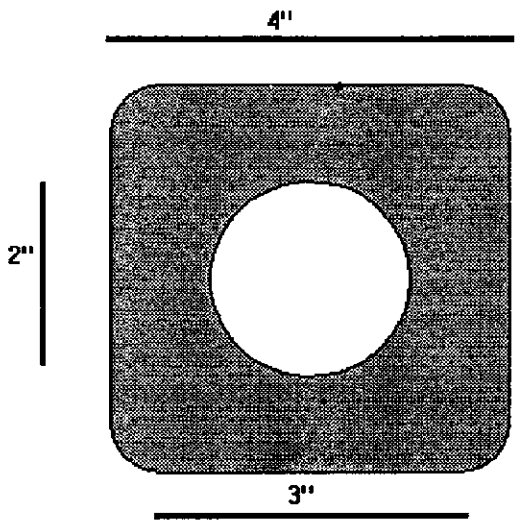
Step-Through Example: Flat Plate with a Single Load

In this example we create a 4 in. square plate with filleted edges (0.5 in. fillet radius) and a 2 in. diameter circular hole in the center. The plate is made of steel, 0.1 in. thick. The model is simply supported at the corners, and a 10g gravity load is applied normal to the plate. The plate is modeled with flat plate elements. Grid point displacements and element stresses are computed.



This example uses English units: inches (abbreviated in.) for length, pounds (abbreviated lb) for force, and seconds (abbreviated sec) for time. Note that MSC/NASTRAN for Windows assumes a consistent set of units, so you need to be consistent and not mix units (i.e., do not mix feet and inches).

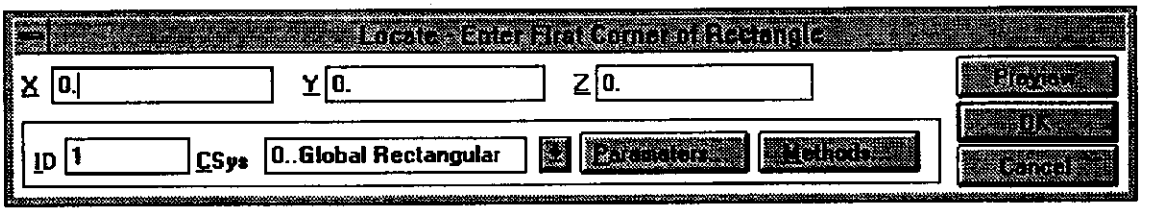
Follow the steps described below to create, analyze, and postprocess the model. Input entities are shown in **bold**.

Creating the Geometry

Start MSC/NASTRAN for Windows by double-clicking on the MSC/NASTRAN icon. The Open Model File dialog box appears; choose **New Model**.

Create the 4 in. Square

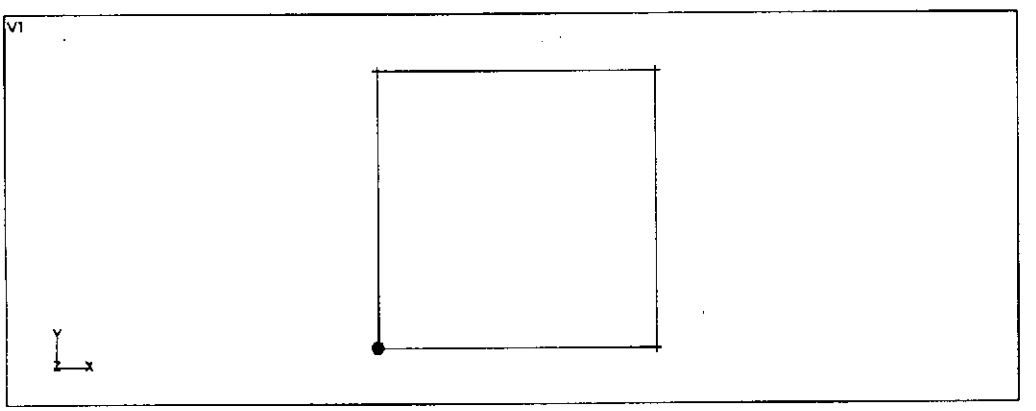
Choose **Create Line Rectangle**. (To do so, choose the **Create** pull-down menu and scroll down to **Line**. This will open another menu, from which you choose **Rectangle**.)



Enter the coordinates for the first corner of the rectangle. Enter 0 (zero) for the **X** coordinate, 0 for the **Y** coordinate, and 0 for the **Z** coordinate; then choose **OK**. For the diagonally opposite corner, enter 4 for **X**, 4 for **Y**, and 0 for **Z**; then choose **OK**. A 4 in. by 4 in. rectangle is now created.

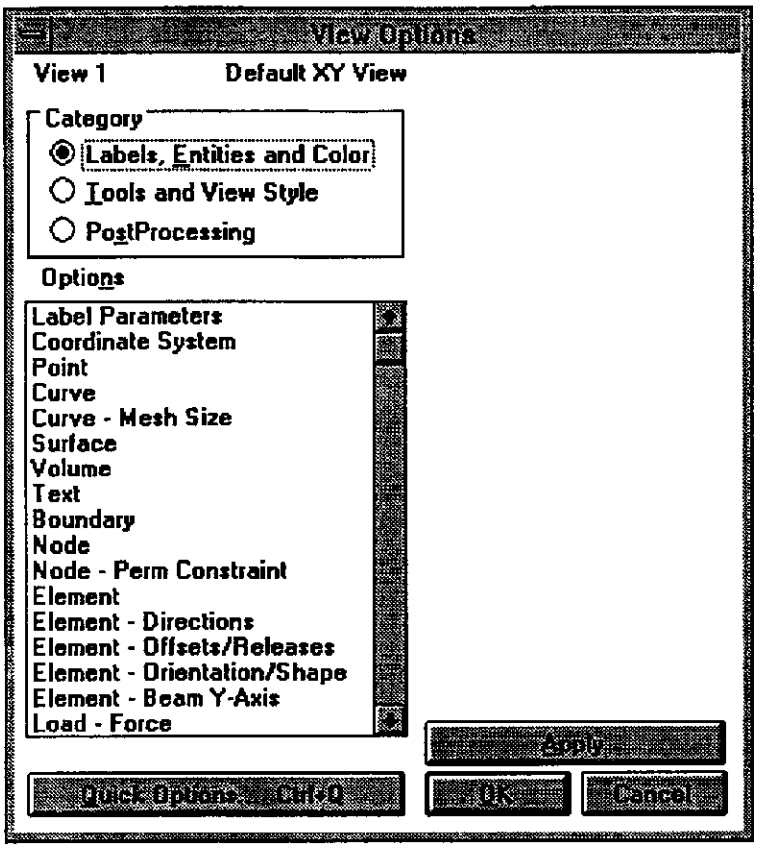
We could create another rectangle at this point, but will not, so choose **Cancel** to exit the rectangle creation dialog box.

