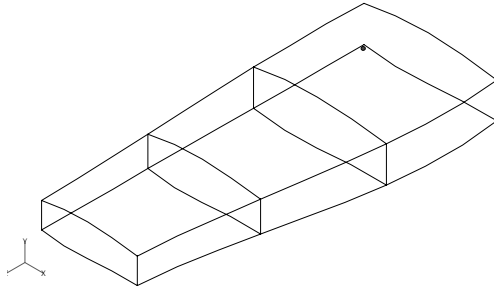
Once you become proficient in FEMAP, you will probably find yourself creating groups as you build a finite element model to keep important areas of the model together for use downstream. If you do not do this, or if you import an existing model, FEMAP has several tools for automatically grouping together portions of your model based on changes in material properties, element properties or even geometric regions. To automatically split our the wing model up into groups based on properties:

- Select **GROUP - OPERATIONS - GENERATE PROPERTY** from the FEMAP menu. Press Select All, and then OK to create groups based on distinct properties in the model.

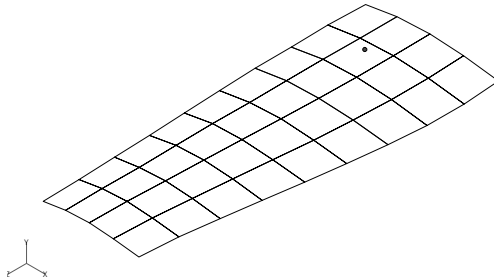


FEMAP creates seven groups of elements corresponding to the seven different properties in this model. To cycle through them, use the right-button shortcut in the graphics windows, go to Model Data, and change the selected group, press OK to redraw the display. Do this several times and you will see the various pieces that make up this model based on property.

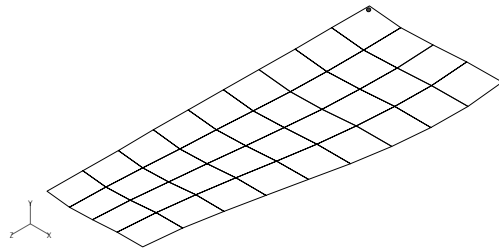
## 1 Stiffeners



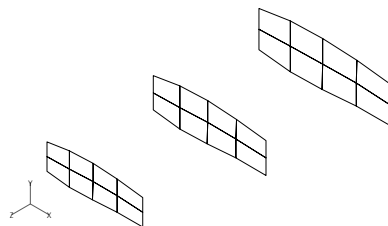
## 2 Upper Wing Skin



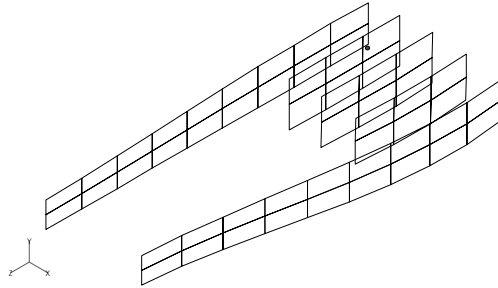
## 3 Lower Wing Skin



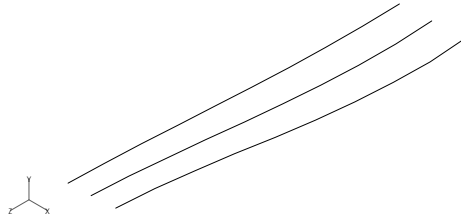
## 4 Ribs



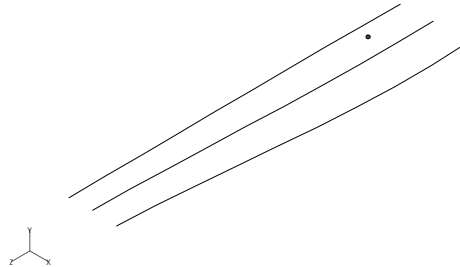
### 5 Spar Webs



### 6 Lower Stringers



### 7 Upper Stringers

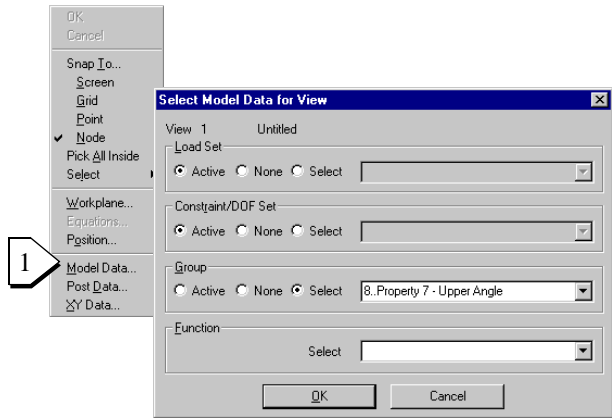




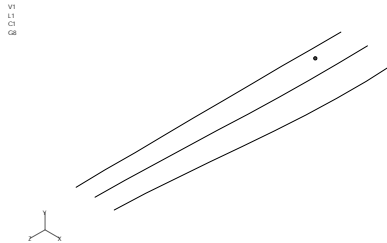
### Another Example of Using the Group Information

In this example we want to examine the axial forces in the upper stringers.

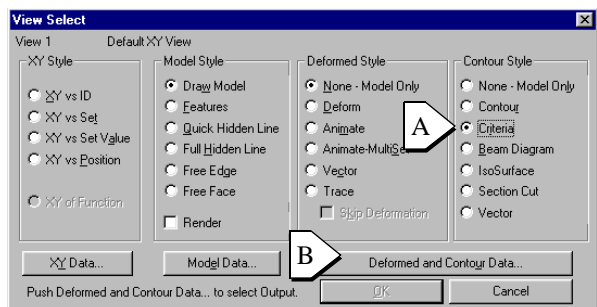
- 1 Click the right mouse button in the graphics window. The shortcut menu will appear, select the Model Data button.
- 2 FEMAP displays the Select Model Data for View dialog box. Change the selected group to the group that was created using Property 7. Press OK to continue.



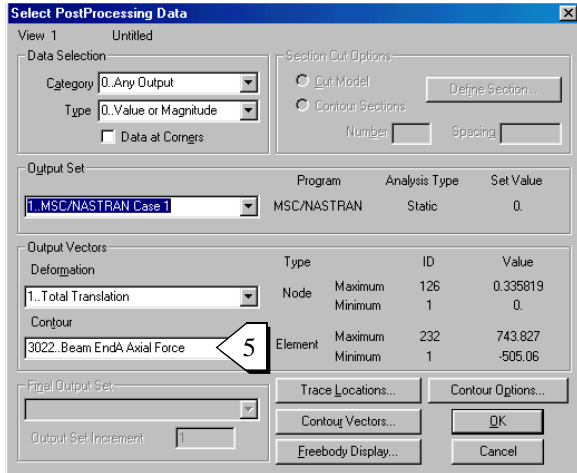
- 3 FEMAP will now display only the upper stringers.



- 4 To view the axial forces in these elements, first select **VIEW - SELECT** (F5 or Ctrl-S).  
A.) Change the Contour Style to Criteria, and then,  
B.) Press the Deformed and Contour Data button to choose what data to use in the post-processing plot.

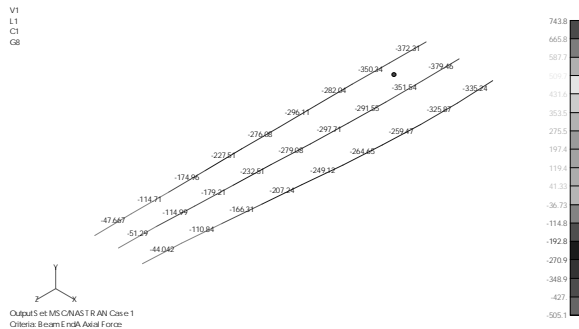


- 5 FEMAP now displays the Select PostProcessing Data dialog box. Change the Contour selection to point to Data Vector number 3022...Beam End A Axial Force. Press OK to return to View - Select, and then press OK in View - Select to initiate the plot.

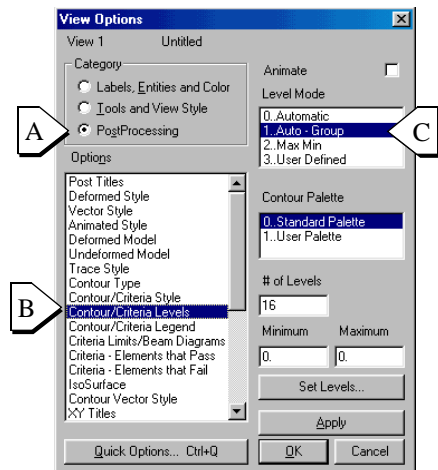


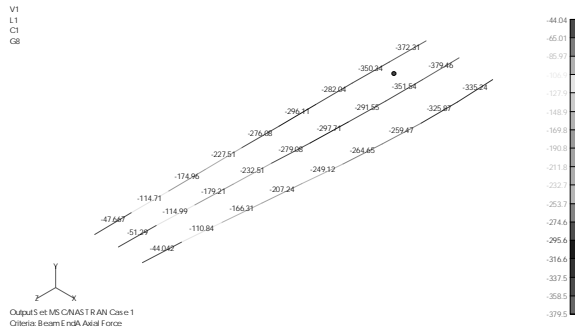
FEMAP now displays a Criteria Plot of the axial forces for the elements displayed.

You will notice that the legend on the right side of the screen does not correspond to the maximum and minimum values of the elements on screen. This is because the legend is drawn based on the max/min of the entire data vector, that is, all beam elements in this model. To adjust the legend to reflect the current group



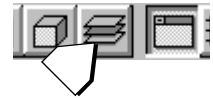
- 1 Select **VIEW - OPTIONS** (F6 or Ctrl-O) from the FEMAP menu.
- Set the Category to PostProcessing.
  - Set the Option to Contour/Criteria Legend.
  - Set the Level Mode to Auto-Group.
- Auto-Group will now automatically set the max/min values of the legend to correspond to elements in the group being displayed. Press OK to update the graphics window.





## 7.4 Working with Layers

As long as you understand the basic layering concept, that every entity in FEMAP has a reference to a layer, and that layer is either on or off, working with layers is quite easy. Control of which layers are displayed is provided in the View - Layers command. View - Layers can be accessed from the FEMAP menu, or with the View - Layers button on the toolbar.

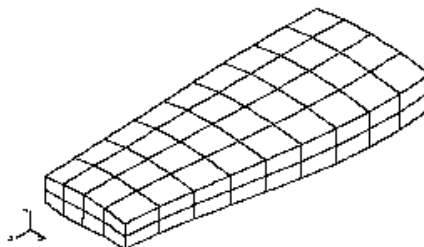


### Layer Example

If you have been working along in this chapter with CH7WING.MOD, the display still contains the Criteria Plot of the upper stringers axial forces. To reset the display, do the following:

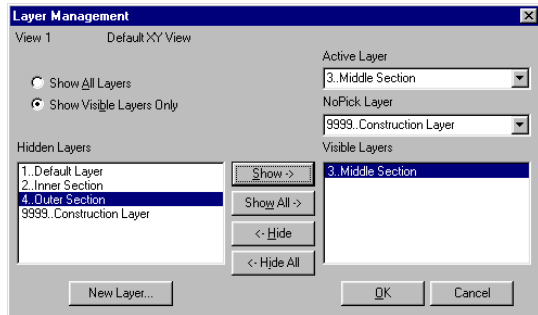
1. Go to View - Select (Ctrl-S), and turn the Contour Style to None.
2. Still in View - Select, press the Model Data button, and change the Group display option to none.
3. Press OK to return to View - Select.
4. Press OK to update the display.

The model will look like:

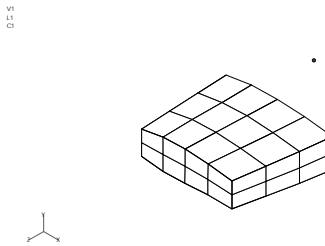


Viewing different layers:

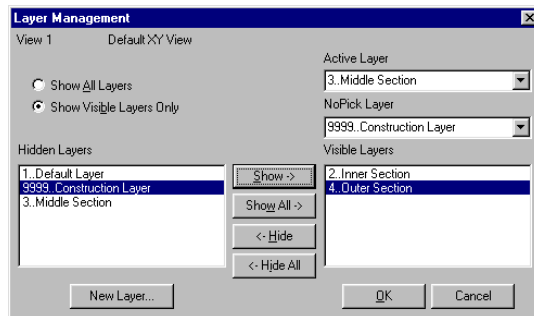
- 1 Select View - Layers from the FEMAP menu, or press the **VIEW - LAYERS** button on the toolbar. By default, the layer option is set to Show All Layers. As long as this option is set to Show All Layers, the list of Hidden and Visible layers will have no effect on the graphics window. Once the Layer Management is set to Show Visible Layers Only, only those layers in the right column, Visible Layers, will be displayed. The layers themselves can be moved from the right column to the left column and vice-versa using the four buttons in the middle of the dialog box. In addition, you can double-click on a layer and it will be transferred to the opposite column. Set up the dialog box as shown and press OK to activate the layer configuration.



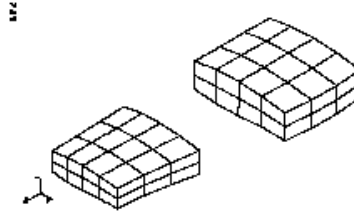
- 2 The display will be updated to:



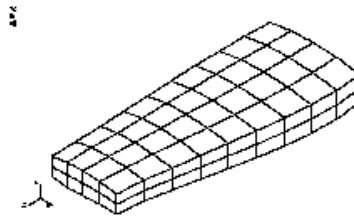
- 3 Access View - Layers again, and change the layer options as shown, press OK to update the display.



- 4 The display now shows:



- 5 Access View - Layers one more time and change the mode back to Show All Layers.



## 7.5 Combining Grouping, Layers and View Options

Between Grouping, Layers, and the wide array of View Options, there is a tremendous amount of control over how your model will be displayed on screen. However, with all these different methods of control you can run into problems. These three methods of view controls are not exclusive. They each affect how the others work.

For example, say you create a new group, add all elements of property 1, and all the nodes associated with these elements, and then use View - Select/Model Data to display just that group. What you would expect to see is the exact entities that you just put in the group. The problem arises out of the fact the layering and the options picked in View - Options also come into play. If all the nodes that were added into this group were on a layer that is not currently being displayed, they will still not be displayed. Similarly, if the nodes have been turned off in View - Options, they will not be displayed.

If you ever get into the situation where you think something should be visible on the display and it is not, first check View - Options and verify that it is on, next check View - Layer and verify that the layer associated with the missing entities is being displayed, and finally, make sure that if a group is being used for the view, that the missing entities are actually in the group. Once you become more familiar with FEMAP and the various options which control the model display, the benefits of the multiple view options methods will become apparent.

## 7.6 Printing

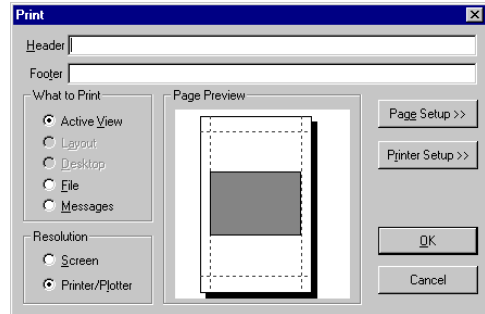
As a Windows application, FEMAP provides What You See is What You Get printing. By default, graphics sent to any printer are vector images, the actual lines, curves and polygons that comprise the graphical representation of your model on screen. As a vector image, the printer driver will break the components down into the colored (or gray scale) dots that form that actual print out. In this manner, FEMAP takes full advantage of the resolution of the output device. Traditional DOS-based FEA (and some Windows ones too), simply dump the bit-

map of the screen to the printer. By doing so you are limited to the resolution of the screen, and not that of the printer.

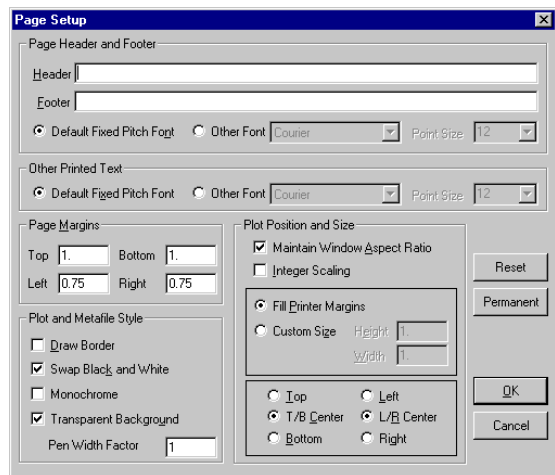
To print any graphics window, select **FILE - PRINT** from the FEMAP menu. If you have more than one graphics windows displayed, you will need to make the window that you want to print from the current graphics window, to do so, click the mouse in the window.

File - Print will display the following dialog box that provides control over how your FEMAP graphics will be printed.

Here you can quickly add a Header and Footer to describe the plot being made in more detail and adjust several aspects of the print including the Page Setup and Printer Setup. Printer Setup is most useful for changing the orientation of the plot between landscape and portrait and for controlling aspects particular to your printer.



Page setup makes it possible to control aspects of you print more closely related to FEA. The most important being in the Plot and Metafile Style. Here, you find the Swap Black and White very useful if you work in the FEMAP default black background with white elements. Without Swap Black and White any prints made would be What You See Is What You Get including the black background. With Swap Black and White, all black entities are switched to white and vice-versa, saving you both toner and making the plot easier to see.



The first step in post processing is to obtain the results. If your analysis program does not launch from FEMAP and automatically return the results, you must read them in. Use File - Translate, select the proper format and press Read Analysis Results. Select the results file for your model from the standard file selection box using the default file extension for your analysis program. Similar to Loads and Constraints, output data is also stored in sets. If you run your model with several different loading conditions or through several different analysis types, FEMAP will keep the output data from each analysis, each mode shape, or each time step in a different output set. Post-Processing can be divided into two main categories, graphical and report. Graphical post-processing can be further divided into:

1. Deformation Plots
2. Contour/Criteria Plots

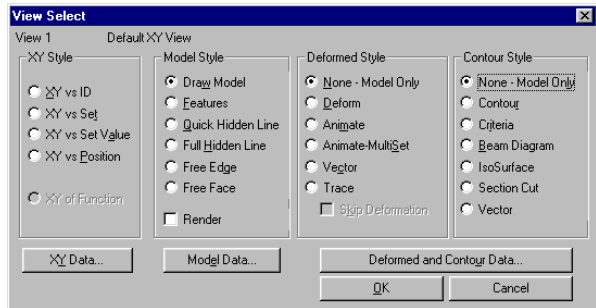
### 3. XY Plots

Deformation and Contour/Criteria plots can be combined in the same view. All model style options (such as Hidden Line) are available for deformed and contour styles.

Report based post-processing is fairly straight forward, providing text output of results data in a variety of formats, printing options, and sorting options.

## 7.7 Graphical Post-Processing

The first step in post-processing is to define the type of plot desired, and the data to be used in the post-processing display. The main control for how your model is displayed including what post-processing options are being used is the View - Select menu option.



From View Select you can invoke five different types of deformed style plots:

1. Deformed - Show a plot of the deformed shape.
2. Animate - Animate the deformed shape.
3. Animate Multi-Set - Perform animation across several sets. Good for transient, nonlinear and frequency response analyses.
4. Vector - Show vectors representing direction and magnitude of output.
5. Trace - Similar to Multi-Set Animation except displays trace lines connecting historical positions of nodes.

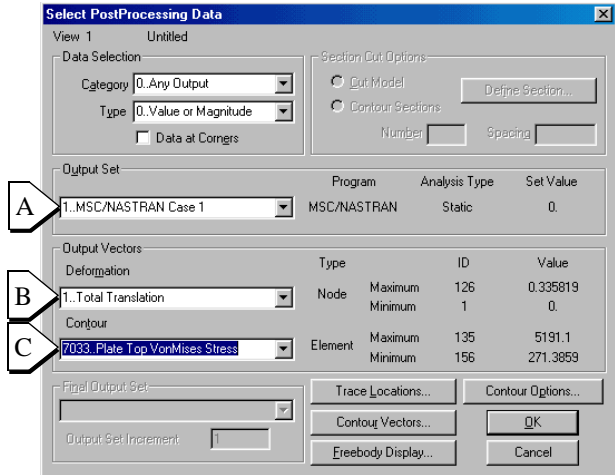
For Multi-Set Animation and Trace plots, you may also decide to only animate the contours by selecting the Skip Deformations plot. This can be extremely useful for heat transfer and similar types of analyses.

From View Select you can invoke six general contour style options:

1. Contour - Provides smooth representation of data.
2. Criteria - Elemental values displayed at centroid of element.
3. Beam Diagram - Similar to 3-D shear and bending moment diagrams. Display results along the length of Line Elements.
4. IsoSurface - Provides interior surfaces of constant values in solid models.
5. Section Cut - Shows contours through any planar cut of a solid model.
6. Vector - Vectors at centroids of elements.

## Selecting the Data to use for Post-Processing

Control over what data is used in deformed or contour plots is provided by the Select PostProcessing Data dialog box. This dialog box is accessed through the View - Select command or the shortcut (right mouse) menu as Post Data. It allows you to control the output set and output vectors shown with the deformed and contour plots. To choose what data is used in the display, simply choose A.) the output set, B.) the data vector to use for deformation, and C.) the data vector to use for contouring. You can limit the category and type of output you see in the drop down lists with the data selection area. If you are animating a multi-set you can choose the final output set to animate as well. Section cuts are defined with the standard plane definition dialog box and are accessed here by pressing the Define Section button.

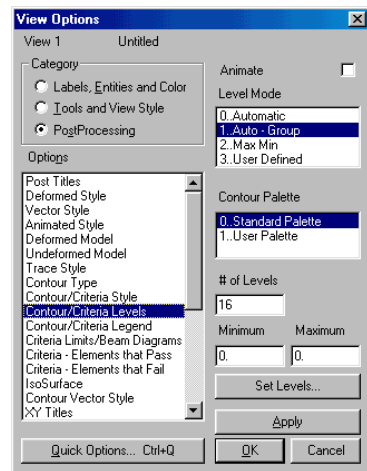


## Specifying Detailed Post-Processing Display Options

Options for controlling the detailed aspects of post-processing can be found in the FEMAP View Options command. Each graphics window can have its view options modified independent of other views. The number and depth of the various view options is such that a full discussion of each is not possible in this manual. Please refer to the full FEMAP Commands manual for a detailed description of what each option does, and how to use it. You are encouraged to go to View - Options, and set the Category to PostProcessing and review the various options. A few of the more important View Options are explained more fully below.

### Deformed Style

For all deformed styles, FEMAP uses these settings to determine the on-screen scale of the deformations. If the “% of Model” option is checked, FEMAP will scale all deformations so that the largest one is equivalent to the percentage of the model size you specify as “Scale %”. If “% of Model” is not checked, FEMAP will deform your model by the amount of the actual deformations.



### Hint:

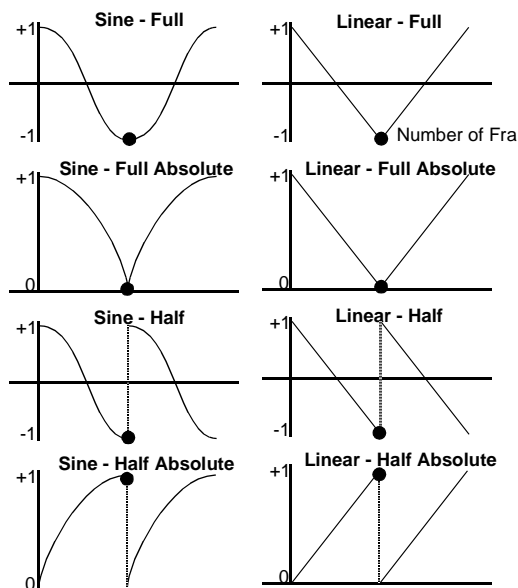
For large displacement analysis, the Deformed Style should almost always be on Actual to obtain an accurate representation of the deformations.



## Animated Style

Many aspects of animation are controlled by the settings for this option. If you choose Single Step, the view will calculate all of the animation frames and then wait. The animation will not begin until you choose the View-Advanced Post-Animation command and press Start. Shape controls the deformations in the frames that are calculated. Full cycle shapes smoothly return to their starting position, half cycle shapes jump back.

By selecting the number of frames in the animation, you control both the animation quality and speed. More frames take longer to calculate, and produce a slower, but smoother animation. Fewer frames are desirable if you want a quick look, or fast animation. If you are using the “Animate” setting, for the Contour/Criteria Levels option, best results are obtained with a larger number of frames.



The Delay factor specifies the initial speed of the animation. This can be varied using the View-Advanced Post-Animation command. Larger numbers result in slower animations.

### Hint:

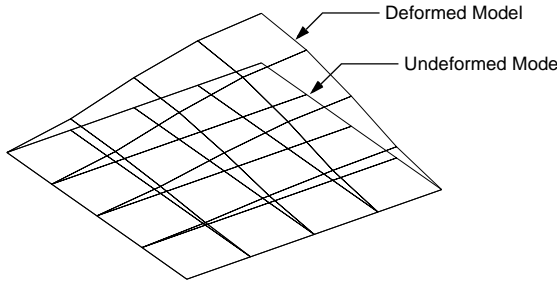
Here are a few suggestions that can help when you are doing animations:

- FEMAP retains all of the frames that you calculate in memory. You can specify a very large number of frames but you must have enough memory to hold those images.
- You can simultaneously animate multiple windows, even at different speeds, but your computer and graphics adapter need to be fairly fast. It takes the combination of a fast computer and a good graphics adapter to adequately handle animation.

If animations are not as fast as you would like, check the following:

- Make sure you are not running other applications in the background on your computer.
- Reduce the number of frames.
- Reduce the size of your graphics window. This may be the biggest savings - although at a price. It can dramatically reduce the amount of data needed for an animation, and hence increase the speed.
- Once an animation has been created, you can control it with the View Animation command.

## Deformed Model



Controls the colors that will be used for a deformed style display. Refer to Undeformed Model for additional information.

## Undeformed Model

Allows you to display your undeformed model, along with a deformed or animating style model. This option should not

be turned on for filled or hidden line view styles. If you do, the deformed and undeformed models will obscure each other.

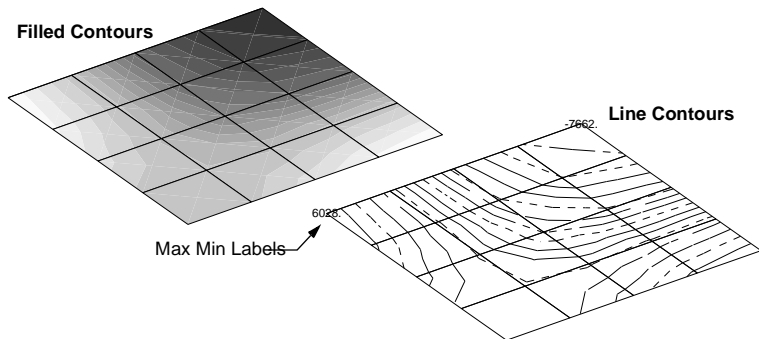
## Contour Type

These options enable you to pick between elemental and nodal contouring. Nodal contouring typically provides a smoother pattern, but can misrepresent values at geometric and material breaks. Elemental contouring provides a more accurate representation in these circumstances, and also provides the capability to plot top and bottom stresses on the same graph (with an additional solid value if solids exist in the model).

You can also choose to have continuous contours or level contours in Render mode.

## Contour/Criteria Style

Allows you to choose whether solid/filled contours will be drawn or just the contour lines. The same setting also applies to filled or unfilled elements for criteria displays. Refer to the Fill Element and Fill-Edges options (earlier in this section) for more ways to customize the appearance of contour and criteria plots.



If you select the “Max Min” labelling option, the two locations with the maximum and minimum output values will be labelled. ID labelling is not used for filled contours. For line contours, the lines are labelled with letters that correspond to those in the Contour Legend. Label Freq controls how many of the lines are labeled. If Label Freq is 5, every fifth contour line will be labeled.

The Data Conversion options control how FEMAP will calculate the Nodal data that is required for contours when you select an Elemental output vector. By default, all elemental data is averaged at the nodes. If you would rather use the maximum values, choose Maximum Value. If you have recovered or calculated elemental corner output, but do not want it to be considered in the contour, choose one of the Skip Corner options.

Contour/Criteria Levels

Specifies the number of contour levels that will be displayed. FEMAP supports up to 255 levels. This option is also used to select the output values where contours will be calculated, and the contour colors.

If you choose the Automatic or Auto-Group Level Modes, FEMAP will determine the maximum and minimum contour values (and the intermediate ones) from the maximum and minimum output values in the output vector you select. Automatic considers data from the entire output vector. Auto-Group is identical, unless you have selected a group. If a group is selected, the maximum and minimum output values of only the entities in the group will be used.

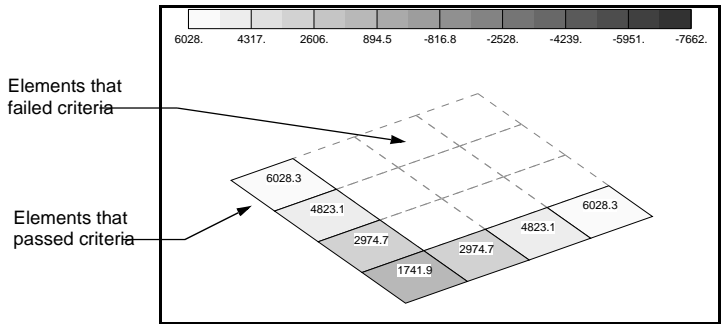
Criteria Limits/Beam Diagrams

Although criteria displays can be used simply as an alternative to contours, where each element is colored based on its output value, their primary purpose is to limit the display based on a selected criteria. This option selects the criteria. You select the type of criteria from the Limits Mode list, and then specify the appropriate values in Minimum and Maximum. The following table lists the available modes and their uses:

Table 2:

Limits Mode	Min	Max	Result
No Limits	-	-	No Criteria. All Elements Pass.
Above Maximum	-	Yes	Elements with output values greater than Maximum Pass.
Below Minimum	Yes	-	Elements with output values less than Minimum Pass.
Between	Yes	Yes	Elements with output values between Minimum and Maximum pass.
Outside	Yes	Yes	Elements with output values less than Minimum or greater than Maximum pass.

If you choose Abs Value, the absolute value of the output data is compared to your selected criteria. The “Criteria - Elements that Pass”, and “Criteria - Elements that Fail”, options control how elements that pass or fail the criteria will be displayed.



Beam Diagrams are also controlled through this option. The Default Direction option sets the elemental or global plane where the beam diagram will be drawn. FEMAP always draws the diagram in the plane that you choose, even if the output is actually based on forces/stresses/... in a different plane. The FEMAP translators should automatically setup the proper information in your model to draw the correct Beam Diagram as you read the output from one of the supported programs. If you create output through some other means, or if sign conventions change in the analysis programs, the “RevB” Default Directions can be used. If you see a

Beam Diagram where End A and End B have reversed signs, when they should be of the same sign, choose one of these options - otherwise use the regular options. The Beam Diagram Color sets the color that will be drawn around the outer edges, and between elements along the diagram.

### Criteria - Elements that Pass and Elements that Fail

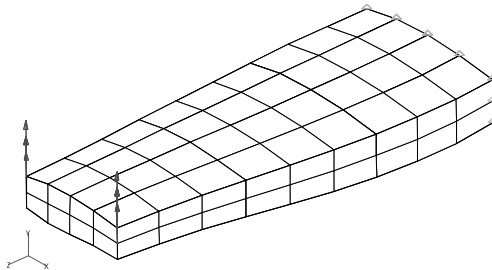
Are both used for criteria style displays. The settings for the first option are used for all elements that pass your selected criteria, or for all elements when you do not specify any criteria. The second option settings are used only for elements that fail the criteria. You can skip displaying either category of elements by turning off the appropriate option. If you select Output Value labeling, the output values will be displayed as a label near the center of the elements.

For criteria displays, element colors are also determined by this option. By default, elements that pass the criteria will be colored using Contour Colors. For this setting, FEMAP compares the elemental output value to the specified contour levels. The element color is then set to the color for the appropriate contour level. By default, elements that fail the criteria are not displayed. If you simply turn them on, the default View Color will cause them to be displayed as dashed/phantom lines.

### Example of using the Post-Processing Option in FEMAP

Open CH7WINGPOST.MOD in the /examples subdirectory where you installed the sample files that accompanied this manual.

