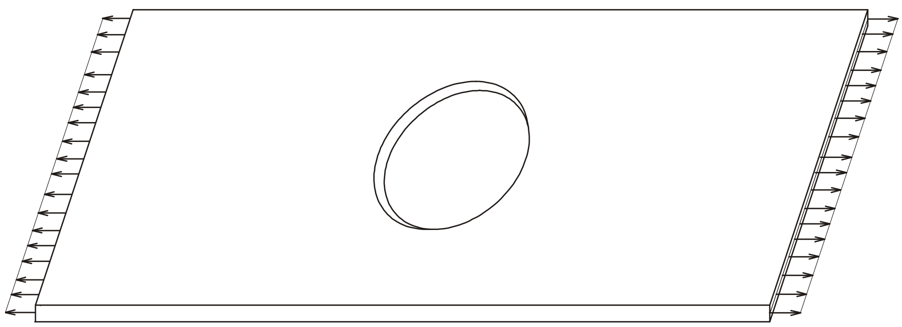
Workshop 3

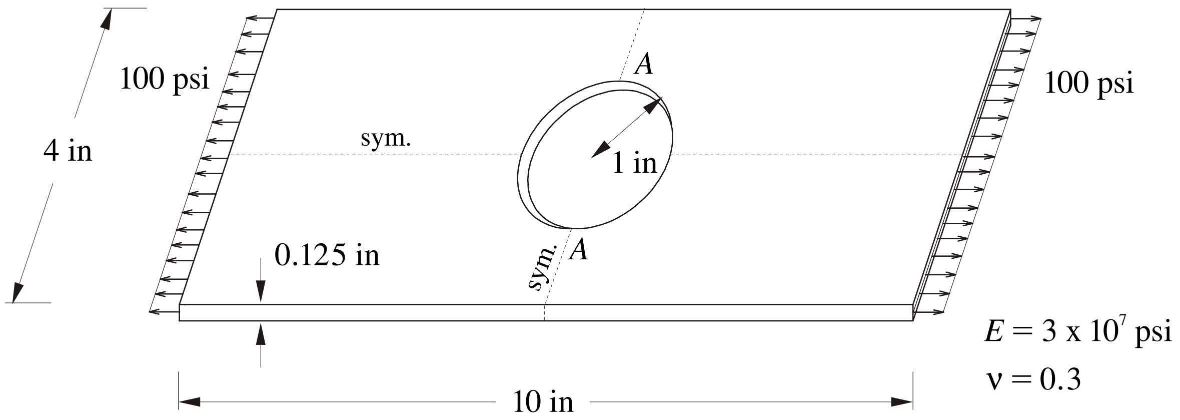
Linear Static Analysis of a Plate with a Circular Hole under Tension



Objectives

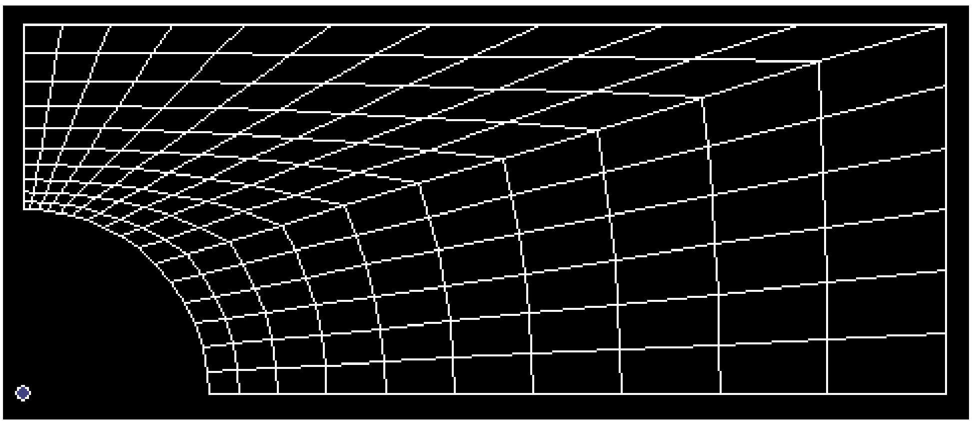
- Manually define material and element properties.
- Manually create the geometry for the plate using the given dimensions.
- Apply symmetric boundary constraints.
- Convert the pressure loading into nodal forces.
- Run the analysis.
- Compare results.