The rectangle may be displayed in the corner of the display window. If so, you can resize and center the display by choosing **View Autoscale**.

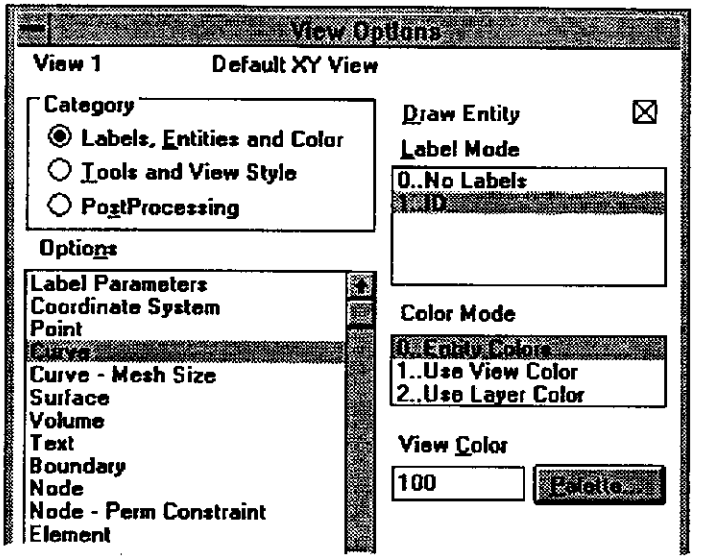


Display the Curve Labels

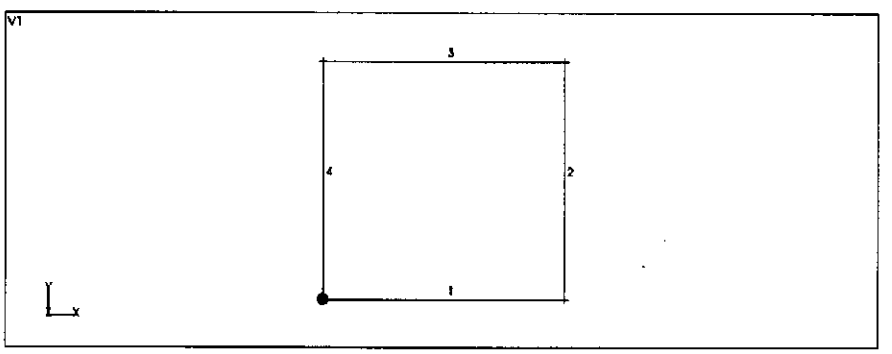
The default for the curve and other geometry labels is to be turned off. We will turn them on here, however, to assist in subsequent operations. To turn them on, choose **View Options**, which brings up a dialog box. Select **Labels, Entities, and Color**.



Choose **Curve** from the Options box, which brings up additional small boxes to the right of the Options box.

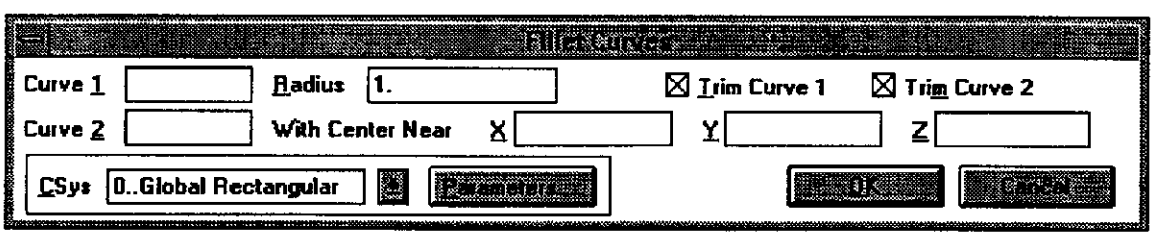


In the Label Mode box, select **ID** to display curve identifier numbers. Then choose **OK** to apply the labels and exit the View Options.

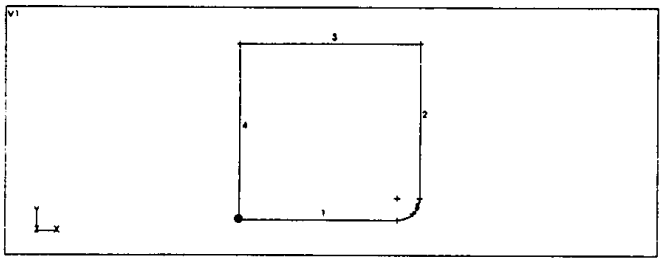


Fillet the Corners

To fillet the corners choose **Modify Fillet**, which brings up the Fillet Curves dialog box.



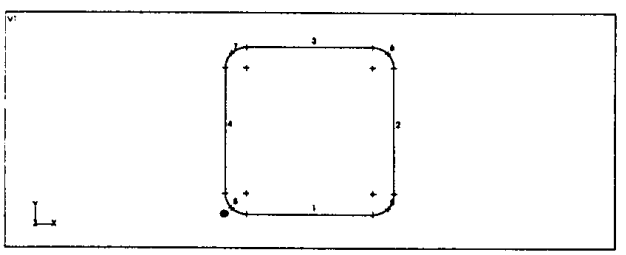
Enter 0.5 for the fillet **Radius**. To fillet the corner between curves 1 and 2, enter 1 for **Curve1** and 2 for **Curve2**; then, to indicate the fillet direction, enter 3 for **X** and 1 for **Y** to define an approximate fillet center of 3,1; then choose **OK**.



Now, change **Curve1** to 2 and **Curve2** to 3, and change **X** to 3 and **Y** to 3, to create a filleted corner between curves 2 and 3; choose **OK** to apply the fillet.