The model will look like:

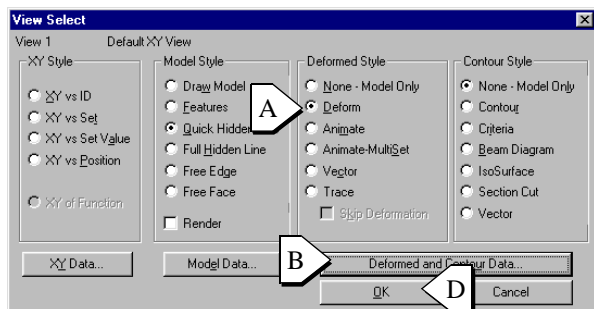


### Viewing the Deformed Model

Choose **VIEW - SELECT** from the FEMAP Menu (Ctrl-S or F5).

A.) Choose Deform for the Deformed Style.

B.) To select the data vectors to use for the deformed plot, press the Deformed and Contour Data button.

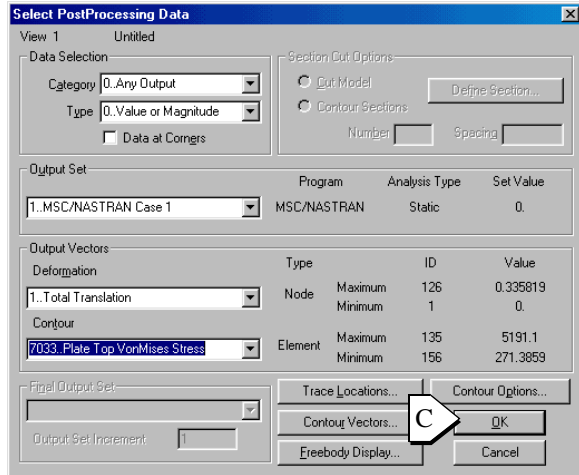


FEMAP now displays the Select PostProcessing Data dialog box. By default, the only Output Set is selected, and the Deformation data vector is set to Total Translation and the Contour data vector is set to Plate Top VonMises Stress.

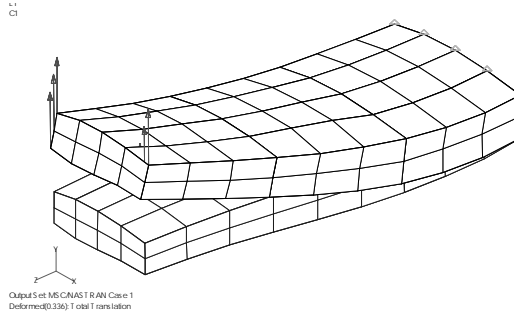
C.) Press OK to accept the defaults.

FEMAP returns you to the View - Select dialog box.

D.) Press OK to accept all options and generate the plot.



FEMAP displays the deformed model (exaggerated so that you can visualize the deformation) on top of the undeformed model.



To turn off the undeformed plot:

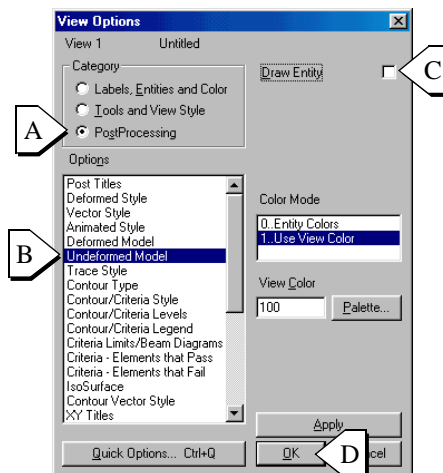
Select **VIEW - OPTIONS** from the FEMAP menu (Ctrl-O or F6).

A.) Change the Category to PostProcessing.

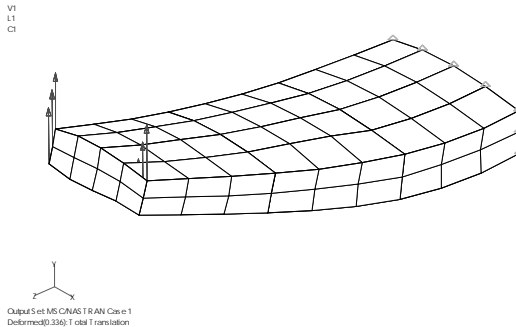
B.) Select the Undeformed Model option.

C.) Toggle the Draw Entity switch.

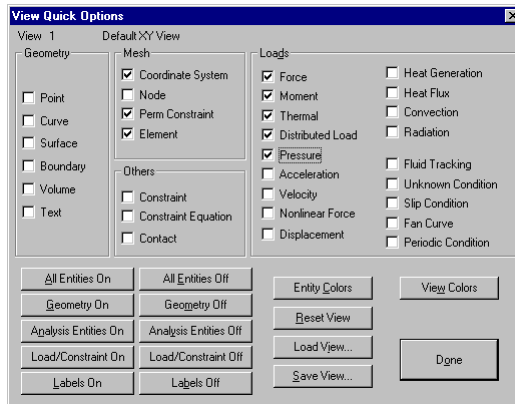
D.) Press OK to continue.



Only the Deformed model will be drawn:



Use the Quick View Options (**CTRL-Q**) and turn off the Nodal Forces. Press the Done button in View Quick Options to update the display and return to FEMAP.



## Viewing Stress Contours

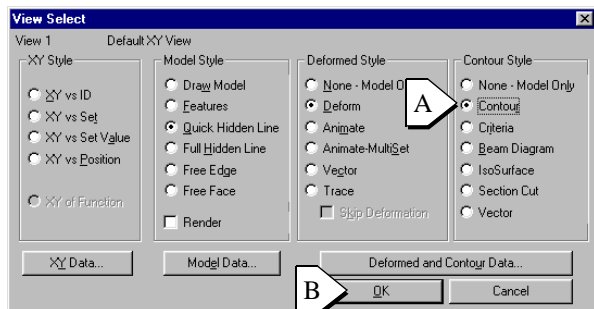
To turn on the display of stress contours, we will again use the View - Select command.

Select **VIEW - SELECT** from the FEMAP menu.

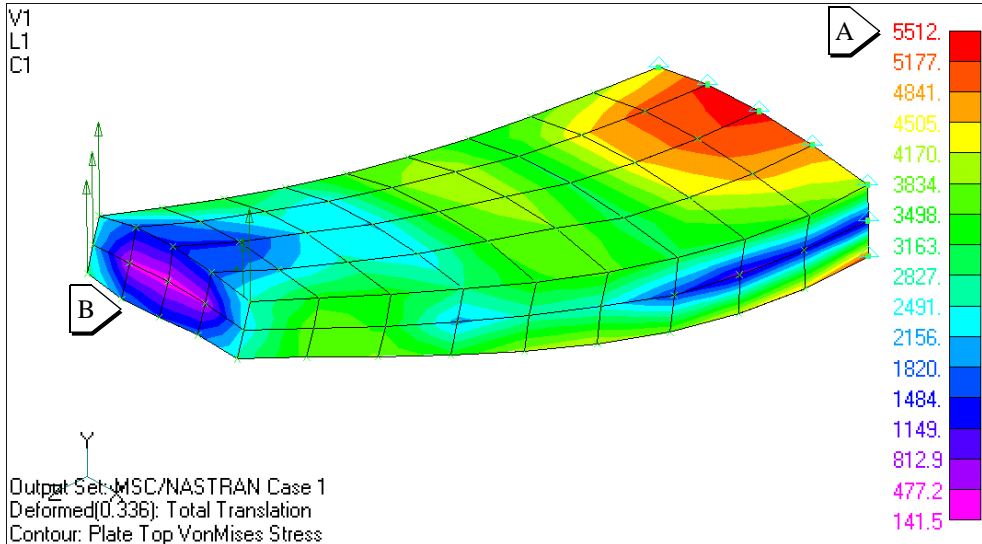
A.) Changed the Contour Style to Contour.

We do not need to enter the Deformed and Contour Data section since the contour data vector was set to Plate Top VonMises Stress when we created the deformed plot. To view the stress contours,

B.) Press OK.

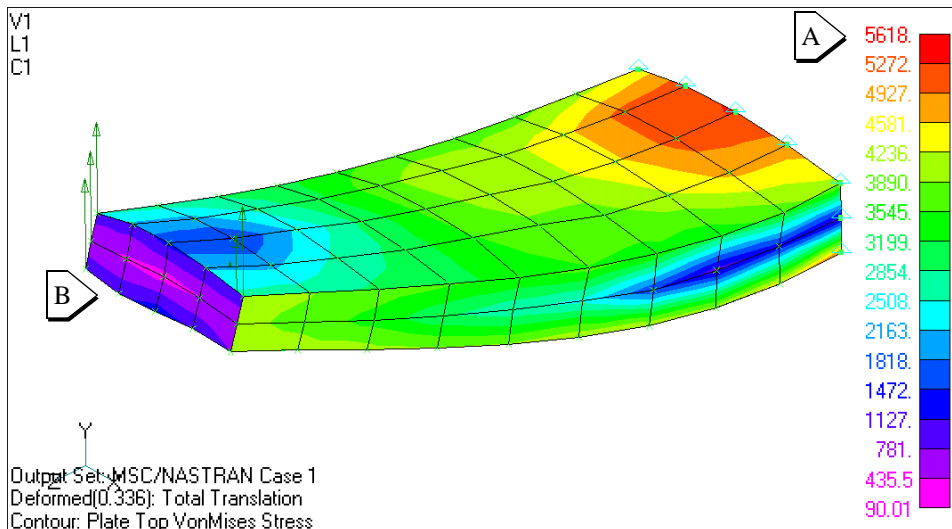


The model is now displayed with both the deformed and stress contours shown at the same time



The above display used Nodal Contouring, which averages across both property and geometric breaks. To eliminate this averaging, and produce a more accurate plot, go to **View Select - Deformed and Contour Data - Contour Options** and select **Elemental** under Contour Type. The defaults can be accepted for the element discontinuities.

The resulting display (after hitting three OKs) is shown below. As you will notice, the stress levels are slightly higher (A), and the distribution along the edge connections are changed significantly (especially at B). The elemental contouring option provides a more accurate representation by “smart averaging” the results only at locations where appropriate.



### Setting up an Animated Contour Plot

To view an animation, we will first enter the number of frames of the animation, and set the option to make the contour levels animate with the deformation.

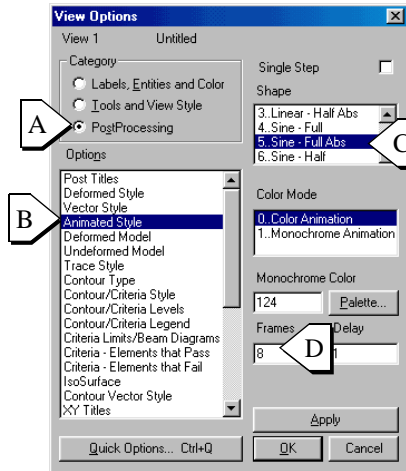
- 1 Select **VIEW - OPTIONS** from the FEMAP menu. Set the animations options as shown:

A.) Set the Category to PostProcessing.

B.) Select the Animated Style Option.

C.) Set the Animation Shape to 5..Sine - Full Abs.

D.) Set the Frames to 8.

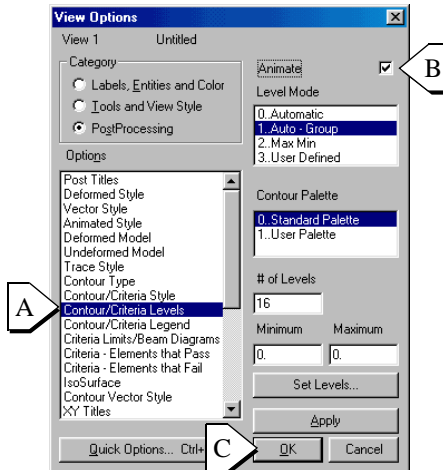


- 2 The color contours can either animate with the deformation, or remain constant. By default, they will remain constant. To cause them to animate:

A.) Change to the Contour/Criteria Levels option.

B.) Toggle the Animate Box to On.

C.) Press OK to continue.



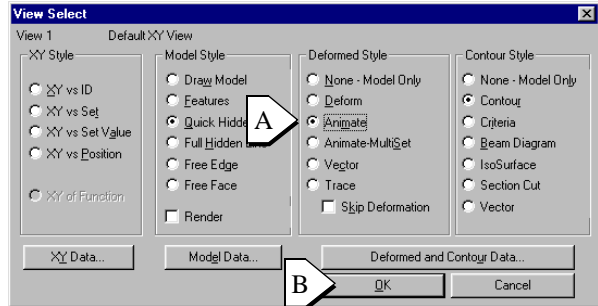
At this point we have only established the animation options. We must now enter **VIEW - SELECT** to change the Deformed Style from Deform to Animate to actually create the animation.



- 3 Choose **VIEW - SELECT** from the FEMAP menu.

A.) Change the Deformed Style to Animate.

B.) Press OK to view the animation.



FEMAP will first calculate each of the eight frames of animation, scaling the deformations and contour levels based on a sine-wave distribution. Once the individual frames have been created in memory, they will be played back at full speed. If the animation is going to fast, use the **VIEW-ADVANCED POST-ANIMATION** command and press the Slower button to slow it down. Once you have adjusted the animation to a comfortable speed, press the OK button in the Animation Control dialog box.

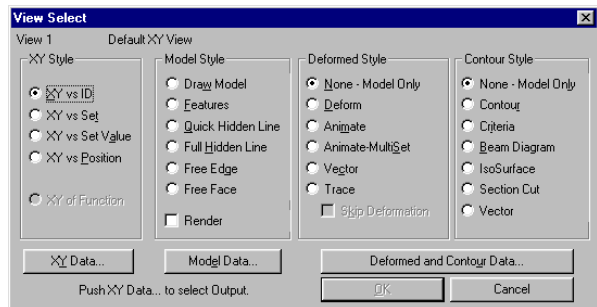
Like the other two categories of View - Options, the Post-Processing category provides a significant number of options that control how your model results are displayed. You are encouraged to experiment with the various options in the Post-Processing section and see how they affect the display of FEA results.

## 7.8 XY Style

FEMAP can also provide XY plots of your results. Just like the Deformed and Contour options discussed above, View - Select also controls (1) whether or not an XY Plot is displayed at all, and (2) what type of XY Plot will be shown.

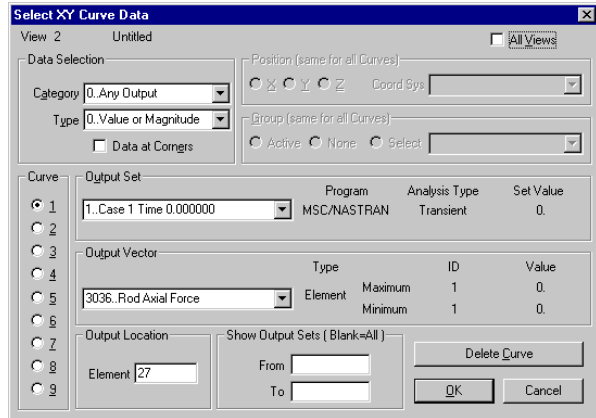
The available types are:

- XY vs. ID - Plots XY Data as a function of ID number for an Output Vector in one Output Set.
- XY vs. Set - Plots XY data versus the output set number for an Output Vector across several Output Sets.
- XY vs. Set Value - Similar to XY vs. Set except uses Output Set value for X.
- XY vs. Position - Plots XY data versus the position of nodes or elements in an axis direction for an Output Vector in one Output Set.
- XY as a Function - Plots XY data for functions. Not a Post-Processing option.



## Controlling an XY Plot

Control over the contents of an XY plot is provided by the Select XY Curve Data Dialog Box, accessed by pressing the XY Data button in View Select, or from the right mouse button shortcut menu as XY data. It allows you to control the output set and output vectors shown on the XY data plots. Simply choose the output set and the output vector from the appropriate drop down boxes. You can limit the category and type of output you see in the drop down lists with the data selection area.



If you are plotting by position you can choose which direction. If you are plotting multiple sets you can choose a starting and ending set and a node or element to plot. You can also choose a group to limit the data viewed. You can plot up to nine curves on the same plot.

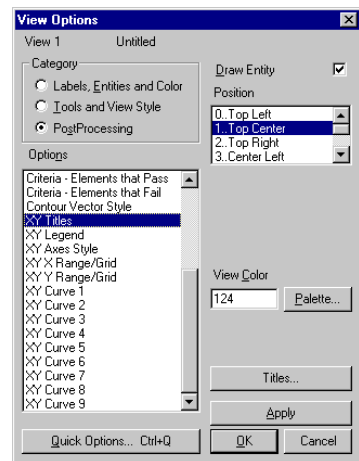
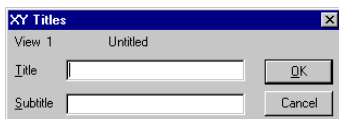
## View Options Post-Processing

View - Options (F6) Post-Processing is where you control the appearance of the XY plots.

Title and legend control the appearance to these item. Axes style change the axis from linear to semi-log or log-log. Range/Grid controls the range of the axes and the grid style. The curve #'s control the color and style of the curve.

### XY Titles

... specifies a title and subtitle for an XY display, and to locate those titles. You can choose any of the standard eight locations for the Titles. In general, Center Left and Center Right are not good choices. Unless you use very short titles, these positions will significantly reduce the size of the graph. The titles are always displayed in the View Color. The title View Color is also used for all axis labels.



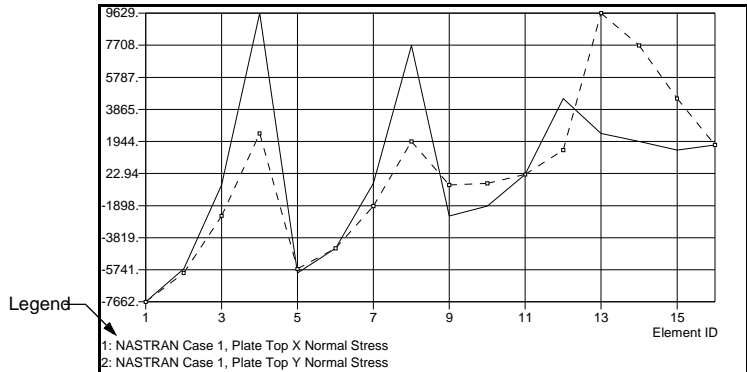
When you press Titles, FEMAP will display the XY Titles dialog box. You can specify any titles, each up to 25 characters in length.

## XY Legend

... controls the location, and format of the XY legend. This legend contains one line for each active curve. It defines the output data which is selected for the curve. The format of the lines in the legend is:

Curve Number: Output Set, Output Vector (Curve Scale Factor)

Output Set and Output Vector are either the IDs or Titles, depending on the labeling option you choose. Each line of the legend is drawn in the same color as the curve it defines.



You can position this legend in any of the eight standard locations. Make certain you do not locate the legend at the same position as the XY Titles, or they will overwrite each other.

## XY Axes Style

... defines the colors of the X and Y axes, and the number of axis divisions (tics). This option controls the color of the axis lines. Refer to XY Titles, if you want to change the colors of the axis labels.

You can also change the plot type between Rectilinear (the normal default), SemiLog and Log-Log, which are often used for Dynamic Analyses.

When you specify zero X Tics or Y Tics, FEMAP will automatically calculate the number of axis divisions. The number will be chosen so the labels do not overwrite one another. If you want a specific number of divisions, specify that number plus one. You must add one because there is always one more tic than division, for the end of the axis.

## XY X Range/Grid

... controls the minimum and maximum X axis values, and the display of the vertical grid lines.

If you choose Automatic, FEMAP will set the minimum and maximum axis values equal to the smallest and largest X values from your entire model. The nature of these values depends on the type of XY plot (vs. ID, vs. Set. . .) you are performing. Auto-Group is similar, but only considers values which are in the group you chose in the View Select command. If you pick "Max Min", you must manually set the Minimum and Maximum axis values.

Your Axis Range choices can be automatically updated by the View Autoscale, Pan, Zoom, or Magnify commands. You can use these commands for XY-plots, just like they are used for model displays.

### XY Y Range/Grid

... is identical to XY X Range/Grid, except that it controls the Y axis and the horizontal grid lines.

### XY Curve 1 through XY Curve 9

... controls the visibility, style, color, and labeling of the data curves for an XY-plot. By default, any curve that you select in the View Select command will be drawn. You can selectively skip curves, by turning off these options. ID and Output Value labels will be drawn at every data point on the curve. Only two labels will be drawn for Max/Min ID and Max/Min Value labeling. These labels will be drawn at the data point with the minimum and maximum output values. The Curve Style setting controls the type of curve or points that will be drawn.

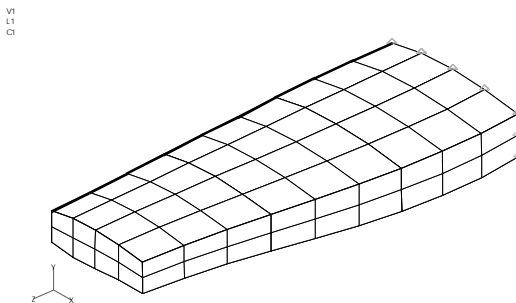
The Scale factor multiplies the actual output values. You can use this factor to display several curves, that have very different magnitudes, in the same Y range. When you specify a scale factor other than one, the position of the curve will be updated appropriately. The output value labels however, will still show the actual, unscaled output values. In addition, any scale factors, other than 1.0, will be shown in the XY Legend.

### Example of Using XY Plots

If you have been following along in this chapter, and have CH7WINGPOST.MOD animating on screen, use **VIEW - SELECT**, set the Deformed Style to None and the Contour Style to None. If you have not been working along, open CH7WINGPOST.MOD in the /examples sub-directory of the directory where the samples for this manual were installed.

This model has saved with it a group that includes the beam elements along the upper cap of the front spar. To see which elements we will use in this XY plot, we will use the View - Show command.

1. Select View - Show from the FEMAP menu.
2. Change the Entity Type to Elements, Press OK.
3. FEMAP displays the standard entity selection box. In the Group Field, select Group number 9. Press OK to View the elements.



To set up the XY Plot:

- 1 Choose **VIEW - SELECT** from the FEMAP menu.

A.) Change the XY Style to XY vs Position..

B.) Press the XY Data button to define the XY Plot.

The Select XY Curve Data dialog box is now displayed.

C.) Change Position to the Global Z-Axis.

D.) Set the Group Option to Select, and then select Group 9.

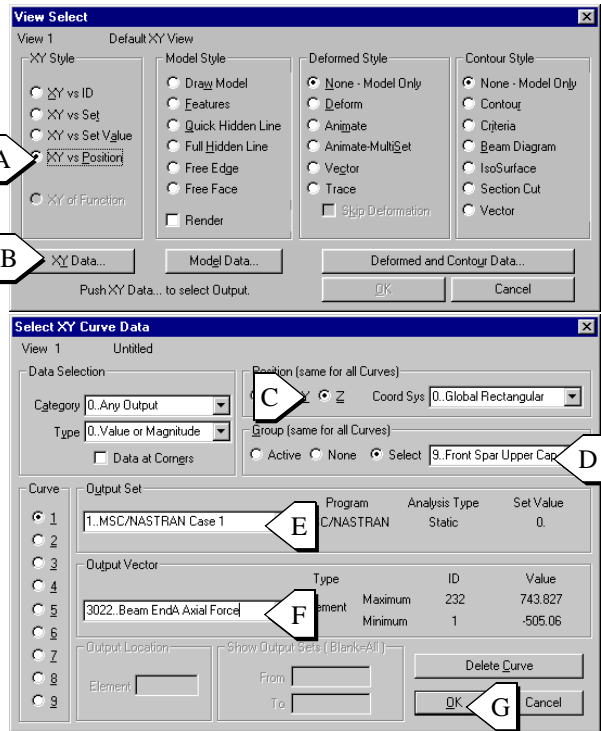
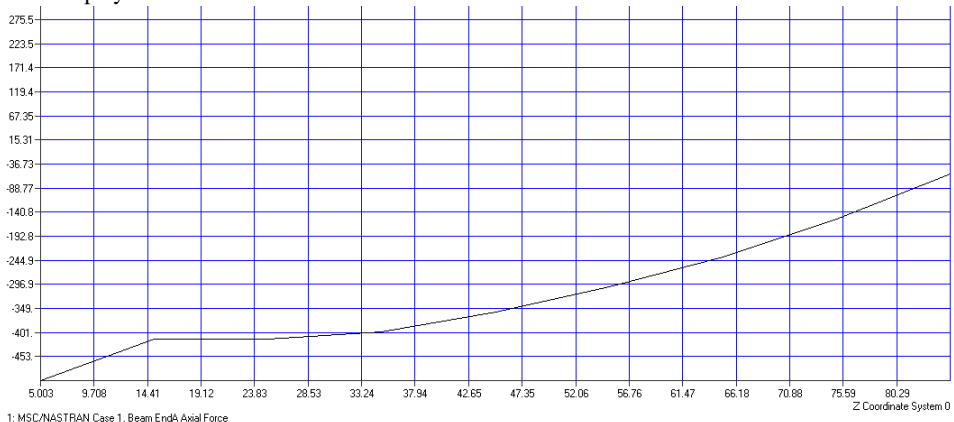
E.) Pick Output Set 1.

F.) Pick data vector number 3022..Beam End A Axial Force.

G.) Press OK to return to View - Select.

Press OK in View Select to accept the options and create the plot.

An XY Plot representing the axial force in the beam elements of the upper spar is now displayed.



XY Plots are just like any other FEMAP graphics window. You can use the View Zoom and Pan controls to adjust the range of display, and File - Print to send this information to a printer. And, just like other graphics windows, you can use File - Picture - Copy, copy the XY plot as vector graphics to the Windows clipboard, and then paste this graph into any other Windows application.

## 7.9 Reporting Results

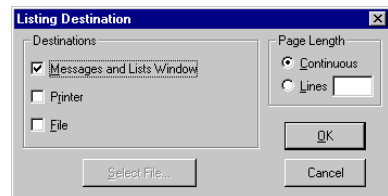
In addition to the graphical post-processing capabilities of FEMAP, there is also a powerful set of report based tools for examination of FEA results.

### Directing Output

Reports are created using the command in the List Output SubMenu



By default, all listings go to the Messages and Lists Window. You can also direct listings to a printer and/or a file. To control where listings appear, choose List - Destination from the FEMAP menu. Here you can toggle on and off the listing destination.



### Note:

Make certain to toggle off listings to a printer or a file when you are finished listing the desired information. FEMAP will continue to send all listings to whatever destinations have been chosen until they are turned off.

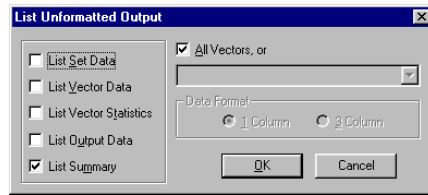
### Example of Using Output Listings

We will once again use CH7WINGPOST.MOD. One very handy feature of List Output is the ability to list a summary of the output data we read from an FEA analysis.

- 1 Select List - Output - Unformatted from the FEMAP Menu. FEMAP then prompts for the Output Set(s) to use. Press OK to use Set 1 and continue.



- 2 Options associated with listing output are now displayed. Toggle off all listing options except for List Summary, then press OK to continue.



FEMAP lists a summary of the Output Data for the Sets selected. This summary contains maximum and minimums for the entire output set that will help you quickly locate high levels of displacement, force and stress.

#### Displacement Summary

Maximum Value 0.33584 Output Vector 1 - Total Translation

Minimum Value -0.038524 Output Vector 6 - R1 Rotation

#### Force Summary

Maximum Value 2990.69 Output Vector 51 - Total Constraint Force

Minimum Value -2946.71 Output Vector 54 - T3 Constraint Force

#### Stress Summary

Maximum Value 7022.13 Output Vector 3154 - Beam EndB Pt4 Comb Stress

Minimum Value -6007.5 Output Vector 100227 - PltC1 Top MinorPrn Stress

### ***Listing Formatted Output***

The most powerful commands associated with listing output are List - Output - Standard and List - Output - Use Format. Both are used to process the nodal and elemental data recovered from a finite element analysis and repackage that data into standard formats or ones you define, and then list that data in printed format. An example of this was presented earlier in the chapter. A report was created listing element forces to the Messages and Lists Window, sorted by element shear force. Instead of duplicating that example here, please go back to the previous one and review it.

Describing how to create your own report formats is beyond the scope of this manual. Details are provided in the other FEMAP manuals and in the on-line help. One of the easiest methods of creating your own format is to load a standard format and then edit it.

### ***Querying Your Model***

List - Output - Query provides a quick method for retrieving the output results for a particular node or element, or group of nodes or elements in your model.

### Query Example

Choose List - Output - Query from the FEMAP Menu. Simply set the Output to List options to the Output Set and type of data



you wish to query, and then pick a node or element of your choosing. Remember, move the mouse to the ID field and then click in it to make it current. You can then type a node or element number, or pick one from the graphics window with the mouse. Once a node or element is selected, press the More button and the output data associated with that node or element will be sent to the List Destination(s). Press Cancel when finished.

### Dynamic Query

You can obtain results for a given deformation and/or contour plot at specific nodes and elements very rapidly. Simply turn on Dynamic Query (usually says Off and is on the right corner of the Status Bar/Tray) to Node or Element, and then simply hold the cursor at a node or element. FEMAP will then show a box providing information on the given node or element, as well as the data at that location which corresponds to the deformed contour plot (if any).

You can even right click in the information to create that exact text to annotate the model at that location, or left click to send the information to the List Destination, typically the Messages and Lists Window.

## 7.10 Getting Your Results to Other Programs

There are two ways you can report results into other programs. You can copy or save the screen views, or you can create lists of the actual data. **FILE - PICTURE - COPY** (Ctrl-C) copies the active view to the clipboard. As previously described in this manual, the data copied to the clipboard is a vector image, and will paste as a vector image into any other Windows program.

The image can also be transferred in a standard Windows Bitmap, or as a Windows Device Independent Bitmap. The File - Picture - Save command will save the active view as a file of any of the above types.

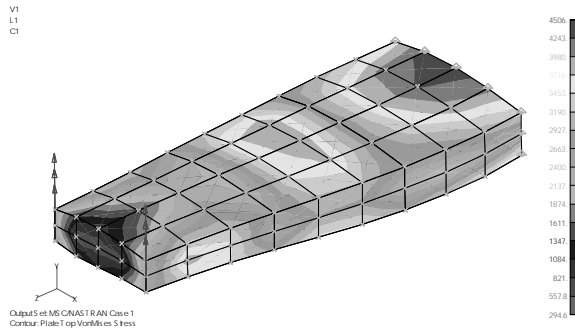
If the selected view is animating, a special bitmap format that can be displayed with the FEMAP REPLAY program will be created. You can also create a series of bitmaps that represent each frame of the animation. This series of bitmaps can be processed by other programs to create animated GIF's for the Internet, or to create AVI files.

The current model can also be written out as a VRML file to be shared across the network or web with any standard VRML viewer. You can write out solid geometry or meshes and even deformed and contoured meshes.

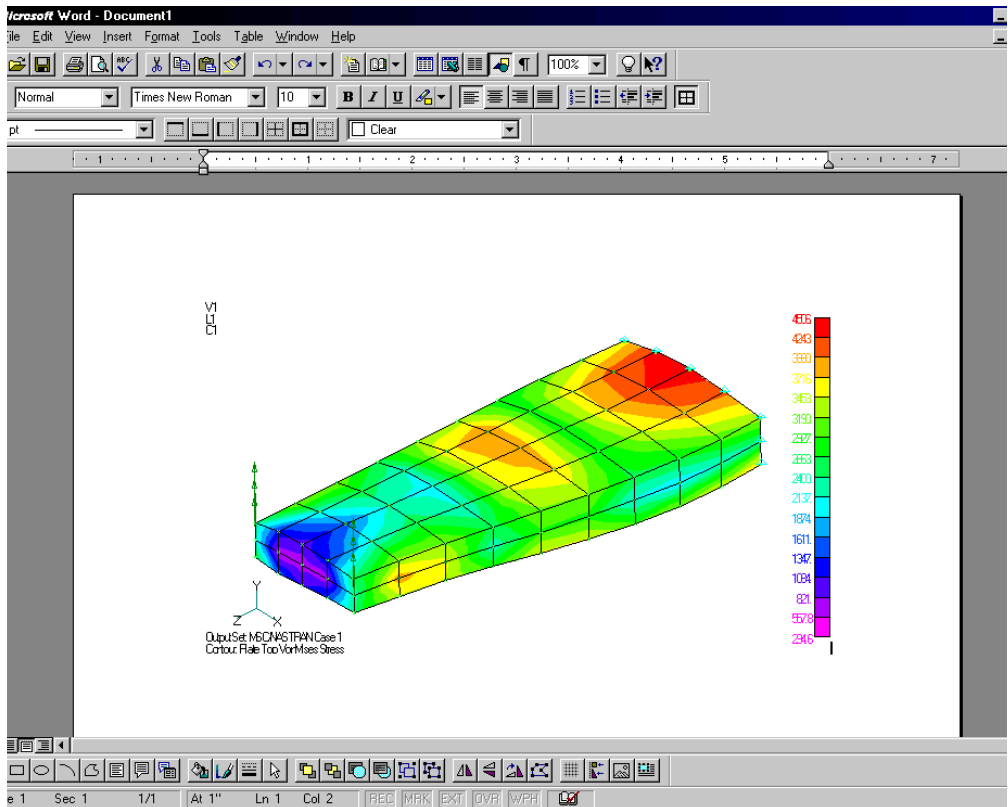
### Example of Copying a Graphics Image

For this example, set up CH7WINGPOST.MOD as a stress contour plot as previously described in this chapter.