Model Description



The plate has a circular hole at the center and is submitted to a uniform tension of magnitude 100 psi at two opposite edges. The presence of the hole changes the uniform stress distribution, making the maximum tensile value, which occurs at A, slightly greater than four times the undisturbed value of 100 psi.

Because we have two axes of symmetry, we need consider only one quadrant. We shall use 160 elements of varying size and orientation with smaller elements appearing around the hole, specifically at A, where we can expect large gradients in the displacement field.



Exercise Procedure

1. Start up **MSC/NASTRAN for Windows 4.5** and begin to create a new model.

   Double click on the icon for the **MSC/NASTRAN for Windows V4.5**.

   On the *Open Model File* form, select **New Model**.

   Turn off the workplane:

   Tools / Workplane (or **F2**) / ☐ **Draw Workplane / Done**

   View / Regenerate (or **Ctrl G**).

2. Create a material called **mat_1**.

   From the pulldown menu, select **Model / Material**.

Title	**mat_1**
Young's Modulus	**3e7**
Poisson's Ratio	**0.3**

   Select **OK / Cancel**.

   NOTE: In the *Messages Window* at the bottom of the screen, you should see a verification that the material was created. You can check here throughout the exercise to both verify the completion of operations and to find an explanation for errors which might occur.

3. Create a property called **prop_1** to apply to the members of the plate.

   From the pulldown menu, select **Model / Property**.

Title	**prop_1**
Material	**mat_1**

   Note that the default element type is **Plate** element, **not parabolic**.

Thickness, Tavg or T_1	**0.125**

   Select **OK / Cancel**.

4. Create the geometry for the inner circle.

   Due to symmetry considerations, only a quarter of the geometry needs to be created.

   Make the geometry in standard form:

   Tools / Advanced Geometry

   Geometry Engine ✓ **Standard**

   Select **OK**.

   First, create two 45° arcs with radii of 1 in:

   Geometry / Curve-Arc / Radius-Start-End

CSys		**Basic Cylindrical**					
Start of Arc	*R*	*1*	*T*	*0*	*Z*	*0*	**OK**
End of Arc	*R*	*1*	*T*	*45*	*Z*	*0*	**OK**
Radius		*1*		**OK**			

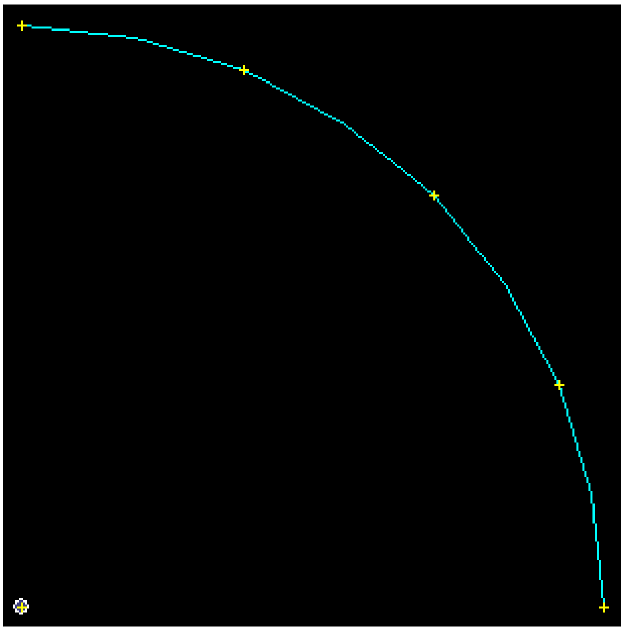
   Create the second arc:

Start of Arc	*R*	*1*	*T*	*45*	*Z*	*0*	**OK**
End of Arc	*R*	*1*	*T*	*90*	*Z*	*0*	**OK**
Radius		*1*		**OK**			

   Select **Cancel**.

   To fit the display onto the screen, select **View / Autoscale / Visible** (or **Ctrl A**).