The filleted corner between curves 3 and 4 can be done in a similar manner, or you could use the mouse to do so graphically. By using the mouse, point a location just below and to the left of the approximate center of curve 3; this operation places 3 in the **Curve1** box. Next, point to a location just above and to the right of curve 4; this places 4 in the **Curve2** box. Note, too, that values for the With Center Near boxes have been filled in. Now, choose **OK** to apply the fillet.

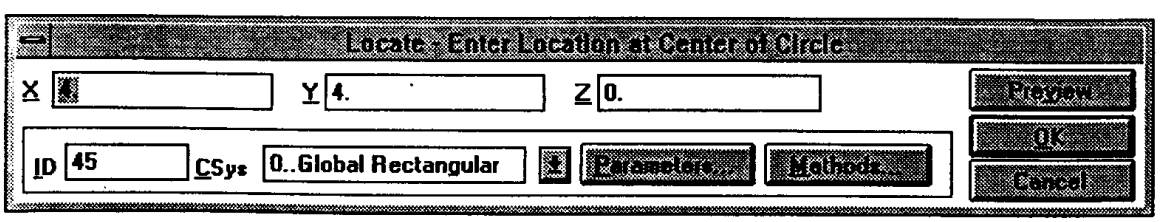
Repeat the same procedure by pointing to a location to the right and below the center of curve 4 (which places 4 in the **Curve1** box) and then above and to the left of curve 1 (which places 1 in the **Curve2** box), and then choose **OK**. This fillets the remaining corner.



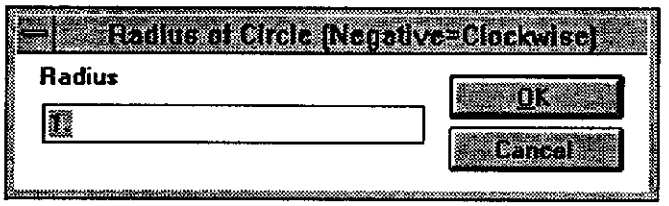
Exit the Fillet Curves box by choosing **Cancel**.

Create the Center Hole

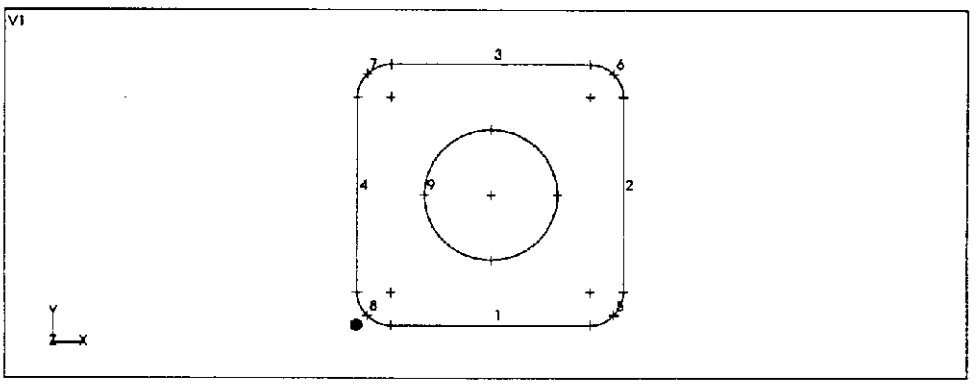
To create the center hole choose **Create Circle Center**, which brings up a dialog box.



Enter 2 for the **X** coordinate of the center and 2 for the **Y** corrdinate of the center; then choose **OK**, which brings up another box.



Enter 1 for the **Radius** and choose **OK** to create the circle. Then, since we do not want to create another circle, choose **Cancel**.



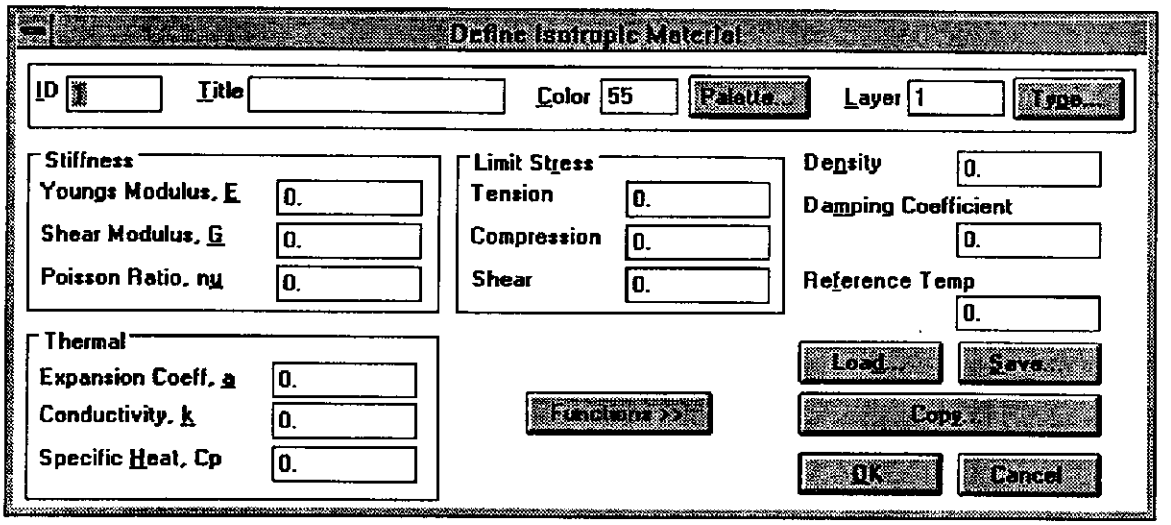
Save the Geometry

It is recommended that after numerous steps in building a complex model that you save your model file. While the steps shown above are not numerous--nor is the model complex--it is instructive, nevertheless, to illustrate how to save the model.