Once the image is displayed on screen, press Ctrl-C to copy it to the clipboard. Now, go to any other windows programs that support graphics (Microsoft Word or Excel, PowerPoint etc.) and paste the image in, usually with the Shift-Ins or Ctrl-V key combinations.

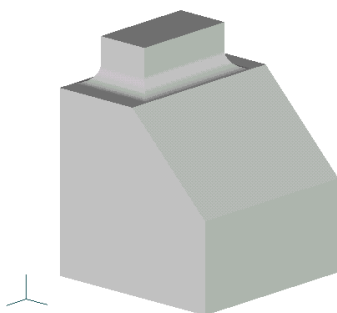


This entire manual was created by cutting and pasting FEMAP graphics and listings into FrameMaker.



# Simple Solid

This example is used to show some simple solid modeling while keeping the number of nodes of the solid mesh under the 300 node limit of the demonstration version.



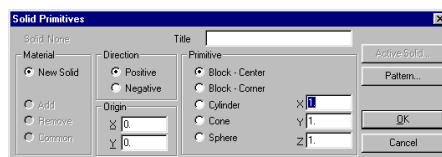
Again, first start FEMAP and create a new model, or if FEMAP is already running, select **FILE - NEW** from the menu.

## 8.1 Create the Geometry

### The Base Primitive

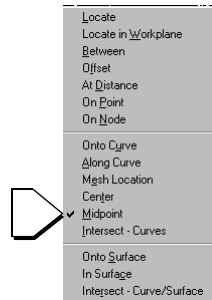
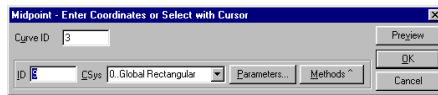
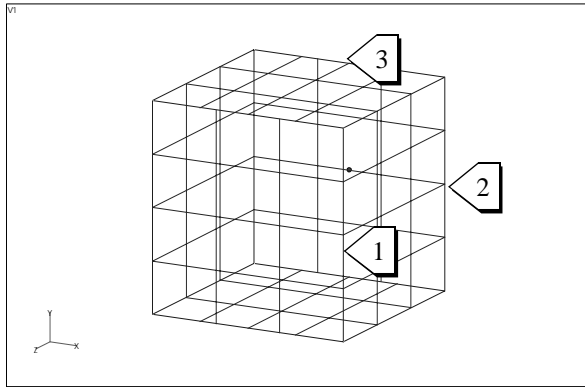
- 1 Select **GEOMETRY-SOLID-PRIMITIVES**.

The solid primitives dialog box gives you a choice of five different solids to create. Two types of blocks, a cylinder, a cone or a sphere. You can control the position of the origin and the direction, both relative to the workplane. For this example the default values of a unit cube centered at the origin are fine, so press OK.



***Slice the Base***

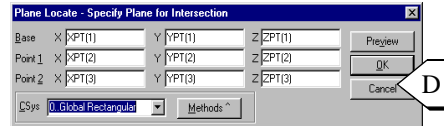
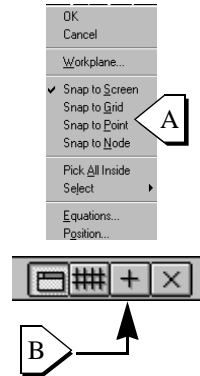
- 1 First we will set up points to use to define our cutting plane. Choose **GEOMETRY-POINT**. The standard coordinate location dialog box is shown. Press the methods button and select midpoint. Select curves 1,2 and 3 pressing OK after each selection. Press cancel to exit the command.



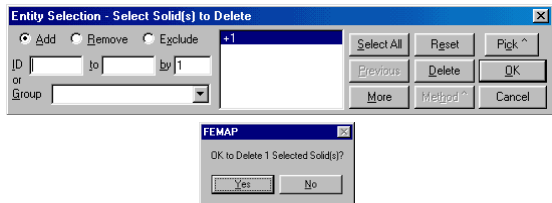
- 2 Select **GEOMETRY-SOLID-SLICE**. Pick the solid and press OK.

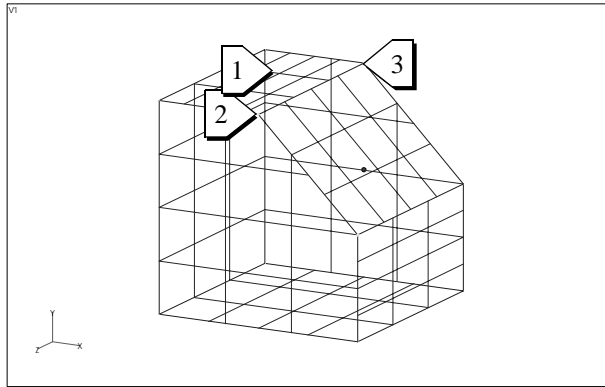


- 3 The standard plane definition dialog box is shown. First, change the snap mode to point, by either  
 A) Pressing the right mouse button in the graphics window, and then clicking the point option under Snap To, or  
 B) Using the toolbar Snap To Point icon, or C) Using the Ctrl-P keyboard shortcut. D) Choose the three points you created in the previous step and press OK.

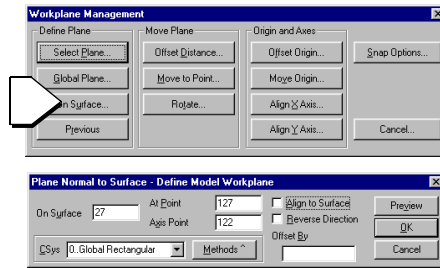


- 4 Select **DELETE-GEOMETRY-SOLID**. Pick the small corner section and press OK. Press Yes to delete the solid.

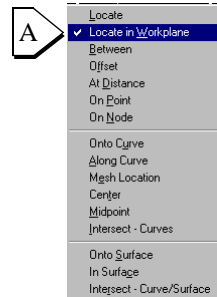
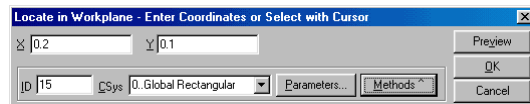


**Add the Rectangular Boss**

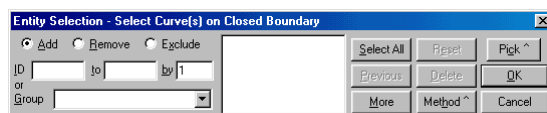
- 1 Select **TOOLS-WORKPLANE**. Press the on Surface button. Pick surface 1, and points 2 and 3 as shown in the above diagram and press OK.



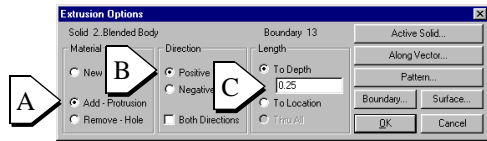
- 2 Choose **GEOMETRY-CURVE LINE-RECTANGLE**. A) Press the methods button and choose Locate in Workplane. B) Enter points at 0.2, 0.1 and 0.8, 0.4 pressing OK after each one.



- 3 Select **GEOMETRY-BOUNDARY SURFACE**. Select the four curves of the rectangle and press OK.



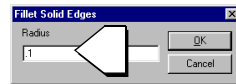
- 4 Select **GEOMETRY-SOLID-EXTRUDE**. A) Make it add-protrusion in B) The positive direction and C) To a depth of 0.25.



- 5 Select **GEOMETRY-SOLID-FILLET**. Again select the four curves of the rectangle and press OK.

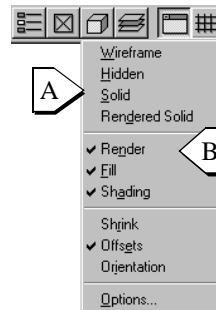


- 6 Enter a radius of 0.1 and press OK.

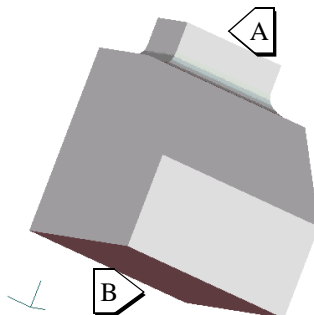


### View as Solid

- 1 A) Press the view style button on the toolbar and choose solid. B) Press the view style button on the toolbar and choose Render. This puts you in the OpenGL graphics mode.

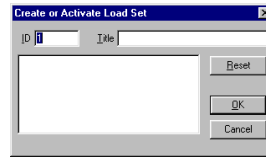


## 8.2 Loads and Constraints

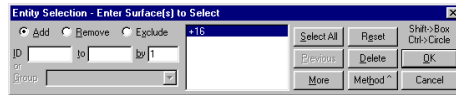


## Loads

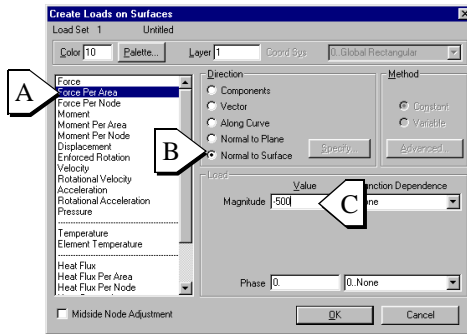
- 1 Select **MODEL-LOAD-ON SURFACE**. FEMAP prompts you to create a load set. Enter a name and press OK.



- 2 Select surface A as shown and press OK

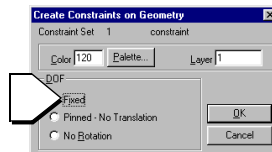
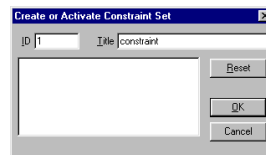


- 3 A) Select the load type force/area. B) Make the direction Normal to Surface and C) enter a value of -500. Press OK.



## Constraints

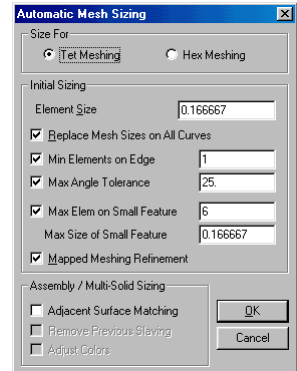
- 1 Select **MODEL-CONSTRAINT-ON SURFACE**. FEMAP prompts you to create a constraint set. Type a name and press OK.
- 2 Select the bottom surface of the solid, surface B, and press OK. Fix this surface.



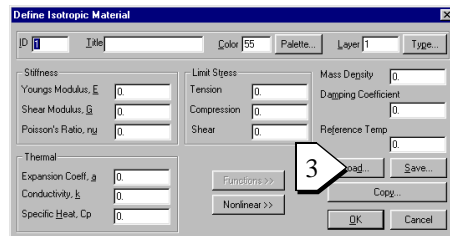


### 8.3 Meshing the Solid

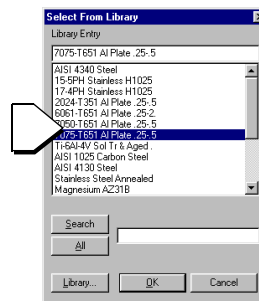
- 1 Choose **MESH-GEOMETRY-SOLIDS**.
- 2 The default values for solid meshing determined by FEMAP are usually adequate to produce a good mesh, especially when Tet Meshing. However, in this case we are trying to keep the number of nodes below 300. Enter an element size of 0.2. This will produce a coarse mesh, but again this is just a demonstration problem.



- 3 Since no material has been created FEMAP prompts you to make one. You can enter in values or press the Load button to bring up the material library.



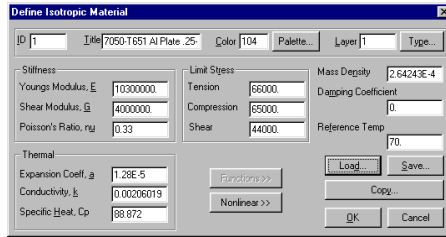
- 4 The material library shipped with FEMAP contains material properties using English units (lb, in, sec). You can create your own materials and store them in this library or create your own library. Select a material and press OK.



**Note:**

Remember, there are no units in FEMAP. All dimensions must be kept consistent with the unit system you use to define your material properties. Always make sure this is correct from the beginning because it is extremely difficult to correct any inconsistencies in units once the model is built.

- Press OK in the Define Material dialog box when the properties have been loaded.



- The automesh solids dialog box appears. Deselect the midside nodes box to turn off automatic insertion of midside node. **This is not a recommended practice for any model, as four noded tetrahedral elements behave too stiffly to get accurate answers. It is only done here to keep the number of nodes below 300.** Press OK.



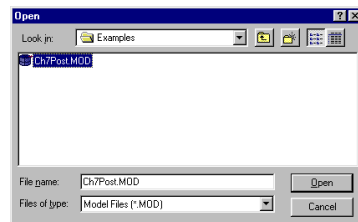
The model is now ready for analysis

## 8.4 PostProcessing

### Reading in Results

For this example we have included a FEMAP model file with results that you can use for post-processing. If you have run your own analysis you may use those results.

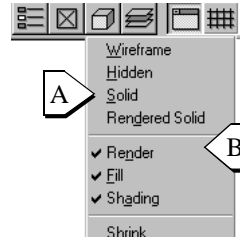
- Select **FILE - OPEN**. The standard file open dialog box appears. Navigate to the /Examples directory and choose the CH8Post.Mod file. Press open.



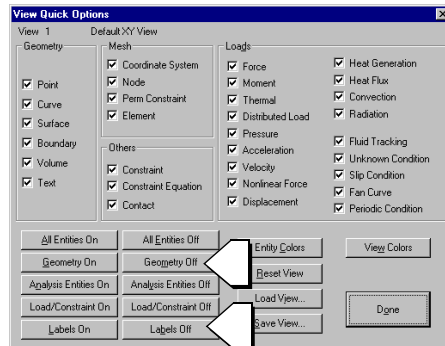
### Postprocessing

This section will take you through some of the ways you can use FEMAP to view analysis results.

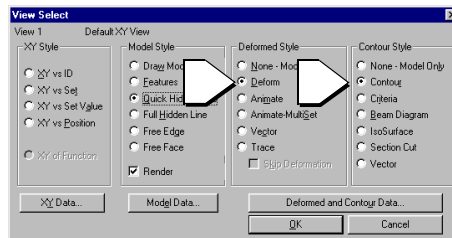
- 1 A) Press the view style button on the toolbar and choose solid. B) Press the view style button on the toolbar and choose Render.



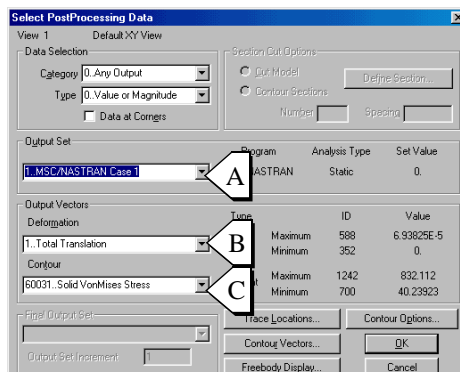
- 2 Press Ctrl-Q or the view quick options toolbar button to bring up the view quick options menu. Press the geometry off button and the labels off button.



- 3 Select **VIEW-SELECT**, press F5 or the view select button on the toolbar to bring up the view select dialog box. Set the Deform Style to Deform and the Contour Style to Contour. Press the Deformed and Contour Data Button.



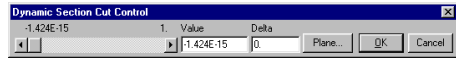
- 4 This brings up the Select PostProcessing Data dialog box. A) select an output set. B) select an output vector to use for the model deformation. C) select an output vector to use for the contour plot. Press OK. Press OK in the view select dialog box.



- 5 Press the Dynamic Display button on the toolbar. Rotate the view to see the deformations and contours on all sides of the model.

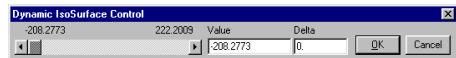


- 6 Select **VIEW-SPECIAL POST-DYNAMIC CUTTING PLANE**. Move the slider bar to move the cutting plane through the model.



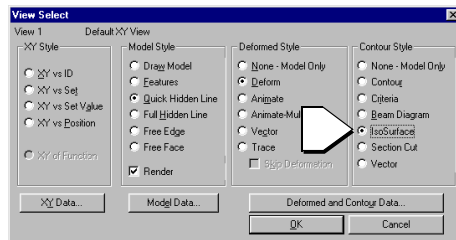
Press the plane button to define a different cutting plane using the standard plane definition dialog box. Press the Dynamic Display button to rotate the view of the cutting plane. Press OK when done.

- 7 Select **VIEW-SPECIAL POST-DYNAMIC ISO SURFACE**. Move the slider bar to change the value of the isosurface being shown. The isosurface itself is calculated

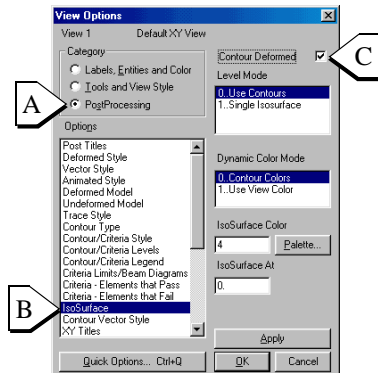


from the output vector chosen for the contour vector. Put the cursor in the value box, enter a value and press apply to see an isosurface at that value. Rotate the view if you would like. Press OK when done.

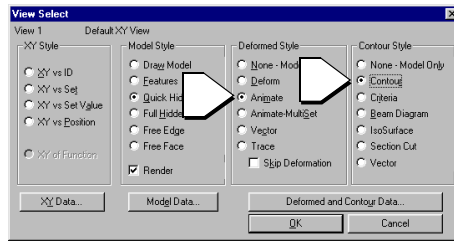
- 8 Select **VIEW-SELECT**, press F5 or the view select button on the toolbar to bring up the view select dialog box. Set the Contour Style to IsoSurface. Press OK.



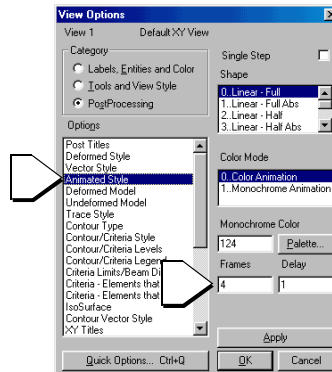
- 9 Select **VIEW-OPTIONS** or press F6. A) Pick postprocessing as the category. B) pick IsoSurface as the option. C) Check the Contour Deformed box to see the deformed output vector contoured on the isosurface. Press Apply. Change the IsoSurface At value and press Apply to see a different isosurface. Press OK. Use **VIEW-SPECIAL POST-DYNAMIC ISO SURFACE** to dynamically change the isosurface value.



- 10 Select **VIEW-SELECT**, press F5 or the view select button on the toolbar to bring up the view select dialog box. Set the Deform Style to Animate and the Contour Style to Contour. Press OK.



- 11 Select **VIEW-OPTIONS** or press F6. In the Postprocessing category select Animated Style. Change the number of frames to get a smoother animation. Increase the delay to slow down the animation. Select the Contour/Criteria Levels option. Check the animate box to animate the contour colors as well as the deformation.



This is the end of this example. Before moving on, experiment with some of the different post-processing options on your own.



# Turbine Blade



This example will illustrate some simple solid modeling in FEMAP. First we will read in an IGES trimmed surface part. Then in FEMAP we will stitch the surfaces into a solid, and then modify the solid so we can get a proper FEA mesh. We can then add loads and constraints directly to the geometry of the solid model. The final step in analysis preparation will be to create a material and then use the automatic tetrahedral element mesher to mesh the solid. To complete this example you must have purchased FEMAP Professional.

## Note:

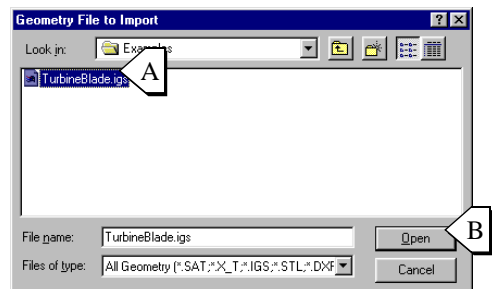
You will not be able to save your model file or export to an analysis program if you are using the 300-Node Demonstration version. A file with results is provided to use for the postprocessing section of this example.

First start FEMAP and create a new model, or if FEMAP is already running, select **FILE - NEW** from the menu.

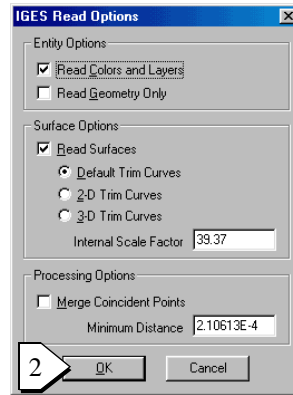
## 9.1 Creating the Geometry

### Reading the IGES File

- 1 Select **FILE-IMPORT-GEOMETRY** from the FEMAP menu. The Windows File Open Common Dialog Box appears. Navigate to the \Examples directory and  
A) Select the TurbineBlade.igs file and  
B) Press open.



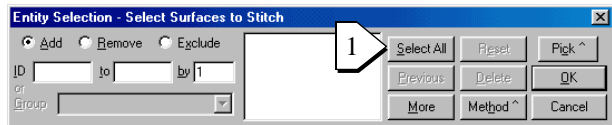
- 2 The IGES Read Options dialog box appears. The default values should work for most IGES files and they will here, so press OK.



The messages and lists window will tell you what has been read and also log any errors in the import operation.

### ***Stitch the IGES trimmed surfaces into a solid***

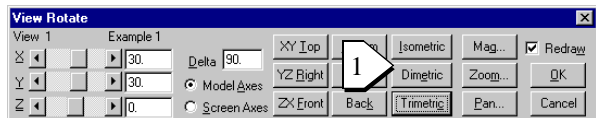
- 1 Choose **GEOMETRY-SOLID-STITCH**. The Standard entity selection dialog box appears and prompts you to select the surfaces to stitch. For this model A) select all surfaces and B) press OK.



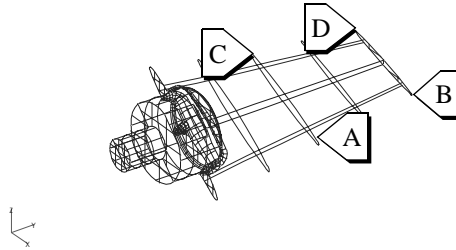
The messages and lists window informs you that the model has been stitched and conforms to Parasolid modeling tolerances. This simply means that the stitching operation was successful and the model is now a FEMAP solid that can be operated on with FEMAP geometry commands.

### ***Rotate the View***

- 1 Choose **VIEW - ROTATE** from the FEMAP Menu (or use the Ctrl-R or F8 short-cut keys) and you will see the View Rotate Dialog Box. There are several pre-defined 3-D views that you can select from, you may want to experiment and press some of them. Before leaving View Rotate, press Dimetric and then OK to dismiss the View Rotate Dialog Box.



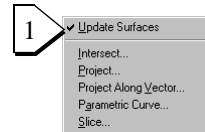




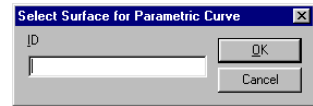
### Split Surface Along Constant UV Line

We now want to split the two large surfaces that make up the blade at the blade's edges so that the mesh will not wrap across the edges. We will do this by imprinting a curve made from a constant parametric value of the surface.

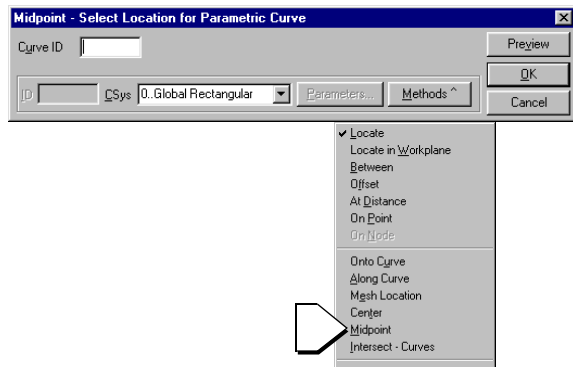
- 1 Choose **GEOMETRY-CURVE FROM SURFACE-UPDATE SURFACES**. This toggles whether or not the created curve imprints on and splits the surface or surfaces it intersects. We want this option on.



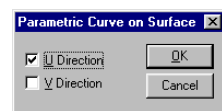
- 2 Choose **GEOMETRY-CURVE FROM SURFACE-PARAMETRIC CURVE**. FEMAP prompts you to select a surface. Move the cursor to highlight surface A and press the left mouse button to pick it. Press OK.



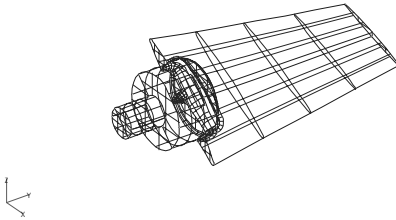
- 3 FEMAP prompts you for a location for the curve. Press the methods button and choose Midpoint from the list. Move the cursor over curve B in the diagram above to highlight it and select it with the left mouse button. Press OK.



- 4 FEMAP prompts you for a parametric curve direction. You can use the surface lines to determine the proper direction. By default FEMAP draws 3 divisions in the U direction and 4 in the V direction. So in this case select the U direction and press OK.



5

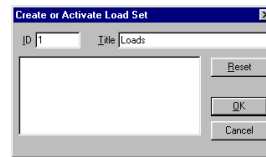


Repeat the above procedure this time using surface C and curve D. There should now be curves along the edges as shown here.

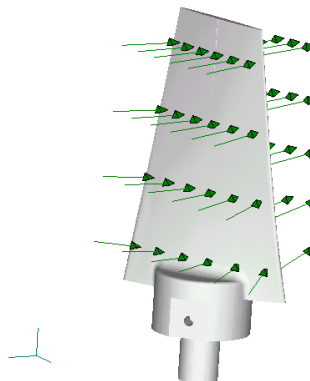
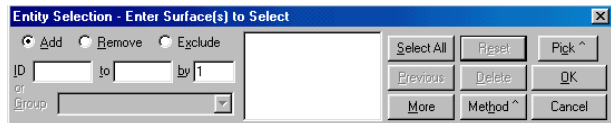
## 9.2 Loads and Constraints

### Add Loads on Geometry

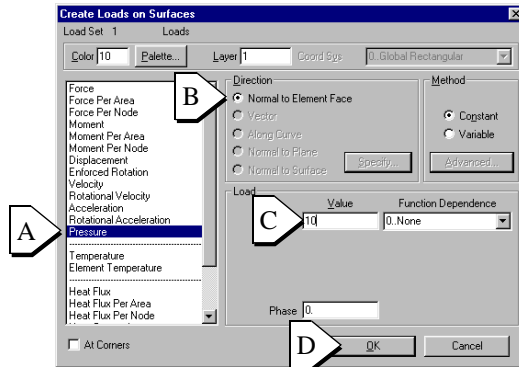
- 1 Choose **MODEL-LOAD-ON SURFACE**, FEMAP prompts you to select a load set or create a new one. Type in a title and press OK.



- 2 FEMAP now asks you to select the surfaces to apply the load. Pick the two large surfaces on the top of the blade and press OK.



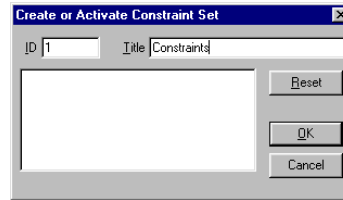
- 3 FEMAP displays the Load on Surfaces dialog box. A) Select pressure as the load type B) Leave the direction as normal to element face C) Enter a value of 10 and D) Press OK.



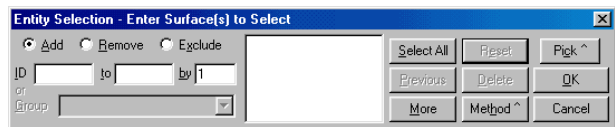
- 4 Repeat the procedure above, this time selecting the bottom surfaces of the blade and entering a pressure of negative ten (-10).

### Add Constraints on Geometry

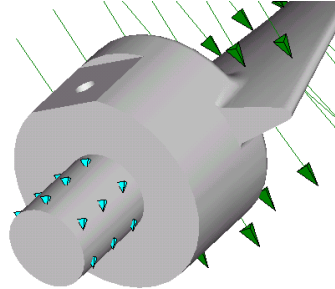
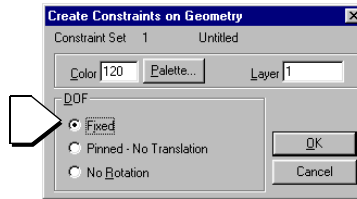
- 1 Choose **MODEL-CONSTRAINT-ON SURFACE**. FEMAP prompts you to select a constraint set or create a new one. Type in a name and press OK.



- 2 FEMAP prompts you to select surfaces. Pick the two halves of the cylinder at the bottom of the blade and press OK.



- 3 Constraints on surfaces are always relative to the global coordinate system and can only be fixed, pinned or have no rotations. Make these surfaces fixed. Press OK to create the constraints, press cancel to end the command.

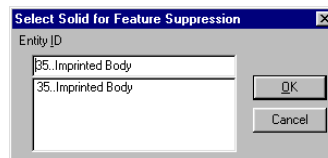


## 9.3 Meshing the Solid

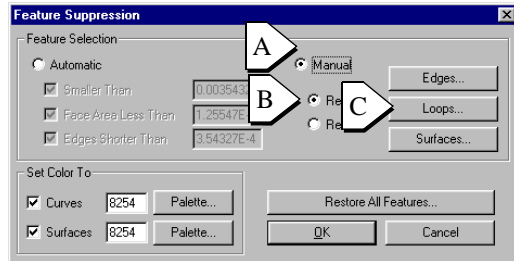
### Suppress Small Features

The small hole in the base of the turbine blade will not affect the results of the analysis but will cause the mesh to condense in that area. To reduce the number of elements and our overall problem size we will suppress this small hole before meshing.

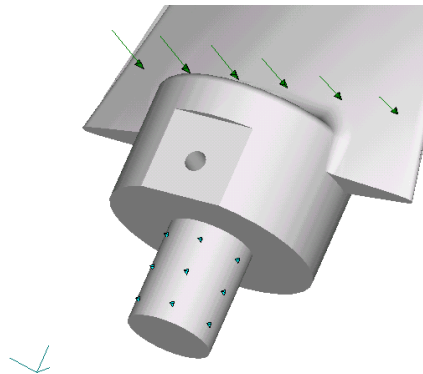
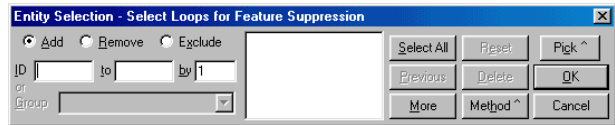
- 1 Choose **MESH-MESH CONTROL-FEATURE SUPPRESSION**. Select the turbine blade as the solid model.



- 2 In the feature suppression dialog box, A) Select manual, B) Select remove and C) Press the Loops button.

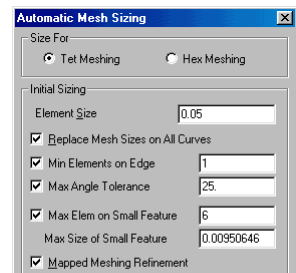


- 3 Select one of the two curves that make up the top of the small hole and press OK. The surfaces of the hole should be grayed out indicating that they are suppressed.

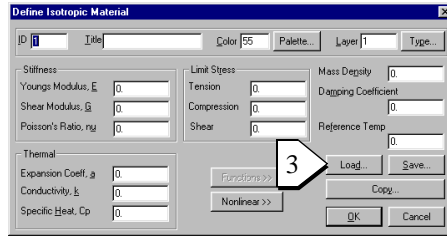


## Meshing the Model

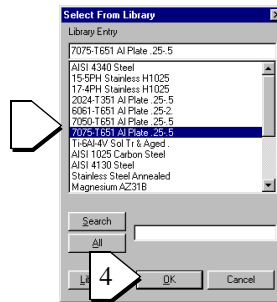
- 1 Choose **MESH-GEOMETRY-SOLIDS**.
- 2 Change the Element Size to 0.05 and press OK. This element size is determined by the shape and size of the various features of the model. The default values determined by FEMAP are usually adequate to produce a good mesh. However, as you gain experience with the solid mesher you may find that a slightly larger element size will still give you a good mesh but greatly reduce the number of elements. On the other hand some parts may need a smaller element size to produce a good mesh in certain areas. Also, keep in mind that you can specify mesh spacing and mesh hard points on all curves and surfaces individually. This is often the best way to get the best mesh although it does take more time and careful planning.