   On your display, there should now be a quarter of a circle from 0° to 90°.



5. Create the geometry for the outer perimeter.

   Create a first line curve:

   Geometry / Curve-Line / Project Points

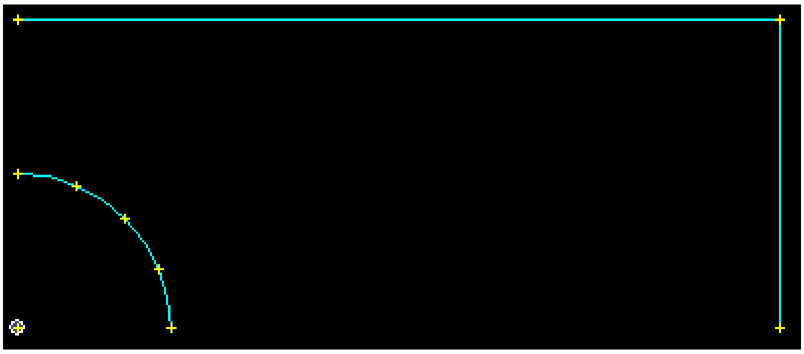
CSys	**Basic Rectangular**						
First Location	*X*	*5*	*Y*	*0*	*Z*	*0*	**OK**
Second Location	*X*	*5*	*Y*	*2*	*Z*	*0*	**OK**

   Create a second line curve:

First Location	*X*	*5*	*Y*	*2*	*Z*	*0*	**OK**
Second Location	*X*	*0*	*Y*	*2*	*Z*	*0*	**OK**

   Select **Cancel**.

   To fit the display onto the screen, select **View / Autoscale / Visible** (or **Ctrl A**).



6. Create the geometry for the surface of the plate.

   First, turn on the curve labels:

   View / Options (or **F6**).

Options	**Curve**
Label Mode	**ID**

   Select **OK**.

   Create 2 surfaces:

   Geometry / Surface / Ruled

From Curve	*2*	To Curve	*4*	**OK**
From Curve	*1*	To Curve	*3*	**OK**

   Select **Cancel**.

   When we created the surfaces, two coincident curves were generated along the common edges. Merge these curves:

   Tools / Check / Coincident Curves / Select All / OK

Options	✓	**Merge Coincident Entities**

   Select **OK**.

   View / Regenerate (or **Ctrl G**).

Notice that there is now only one curve along the common edge. This will make it easier to apply the mesh control to the surfaces.

7. Define appropriate mesh size for critical edges.

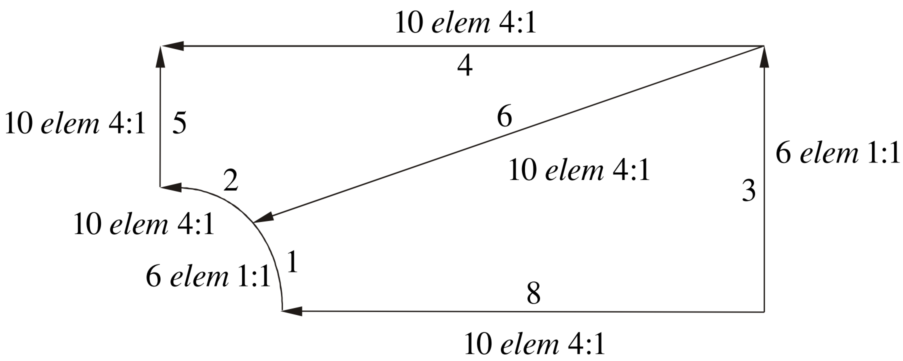
   It is very important to know the direction of each edge of the surfaces to define an irregular mesh:

   View / Options (or **F6**) / ✓ **Tools and View Style**

   Highlight **Curve and Surface Accuracy / Show Arrows**

   Select **OK**.

   The edges are pointing in the direction indicated in the figure.



   Mesh / Mesh Control / Size Along Curve

   Select curves **2**, **4**, **6**, **8** and, then, **OK**.

Number of Elements	**10**
Node Spacing	✓ **Biased**
Bias Factor	**4**
	✓ **Small Elements at End**

   Select **OK**.