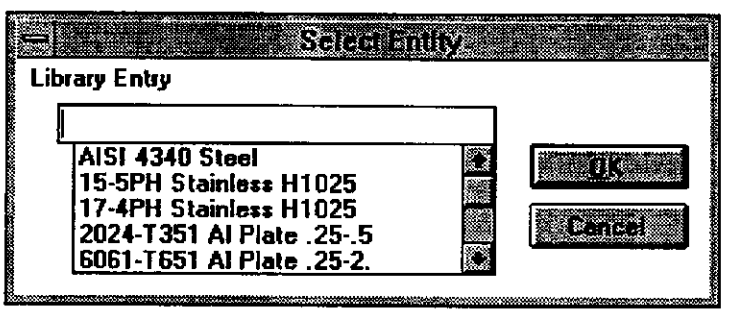
To save your model, choose **File Save**; the Save As dialog box appears ("save as" because the file has not yet been saved). Enter plate1 to save the file as plate1.mod. Note that "mod" is the default filename extension for a model file. Note, too, that this filename is now shown on the MSC/NASTRAN for Windows title bar. Next time you do a save of this model the updated model file will be saved with the same name.

Creating the Material Properties

Now that we have created the basic geometry we will create the properties, beginning with the material properties. Choose **Create Material**, which brings up the Define Isotropic Material dialog box. (Note that the default material type is isotropic.)



Material constants can be individually entered or they can be read from a materials library, which is what we will do in this example. To read the material properties, choose **Load**, which brings up the Select Entity box.

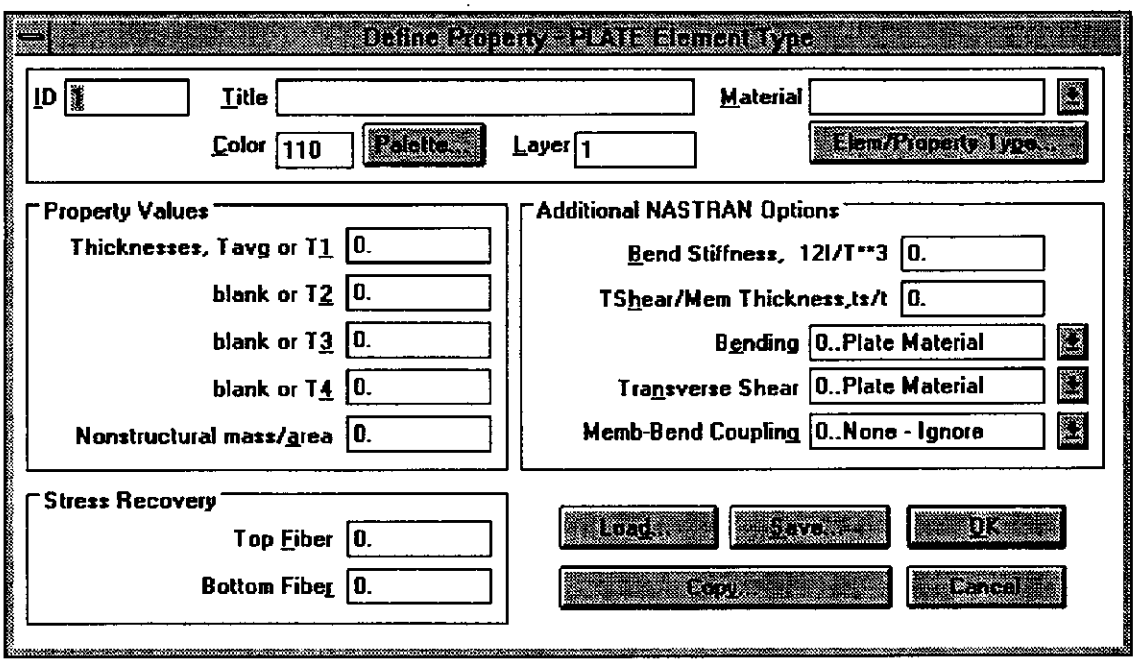


Select **AISI 4340 Steel**, choose **OK**, and note that the isotropic material properties are entered. Note that the Density is mass density, which is in units of lb-sec^2/in.^{4} for English units. Note, too, that the material ID is 1.

Choose **OK** to create the material properties and then **Cancel** to exit the menu.

Creating the Element Properties

Next we create element properties. Choose **Create Property** to bring up the Define Property--PLATE Element Type box. (Note that the default element type is plate elements.)



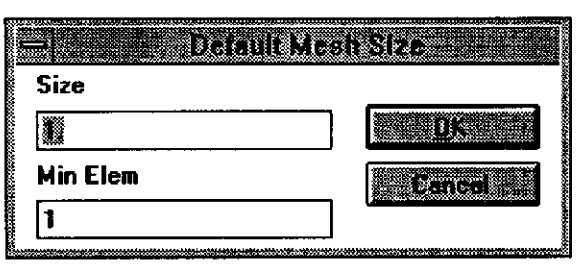
Enter 1 in the **Material** box to assign material 1 to this plate property. (Alternatively, you could scroll down the available material types by using the down arrow on the side of the Material box.)

Next, enter the plate properties. Under Property Values enter 0.1 for **T1**; this is the thickness of the plates. Defaults for Additional Options are fine for this example.

Choose **OK** to create the plate properties and then **Cancel** to exit the menu.

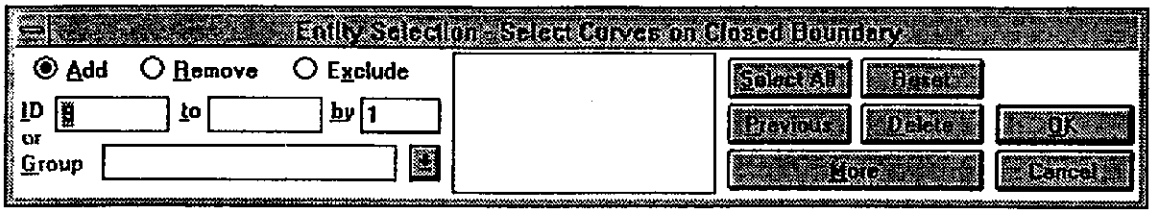
Generating the Mesh

After we define the engineering properties we create the element mesh. First, choose **Generate Mesh Size Default**, which brings up a dialog box.

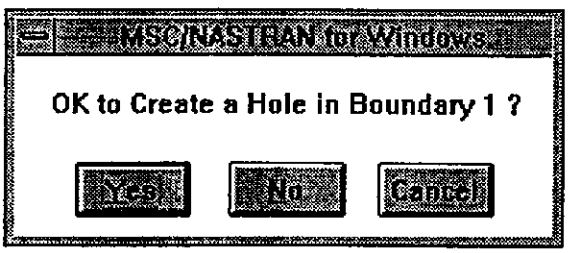


Enter 0.4 for **Size** (the default mesh size) and then choose **OK**.