- 3 Since no material has been created FEMAP prompts you to make one. You can enter in values or press the Load button to bring up the material library.



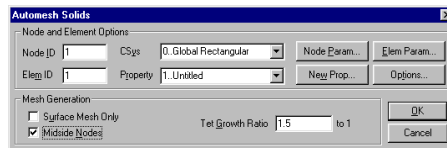
- 4 The material library shipped with FEMAP contains material properties using English units (lb, in, sec). You can create your own materials and store them in this library or create your own material library. For this example select a material from this library and press OK.



## Note:

Remember, there are no units in FEMAP. All dimensions must be kept consistent with the unit system you use to define your material properties.

- 5 Press OK in the define material dialog box when the properties have been loaded.
- 6 The automesh solids dialog box appears. Leave the values as the defaults and press OK.



When the model finishes meshing it will be ready for analysis.

## Note:

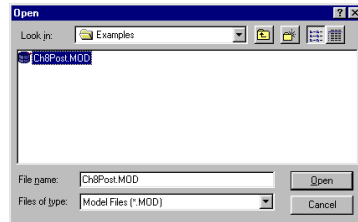
You should always check the shapes of your elements before you run an analysis. Badly distorted elements can cause incorrect results and analysis failure. For information on checking element distortion refer to the FEMAP Command Reference.

## 9.4 PostProcessing

### Reading in Results

For this example we have included a FEMAP model file with results included that you can use for Postprocessing. If you have run your own analysis you may use those results.

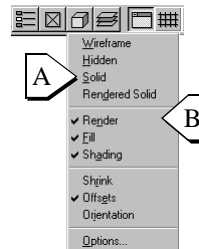
- 1 Select **FILE - OPEN**. The standard file open dialog box appears. Navigate to the /Examples directory and choose the CH9Post.Mod file. Press open.



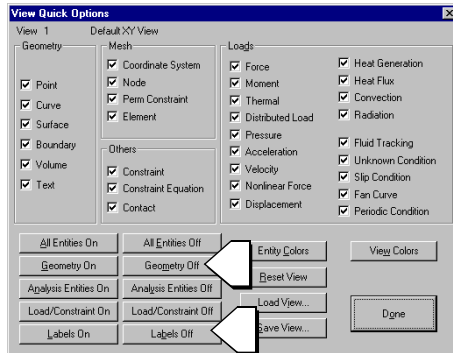
### Graphical Postprocessing

This section will take you through some of the ways you can use FEMAP to view analysis results.

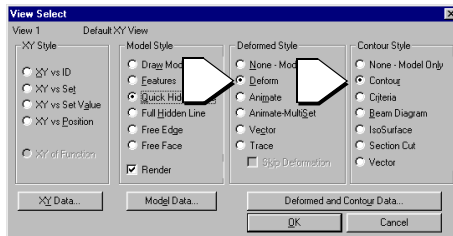
- 1 A) Press the view style button on the toolbar and choose solid. B) Press the view style button on the toolbar and choose Render. This puts you in Render mode which allows dynamic pan, zoom and rotate of solid contoured models as well as dynamic isosurfaces and sections cuts and also speeds up general graphics.



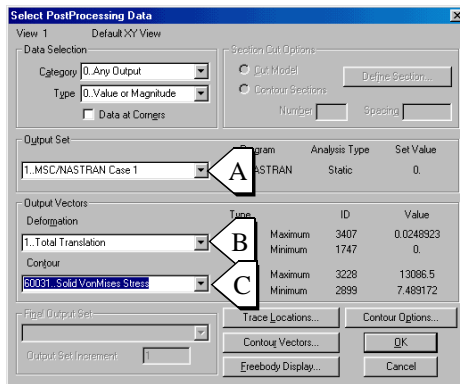
- Press Ctrl-Q or the view quick options toolbar button to bring up the view quick options menu. Press the geometry off button and the labels off button.



- Select **VIEW-SELECT**, press F5 or the view select button on the toolbar to bring up the view select dialog box. Set the Deform Style to Deform and the Contour Style to Contour. Press the Deformed and Contour Data Button.



- This brings up the Select PostProcessing Data dialog box. A) Select an output set. B) Select an output vector to use for the model deformation. C) Select an output vector to use for the contour plot. Press OK. Press OK in the view select dialog box.

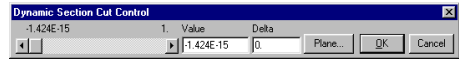


- In Render mode simply click the left mouse button in the graphics window and drag the cursor. The model will rotate in XY. You can also use the same Dynamic Display options with the Alt, Ctrl, and Shift (Rotate Z, Pan, Zoom)



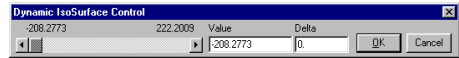
- 6 Select **VIEW-SPECIAL POST-DYNAMIC CUTTING PLANE**. Move the slider bar to move the cutting plane through the model.

Press the plane button to define a different cutting plane using the standard plane definition dialog box. Press the Dynamic Display button to rotate the view of the cutting plane. Press OK when done.

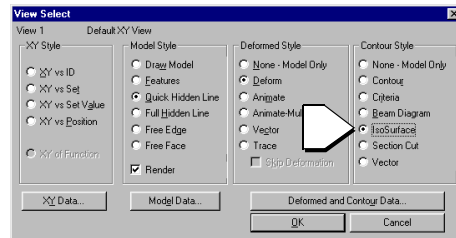


- 7 Select **VIEW-SPECIAL POST-DYNAMIC ISO SURFACE**. Move the slider bar to change the value of the isosurface being shown. The isosurface itself is calculated

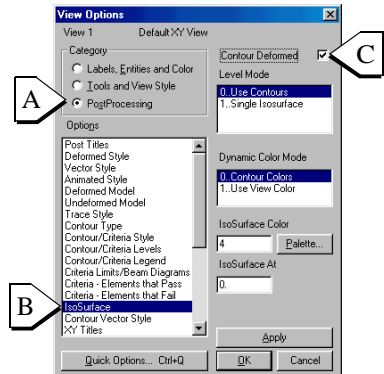
from the output vector chosen for the contour vector. Put the cursor in the value box, enter a value and press apply to see an isosurface at that value. Rotate the view if you would like. Press OK when done.



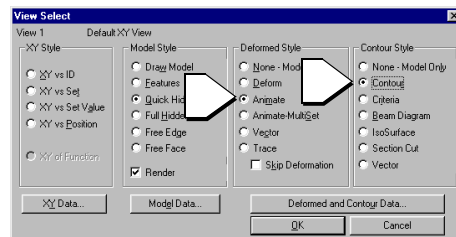
- 8 Select **VIEW-SELECT**, press F5 or the view select button on the toolbar to bring up the view select dialog box. Set the Contour Style to IsoSurface. Press OK.



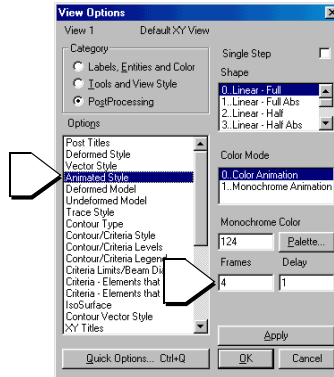
- 9 Select **VIEW-OPTIONS** or press F6. A) Pick post-processing as the category B) Pick IsoSurface as the option. C) Check the Contour Deformed box to see the deformed output vector contoured on the isosurface. Press Apply. Change the IsoSurface At value and press Apply to see a different isosurface. Press OK. Use **VIEW-SPECIAL POST-DYNAMIC ISO SURFACE** to dynamically change the isosurface value.



- 10 Select **VIEW-SELECT**, press F5 or the view select button on the toolbar to bring up the view select dialog box. Set the Deform Style to Animate and the Contour Style to Contour. Press OK.

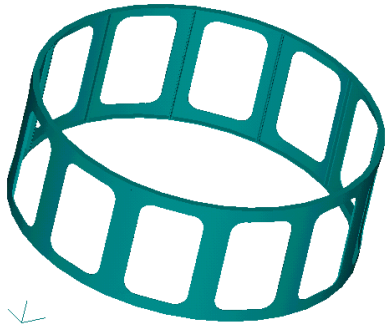


- 11 Select **VIEW-OPTIONS** or press F6. In the Postprocessing category select **Animated Style**. Change the number of frames to get a smoother animation. Increase the delay to slow down the animation. Select the **Contour/Criteria Levels** option. Check the animate box to animate the contour colors as well as the deformation.



Experiment with some of the other postprocessing and viewing options on your own.

# Cylindrical Support

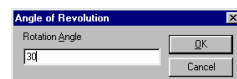
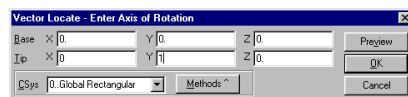
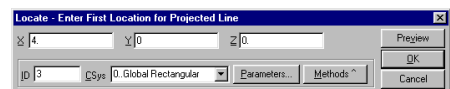


This example will demonstrate some basic line and surface modeling used to produce beam and plate elements using the **PARASOLID** modeling engine. (You can also use the ACIS solid modeling engine to complete this problem, but some aspects of the mesh may be slightly different due to differences in the parametric definitions of the created surfaces in ACIS and Parasolid). It will also show you how to build, orient and view beams so you can be certain they correctly represent your model. Again, first start FEMAP and create a new model, or if FEMAP is already running, select **FILE - NEW** from the menu.

## 10.1 Creating the Geometry

### Making the Underlying Surface

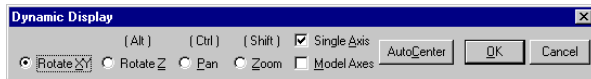
- 1 Choose **GEOMETRY-CURVE LINE-PROJECT POINTS**. The standard coordinate location dialog box appears. Enter a point of 4,3,0 and press OK. Enter a point of 4,0,0 and press OK. Press cancel to exit the command. Press Ctrl-A to autoscale the view.
- 2 Choose **GEOMETRY-SURFACE-REVOLVE**. The standard entity selection dialog box appears. Select the line you just drew and press OK.
- 3 You are now prompted for a vector to rotate about. Enter coordinates of 0,0,0 and 0,1,0 and press OK. Enter a Rotation Angle of 30 degrees



- 4 You should have created 1/12 of a cylindrical surface 3 units high with a radius of 4. Rotate the view to check you model.
- 5 Press the Dynamic Display button on the toolbar.

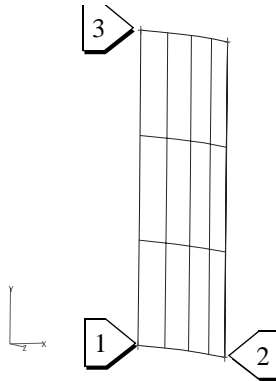


- 6 FEMAP displays the Dynamic Display dialog box at the bottom of the screen.

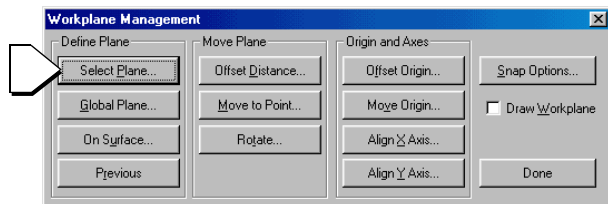


- 7 To dynamically rotate your model, move the cursor inside the graphics window, then press and drag it left to right and up and down. This will dynamically rotate the model. By pressing and holding the Shift key, and pressing and dragging the mouse up and down you can scale the view dynamically. Using the Ctrl key you can dynamically pan the view. Orient your model similar to the view below and press OK or the Return key to leave the Dynamic Display

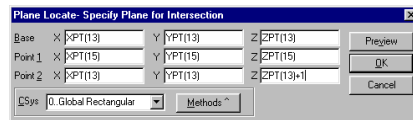
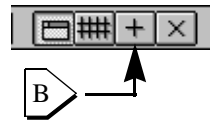
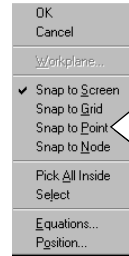
### Creating the Cutout Geometry



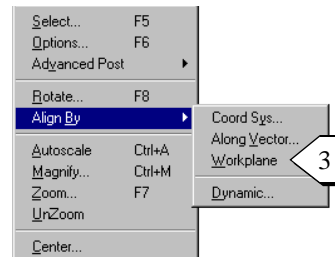
- 1 Choose **TOOLS-WORK-PLANE**. Press the select plane button. The standard plane definition dialog box appears.



- 2 First, change the snap mode to point, by either  
 A) Pressing the right mouse button in the graphics window, and then clicking the point option under Snap To, or  
 B) Using the toolbar Snap To Point icon, or C) Using the Ctrl-P keyboard shortcut. D) Choose point 1, then 2, then 3 as shown in the previous diagram and press OK.



- 3 Choose **VIEW-ALIGN BY-WORKPLANE** to orient the view so you are looking directly at the workplane.

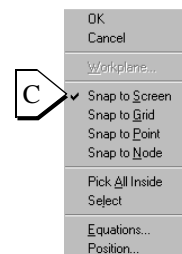
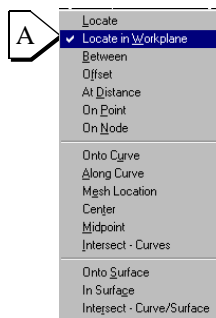


- 4 Choose **GEOMETRY-CURVE LINE-RECTANGLE**.

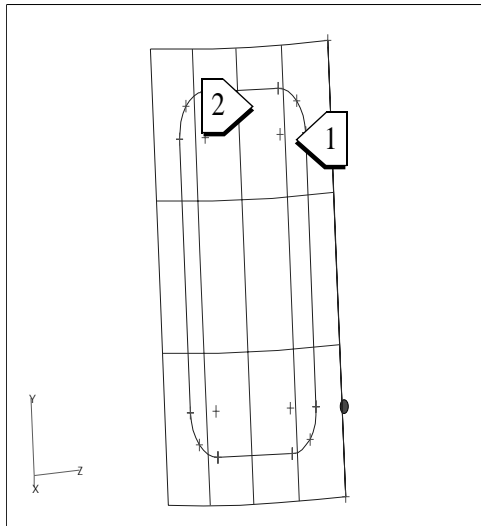
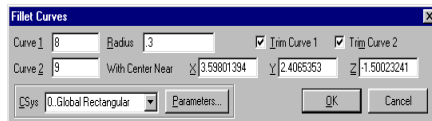
A) Press the methods button and choose Locate in Workplane.

B) Enter points at 0.3,0.3 and 1.77,2.7 pressing OK after each one.

C) Set Your Snap Mode back to Screen.

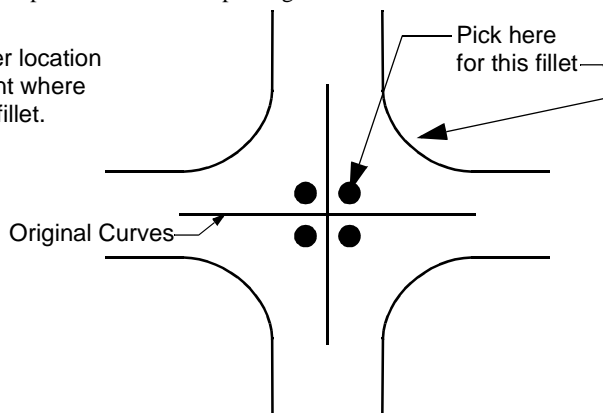


- 5 C) Select **MODIFY-FILLET**. The Fillet Curve Dialog Box requires input of the two curves and a location. FEMAP uses the location on screen that you select the curve at to determine which of the four possible fillets between two curves should be used. When picking the first curve on the right side, move the mouse to a location slightly inside the rectangle, towards the left side of the curve. You will notice the line highlighting, giving you a preview of exactly which curve will be picked. When you have the mouse in the position indicated, press the left mouse button to pick the curve. Pick the top curve, again from inside the rectangle. Make the fillet radius 0.3. Continue until all four corners are filleted.



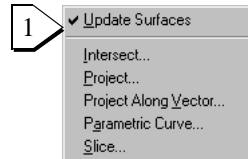
### Note:

By picking inside the lines you specify a fillet radius whose center will be toward the sides of the picks. The effect of picking on other sides is illustrated below:

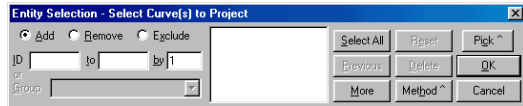
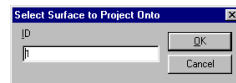


**Project onto Surface**

- 1 Choose **GEOMETRY - CURVE-FROM-SURFACE - UPDATE SURFACES** if it is not already checked. This is so that the curves projected onto the surface will imprint on the surface.

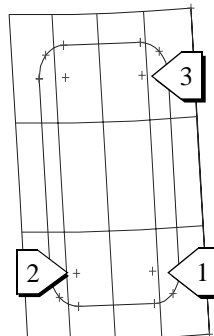
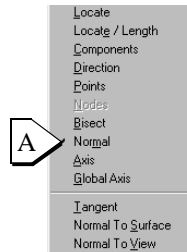
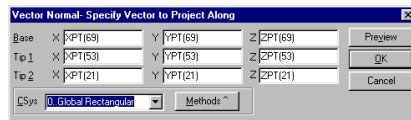


- 2 Choose **GEOMETRY - CURVE FROM SURFACE - PROJECT ALONG VECTOR**. Select the surface and press OK.



- 3 The standard entity selection dialog box appears and prompts you for curves to project. Select the eight curves that form the cutout (4 lines and 4 arcs) and press OK.

- 4 The standard vector dialog box appears. A) press the methods button and choose normal. You now pick three locations to define a plane, the normal of which will be the vector of projection. B) Set your snap mode to points and C) Choose points 1, 2, and 3 as shown. Rotate the view if necessary and press the preview button to be sure the vector is pointing at the surface. If not, re-select the points remembering the right hand rule to define the vector direction. Press OK when done.

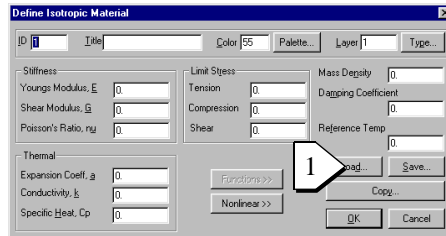


## 10.2 Materials, Properties and Elements

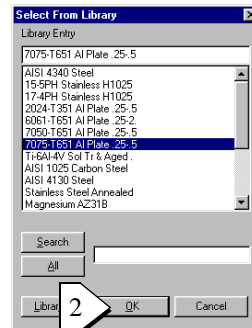
### Mesh the Surface

#### Create the Material and Property

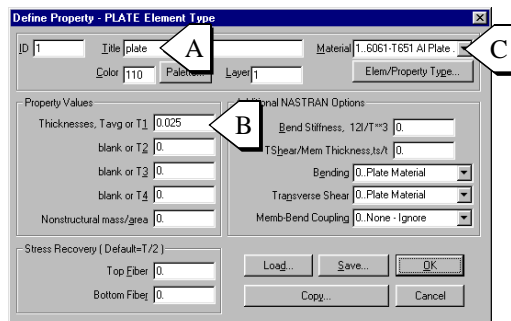
- 1 Choose **MODEL-MATERIAL**. You can enter in values or press the load button to bring up the material library.



- 2 The material library shipped with FEMAP contains material properties using English units (lb, in, sec). You can create your own materials and store them in this library or create your own library. For this example select a material from this library and press OK.



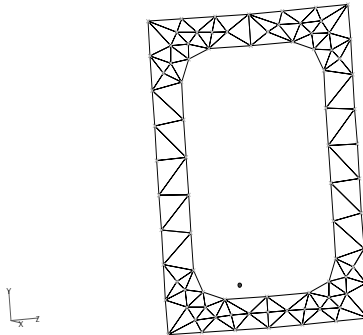
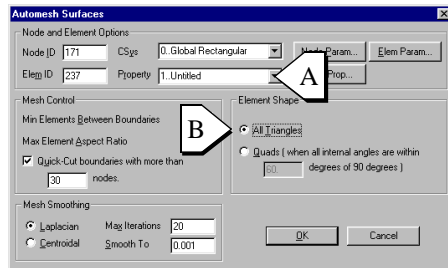
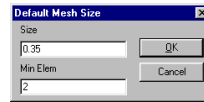
- 3 Press OK in the define material dialog box to create the material. The dialog box stays up and allows you to create another material. We only need one for this example so press cancel to exit the command.
- 4 Choose **MODEL-PROPERTY**. A) name the property plate. B) Enter a thickness of 0.025 C) Select material 1 from the drop down list. D) Press OK to create the material. The dialog box stays up and allows you to create another property. Press cancel or ESC to exit the command.





### Mesh the Surface

- 1 Choose **MESH-MESH CONTROL-DEFAULT SIZE**. Make the size 0.35 and the minimum elements 2 and press OK.
- 2 Choose **MESH-GEOMETRY-SURFACE**. Select the outer surface and press OK.
- 3 The automesh surface dialog box appears. A) Choose the plate property from the drop down list. B) Make the element shape all triangles and press OK.



## Add Beam Stiffeners

### Create the Beam Property

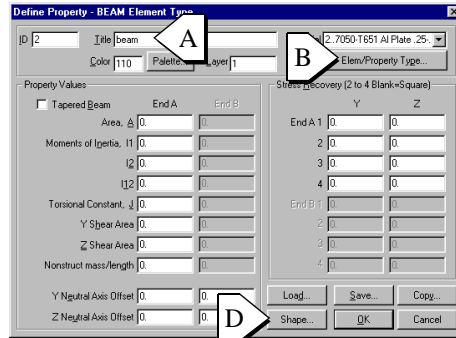
- 1 Choose **MODEL-PROPERTY**.

A) Name the property beam.

B) Press the Elem/Prop. button

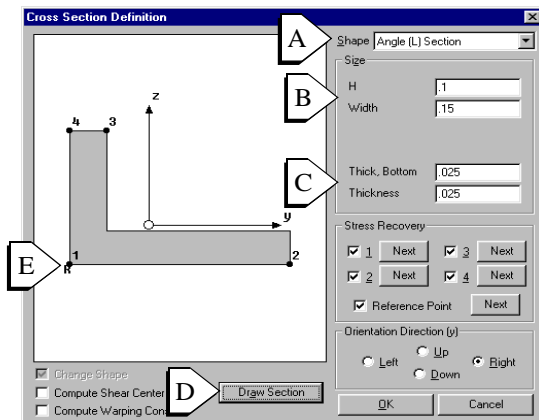
C) Select beam as the property type and press OK.

D) Press the shape button in the Define Property dialog box.



- 2 A) From the drop down list select Angle (L) Section for the shape.  
B) Enter values of 0.1 and 0.15 for height and width.  
C) Make both thicknesses 0.025.  
D) Press the draw section button to draw a cross section of the shape.  
E) Take note that the Reference Point is in the bottom left corner. This will be important when we mesh the curves

Press OK to exit to the Main Beam Element Type selection screen and say OK to create the property.



- 3 We now want to set the offsets so that the beam will sit on top of the plate as shown below. This could be difficult for each individual element, but we will use Mesh Attributes to automatically assign the proper orientations to curves that we will automesh.

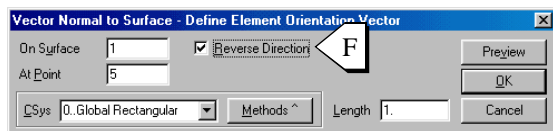
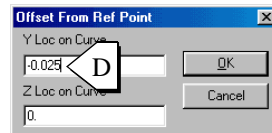
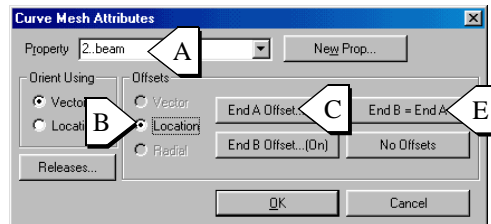
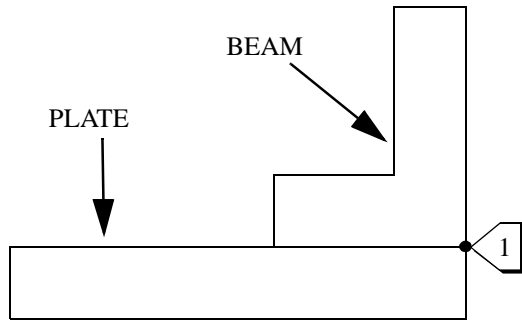
Select **Mesh-Mesh Control-Attributes Along Curve**, select the three curves around the surface, one straight and the two arcs. In the Curve Mesh Attributes dialog box

- (A) pick the beam property we just created  
 (B) pick Location as the type of offset  
 (C) press End A Offset.  
 (E) press End B = End A.

This brings up the Offset from Ref Point dialog box which allows you to locate the beam with respect to its Reference Point. Enter (D) a Y Loc on Curve value of  $-0.025/2$  (half the plate thickness) and press OK.

Back in the Curve Mesh Attributes box press End B = End A (E) and then press OK.

The vector locate dialog box will appear. Change the method to **Normal to Surface**, pick the surface and a point on the straight curve you selected. Check the reverse direction button (F) so that the vector points toward the center of the cylinder.



Repeat the above process on the straight curve you did not select before. **Refer to the command reference for more information on mesh attributes and offsets.**

5 Choose **VIEW-OPTIONS**.

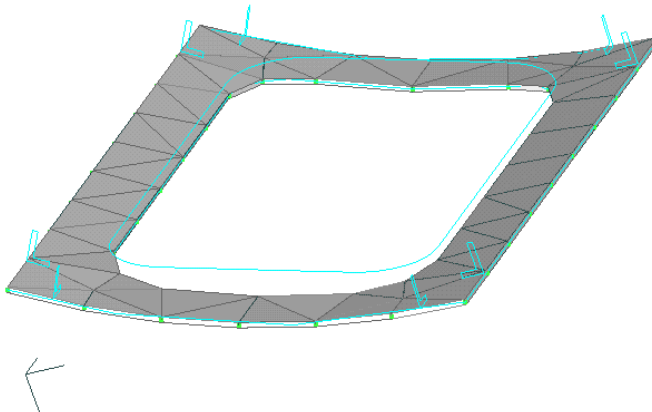
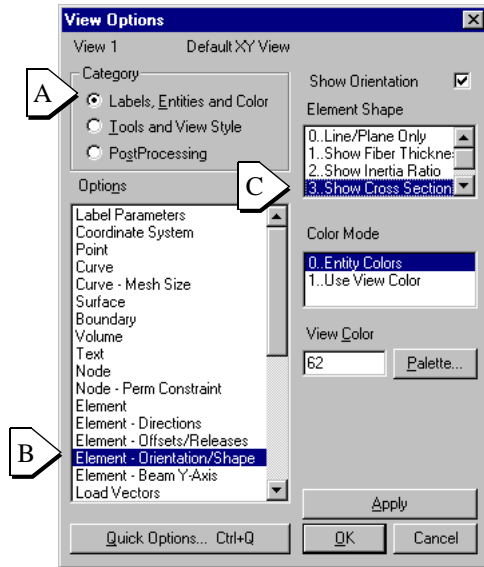
The view options dialog box is displayed.

A) Select the Labels, Entities and Color category,

B) Choose Element-Orient-ation/Shape as the option, and

C) Select Show Cross Section as the Element Shape option.

Press OK and the Cross Sections of the mesh attributes are drawn. The picture should resemble the one below. Notice, the beam cross sections are on top of the plate.



## Generate the Beam Elements

- Choose **MESH-GEOMETRY-CURVE**. Select the four curves along the outside of the rectangular surface. The Geometry Mesh Options dialog box appears. A) Notice that use-meshing-attributes has been selected for you. Press OK to mesh

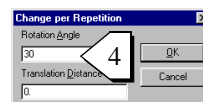
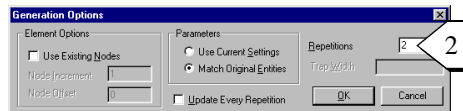
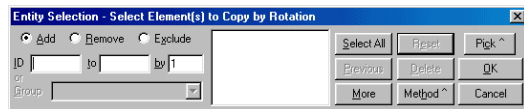


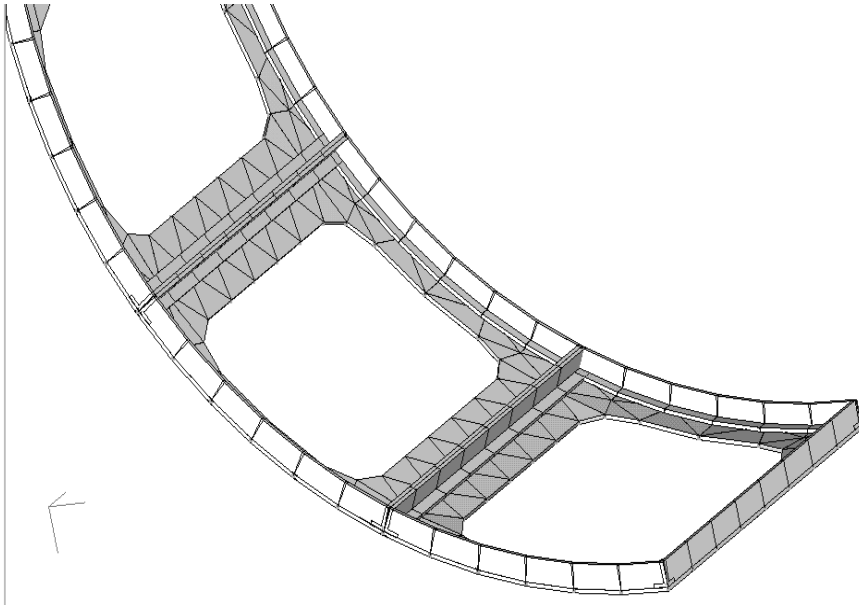
### Note:

Remember, Beam elements have their X-direction defined as going from the first node to the second, the Y-direction defined by the user (usually a vector orientation) and the Z-direction defined by the cross product of the X and Y directions.

## Copy the Elements to Make a Quarter Cylinder

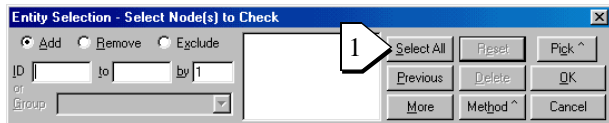
- Select **MESH-ROTATE-ELEMENT**. Select all the elements, beams and plates.
- In the Generation Options dialog box change the number of repetitions to 2 and press OK.
- In the Vector Definition dialog box change the method to locate and enter a vector with a base of 0,0,0 and tip of 0,1,0. Press OK.
- Make the rotation angle 30 degrees. This will make two copies of the elements, each 30 degrees from the previous one.



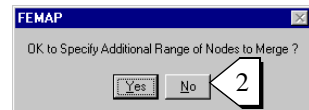


### Merging Coincident Nodes

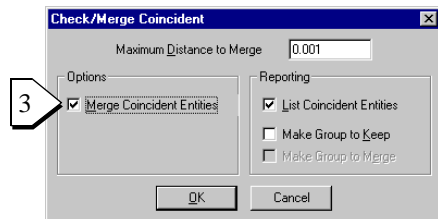
- 1 Select **TOOLS-CHECK - COINCIDENT NODES** from the FEMAP menu. FEMAP prompts you for a list of nodes to check, press **Select All** and then **OK** to continue.



- 2 FEMAP now asks if you would like to select another range of nodes to merge. Answer **No** to continue.



- 3 The **Check/Merge Coincident** Dialog Box is now displayed. Check the **Merge Coincident Entities** box. Press **OK** to continue.



## 10.3 Constraints

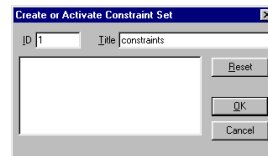
### Adding Constraints

We want to fix the bottom of the cylinder to simulate sitting on the ground. We also need to add symmetry constraints to the ends of the quarter cylinder so it solves as if it were a complete cylinder.

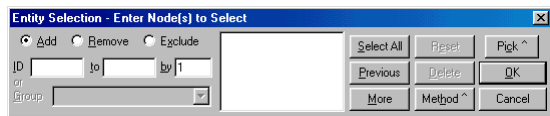
- 1 First we need to rotate the view to facilitate easy node selection. Choose View - Rotate from the FEMAP Menu (or use the Ctrl-R or F8 shortcut keys) and you will see the View Rotate Dialog Box. There are several pre-defined 3-D views that you can select from, you may want to experiment and press some of them. Before leaving View Rotate, press X-Y Top and then OK to dismiss the View Rotate Dialog Box.