   The *Bias Factor* controls the spacing of nodes. Setting it to 4, with linear bias (✓ **Biased**), will make the first element four times as big as the last if you select

"Small Elements at End". The first element refers to the element at the first point of the curve, according to the directions shown in the figure.

Select curve **5** and, then, **OK**.

Number of Elements	**10**	
Node Spacing	✓	**Biased**
Bias Factor	**4**	
	✓	**Small Elements at Start**

Select **OK**.

Select curves **1**, **3** and, then, **OK**.

Number of Elements	**6**	
Node Spacing	✓	**Equal**

Select **OK** / **Cancel**.

8. Generate the finite elements.

Mesh the two surfaces:

Mesh / Geometry / Surface

Select both surfaces and, then, **OK**

Property	**prop_1**

Select **OK**.

9. Remove coincident grids from the model.

Additional nodes were created while generating QUAD4 elements. To eliminate these coincident nodes, do the following:

Tools / Check / Coincident Nodes / Select All / OK

When asked if you wish to specify an additional range of nodes to merge, respond **No**.

Options ☑ **Merge Coincident Entities**

Select **OK**.

As the *Messages Window* states, 11 nodes have now been merged.

To better view the display, do the following to remove the unnecessary labels:

View / Options / Quick Options (or **Ctrl Q**) / ☐ **Point** / ☐ **Curve** / ☐ **Surface** / ☐ **Node** / **Labels Off** / **Done** / **OK**.

On your display, there should now be the figure shown at the bottom of page 3-2.

10. Create the model constraints.

Before creating the appropriate constraints, a constraint set needs to be created. Do so by performing the following:

Model / Constraint / Set

Title **constraint_1**

Select **OK**.

This constraint set will have 3 different constraints.

Now, define the first relevant constraint for the model.

Model / Constraint / Nodal / Method^/ on Curve

Select the left edge / **OK**.

On the *DOF* box, select **X Symmetry / OK**.

Next, define the second relevant constraint for the model.

Method^/ on Curve

Select the bottom edge / **OK**.

On the *DOF* box, select **Y Symmetry / OK**.

Finally, define the third relevant constraint for the model.

Select the node on the bottom left corner / **OK**.

On the *DOF* box, fix TZ to restraint Z direction movement (just to remove the rigid body motion):

✓	**TZ**

Select **OK**.

A warning message will appear: *Selected Constraints Already Exist. OK to Overwrite (No = Combine)?* Select **No** to combine and, then, **Cancel**.

11. Create the loading conditions.

Like the constraints, a load set must first be created before creating the appropriate model loading.

Model / Load / Set (or **Ctrl F2**)

Title	**load_1**

Select **OK**.

Since the type of the given load (pressure) is not an available option for the edge of the plate, it must first be converted into nodal forces or distributed along the edge length and, then, applied to the model.

In this model, a 100 psi pressure force acting over the 0.25 in^2 (2 in $\times$ 0.125 in) can be converted to a total equivalent nodal force of 25 lbf. Since we are going to distribute this force over 2 in of edge length, the force per length will be 12.5 lbf/in.

Model / Load / On Curve

Select the right edge / **OK**.

Highlight **Force Per Length**

Load	**FX**	✓	**12.5**

Select **OK / Cancel**.

To visualize nodal forces:

Model / Load / Expand / OK

View / Options (or **F6**)

Category	✓ **Labels, Entities and Color**
Options	**Load Vectors**
Vector Length	**Scale by Magnitude**
Options	**Load-Force**
Label Mode	**Load Value**

Select **OK**.

View / Regenerate (or **Ctrl G**).

Note that the nodes at the corners are loaded half as much as the inner nodes because they are surrounded by half as much area.

12. Run the analysis.

File / Analyze

Analysis Type	**Static**
Loads	✓ **load_1**
Constraints	✓ **constraint_1**
	✓ **Run Analysis**

Select **OK**.

When asked if you wish to save the model, respond **Yes**.

Be sure to set the desirable working directory.

File Name	**work_3**

Select **Save**.

When the MSC/NASTRAN manager is through running, MSC/NASTRAN for Windows will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran successfully, we will not bother with the details this time. Then, select **Continue**.