Next, choose **Generate Define Boundary** to select the boundaries of the mesh. This brings up the Entity Selection box.

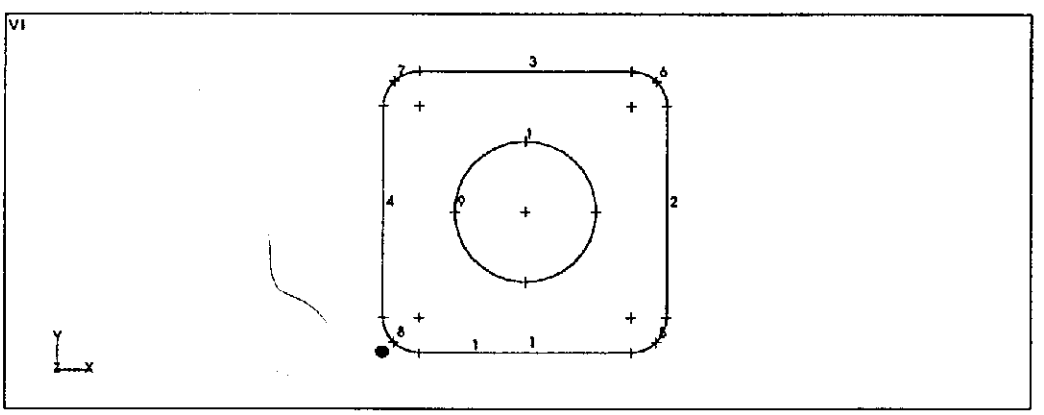


Select curve 1 (position the cursor near the center of the curve and select it), then in the same manner select curve 5 (the fillet on the right), curve 2, curve 6, curve 3, curve 7, curve 4, and finally curve 8--in that order. Then choose **OK**. The outer edge of the boundary is defined, and we have to now have to define the inner edge given by the hole.

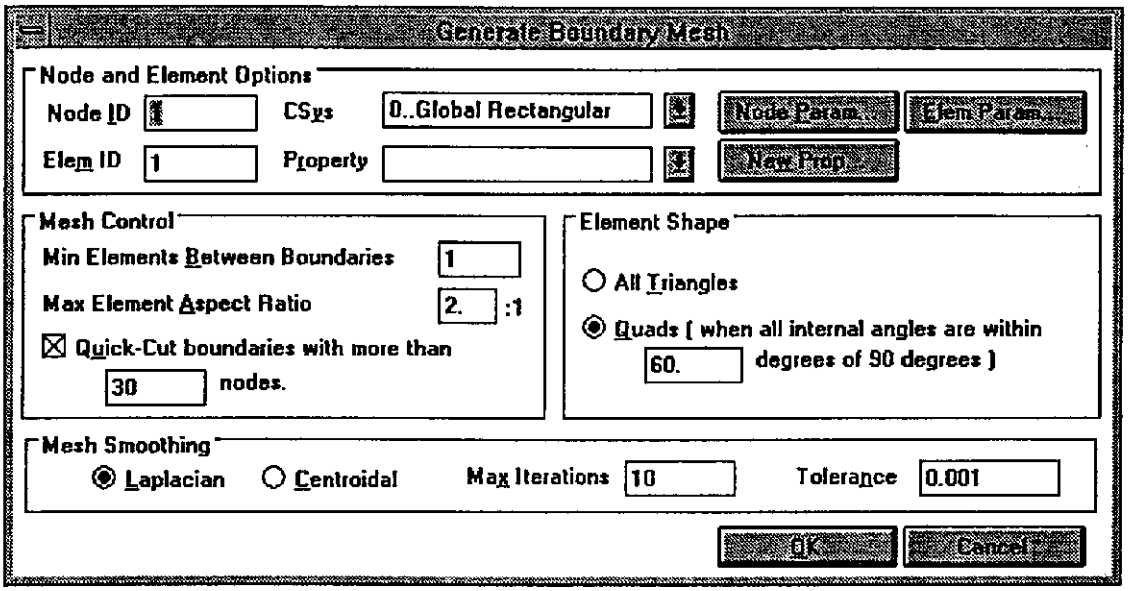


In the OK to Create a Hole in Boundary 1 box, choose **Yes**, and then select curve 9 (position the cursor over curve label 9 and click the mouse). Then choose **OK** to define the hole as curve 9. The box now asks, "OK to Create another Hole in Boundary 1?" and you choose **No**, since there is only one hole in this example.

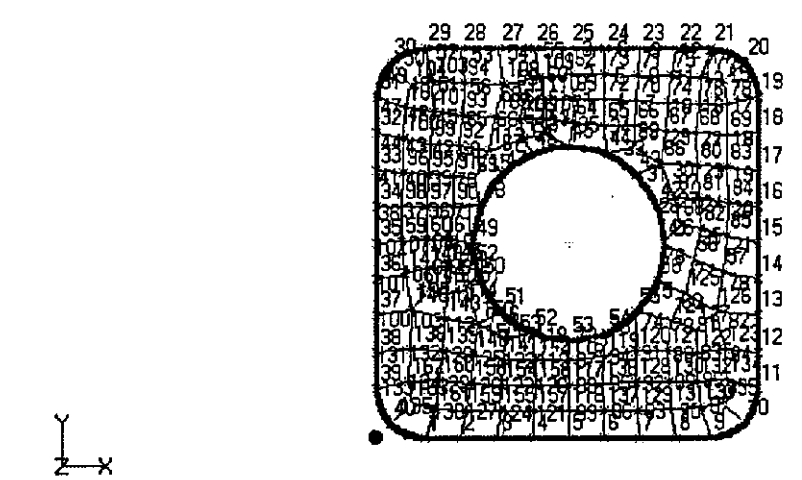
When the Boundary Definition box appears, choose **OK** to use the defaults, and note that the boundary edges are then highlighted.



Next, choose **Generate Boundary Mesh** and then choose **OK**, which brings up the Generate Boundary Mesh box.