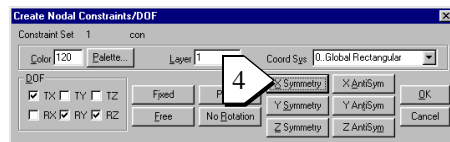
- 2 Select **MODEL-CONSTRAINT**. FEMAP prompts you to create a constraint set. Type in a name and press OK.



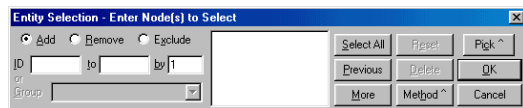
- 3 Hold down the shift key to enable box picking (or use the Pick Menu) and carefully select all the nodes on the left edge of the model.



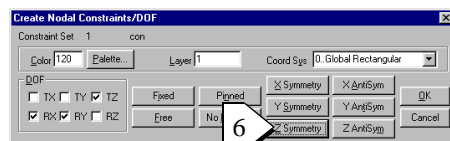
- 4 A) Constrain these nodes by pressing the X-Symmetry button. Press OK.



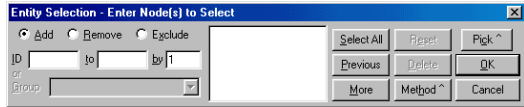
- 5 FEMAP prompts you to select more nodes. Again make a box pick but this time choose the nodes on the right edge of the model. You may need to re-orient your view to do this.



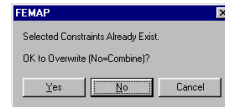
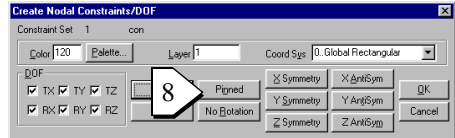
- 6 A) Constrain these by pressing the Z-Symmetry button. Press OK.



- 7 Again FEMAP prompts you for more nodes. This time we want to select all the nodes along the bottom edge. You may need to re-orient the view to make the picking easier. When you have finished picking press OK.



- 8 Completely constrain these nodes by pressing the Fixed button. Press OK. Since some of the nodes on the bottom are the same as those on the side (the corner nodes) FEMAP will ask you if it is OK to overwrite the existing constraints. In this case it does not matter whether you overwrite or combine since the constraints you are adding also constrain the DOF you are overwriting. However, in other cases you may have to be sure one way or the other.

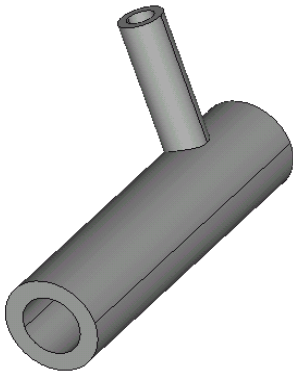


The model is now ready for modal analysis. Try adding a distributed load on the top of the model and do a structural analysis.



# Pipe Intersections

The following example is actually two examples in one. The first will show you how to intersect pipes using surfaces for meshing with plate elements. The second one will make a solid model for solid elements. These will not be complete examples, no loads or constraints will be added, and the models are not meant to resemble any actual parts. These examples are mainly meant to present some new modeling techniques available in FEMAP. To perform this example, you will need to have either the ACIS or Parasolid modeling engine active. If you have the 300-Node version, you will not be able to save your model file or change the model after meshing due to size limitations.



First start FEMAP and create a new model, or if FEMAP is already running, select **FILE - NEW** from the menu.

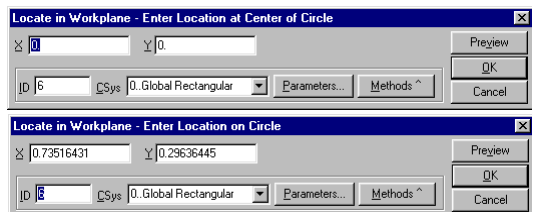
## 11.1 Surface Intersection

### 11.1.1 Geometry

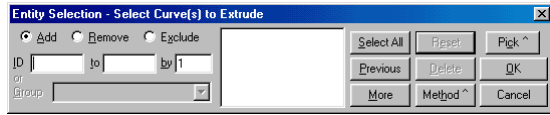
In this example we will create a number of intersecting cylindrical surfaces. We will break the surfaces along their curves of intersection and then mesh the surfaces we need.

#### Create Surfaces

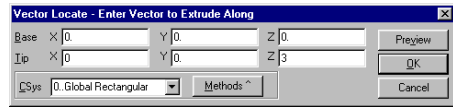
- 1 Choose **GEOMETRY-CURVE-CIRCLE-RADIUS**. The coordinate location dialog box appears and prompts you to enter the center of the circle. Pick a point on the screen and press OK. The box appears again and asks you for a location on the circle. Pick a point about half a unit away from the first point and press OK. A circle should be drawn.



- Choose **GEOMETRY-SURFACE-EXTRUDE**. FEMAP prompts you to select a curve to extrude. Select the circle and press OK.



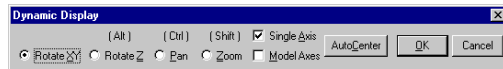
- FEMAP prompts you for a vector to extrude along. Enter a base of 0,0,0 and a tip of 0,0,3 and press OK. This will produce a cylindrical surface 3 units long in the z direction. Press cancel to exit the command.



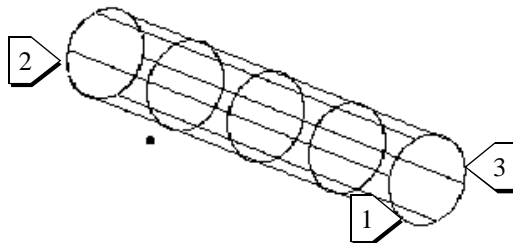
- Press the Dynamic Display button on the toolbar.



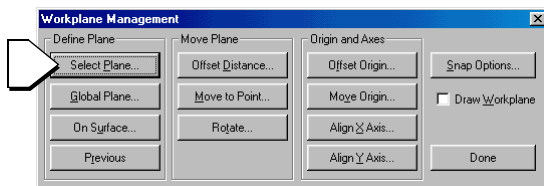
- FEMAP displays the Dynamic Display dialog box at the bottom of the screen.



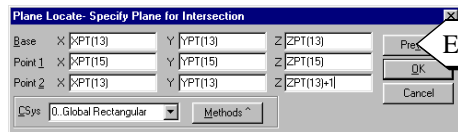
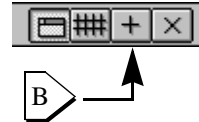
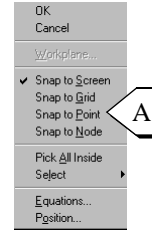
- To dynamically rotate your model, move the cursor inside the graphics window, and then press and drag it left to right and up and down. This will dynamically rotate the model. By pressing and holding the Shift key, and pressing and dragging the mouse up and down, you can scale the view dynamically. Using the Ctrl key in combination with pressing the left mouse button and dragging, you can dynamically pan the view. When you get the model in an orientation similar to the one shown, press OK or the Return key to leave Dynamic Display.



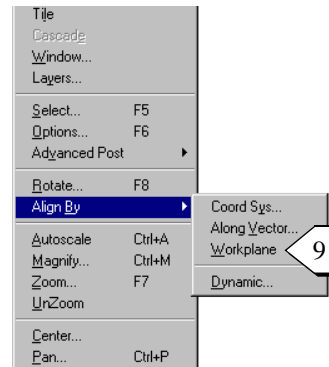
- Choose **TOOLS-WORKPLANE**. Press the select plane button. The standard plane definition dialog box appears.



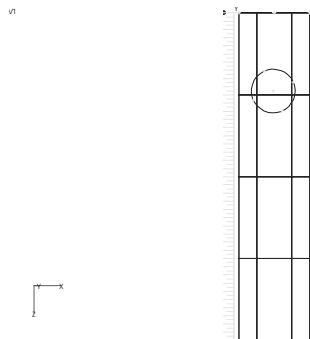
- 8 First, change the snap mode to point, by either  
 A) Pressing the right mouse button in the graphics, and then clicking the point option under Snap To, or B) Using the toolbar Snap To Point icon, or C) Using the Ctrl-P keyboard shortcut. D) Choose point 1, then 2, then 3 as shown in the above diagram. E) Press the preview button to be sure the plane is slicing the cylinder in half. Now press OK.



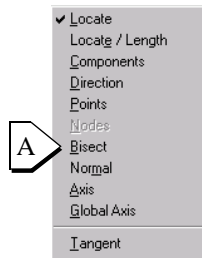
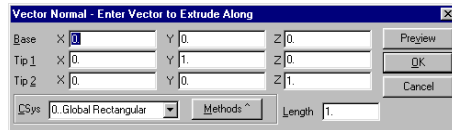
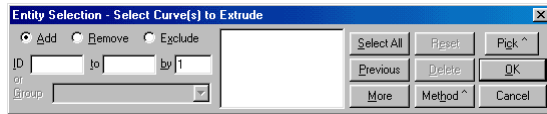
- 9 Choose **VIEW-ALIGN BY-WORKPLANE** to orient the view so you are looking directly at the workplane.



- 10 Set your snap mode back to screen. Repeat the **GEOMETRY-CURVE-CIRCLE-RADIUS** command and put a circle on the workplane somewhere inside the boundary of the cylinder as shown.

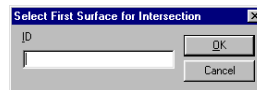
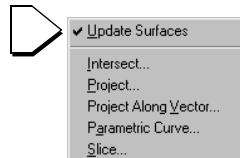


- 11 Choose **GEOMETRY-SURFACE-EXTRUDE**. FEMAP prompts you to select a curve to extrude. Select the circle and press OK.
- 12 FEMAP prompts you for a vector to extrude along.
  - A) Press the methods button and choose normal.
  - B) Set your snap mode back to points.
  - C) Pick three points on the circle you are extruding to define a vector normal to the circle.
  - D) Change the length to 1.5 and press OK. Press cancel to exit the command. You should now have two cylindrical surfaces.

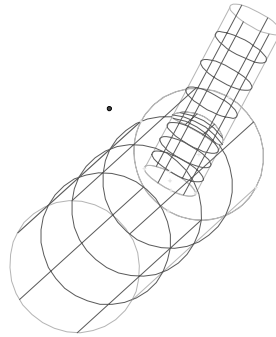
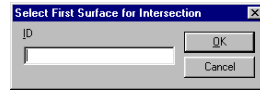


## Intersect Surfaces

- 1 Choose **GEOMETRY - CURVE-FROM-SURFACE-UPDATE SURFACES** if it is not already checked. This is so that the curves created on the surface will split the surface.
- 2 Choose **GEOMETRY-CURVE FROM SURFACE-INTERSECT**, select one of the surfaces and hit OK.

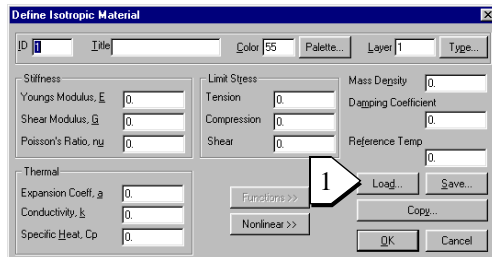


- 3 Select the other surface and press OK. A curve should be created where the two surfaces intersect and the surfaces will be split along this curve. We now have surfaces we can mesh to form intersecting pipes.

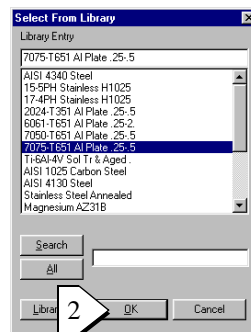


## Create Material and Properties

- 1 Choose **MODEL-MATERIAL**. You can enter in values or press the load button to bring up the material library.

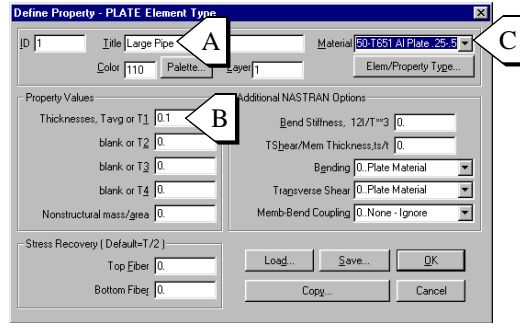


- 2 The material library shipped with FEMAP contains material properties using English units (lb, in, sec). You can create your own materials and store them in this library or create your own library. For this example select a material from this library and press OK.

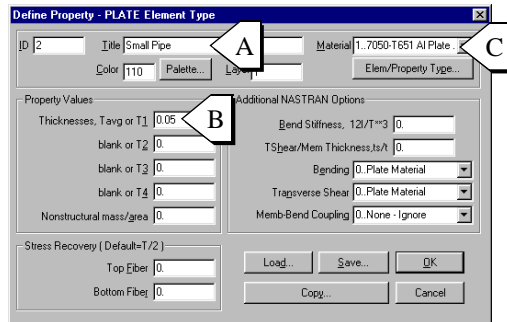


- 3 Press OK in the define material dialog box to create the material. The dialog box stays up and allows you to create another material. We only need one for this example so press cancel to exit the command.

- Choose **MODEL-PROPERTY**. A) Name the property large pipe. B) Enter a thickness of 0.1. C) Select material 1 from the drop down list. D) Press OK to create the material. The dialog box stays up and allows you to create another property.



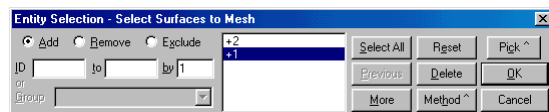
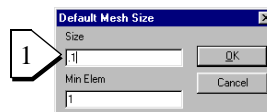
- A) Name the second property small pipe. B) Enter a thickness of 0.05. C) Select material 1 from the drop down list and press OK to create the material. The dialog box stays up and allows you to create another property, but we are done so press cancel to exit the command.



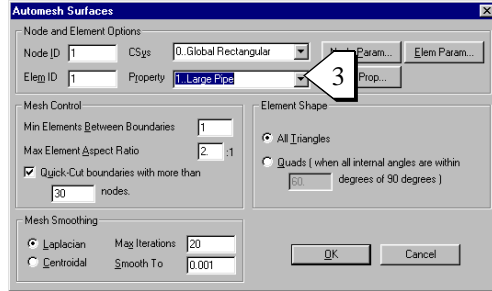
## 11.1.2 Materials, Properties and Elements

### Meshing the Surfaces

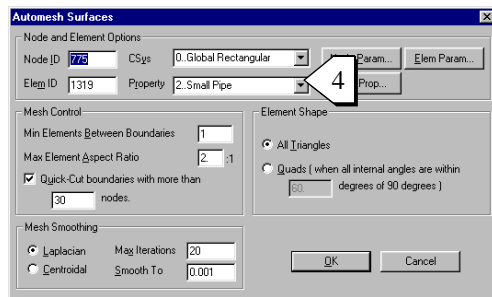
- Choose **MESH-MESH CONTROL-DEFAULT SIZE**. Enter a size of 0.1 and press OK. This size will give us an adequate mesh.
- Choose **MESH-GEOMETRY-SURFACE**. Select the two large surfaces of the longer pipe and press OK.



- 3 The automesh surfaces dialog box appears. A) Select the Large Pipe property from the drop down list and press OK.



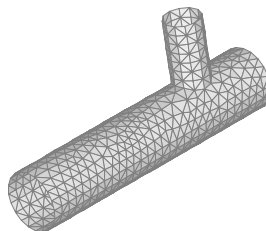
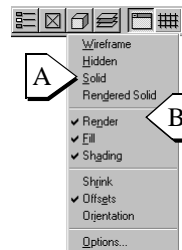
- 4 Choose **MESH-GEOMETRY-SURFACE**. Select the two sections of the surfaces of the shorter pipe that stick out above the longer pipe and press OK. Select the Small Pipe property from the drop down list and press OK.



### Clean up the View

Press Ctrl-Q to bring up the FEMAP Quick View Options Dialog Box. Select Geometry Off, Labels Off and press Done.

- 1 A) Press the view style button on the toolbar and choose solid. B) Press the view style button on the toolbar and choose Render. You can now dynamically rotate the model in render mode simply by holding the left mouse down and dragging it in the graphics window.

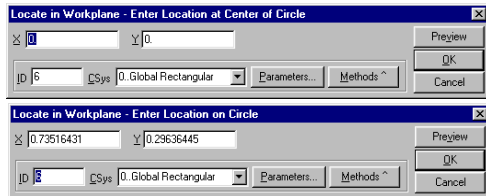


## 11.2 Solid Intersection

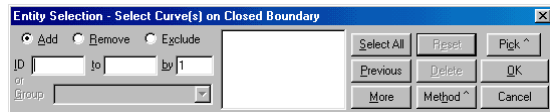
### 11.2.1 Geometry

#### Create Solid

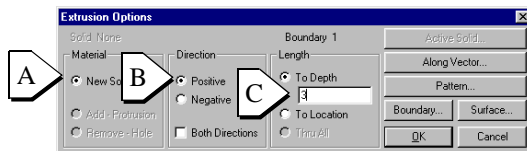
- 1 First start a new model with the **FILE-NEW** command. Choose **GEOMETRY-CURVE CIRCLE-RADIUS**. The coordinate location dialog box appears and prompts you to enter the center of the circle. Pick a point on the screen and press OK. The box appears again and asks you for a location on the circle. Pick a point about half a unit away from the first point and press OK. A circle should be drawn. Press cancel to exit the command.



- 2 Choose **GEOMETRY-BOUNDARY SURFACE**. Select the circle you just drew and press OK.



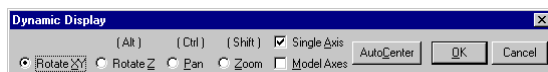
- 3 Choose **GEOMETRY - SOLID - EXTRUDE**. A) Make it a new solid. B) The direction should be positive. C) Make it to a depth of 3.



- 4 Press the Dynamic Display button on the toolbar. (If you switch to Render mode first, you can dynamically rotate the model without going into Dynamic Display - all other options remain the same).

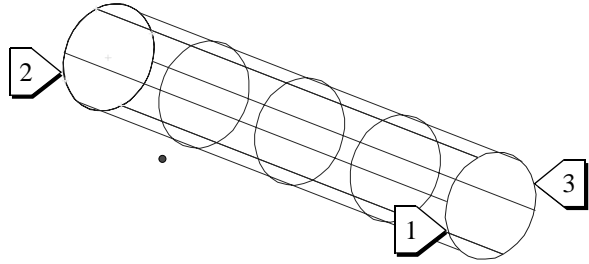


- 5 FEMAP displays the Dynamic Display dialog box at the bottom of the screen.

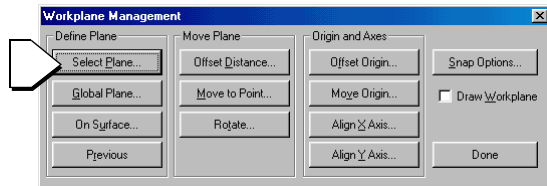




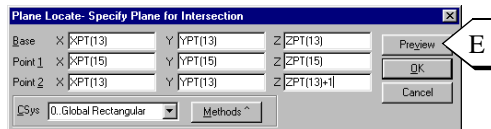
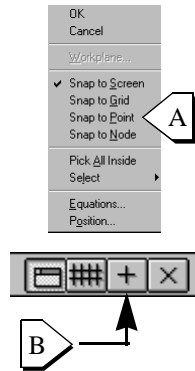
- 6 To dynamically rotate your model, move the cursor inside the graphics window, and then press and drag it left to right and up and down. This will dynamically rotate the model. By pressing and holding the Shift key, and pressing and dragging the mouse up and down, you can scale the view dynamically. Using the Ctrl key in combination with pressing the left mouse button and dragging, you can dynamically pan the view. When you get the model in an orientation similar to the one shown, press OK or the Return key to leave Dynamic Display.



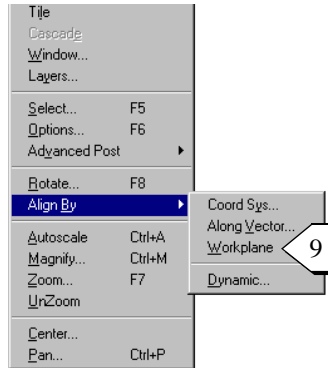
- 7 Choose **TOOLS-WORKPLANE**. Press the select plane button. The standard plane definition dialog box appears.



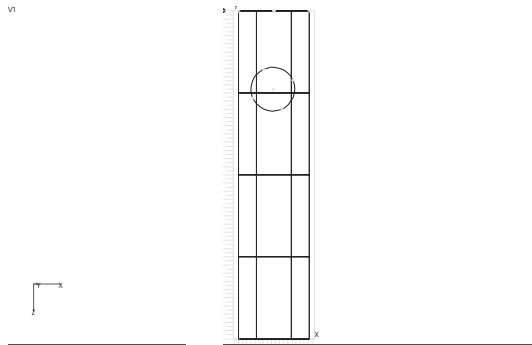
- 8 First, change the snap mode to point, by either A) Pressing the right mouse button in the graphics, and then clicking the point option under Snap To, or B) Using the toolbar Snap To Point icon, or C) Using the Ctrl-P keyboard shortcut. D) Choose point 1, then 2, then 3 as shown in the above diagram. E) Press the preview button to be sure the plane is slicing the cylinder in half. Now press OK.



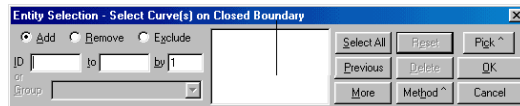
- 9 Choose **VIEW-ALIGN BY-WORKPLANE** to orient the view so you are looking directly at the workplane.



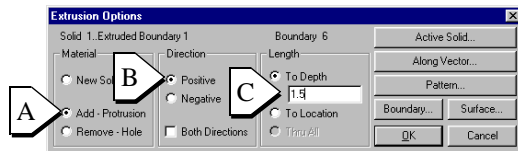
- 10 Set your snap mode back to screen. Repeat the **GEOMETRY-CURVE-CIRCLE-RADIUS** command and put a circle on the workplane somewhere inside the boundary of the cylinder as shown.



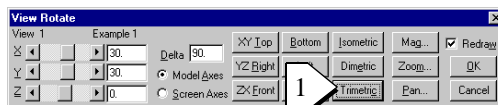
- 11 Choose **GEOMETRY-BOUNDARY SURFACE**. Select the circle you just drew and press OK.



- 12 Choose **GEOMETRY - SOLID - EXTRUDE**. A) Make it add. B) The direction should be positive C) Make it to a depth of 1.5.

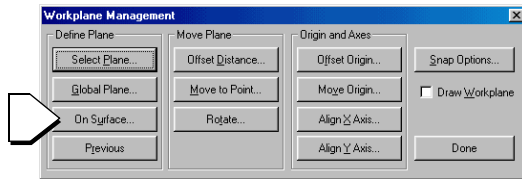


- 13 Choose **VIEW - ROTATE** from the FEMAP Menu (or use the Ctrl-R or F8 shortcut keys) and you will see the View Rotate Dialog Box. There are several pre-defined 3-D views that you can select from. You may want to experiment and press some of them. Before leaving View Rotate, press Isometric and then OK to dismiss the View Rotate Dialog Box.

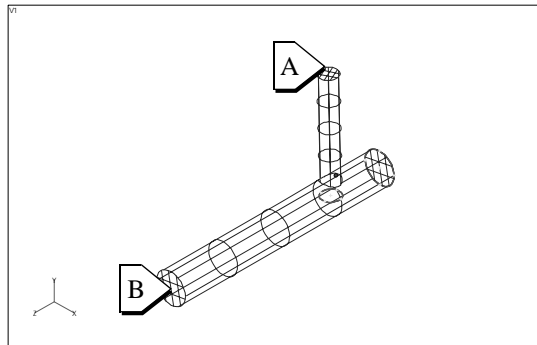
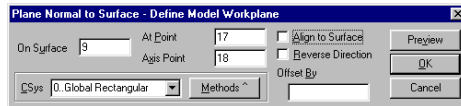


## Removing the inner Material

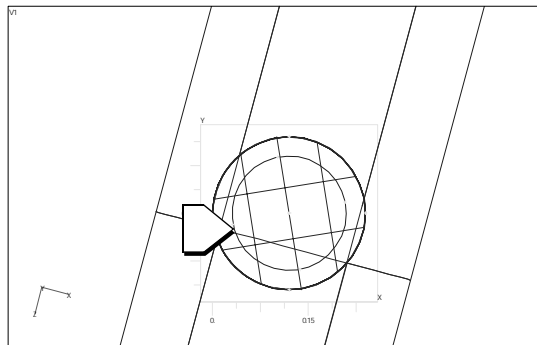
- 1 Choose **TOOLS-WORK-PLANE**. Press the on surface button. The standard plane definition dialog box appears.



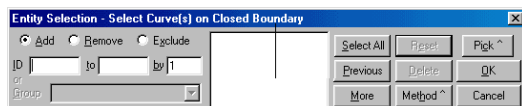
- 2 The Plane Normal to Surface dialog box appears. A) Select surface A in the diagram below and choose two points on the circle. Align the view to the workplane.



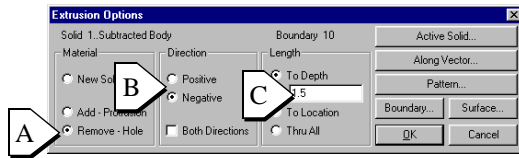
- 3 Choose **GEOMETRY-CURVE-CIRCLE-RADIUS**. Change the method to Center, pick the circle, and FEMAP will select the center of the current circle to be the center of the new circle. Then change the method back to Locate, and with the Snap To on Screen pick a point inside the circle to give you the proper wall thickness. Something similar to that shown.



- 4 Choose **GEOMETRY-BOUNDARY SURFACE**. Select the circle you just drew and press OK.



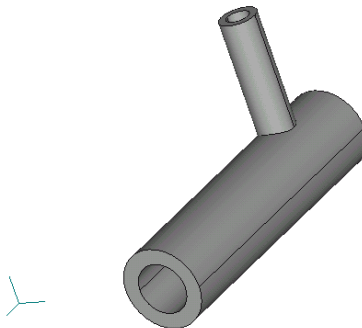
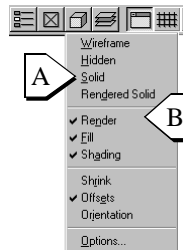
- 5 Choose **GEOMETRY - SOLID - EXTRUDE**. A) Make it remove. B) The direction should point into the solid, either positive or negative. Make it both if you cannot tell. C) Make it to a depth of 1.5.



- 6 Repeat the above procedure on surface B but when extruding make the length through all. You should now have a complete solid model of two pipes intersecting

### Change the View

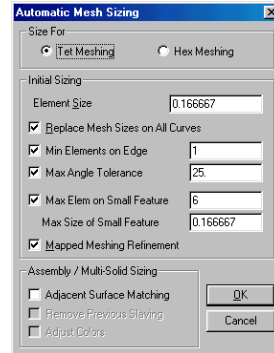
- 1 A) Press the view style button on the toolbar and choose solid. B) Press the view style button on the toolbar and choose Render.



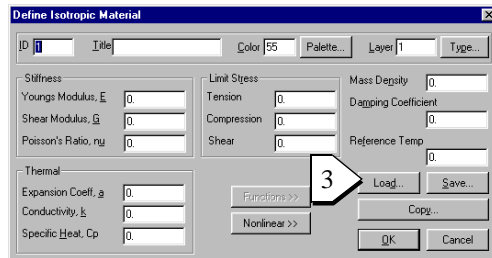
## 11.2.2 Meshing the Solid

- 1 Choose **MESH-GEOMETRY-SOLIDS**.

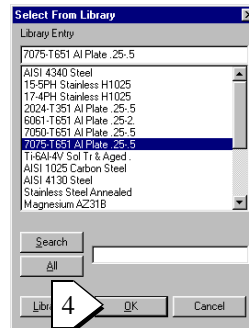
- 2 The default values FEMAP calculated for Tet meshing this model are fine. Press OK to accept them. Feel free to experiment with different mesh sizes to become familiar with the meshing process.



- 3 Since no material has been created FEMAP prompts you to make one. You can enter in values or press the Load button to bring up the material library.



- 4 The material library shipped with FEMAP contains material properties using English units (lb, ft, sec). You can create your own materials and store them in this library or create your own library. For this example select a material from this library and press OK.

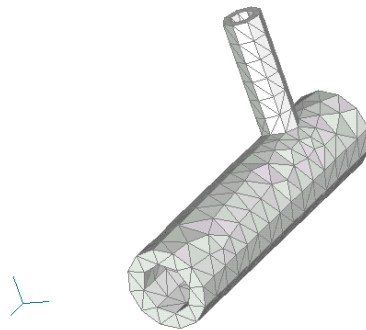


**Note:**

Remember, there are no units in FEMAP. All dimensions must be kept consistent with the unit system you use to define your material properties. Always make sure this is correct from the beginning because there is no way to correct inconsistencies in units once the model is built.

- 5 Press OK in the define material dialog box when the properties have been loaded.

- 6 The automesh solids dialog box appears. Leave the values as the defaults and press OK.

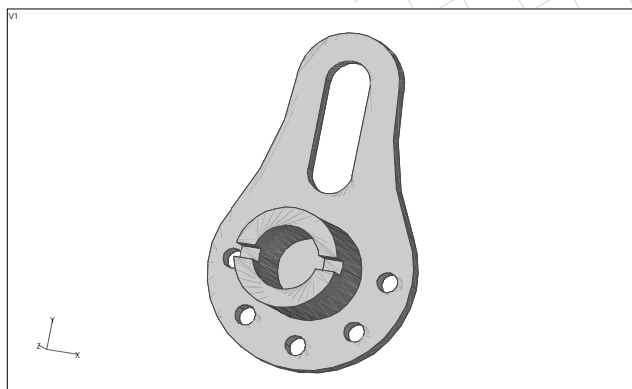


# Slotted Guide

# 12

This example will build a solid model from beginning to end completely in FEMAP. We will then run the solid mesher and add loads and constraints.

To perform this example, you will need to have either the ACIS or Parasolid modeling engine active. If you have the 300-Node version, you will not be able to save your model file or change the model after meshing due to size limitations.



First start FEMAP and create a new model, or if FEMAP is already running, select **FILE - NEW** from the menu.

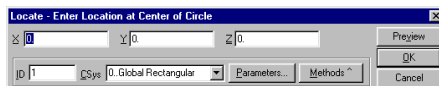
## 12.1 Creating the Geometry

### The Base Curves

- 1 Select **GEOMETRY-CURVE CIRCLE-RADIUS**.

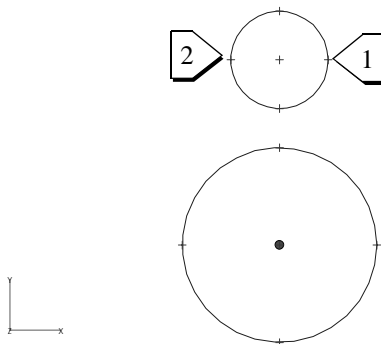
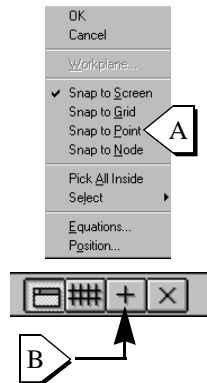
Enter a point of 0,0,0 for the center, press OK.

Enter 50,0,0 for the radius and press OK.

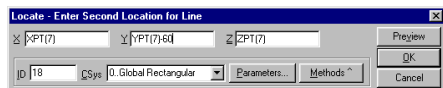
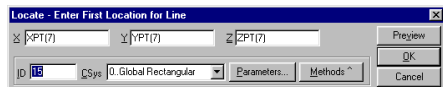
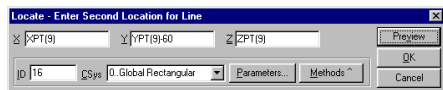
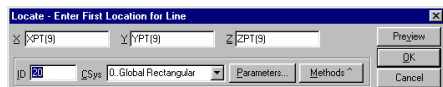


- 2 The command allows you to enter another circle. Enter a point of 0,95,0 for the center and press OK. Enter 25,95,0 for the radius and press OK. Press cancel after both circles are drawn to exit the command.

- 3 We need to change the snap mode to point. Do this by either A) pressing the right mouse button in the graphics window, and then clicking the point option under Snap To, or B) Using the toolbar Snap To Point icon, or C) Using the Ctrl-P keyboard shortcut.