13. View the results of the analysis.

First, however, you may want to remove the load and boundary constraint markers.

View / Options / Quick Options (or **Ctrl Q**)

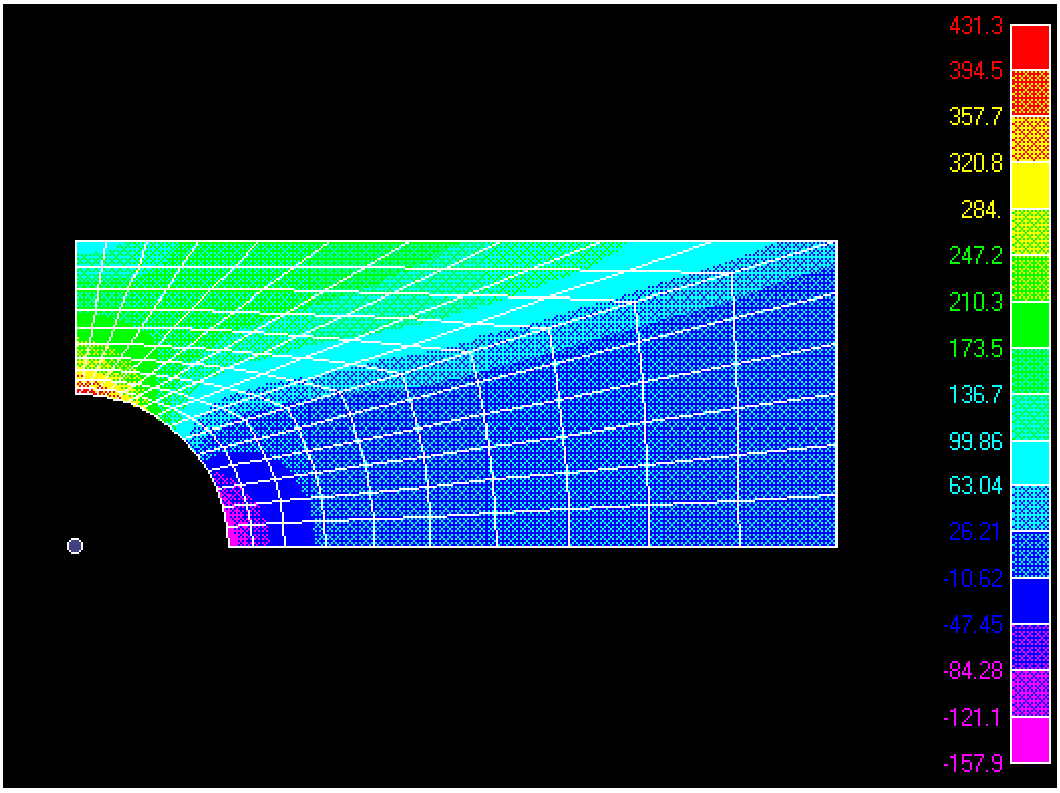
□ **Force** / □ **Constraint / Done / OK**

View the X Normal Stress Fringe Plot:

View / Select (or **F5**)

Deformed Style	**None - Model Only**
Contour Style	**Contour**
	Deformed and Contour Data
Output Vectors / Contour	**7020 Plate Top X Normal Stress**

Select **OK / OK**.



From the Stress Scale, what is the maximum σ_x stress and where it occurs?

Confirm the answer clicking **Off**, at the right bottom side of the screen, and then selecting **Node**.

Leaving the cursor next to the top of the circular hole, the stress value that comes out coincides with the maximum value shown in the Stress Scale.

This concludes the exercise.

File / Save

File / Exit.

Answer

	MSC/NASTRAN	**Theoretical value**
Maximum stress	431.3	430

Improved results could be obtained by making a finer mesh, particularly near the hole.

NOTE: The QUAD4 element provides the stress concentration factor 4.31 ($\approx$ 431.3/100), which agrees accurately with 4.30 (= 430/100) given by the theory of elasticity (Timoshenko, S. P., and Goodier, J. N., 1970, *Theory of Elasticity*, 3rd edition, McGraw-Hill Kogakusha, Tokyo, page 95).