Enter 1 in the **Property** box and then choose **OK**. Automatic quad meshing takes place, and there may be a 10-45 second wait, depending on the speed of your computer. When meshing is completed, 135 elements and 107 grid points are generated.

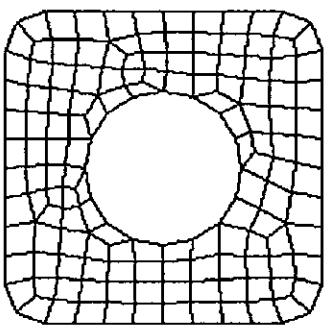


Modify the Display

In order to better view the model, turn off the element and grid labels. To do so, choose **View Options**, and under the Options menu choose **Element**. Select **No Labels** under the Label Mode, then choose **Apply**. Next choose **Node** under the Options menu, select **No Labels** under the Label Mode, turn off **Draw Entity** (by clicking on it), and finally choose **OK**. Element and grid point labels are now turned off, as are the grids themselves.

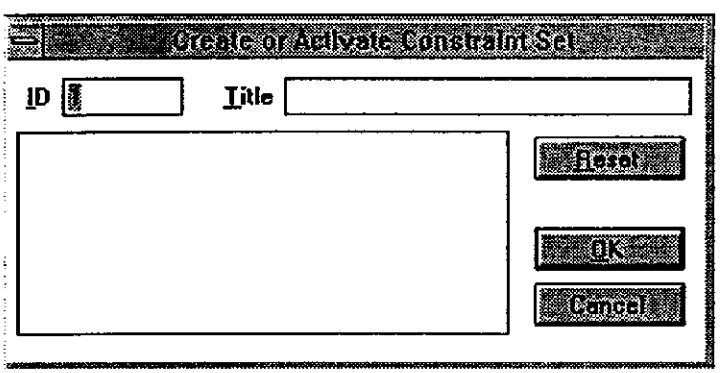
Other entities can also be turned off for clarity. Choose **View Options**, and under the Options menu select **Point**, then turn off **Draw Entity**, and choose **Apply**. Select **Curve**, turn off **Draw Entity**, then choose **Apply**. Select **Boundary**, turn off **Draw Entity**, then choose **OK**. The geometry (curves and points) and boundaries are now turned off.

The filled dot at the origin can also be turned off. Choose **View Options**, then under Category choose **Tools and View Style**, which brings up another set of boxes. Under Options select **Origin**, turn off **Draw Entity**, then choose **Apply**. To turn off the view legend, select **View Legend**, turn off **Draw Entity**, and choose **Apply**. To turn off the view axes, select **View Axes**, turn off **Draw Entity**, and choose **OK**.

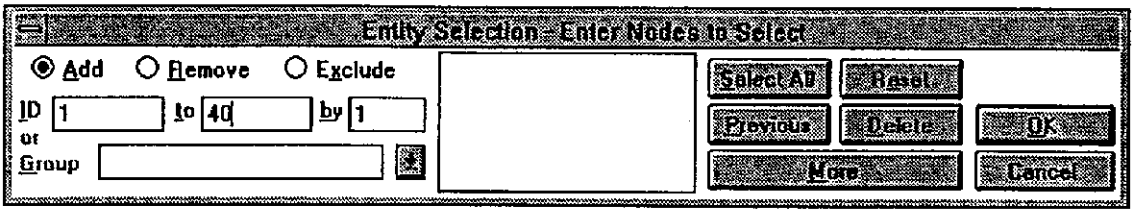


Creating the Boundary Conditions

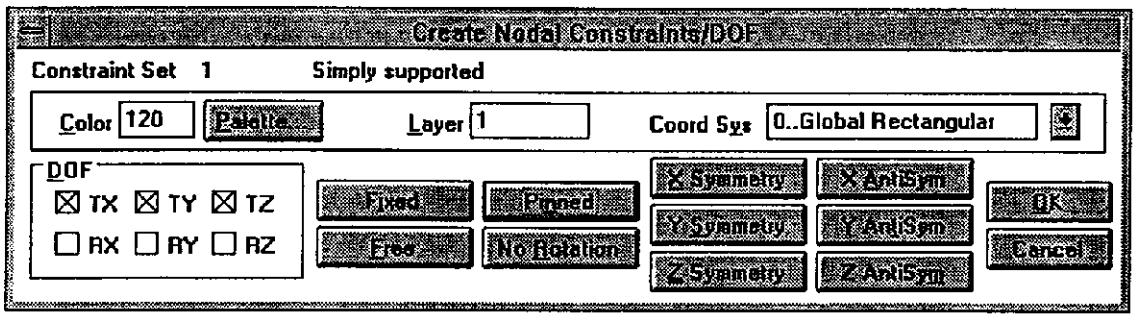
Next, we apply pinned supports to the outer edge. Choose **Create Constraint Set.**



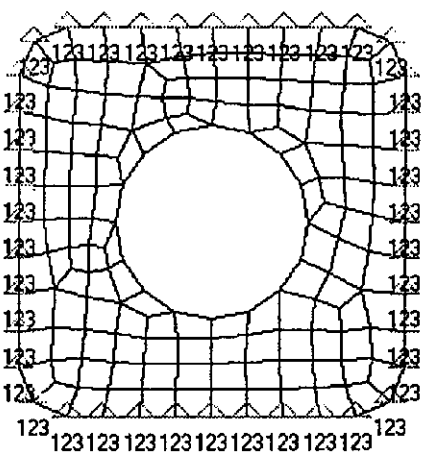
Enter "Simply supported" in the **Title** box and choose **OK**. Choose **Create Constraint Nodal**.



Enter 1 in the **ID** box and 40 in the **to** box, then choose **OK**; this brings up the Create Nodal Constraints dialog box.