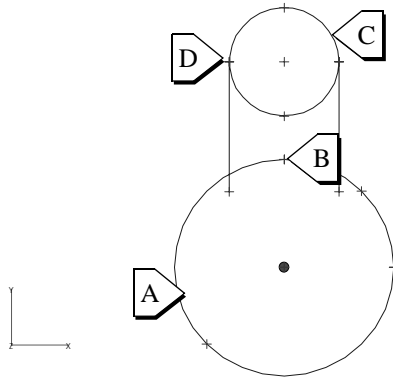


- 4 Select **GEOMETRY-CURVE LINE-PROJECT POINTS**. The standard coordinate location dialog box is shown. Pick point 1 in the previous diagram and press OK.
- 5 FEMAP prompts you for a second point. Pick point 1 again but before you press OK subtract 60 from the Y value in the dialog box.
- 6 The dialog box stays active. Pick point 2 in the previous diagram and press OK.
- 7 FEMAP prompts you for a second point. Pick point 2 again but before you press OK subtract 60 from the Y value in the dialog box.

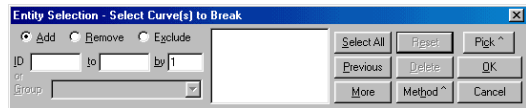




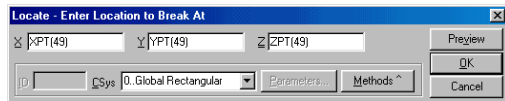
- 8 We now need to break the circles so we can fillet the curves and make this a single boundary.



- 9 Select **MODIFY-BREAK**. Pick circle A from the previous diagram and press OK.



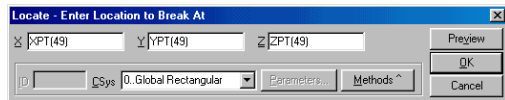
- 10 Select point B as the location to break at and press OK.



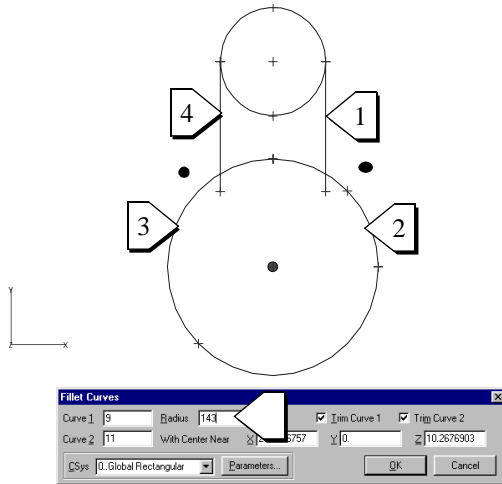
- 11 The command repeats. Pick circle C in the selection box and press OK.



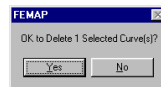
- 12 Select point D as the location to break at and press OK. Press cancel to exit the command.



- 13 Set your snap mode back to screen. Choose **MODIFY-FILLET**. The fillet curves dialog box is shown. Pick curves 1 and 2 from the side shown, enter a radius of 143 and press OK.
- 14 The command repeats. Pick curves 3 and 4 from the side shown and press OK. Press cancel to exit the command.



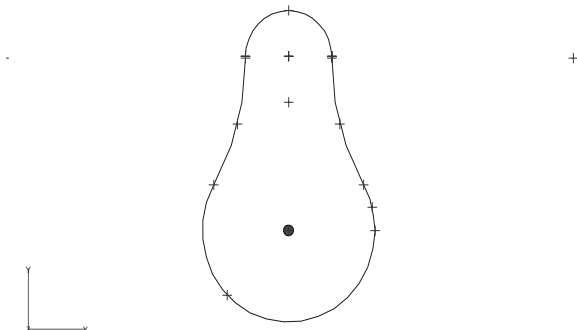
- 15 Select **DELETE-GEOMETRY-CURVE**. Pick the inside half of the top circle and press OK. Press yes to delete the curve.



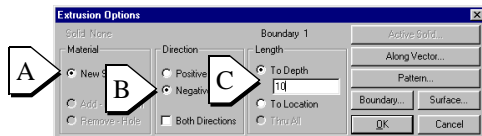
## The Solid

First we must create a boundary from the curves we just created, and then extrude the boundary to form a solid part.

- 1 Select **GEOMETRY-BOUNDARY SURFACE**. Select the five curves and press OK.



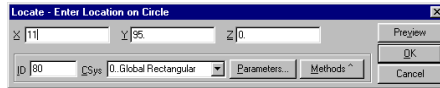
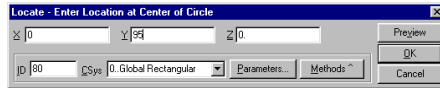
- 2 Choose **GEOMETRY-SOLID-EXTRUDE**. Since there is only one boundary it will be selected automatically. Make it A) New solid, B) Negative and C) To a depth of 10.



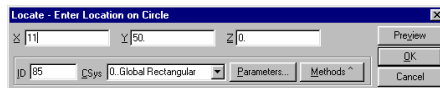
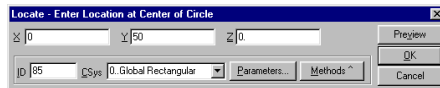
**The Slot at the Top**

- 1 Select **GEOMETRY-CURVE CIRCLE-RADIUS**.

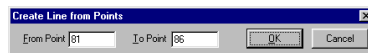
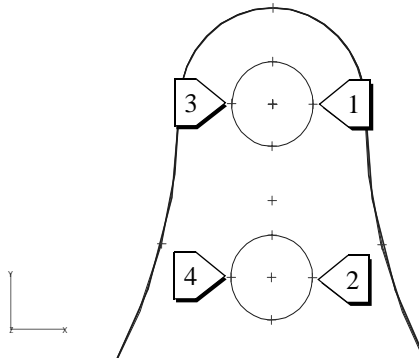
Enter the point 0,95,0 for the center and 11,95,0 for a location on the circle.



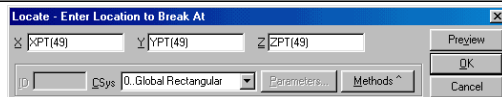
- 2 The command repeats. This time enter the point 0,50,0 for the center and 11,50,0 for a location on the circle. Press cancel when done to exit the command.



- 3 Select **GEOMETRY-CURVE LINE-POINTS**. Pick points 1 and 2 and press OK.
- 4 Pick points 3 and 4 and press OK. Press cancel to exit the command.

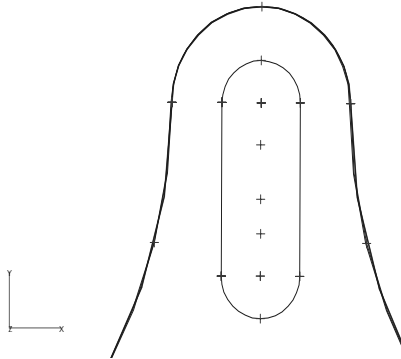


- 5 Set your snap mode to points. Select **MODIFY-BREAK**. Pick the lower circle and break it at point 4. Pick the upper circle and break it at point 3.

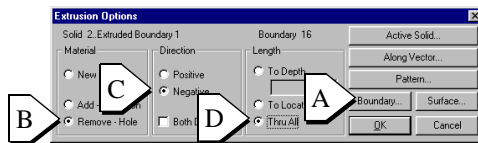


- 6 Select **DELETE-GEOMETRY-CURVE**. Pick the two inside halves of the circles and press OK. Press Yes to delete them.

- 7 Select **GEOMETRY-BOUNDARY SURFACE**. Select the four curves that make up the slot and press OK.



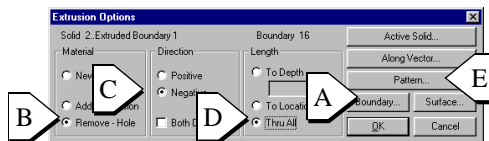
- 8 Choose **GEOMETRY-SOLID-EXTRUDE**. A) Press the boundary button and pick the boundary you just created. Make it B) remove-hole C) negative and D) through all.



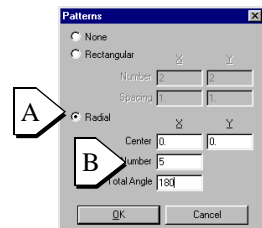
### The Five Bolt Holes

- 1 Select **GEOMETRY-CURVE CIRCLE-RADIUS**. Enter a point of -38,0,0 for the center and 38,5,0 for a point on the circle.
- 2 Select **GEOMETRY-BOUNDARY SURFACE**. Select the circle and press OK.

- 3 Choose **GEOMETRY-SOLID-EXTRUDE**. A) Press the boundary button and pick the boundary you just created. Make it B) Remove-hole C) Negative and D) Through all. E) Press the pattern button.

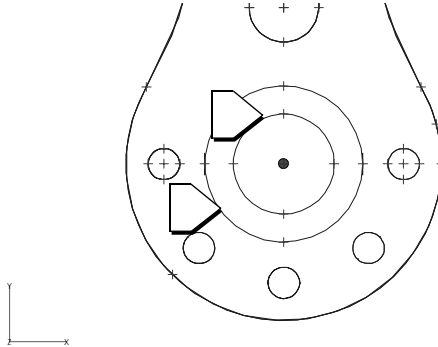


- 4 A) Make the pattern radial with B) a center of 0,0 make 5 copies and through a total angle of 180. The angle is considered positive counter-clockwise around the workplane normal. That is why we made the hole in the negative X. Press OK in both dialog boxes to make the holes.

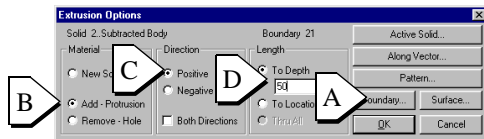


**The Guide Boss**

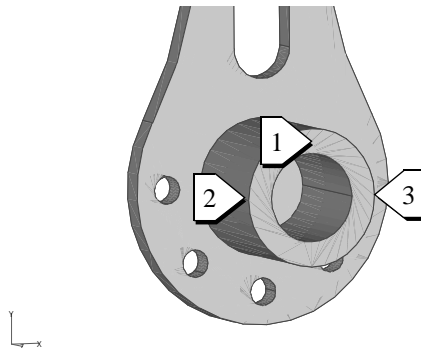
- 1 Select **GEOMETRY-CURVE CIRCLE-RADIUS**. Enter points of 0,0,0 and 25,0,0 for the first circle and 0,0,0 and 16,0,0 for the second circle.
- 2 Select **GEOMETRY-BOUNDARY SURFACE**. Select the two circles and press OK.



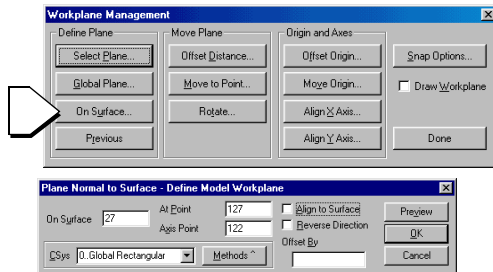
- 3 Choose **GEOMETRY-SOLID-EXTRUDE**. A) Press the boundary button and pick the boundary you just created. Make it B) Add-protrusion C) Positive and D) To a depth of 50.

**Slot in Guide Boss**

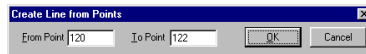
- 1 We want to put our work-plane at the top of the guide boss so we can build the slot. First, change the snap mode to point.



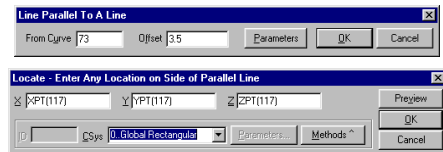
- 2 Select **TOOLS-WORK-PLANE**. Press the On Surface button. Pick surface 1, and points 2 and 3 as shown in the above diagram and press OK.



- 3 Select **GEOMETRY-CURVE LINE-POINTS**. Select points 2 and 3 in the diagram and press OK.



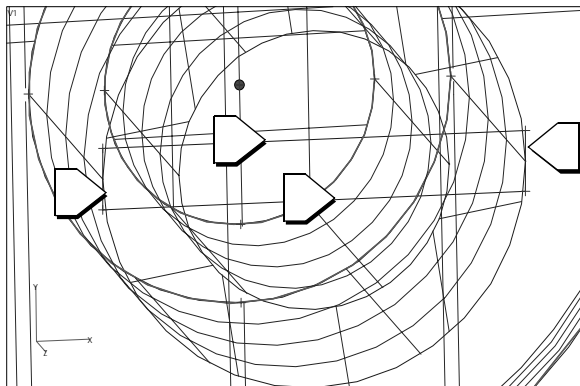
- 4 Select **GEOMETRY-CURVE LINE-PARALLEL**. Select the curve you just drew and enter an offset of 3.5. Press OK. Pick a point to one side of the curve and press OK. The command repeats. Select the same curve, use the same offset but this time pick a point on the other side. Press OK and then cancel to exit the command.



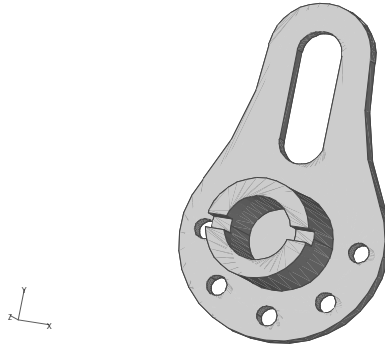
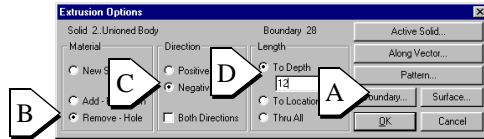
- 5 We want to use the two offset lines we created to form a rectangle. Select **GEOMETRY-CURVE LINE-POINTS**. Select an end point of each line on one side to draw one line and an end point of each line on the other side to draw the other line.



- 6 Select **GEOMETRY-BOUNDARY SURFACE**. Select the four curves of the rectangle and press OK.



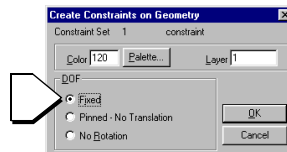
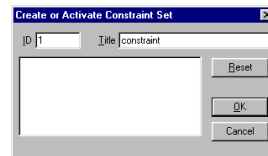
- 7 Choose **GEOMETRY-SOLID-EXTRUDE**. A) Press the boundary button and pick the boundary you just created. Make it B) Remove-hole C) Negative and D) To a depth of 12.



## 12.2 Loads and Constraints

### Add Constraints to Geometry

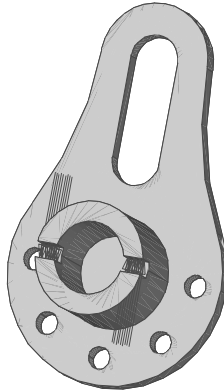
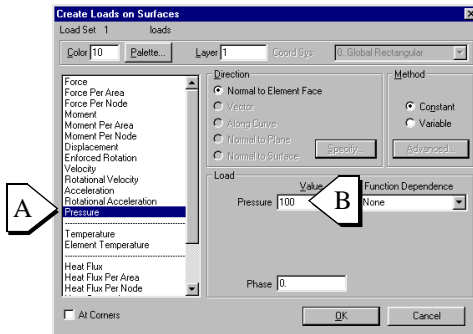
- 1 Select **MODEL-CONSTRAINT-ON SURFACE**. FEMAP prompts you to create a constraint set. Type a name and press OK.
- 2 Select the half cylinder at the top of the slot and press OK. Fix this surface.



### Add Loads to Geometry

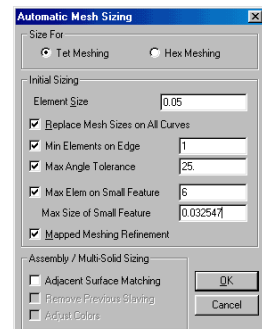
- 1 We want to simulate a load resulting from twisting a rod inserted into the guide boss. Select **MODEL-LOAD-ON SURFACE**. Name the load set and press OK. Select two surfaces, one on each side of the guide boss slot, facing in opposite directions.

- 2 A) Make the load type pressure and B) Give it a value of 100. Press OK. Press cancel to exit the command.



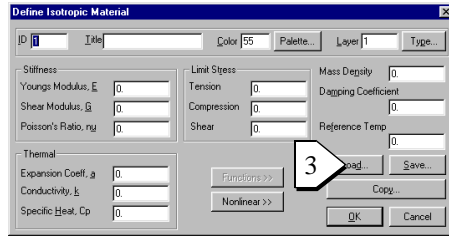
## 12.3 Meshing the Solid

- 1 Choose **MESH-GEOMETRY-SOLIDS**.
- 2 The default values for solid meshing determined by FEMAP are usually adequate to produce a good tetrahedral mesh. However, as you gain experience with the solid mesher you may find that a slightly larger element size will still give you a good mesh but greatly reduce the number of elements. On the other hand some parts may need a smaller element size to produce a good mesh in certain areas. Also keep in mind that you can specify mesh spacing and mesh hard points on all curves and surfaces individually. This is often the best way to get the best mesh although it does take more time and careful planning.

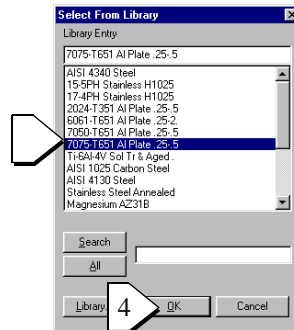




- 3 Since no material has been created FEMAP prompts you to make one. You can enter in values or press the Load button to bring up the material library.



- 4 The material library shipped with FEMAP contains material properties using English units (lb, in, sec). You can create your own materials and store them in this library or create your own library. **If you use a FEMAP material for this example your results will be wrong. The geometry was created in millimeters.** However if you have no metric materials available you may use a FEMAP material to complete the problem as long as you remember any solution will be wrong.

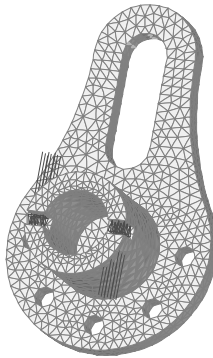
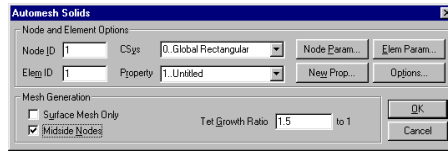


**Note:**

Remember, there are no units in FEMAP. All dimensions must be kept consistent with the unit system you use to define your material properties. Always make sure this is correct from the beginning because it is extremely difficult to correct inconsistencies in units once the model has been built.

- 5 Press OK in the define material dialog box when the properties have been loaded.

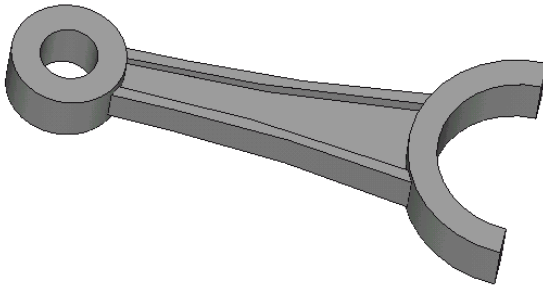
- 6 The automesh solids dialog box appears. Leave the values as the defaults and press OK.



The model is now ready for analysis.

# Connecting Rod

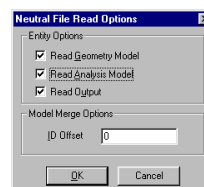
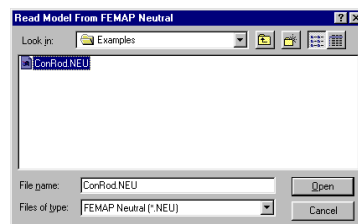
This example will demonstrate some of FEMAP's solid modeling capabilities. We will start by reading in a neutral file that contains the base curves we will use to build the solid. These curves were created in FEMAP and you could build this model from scratch in FEMAP, but to save time and get right to the solid modeling we have provided the curves for you. First start FEMAP and create a new model, or if FEMAP is already running, select **FILE - NEW** from the menu.



## 13.1 Creating the Geometry

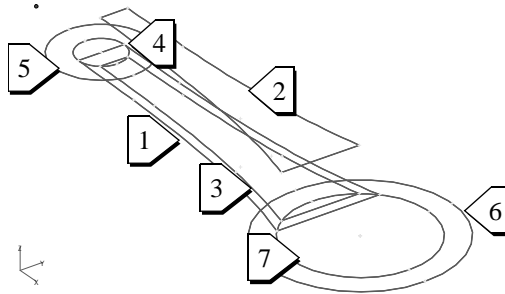
### Reading the Neutral File

- 1 Choose **FILE-IMPORT-FEMAP NEUTRAL**. The Windows File Open Common Dialog Box appears. Navigate to the \Examples directory and A) select the ConRod.neu file and B) press open.
- 2 Press OK to accept the default values in the neutral file read options dialog box.



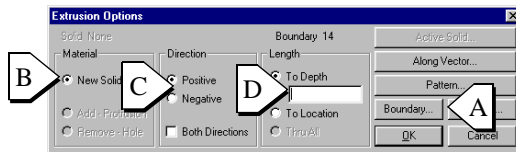
### Creating the Solid

The geometry you read in from the neutral file contains all the existing curves and boundaries you will need to form the solid model. As I said before, this geometry was all created in FEMAP and is not hard to duplicate but those commands are covered in other examples.

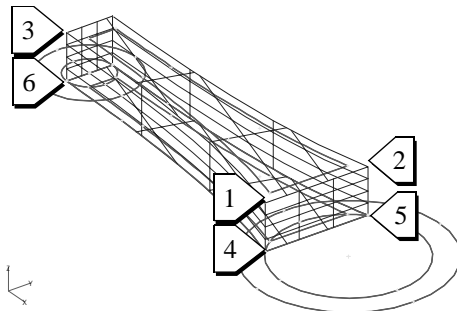
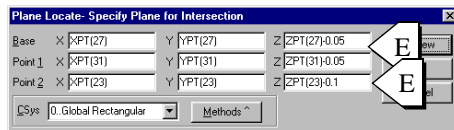
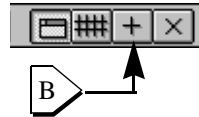
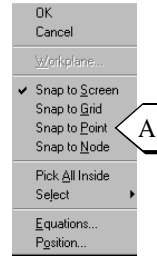


### Extruding the Beam

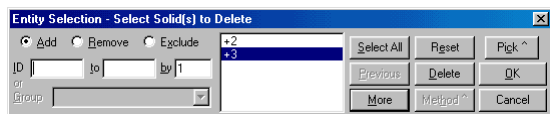
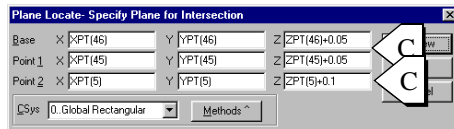
- 1 Choose **GEOMETRY - SOLID - EXTRUDE**. A) Press the boundary button and select boundary 1. B) This should be a new solid, in the C) Positive direction and D) To a depth of 0.5. Check these values and press OK.
- 2 Choose **GEOMETRY - SOLID - SLICE**. Pick the solid just created and press OK.



- 3 Now you need to specify a cutting plane. The standard plane definition dialog box appears. First, change the snap mode to point, by either A) Pressing the right mouse button in the graphics, and then clicking the point option under Snap To, or B) Using the toolbar Snap To Point icon, or C) Using the Ctrl-P keyboard shortcut. In the Plane Definition dialog box make the methods locate. D) Choose points at locations 1, 2, & 3 (not necessarily points 1, 2, and 3) as shown in the diagram. E) In the dialog box subtract 0.05 from the Z values of the first two points and 0.1 from Z value of the third point. Press OK when done.



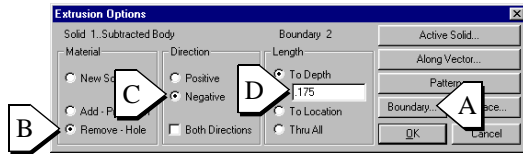
- 4 Repeat the above procedure to slice the other side. A) pick the large section of the split solid. B) pick points at locations 4, 5, & 6 as shown above. C) add 0.05, 0.05 & 0.1 to the Z values of the picked points.
- 5 Choose **DELETE - GEOMETRY - SOLID**. Select the two slivers on the top and bottom of the beam and press OK.



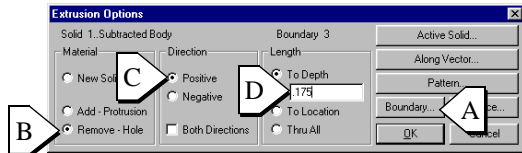
We want to turn this part of our solid into something more like an I-beam instead of a block. To do this we will remove material by extruding boundaries into the top and bottom of the beam.

### Remove Material from the Beam

- 1 Choose **GEOMETRY - SOLID - EXTRUDE**. A) Press the boundary button and select boundary 2, B) Check the remove material box, C) Make the direction negative and check the vector on the model to be sure it is pointing from the boundary towards the beam, D) Make the length to a depth of 0.175.

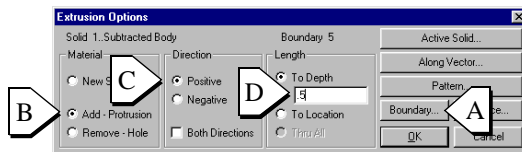


- 2 We now need to do this on the other side. Choose **GEOMETRY - SOLID - EXTRUDE**. A) Press the boundary button and select boundary 3, B) Check the remove material box, C) Make the direction positive and check the vector on the model to be sure it is pointing from the boundary towards the beam, D) Make the length to a depth of 0.175.

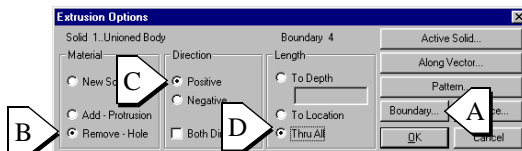


### Forming the Small End of the Rod

- 1 Choose **GEOMETRY - SOLID - EXTRUDE**. A) Press the boundary button and select boundary 5, the larger of the two small circles. B) Add material. C) The direction should be positive but check the arrow to make sure it points in the direction of the beam. D) Make it to a depth of 0.5.

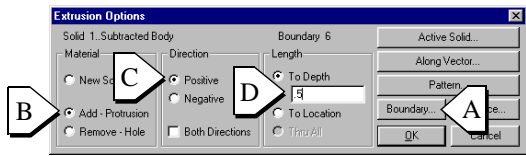


- 2 Choose **GEOMETRY - SOLID - EXTRUDE**. A) Press the boundary button and select boundary 4, the smaller of the two small circles. B) Remove material. C) The direction should be positive but check the arrow to make sure it points in the direction of the beam. D) Make it through all.

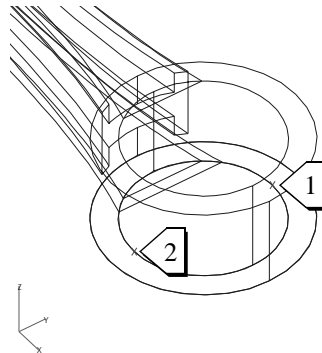
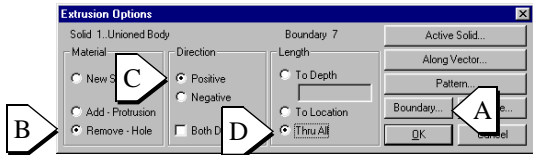


**Forming the Big End of the Rod**

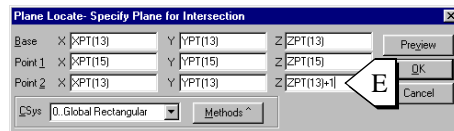
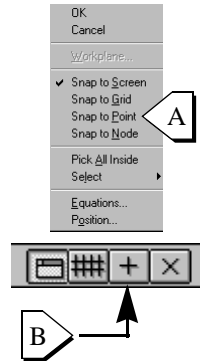
- 1 Choose **GEOMETRY - SOLID - EXTRUDE**. A) Press the boundary button and select boundary 6, the larger of the two large circles. B) Add material. C) The direction should be positive but check the arrow to make sure it points in the direction of the beam. D) Make it to a depth of 0.5.



- 2 Choose **GEOMETRY - SOLID - EXTRUDE**. A) Press the boundary button and select boundary 7, the smaller of the two large circles. B) Remove material. C) The direction should be positive but check the arrow to make sure it points in the direction of the beam. D) Make it through all.
- 3 Choose **GEOMETRY - SOLID - SLICE**. Select the solid and press OK.



- 4 The standard plane location box now appears prompting you for a cutting plane. First, change the snap mode to point, by either A) Pressing the right mouse button in the graphics, and then clicking the point option under Snap To, or B) Using the toolbar Snap To Point icon, or C) Using the Ctrl-P keyboard shortcut. D) Choose point 1, then 2, then 1 again as shown in the above diagram. Before pressing OK, E) Add 1 to the Z value of Point 2 in the plane location dialog box as shown below. Now press OK. The big end of the rod should be split in two. Press OK to continue.



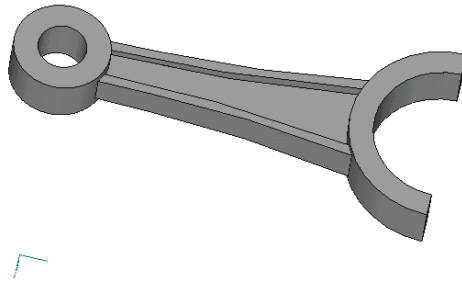
- 5 Choose **DELETE - GEOMETRY - SOLID**. Select the disconnected half of the big end and press OK.

### View as Solid

- 1 A) Press the view style button on the toolbar and choose Rendered Solid.





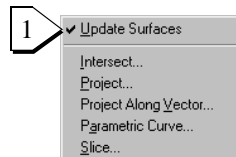


## 13.2 Loads and Constraints

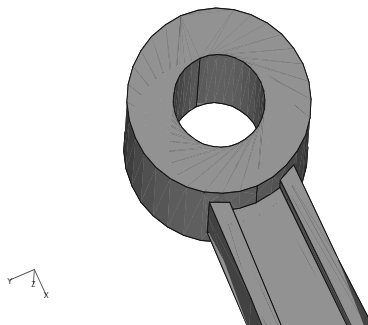
### *Prepare the Small End for Load on Surface*

We only want to load the bottom half of the small end of the rod, but currently it is split side to side. We will update the surfaces with a parametric curve at the midpoint of these two surfaces to split the inside of the small end into quarters. This will allow us to put a load on the two quarters of the bottom part of the small end.

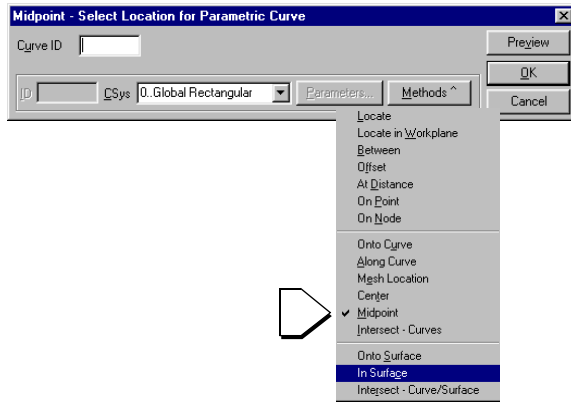
- 1 Choose **GEOMETRY - CURVE-FROM-SURFACE - UPDATE SURFACES** if it is not already checked. This is so that the curves created on the surface will split the surface.



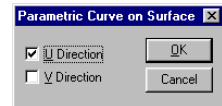
- 2 Choose **GEOMETRY - CURVE-FROM-SURFACE - PARAMETRIC CURVE**. Select one of the inner surfaces of the small end and press OK.



- 3 In the coordinate location dialog box press the methods button and choose midpoint. Select one of the arcs of the surface you picked in step 2 and press OK.



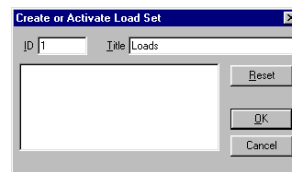
- 4 FEMAP prompts you for a parametric curve direction. You can use the surface lines to determine the proper direction. By default FEMAP draws 3 divisions in the U direction and 4 in the V direction. So in this case select the U direction and press OK.
- 5 Repeat steps 2, 3 & 4 on the other surface of the small end.



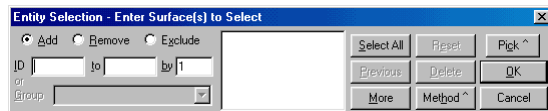
## Adding Loads on Geometry

We will add a slightly angled force on the lower half of the small end to simulate the piston pushing down on the rod.

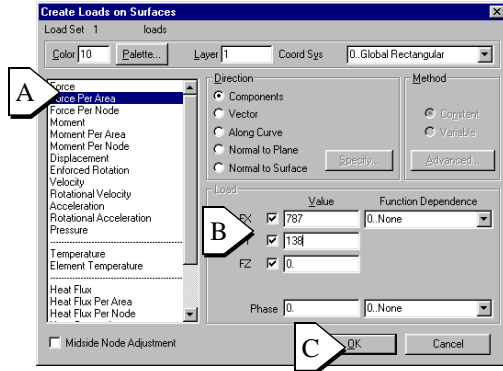
- 1 Choose **MODEL-LOAD-ON SURFACE**, FEMAP prompts you to select a load set or create a new one. Type in a title and press OK.



- 2 FEMAP now asks you to select the surfaces to apply the load. Select the two quarter surfaces of the small end that are near the beam portion of the rod and press OK.

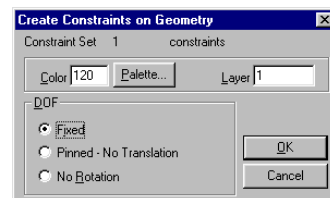
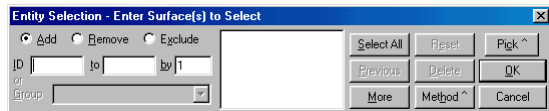
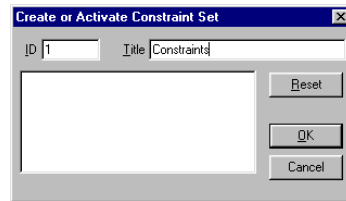


- 3 FEMAP displays the Load on Surfaces dialog box. A) Select force/area as the load type. B) Enter a value of 787 in the x direction and 138 in the y direction and C) Press OK. This corresponds to a force/area of 800 at an angle of 10 degrees with the X-axis.



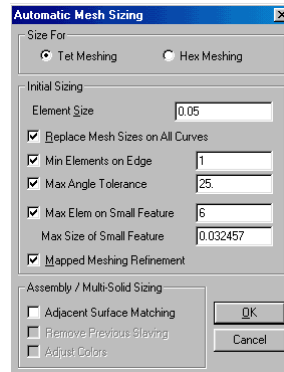
### Adding Constraints on Geometry

- 1 Choose **MODEL-CONSTRAINT-ON SURFACE**. FEMAP prompts you to select a constraint set or create a new one. Type in a name and press OK.
- 2 FEMAP prompts you to select surfaces. Select the two surfaces on the inside of the big end and press OK.
- 3 Constraints on surfaces are always relative to the global coordinate system and can only be fixed, pinned or have no rotations. Make these surfaces fixed. Press OK to create the constraints, press cancel to end the command.

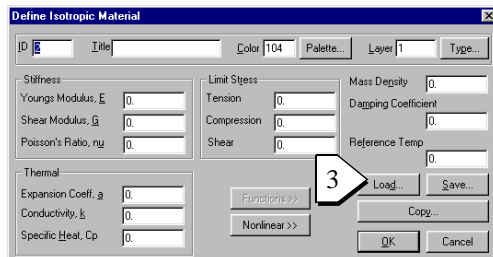


## 13.3 Meshing the Solid

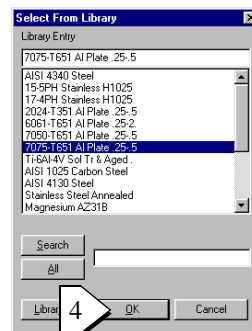
- 1 Choose **MESH-GEOMETRY-SOLIDS**.
- 2 The default values FEMAP calculated for this model are fine. Press OK to accept them. Feel free to experiment with different mesh sizes to become familiar with the meshing process.



- 3 Since no material has been created FEMAP prompts you to make one. You can enter in values or press the Load button to bring up the material library.



- 4 The material library shipped with FEMAP contains material properties using English units (lb, in, sec). You can create your own materials and store them in this library or create your own library. For this example select a material from this library and press OK.



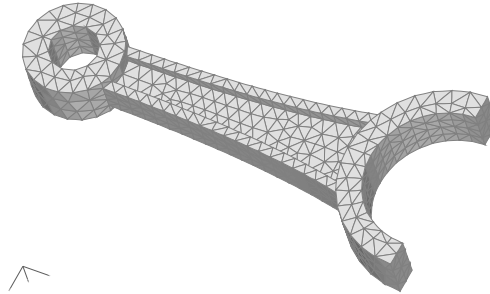
**Note:**

Remember, there are no units in FEMAP. All dimensions must be kept consistent with the unit system you use to define your material properties.

- 5 Press OK in the define material dialog box when the properties have been loaded.
- 6 The automesh solids dialog box appears. Leave the values as the defaults and press OK.



When the meshing completes the model will be ready for analysis.





# Midsurface

# 14

To perform this example, you must have FEMAP Professional with Parasolid active. Furthermore, you will not be able to complete this example with the 300-Node Demo version.

## 14.1 Introduction

The purpose of this example is to demonstrate semi-automatic midsurface extraction capabilities of FEMAP. Existing geometry will be imported into FEMAP using our new STEP interface. The model will then be midsurfaced. There is a small amount of cleanup that must be performed in order to attain the true idealized model of the electrical box.

The Midsurfaced box will be meshed and boundary conditions applied. The model will then be analyzed and finally post-processed. Elemental contouring ("Smart Results") will also be featured in the post-processing.

### Importing the Geometry into FEMAP

- 1 Choose the **File/Import/Geometry** command.

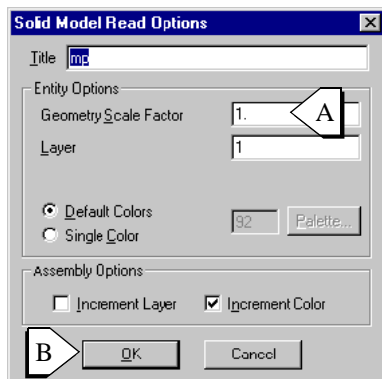
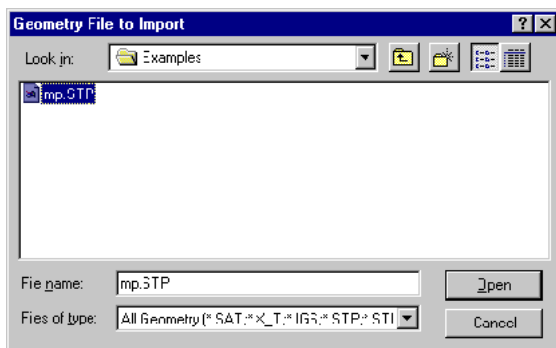
Open the Examples folder in the FEMAP 7 directory.

Choose the STEP file named *mp.STP* and Click **OK**.

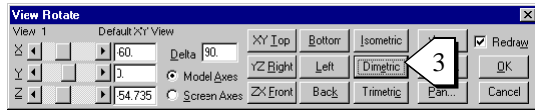
- 2 When the Solid Model Read Options dialog box appear, make certain

A.) the Geometry Scale Factor is set to *1*,

B) then Click OK.

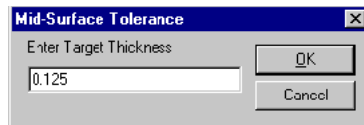
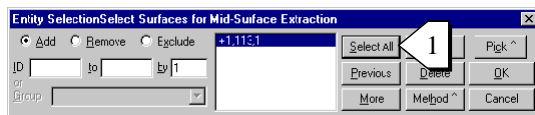


- When the geometry is imported the view needs to be rotated to get a better look at the part. Choose **View - Rotate (F8)** and click *Dimetric*, and then OK.



## 14.2 Creating the Midsurface Model.

- Choose the **Geometry - Midsurface - Automatic** command. Click Select All in the dialog box to choose all the surfaces in the model and hit OK
- The Mid Surface Tolerance Thickness must be entered. For this example enter **0.125**



### Note:

The target thickness is used to determine which surfaces to place a midsurface between. The target thickness should be slightly larger than the largest distance between the planes on the solids which the user wants midsurfaced. If the target thickness is too low, then all of the desired midsurfaces will not be created. If the target thickness is too high, however, then some unwanted midsurfaces will be created between the wrong surfaces.

### Hint:

An easy way to determine a suitable target thickness is to use the Ctrl-D command while the Mid-Surface Tolerance dialog box is on the screen. The Ctrl-D command is used to determine a distance any time a field is highlighted in a dialog box. A dialog box will appear and ask to define a location to measure from and then a location to measure to. When the Ctrl-D command is used when the Mid-surface Tolerance dialog box is on screen it will automatically make the target thickness slightly larger.

### Note:

By choosing the **Geometry - Midsurface - Automatic** command FEMAP is actually going through three commands: (1) **Geometry - Midsurface - Generate**, (2) **Geometry - Midsurface - Intersect**, and (3) **Geometry - Midsurface - Cleanup** commands in that order. If the Automatic midsurfacing command has removed any necessary midsurfaces, then you may want to go through the Midsurfacing commands one at a time, which will enable you to pick

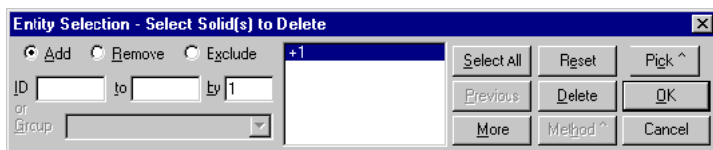


and choose which surfaces should be kept. The **Geometry - Midsurface - Cleanup** command provides an easy way to remove all of the unnecessary midsurfaces manually by placing all of the surfaces it would have deleted onto a separate layer.

The Next step is

to delete the original solid.

Select the **Delete - Geometry - Solid** command, choose Solid 1 (the original geometry) and Click OK.



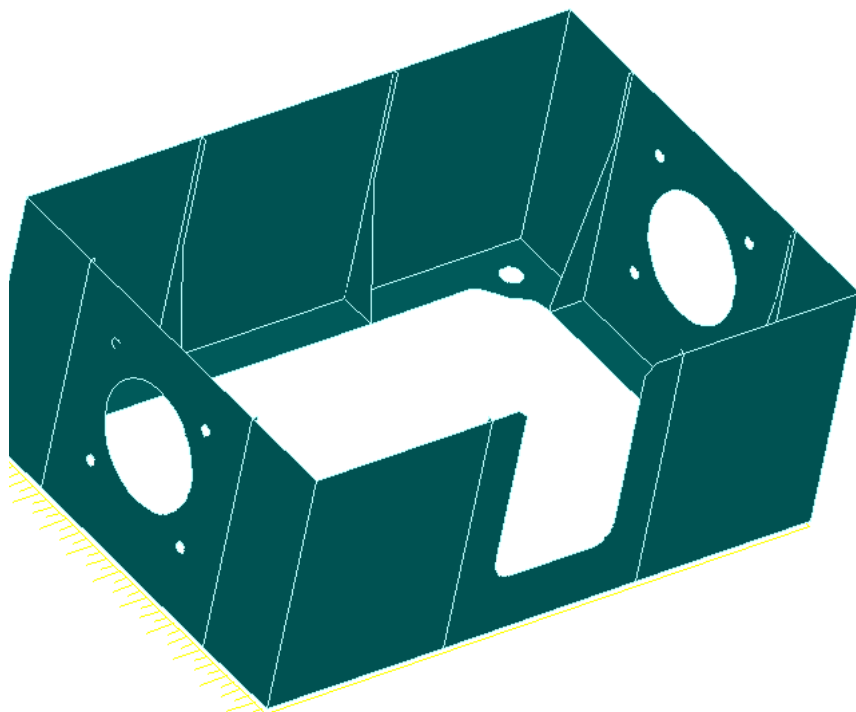
### Note:

Sometimes you may want to keep the original solid for future reference. The original solid and the midsurfaces could be placed on separate layers or the midsurfaces could be placed in a group. See the Commands manual for more information on how to use groups and layers.

Click the View Style button (This button appears in the top toolbar. It is the solid cube with a shaded face) and select Render mode. Click the View Style button again and select Solid mode.

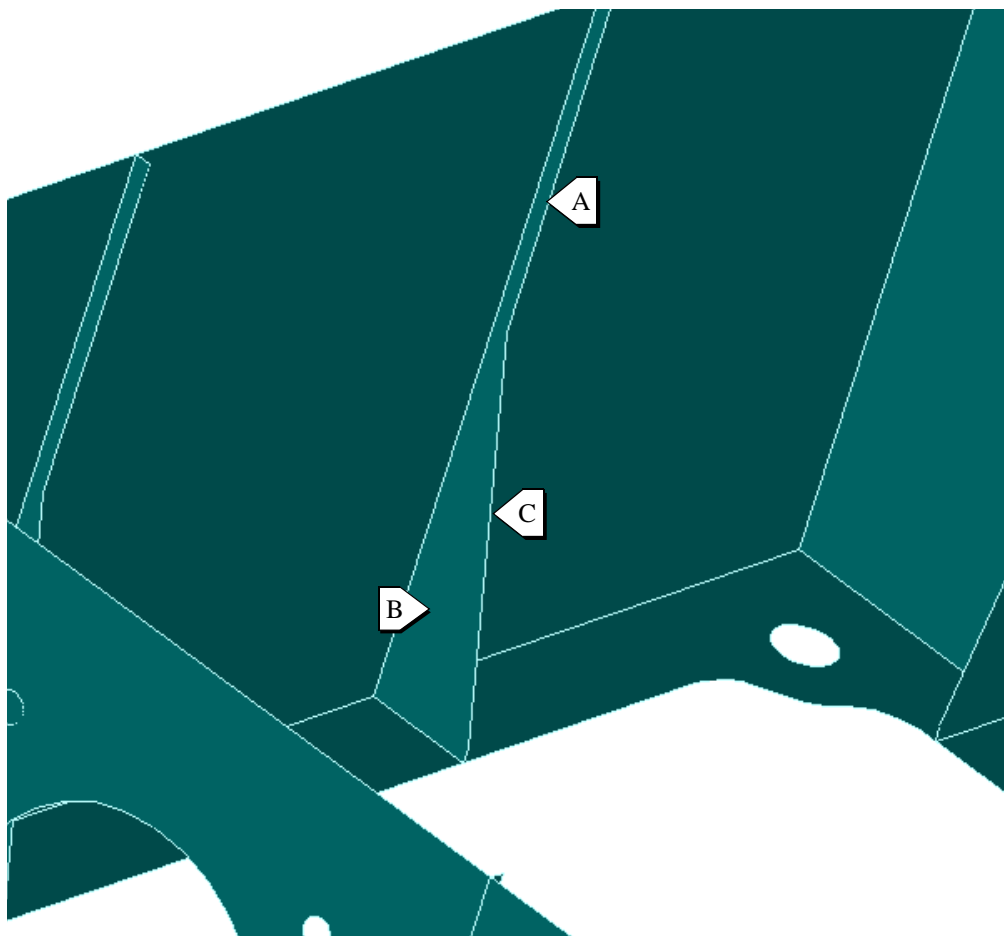


The midsurface geometry should look like this:

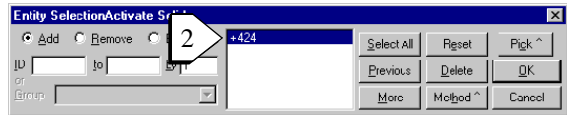
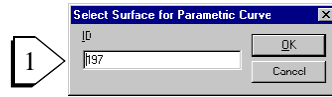


Use the dynamic rotation, accessed by simply clicking and holding the left mouse button while dragging the mouse to rotate and more carefully examine the model. If you look carefully, this geometry still requires some additional manual work using one of the other midsurface commands.

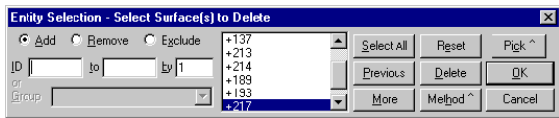
The upper portion of the ribs of the box must be deleted in order to create a more accurate midsurface model. FEMAP offers a specific command which enables a curve to be used to trim a surface, Geometry - MidSurface - Trim with Curve. In the figure below, we would like to remove the unwanted section A). This can be accomplished by simply trimming surface B) with curve C). The specific steps are shown below.



- 3 Choose the **Geometry - Midsurface - Trim with Curve** command. You will be prompted to
  - (1) Select the surface: Select *one of the eight rib surfaces* (A above) that have the “undesired portion”.
  - (2) Select the Trimming entity: Pick the Curve (C above) to cut the surface.
- 4 Click **OK**. Repeat the operation for the other seven surfaces *until all eight surfaces have been trimmed*. After all the surfaces have been trimmed click **Cancel**.



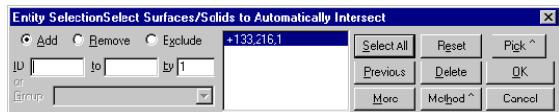
- 5 Select the **Delete- Geometry - Surface** Command. Select the new surfaces that have been created on the top portion each rib and delete them,



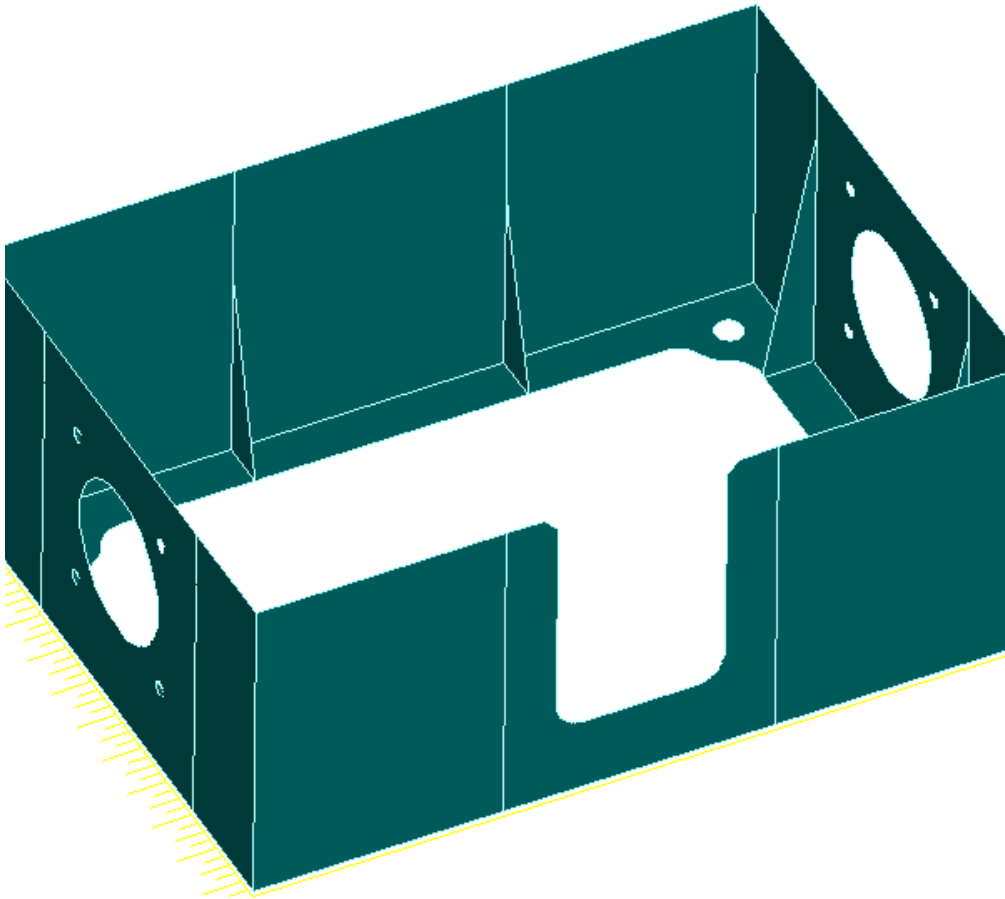
### Note:

The geometry must be intersected again in order for the newly created surfaces to be completely intersected. This causes a point to be placed at the pointed tip of the rib, which facilitates the mesh to be continuous in these areas.

- 6 Choose the **Geometry - Midsurface - Intersect** Command. Push the **Select All** button and hit **OK**.



The Geometry should look like this:

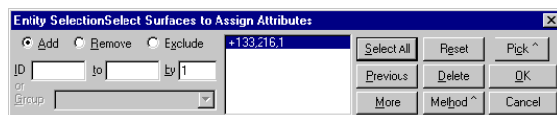


### 14.3 Meshing the Model

#### Note:

The first step before attempting to mesh a midsurfaced model is to assign the mesh attributes for the different surfaces. **THIS IS VERY IMPORTANT.** If the correct attributes are not assigned then the results will not be correct.

- 1 To automatically assign mesh attributes to a model that has been midsurfaced, use **Geometry - Midsurface - Assign Mesh Attributes**.



- 2 Select All, hit OK, and choose a “dummy” material. It can be changed later by using the **Modify/Edit/Property** for the individual properties. Click **OK**. FEMAP then creates a different property for each surface.

### Note:

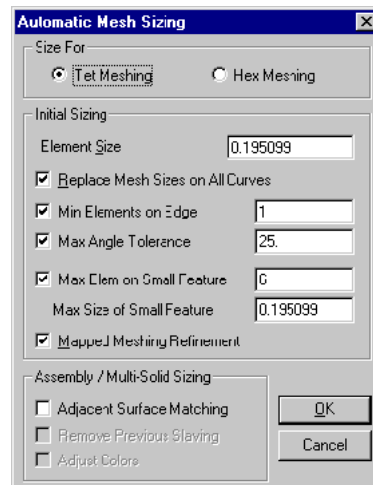
If any midsurfaces are manually created using commands such as **Geometry - Surface - Offset** or **Geometry - Surface - Extrude**, **THESE SURFACES DO NOT HAVE MESH ATTRIBUTES, THEY MUST BE ASSIGNED MANUALLY BY CREATING PROPERTIES OR ASSIGNING EXISTING PROPERTIES WHICH USE THE CORRECT THICKNESS.**

- 3 Choose the **Mesh - Mesh Control - Size on Surface** Command.

Push the **Select All** button.

Click **OK**.

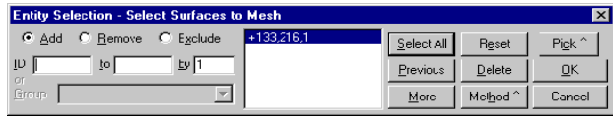
Use the defaults in the Automatic Mesh Sizing dialog box.



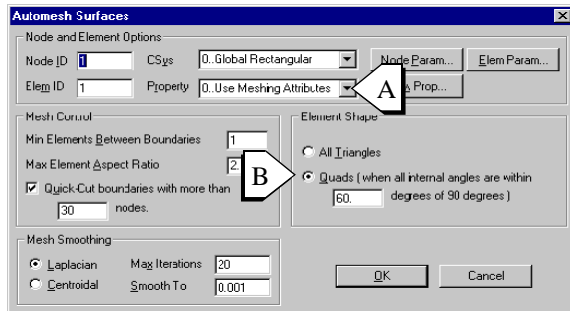
### Note:

If the hex meshing button is selected in the Automatic Mesh Sizing dialog box, FEMAP will display error messages while trying to set the mesh size on these "solids". These errors will not have an effect on a midsurface model that contains only surfaces that are to be meshed with plates. As long as there are no actual solids these error messages can be ignored.

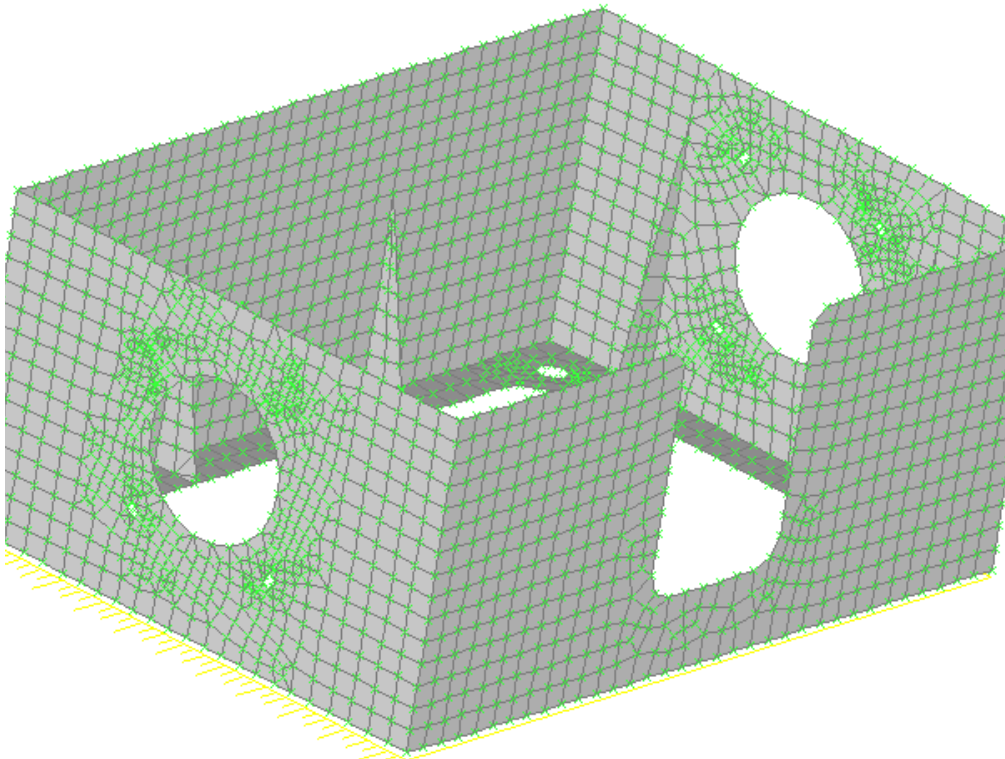
- 4 Choose the **Mesh - Geometry - Surface** Command. Push the Select All button.



- 5 Click **OK**. Notice the Property has already been selected as *0. Use Meshing Attributes* (A). This was assigned because of the **Geometry -Midsurface - Assign Mesh Attributes** Command. Make sure the Quads (B) option is selected.



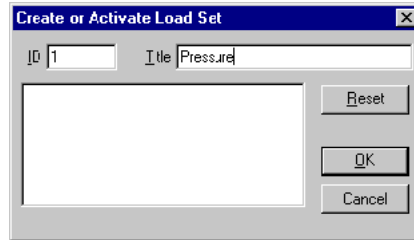
Use the **View Regenerate** (or Ctrl+G) to regenerate the view. The display should look the figure below.i



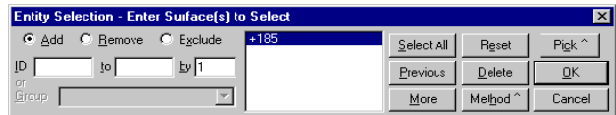
## 14.4 Applying Loads and Constraints

The Model is now ready to have Loads and Constraints applied.

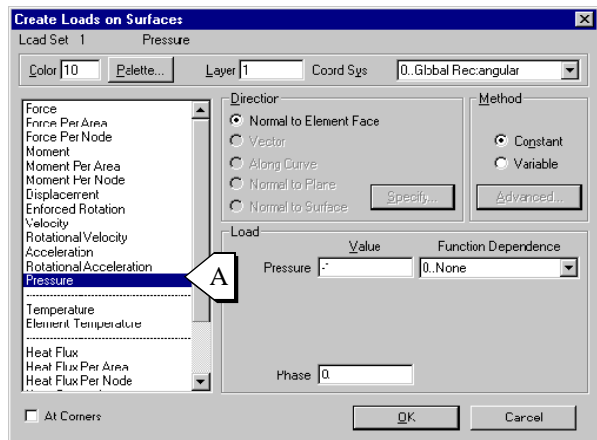
- 1 Use the **Model - Load - Set** Command. Name the Load Set *Pressure*. Click **OK**.



- 2 Choose the **Model-Load-On Surface** Command. Select Surface 185. Click **OK**.

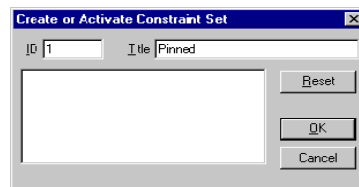


- 3 Choose (A) Pressure and enter a value of  $-1$ . Click **OK**. Click **Cancel**.

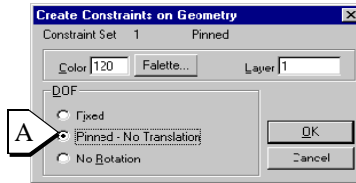
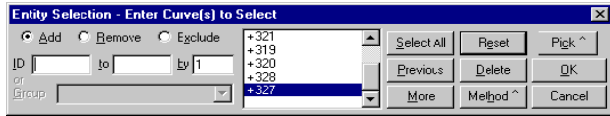


- 4 Regenerate the Model using **Ctrl-G** or the **View - Regenerate** Command.

Choose the **Model - Constraint - Set** Command. Name the Constraint Set *Pinned*. Click **OK**.



- 5 Choose the **Model - Constraint - On Curve** Command. Select the eight curves that comprise the four holes located on the base of the box. There is one hole in each corner. Click OK.
- 6 Select the button marked Pinned - No Translation (A). Click **OK**. Click **Cancel**.



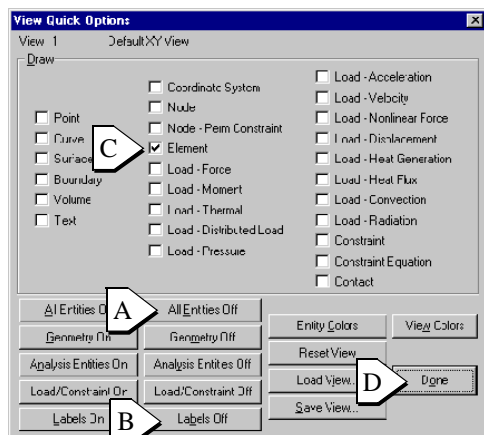
### Note:

If you would like to see exactly on which nodes the Loads and Constraints are applied, simply use the Model - Load - Expand and the Model - Constraint - Expand Command to expand the loads and constraints, respectively to the nodes and elements.

The model is now ready to be sent to a solver. Export an Analysis Model and perform a linear static analysis with an available solver. We used NASTRAN to solve this example. If you do not have a solver we have included in the example directory a finished model along with a set of results, open the finished model mpdone.mod and then import the NASTRAN mp.op2 results file and continue on to postprocessing.

## 14.5 Post-Processing

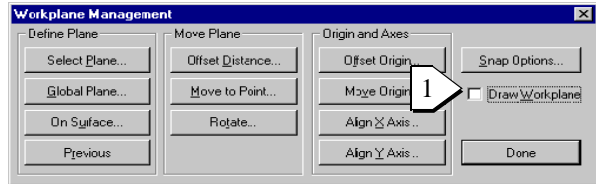
Import the Analysis Results. (For those who did not use a solver, import the mp.op2 file included in the /examples subdirectory). To facilitate the viewing of results the Geometry and any Analysis Entities can be turned off. Press **Ctrl-Q** to bring up the Quick Options dialog box. This can Quick Options dialog box can also be reached by pressing the **F6** key or using the **View - Options** Command and then pushing the **Quick Options** button, or by pressing the button with a square with an X through it on the top toolbar.





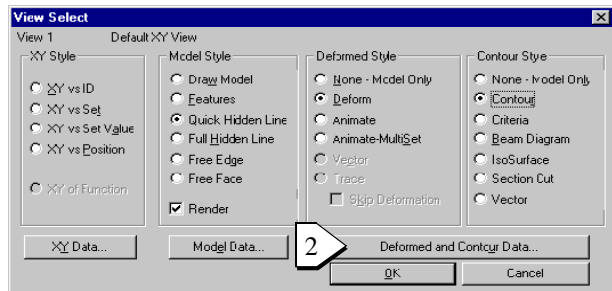
Hit (A) All Entities Off, (B) Labels Off and the (C) Element. Press Done (D) and now only the elements are visible.

- 1 Press the right mouse button anywhere inside the modeling area and a menu will come up. Choose Workplane. When the Workplane dialog box appears click the box next Draw Workplane (1) and Click Done



- 2 Now the Yellow Workplane is no longer visible. To view the results choose the **View - Select** command, or press the **F5** key, or pick on the View Select button in the top toolbar (The View Select dialog box has three rectangles followed by horizontal lines).

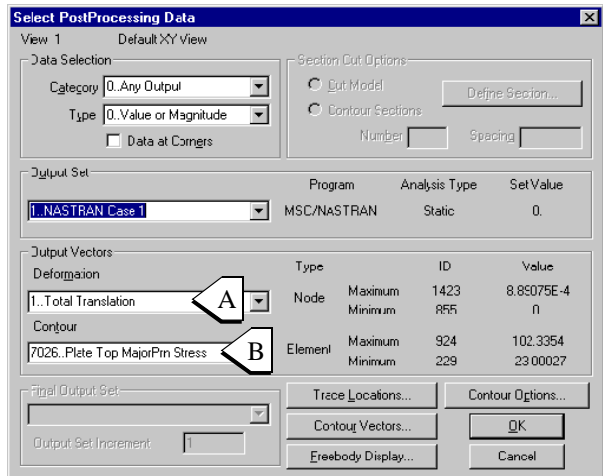
The View Select dialog box will appear.