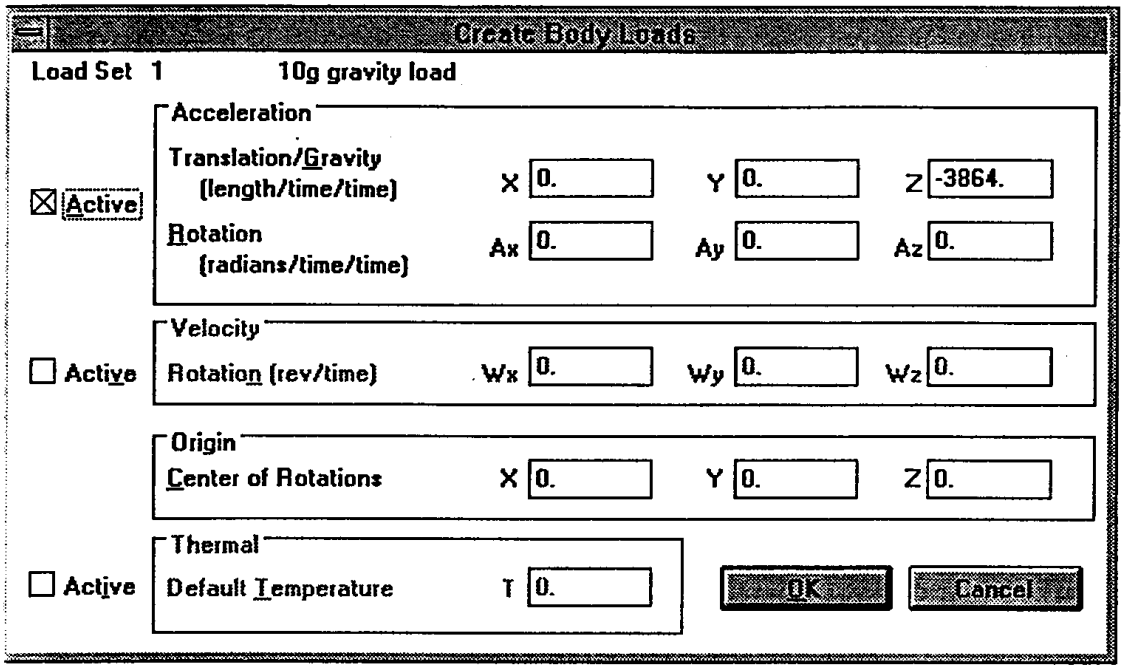
Select **Pinned** to define pinned supports, then choose **OK**. Pinned supports are applied to grid points 1 through 40 (the outer edge).



Choose **Cancel** to exit the constraint creation.

Creating the Applied Load

Next, we apply a 10g vertical load. Choose **Create Load Set**. Enter "10g gravity load" in the **Title** box and choose **OK**. Choose **Create Load Body** to bring up a dialog box.



Select **Active** to the left of Acceleration and enter -3864 for **Z.** (This defines a steady-state acceleration of 3864 in./sec^2--10g--in the -Z direction, which is normal to the plate.) Choose **OK** to apply the acceleration.

Saving the Model

Save the model again by choosing **File Save.**

Analyzing the Model

This demo version does not contain the solver, so we will read the MSC/NASTRAN for Windows results that are supplied with this demo.

Processing the Results

Results processing displays the computed results.

Read the Results

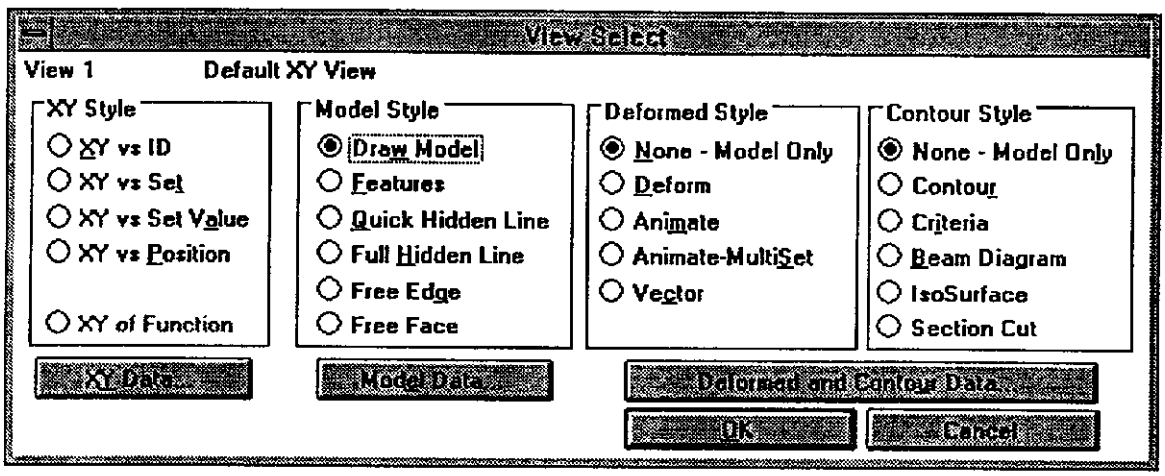
Choose **File Read Output**, select plate1.f06, choose **Open**, and choose **Yes** ("OK to begin reading file...").

Modify the Display

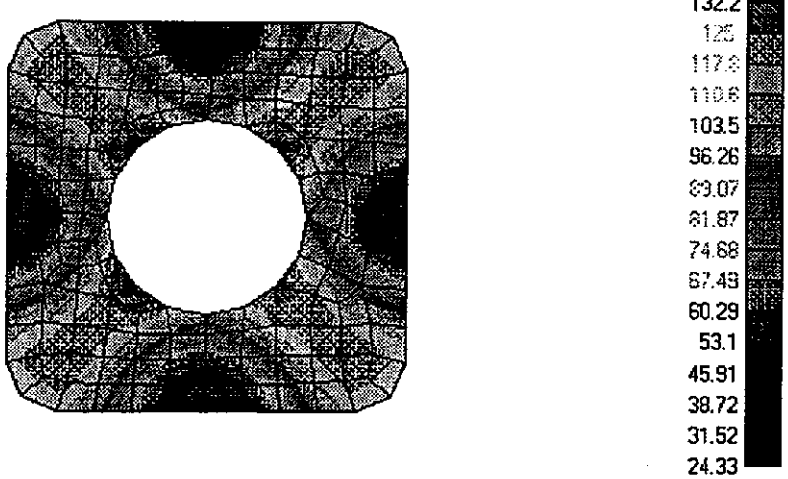
Turn off the constraint entities and labels. Choose **View Options**, select **Labels, Entities and Color**, and under the Options box select **Constraint**. Select **No Labels** and turn off **Draw Entity**, then choose **OK**.

Plot Stress Contours

Choose **View Select**, which brings up a dialog box.