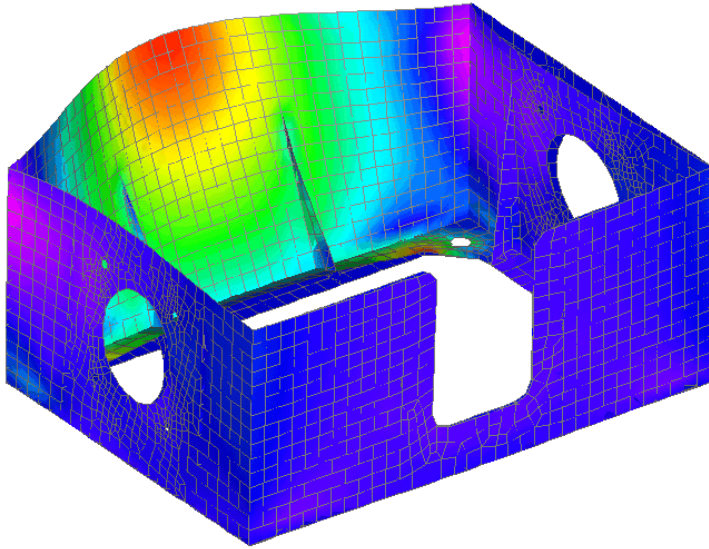
- 3 Press on **Deformed and Contour Data** (2 above). Make certain  
(A) *1..Total Translation* is selected as the Deformation Output Vector and

(B) *7026..Plate Top MajorPrn Stress* as the Contour Output Vector.

Click **OK**. Click **OK** again.

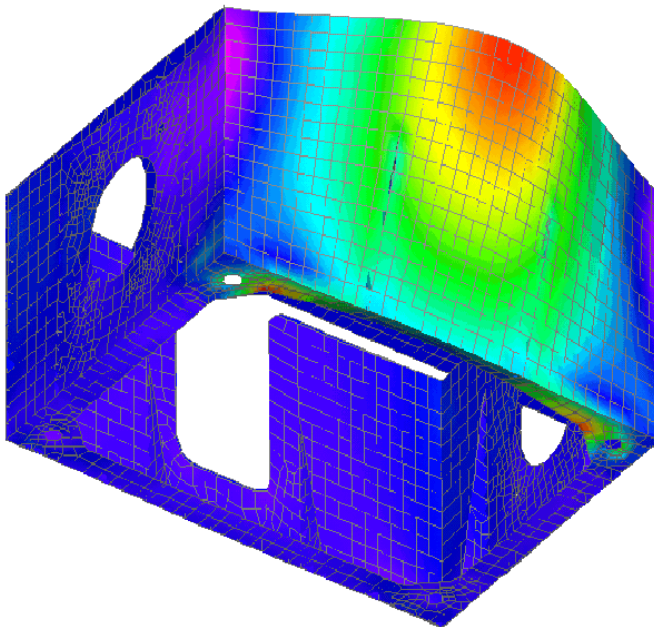


The model should look like the figure on the following page.



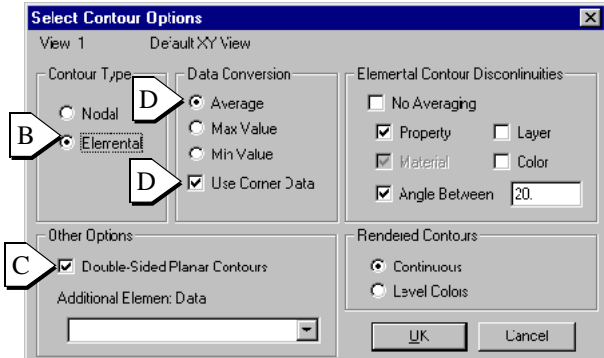
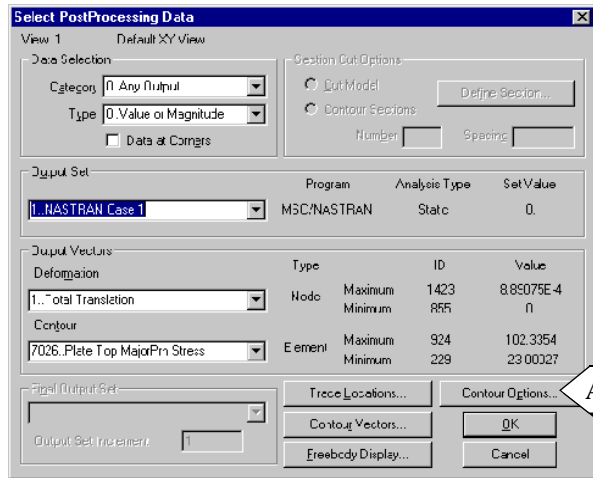
Click the left mouse button inside the modeling area and drag it on the screen. As long as the model is in **Render Mode** the model will Dynamically rotate and the different faces of the plate elements can be viewed.

The back of the model looks like this:



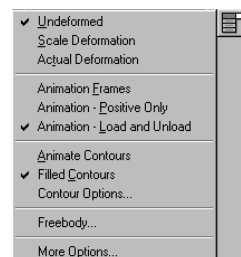
The contour for Top MajorPrn Stress is can be shown on both faces of the plate element

- 4 To view the results choose the **View - Select** command, or press the **F5** key, or pick on the View Select button in the top toolbar. The View Select dialog box will appear.
- 5 As Before, Press on Deformed and Contour Data. Make sure 1..Total Translation is selected as the Deformation Output Vector and 7026..Plate Top MajorPrn Stress as the Contour Output Vector.
- 6 A) Click Contour Options  
B) Click the radio button marked Elemental  
C), then click box marked Double-Sided Planar Contours  
D), Make sure Average and Use Corner Data are selected under Data Conversion.  
Click **OK**, 3 Times.

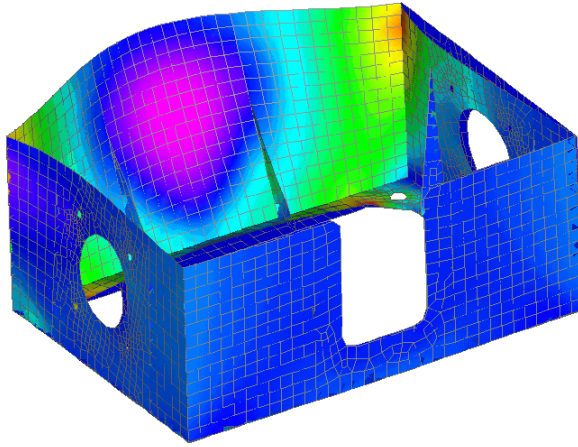


### Hint:

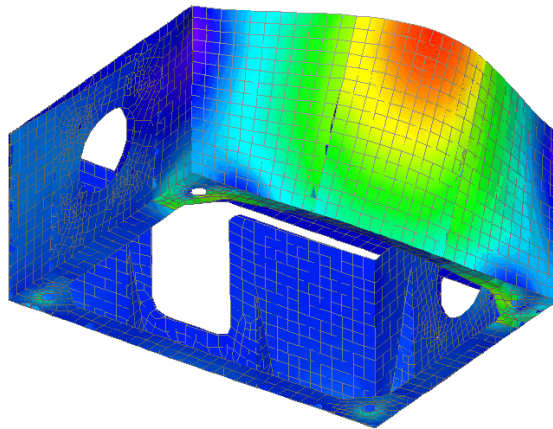
The Contour Options can also be accessed on the Commands Postprocessing Toolbar under Post Options. This provides much quicker access to these commands Contour options instead of going through View Select, Deformed and Contour Data, Contour Options (or even Post Data, Contour Options).



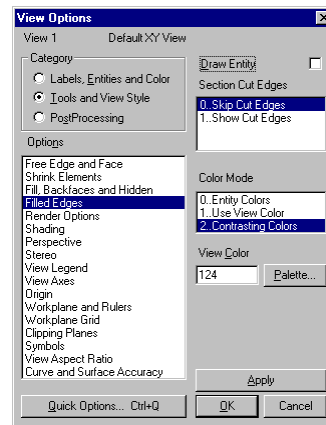
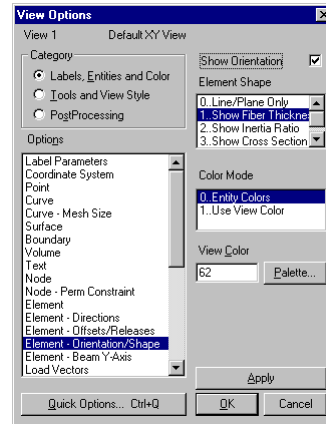
The model should now look like this; the contour shown is for Bot MajorPrn Stress



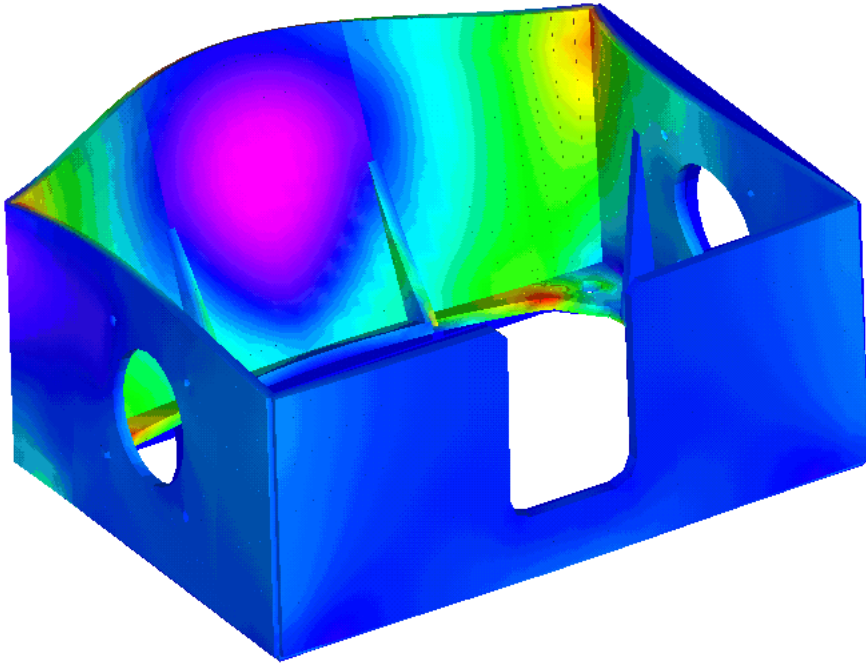
Click the left mouse button inside the modeling area to use the Dynamic Rotate feature again. Remember; as long as the model is in Render Mode the model will Dynamically rotate. Because Plate Top MajorPrn Stress has been chosen as the output vector, FEMAP will automatically choose Bot MajorPrn Stress to use as the output vector on the other side of the plate elements as the default. The backside of the model has a contour of the Top MajorPrn Stress, as shown below.



- 7 The Double-Sided results can be viewed better by showing the element thicknesses. Press the **F6** key or use the **View/Options** command. Choose *Element – Orientation/Shape* and then select *1..Show Fiber*
- 8 Click the button next to Tools and View Style. Choose Filled Edges. Click the box next to Draw Entity. Click OK.



*The stress is now shown through the thickness of the plate elements:*



# Hex Meshing Overview

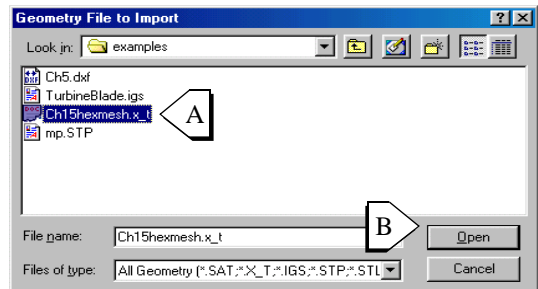
This example requires FEMAP Parasolid modeler to complete. If you have the 300-Node version you will not be able to save or export the model.

## 15.1 Introduction

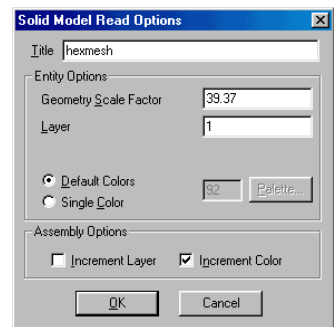
This is an example of how solids can be subdivided to facilitate hex meshing in FEMAP. It assumes you are familiar with FEMAP and do not need step by step instructions for all commands. It is intended to give you one method of approaching the problem of hex meshing solid models, and demonstrates only a few of the commands that can be used to hex mesh in FEMAP. For more descriptions and methods refer to the Commands manual and the User Guide.

## 15.2 Importing the Geometry

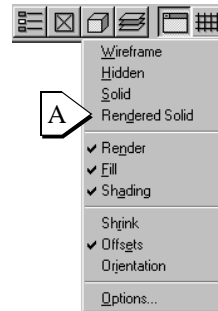
- 1 Select **FILE - IMPORT - GEOMETRY** from the FEMAP menu.
- 2 FEMAP displays the standard Windows File Open Dialog Box. Maneuver to the /examples subdirectory and A.) select the Ch15hexmesh.x\_t file, and B.) Press Open.



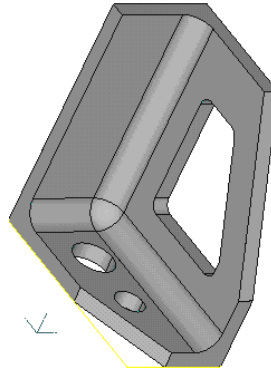
- 3 The FEMAP Solid Model Read Options Dialog Box is displayed, providing several options for how to treat the incoming data. Set the Geometry Scale Factor to 39.37 and press OK.



- 4 A) Press the view style button on the toolbar and choose Rendered Solid.

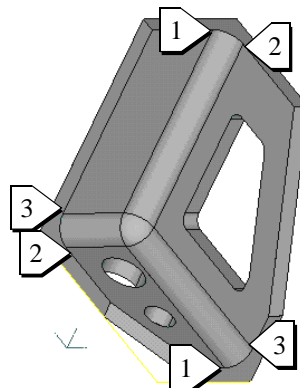


- 5 Left click and drag in the graphics window to dynamically rotate the model.



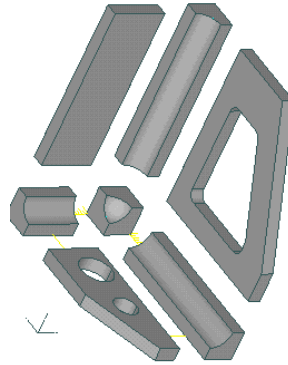
## 15.3 Subdividing the Solid

- 1 We first want to slice the solid with three planes. Use the **Geometry Solid Slice** command three times. Be sure to select all the solids each time. Using whatever method you please, slice the solid along the planes of the curves pointed to by 1, 2 and 3

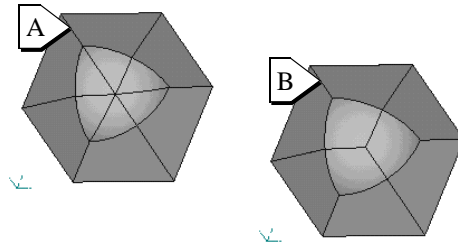




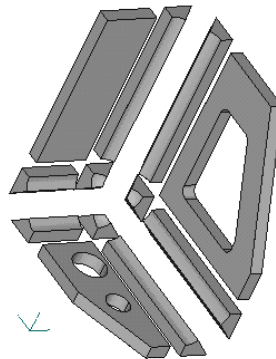
- 2 You should end up with seven separate solids. The picture is an exploded view to clearly show the separate solids. Your view will still look like the previous one.



- 3 The square in the center is not a hex-meshable solid. We have learned from experience that a good way to subdivide this part is to cut it into sixths (A), and then add pieces back together to form three six sided volumes (B) that are easily hex meshed.

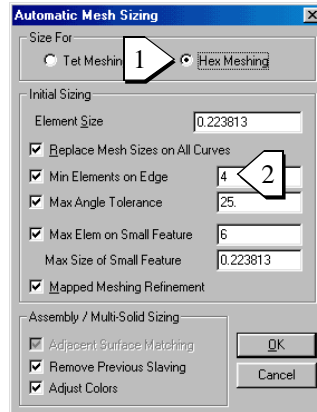


- 4 Keep in mind that you need to have surface meshes that match. The easiest way to ensure this on this model is to also slice the radiused solids. Try to produce the 12 distinct solids at right. The picture is an exploded view to clearly show the separate solids.

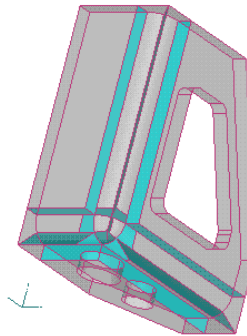


## 15.4 Preparing for Meshing

- 1 Select **Mesh - Mesh Control - Size on Solid** and select all the solids. In the Automatic Mesh Sizing dialog box choose Hex Meshing and set the Min Elements on Edge to 4. The rest of the defaults are fine so press OK.

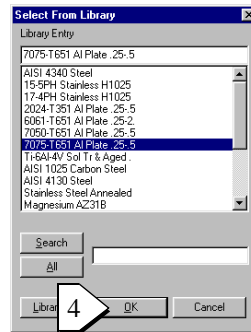
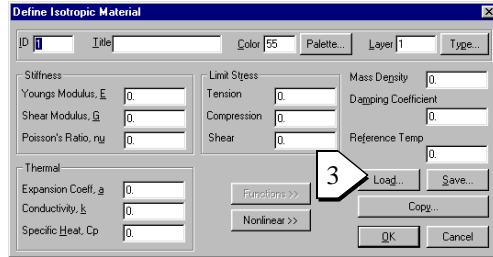


- 2 The sizes are set and colors are updated to show which solids are hex-meshable and which surfaces have been linked.



## 15.5 Meshing

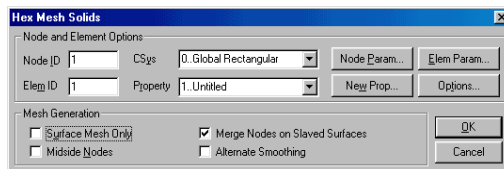
- 1 Select **Mesh Geometry Hex Mesh Solids** and select all the solids.
- 2 Since no material has been created FEMAP prompts you to make one. You can enter in values or press the Load button to bring up the material library.
- 3 The material library shipped with FEMAP contains material properties using English units (lb, ft, sec). You can create your own materials and store them in this library or create your own library. For this example select a material from this library and press OK.



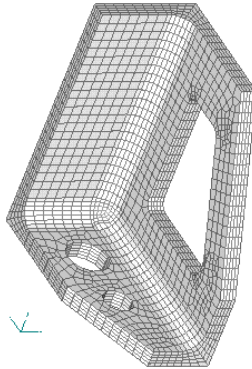
### Note:

Remember, there are no units in FEMAP. All dimensions must be kept consistent with the unit system you use to define your material properties. Always make sure this is correct from the beginning because it is extremely difficult to correct inconsistencies in units once the model is built.

- 4 Press OK in the define material dialog box when the properties have been loaded.
- 5 The Hex Mesh Solids dialog box appears. Leave the values as the defaults and press OK.



Finished mesh with all entities but elements turned off.



Notice that one of the surfaces of the square with the sphere cut out has not meshed well. The reason is that there is a pole on that surface. You could delete all nodes and elements and try to fix that surface and mesh again. We have done it using explode, deleting that surface, making a new one from edge curves, and stitch. Remember to re-run the size on solid command to ensure surface linking.

# Hex Meshing

# 16

This example requires FEMAP Parasolid modeler to complete. If you have the 300-Node version you will not be able to save or export the model.

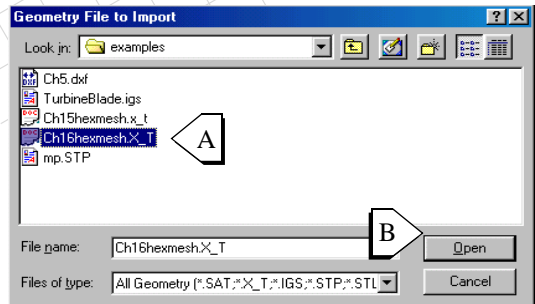
## 16.1 Importing the Geometry

- 1 Select **FILE - IMPORT - GEOMETRY** from the FEMAP menu.

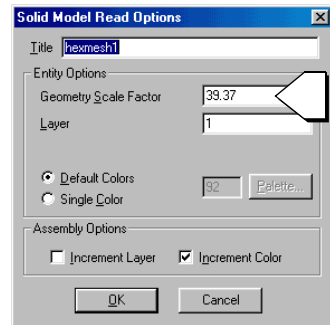
- 2 FEMAP displays the standard Windows File Open Dialog Box. Maneuver to the /examples subdirectory and

A.) Select the  
Ch16hexmesh.x\_t file,

B.) Press Open.

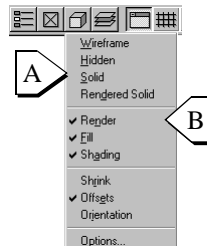


- 3 The FEMAP Solid Model Read Options Dialog Box is displayed, providing several options for how to treat the incoming data. Set the Geometry Scale Factor to 39.37 and press OK.

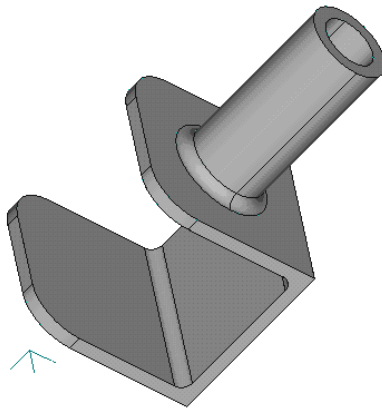


A) Press the view style button on the toolbar and choose solid.

B) Press the view style button on the toolbar and choose Render.

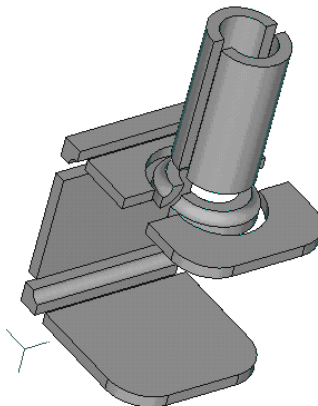


Left click and drag in the graphics window to dynamically rotate the model.

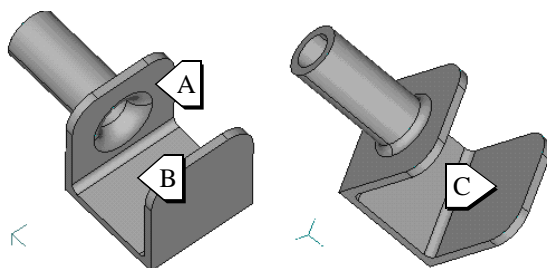


## 16.2 Subdividing the Solid

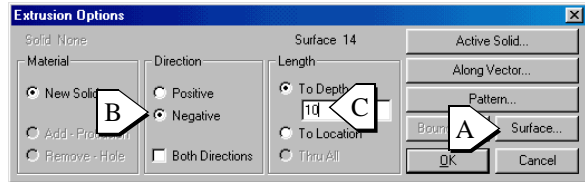
- 1 We want to subdivide this solid into the ten independent solids shown here. The solids are shown exploded for viewing only.



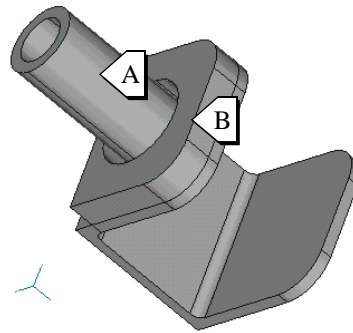
- 2 Start with **Geometry - Solid - Embed Face**, pick surface A. Repeat this command for surfaces B and C.



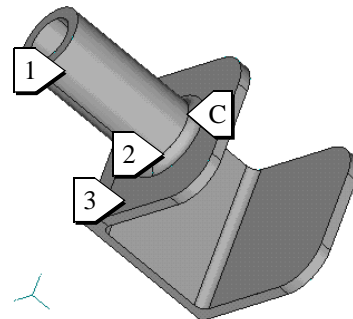
- 3 Now use **Geometry-Solid-Extrude** and press the surface button (A) and pick surface A above. Change the direction to negative (B) and enter a depth of 10 (C).



- 4 Next pick **Geometry-Solid-Embed**. Pick the solid with the tube as the base solid at (A), and the one you just created at (B) as the one to embed.



- 5 You now need to make two slices. Slice the tube off at the top of the radius (C), and slice the three solids (1,2,3) near the tube in half. Refer to the exploded diagram if you have difficulty visualizing the individual solids.



## 16.3 Meshing

We will begin setting up a hex mesh using the default approaches and let FEMAP set up the mesh automatically. You will find that the defaults provide a good hex mesh but we will mesh the solid again using some of the more advanced options to obtain a mapped mesh.

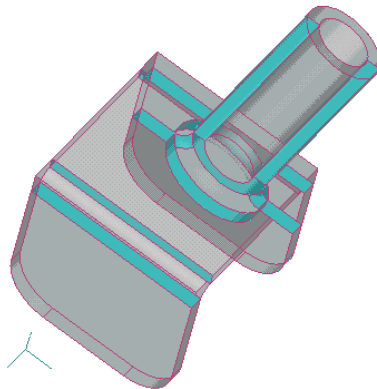
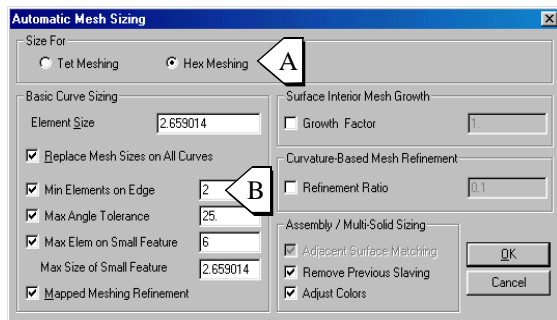
### 16.3.1 Free Meshing

- 1 Use **Mesh-Mesh Control-Size On Solid**, select all solids and

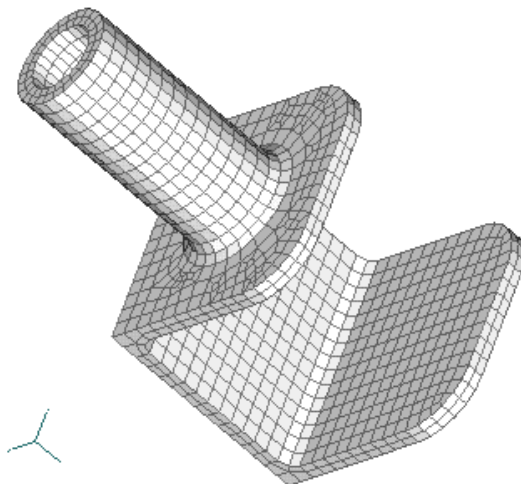
(A) Turn on Size for Hex Meshing, and

(B) Enter a Min Elements on Edge of 2.

This command will also link all the shared surfaces to ensure a consistent mesh.

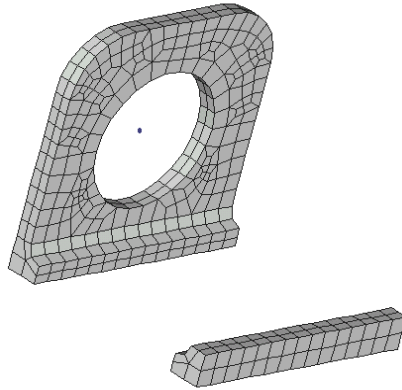


- 2 Use the **Mesh-Geometry-Hexmesh Solids** command and select all the solids to hex mesh. If you have properly set mesh sizes and linked surfaces, the hex mesh should run automatically.





As you can see on the previous page, FEMAP has produced a mesh with good hexahedral elements except for some hexes and wedges shown bellow.



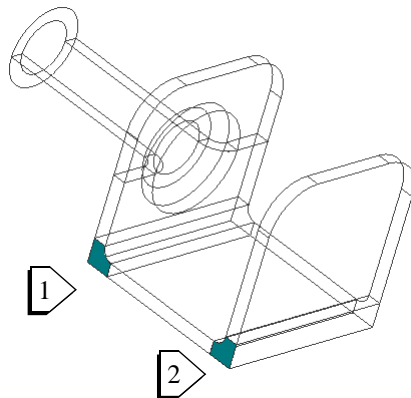
By defining special Approaches on the geometric surfaces in question we can achieve a better mesh.

### 16.3.2 Mapped Meshing

Brick meshing inherently requires the mesh to be propagated throughout the solid geometry. Therefore if the surfaces are mixed meshed with triangles and quads then the final hex mesh will include bricks and wedges. The default free mesher will always use a combination of triangles and quads on any surface that is not a simple 4 sided region. To force FEMAP to map mesh surfaces that are not 4 sided we will use the **Mesh-Mesh Control-Approach on Surface** command.

We must first delete the existing mesh using **Delete-Model-Mesh** and **Select All** to delete the entire mesh.

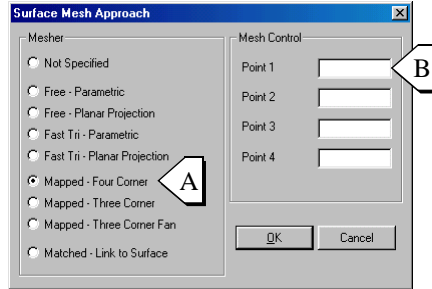
- 1 To force FEMAP to provide a mapped mesh on surfaces we will need to set up approaches on the surfaces that we want mapped.



## 2 Use **Mesh-Mesh Control-Approach on Surface**

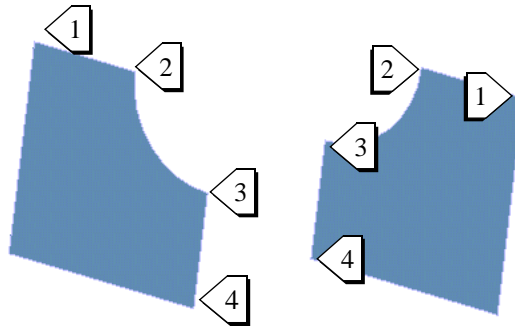
FEMAP will ask you to pick the surface you wish to put a approach on. Choose surface 1 on the previous page.

When the Surface Approach dialog box comes up select A.)  
**Mapped - Four Corner**



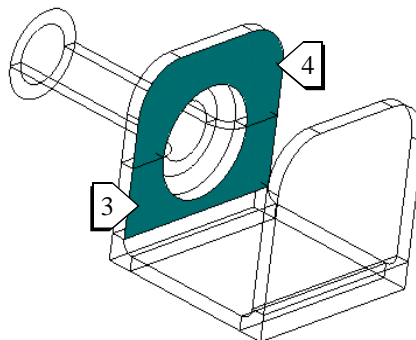
## 3 Since this surfaces has more than 4 corners you must specify which corners you want FEMAP to map between.

B.) Choose the corners as shown for the first surface and say OK. The command will auto repeat allowing you to choose surface 2 on the previous page and select the four corners from the diagram to the right.

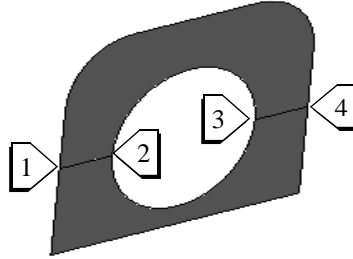


Now we will set up surfaces 3 and 4 for mapped meshing. Use **Mesh-Mesh Control-Approach on Surface** again.

FEMAP will ask you to chose the surface to set the approach on. Select surface 3 and 4 from the diagram to the right.



After selecting the surfaces choose the **Mapped Four Corner** approach and select the four points as shown to the right. Then say OK.

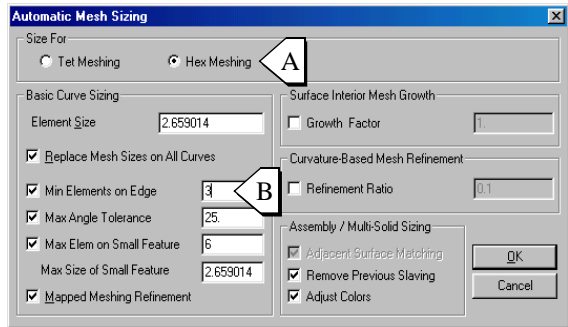


- 4 Mesh all of the solids using **Mesh-Mesh Control-Size On Solid**, select all solids and

(A) Turn on Size for Hex Meshing, and

(B) Enter a Min Elements on Edge of 3.

- 5 Select **Mesh-Geomerty-Hexmesh Solids** and select all of the solids to hex mesh.



By applying approaches on surfaces of the model the quality of the mesh can be greatly improved.