Select **Contour** under Contour Style, and then choose **Deformed and Contour Data.** Under Output Vectors, Contour pick the scroll bar, scroll down, select **Plate Top VonMises Stress**, then choose **OK**. Choose **OK** again to draw the von Mises stress contours.

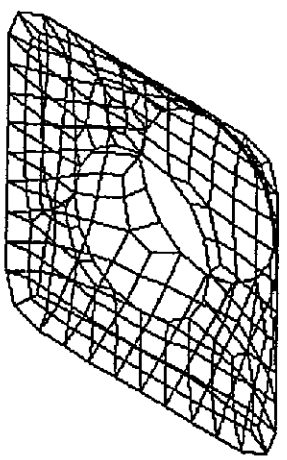


Output Set: MSC/NASTRAN Case 1
Contour: Plate Top VonMises Stress

Plot the Deformed Shape

Choose **View Select** and turn off Contour by selecting **None--Model Only** under Contour Options. Then select **Deform** under Deformed Style. Choose **Deformed and Contour Data**, and under Output Vectors, Deformation pick the scroll bar. Scroll down, select **T3 Translation** (which is Z translation), and then choose **OK**. Choose **OK** again to draw the deformed shape.

Note that the deformed shape is plotted in blue and is overlaid on the undeformed (white) shape. The deformation can better be viewed by rotating the model. Choose **View Rotate**, select **Isometric**, and then choose **OK**. The model is rotated such that the deformation can be seen.



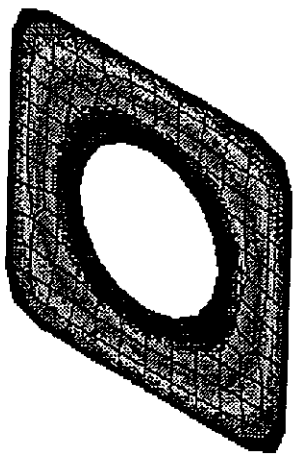
Output Set: MSC/NASTRAN Case 1
Deformed[0.000101]: T3 Translation

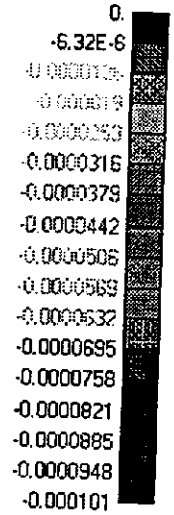
Animate the Deformation

The deformation can be animated. Choose **View Select**, and under Deformed Style choose **Animate**. Choose **OK** to animate the deformation. The animation continues until another menu--any menu--is selected.

Plot Deformation Contours

Choose **View Select**, select **None--Model Only** under Deformed Style, and select **Contour** under Countour Style. Choose **Deformed and Contour Data**, and under Output Vectors, Contour select **T3 Translation**. Choose **OK**, and then choose **OK** again.





Output Set: MSC/NASTRAN Case 1
Contour: T3 Translation

Exiting the Program

Congratulations! You have successfully completed the first example problem. To exit, choose **File Exit** and choose **Yes** to save the current model.

Working with MSC/NASTRAN for Windows

You can use this demo version to explore the graphics, model building, and results processing capabilities of MSC/NASTRAN for Windows, by several means:

Display an Example Model

Several sample models are supplied in the \examples directory. These files can be read by opening an existing model that has a .mod filename extension. For files that have been saved as neutral files (which are denoted with a .neu filename extension), open a **New Model**, choose **File Translate**, scroll down Format and select FEMAP Neutral File, select **Read**, and then **Open** the file.

Edit an Existing Model

To edit an existing model, choose **Open** and select the filename when MSC/NASTRAN for Windows first starts up. Try using different material properties, specifying different boundary conditions, or applying different loads. Remember to save the model with a different filename if you do not want to overwrite the original model.

Create a New Model

Using steps similar to those in the step-through example, create new models (of up to 200 grid points). You can "poke around" the menus when creating your models. If you need help, access the online help by choosing **Help Index**. You can read or print the help information in order to learn more about how to use the capabilities.

We recommend creating your model with the following steps:

Create the geometry (**Create Point, Create Line**, etc.).

Create the material properties (**Create Material**).

Select the element type (**Create Property**, then **Element/Property Type** and select the element type).

Create the element properties (**Create Property**).

Generate the element mesh (**Generate Mesh Size, Generate On Geometry**, etc.).

Create boundary conditions (**Create Constraint Set, Create Constraint Nodal**, etc.).

Create loads (choose **Create Load Set, Create Load Body**, etc.).

Save the model!

Read CAD Geometry

CAD geometry--in the form of DXF and IGES files--can be imported into MSC/NASTRAN for Windows. Choose **File Translate** and under **Format** scroll down and select DXF or IGES; then choose **Read**. The geometry is then read and displayed.

Read a Finite Element Model

Input files from virtually any finite element analysis program can be imported. (Note that for the demo version these models must be limited to a maximum of 200 grid points.) To read a finite element model, choose **File Translate** and under **Format** scroll down and select the program name; then choose **Read**. The model is then read and displayed.

Getting Around the Menus

The display is manipulated via the **View** menu. **View New** sets up multiple display windows, each of which can be resized and moved. When you have multiple windows, display operations take place only for the active window (which is selected by clicking on it). **View Select** selects the model display style and postprocessing options. **View Options** turns entities on and off individually.

Frequently-used commands have an icon on the Toolbar that runs across the top of the display window. As you scroll across these icons the description of each is shown at the top of the screen. For example, the first icon--a circle and an arrow--is dynamic pan, zoom, and rotate, for which you use the mouse to move (pan), zoom (enlarge), and rotate the model.

Many commands can also be selected via color icons in the Toolbox. You can display the Toolbox by choosing **Tools Toolbox**. Operations are performed when the appropriate icon is selected.

Graphics can be saved by choosing **File Picture Save**, which saves the active window as a bitmap image. The entire display--all windows and the toolbar--are saved by choosing **File Picture Save Desktop**. The bitmaps are saved with a white background for better black and white hardcopy; this can be changed by selecting **File Page Setup** and turning off **Swap Black and White**.

A useful command is **Tools Undo**, which undoes the last command. This lets you experiment with various options and get back to where you started if you make a mistake.