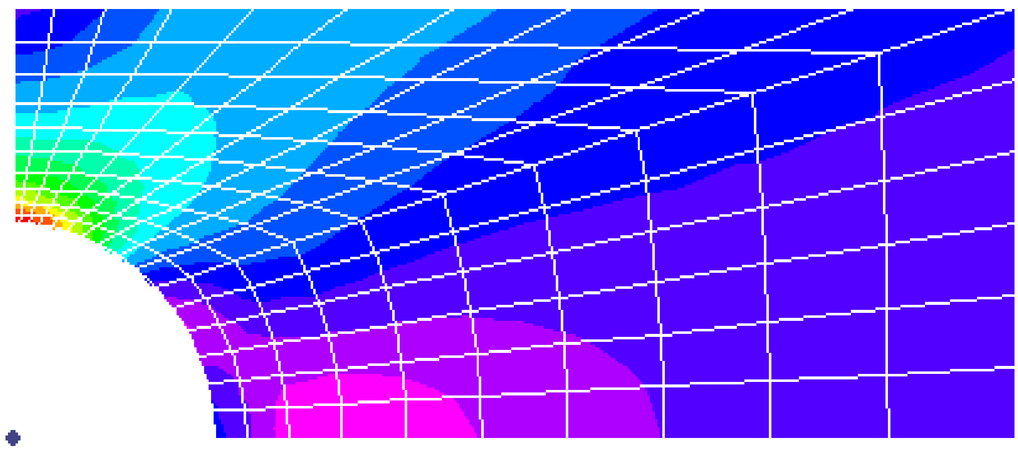
WORKSHOP 14

Analysis of a Tension Coupon



Objectives:

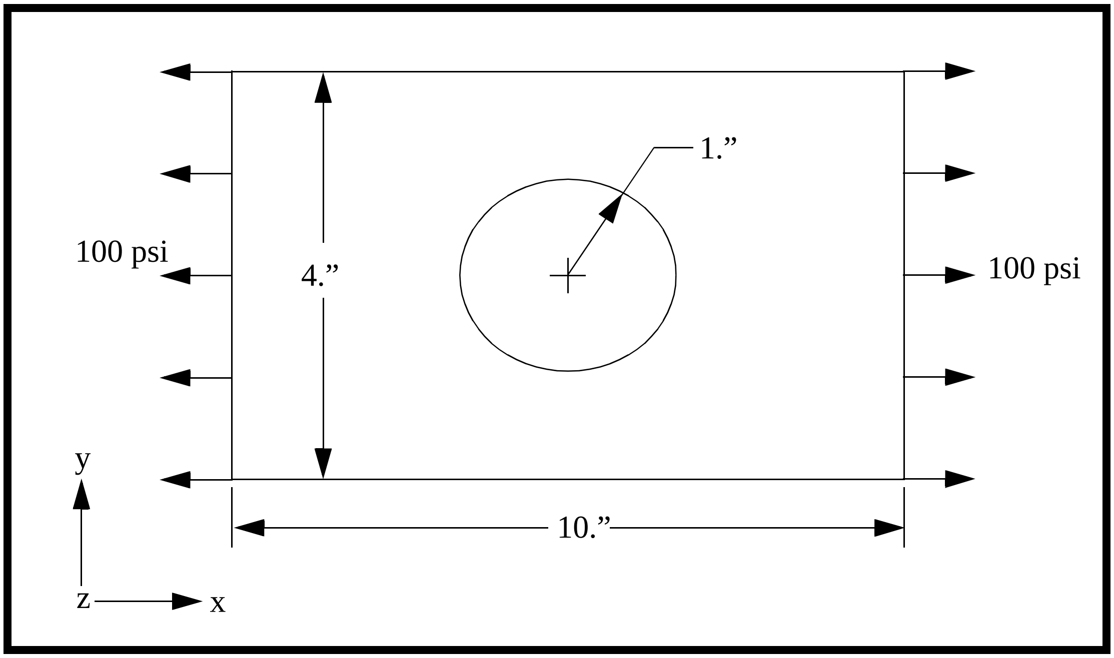
- Manually define material and element properties.
- Manually create the geometry for the tension coupon using the given dimensions.
- Apply symmetric boundary constraints.
- Convert the pressure loading into nodal forces.
- Run the analysis.
- Compare results.

Model Description:

For the following problem, the user is expected to run the complete analysis and verify the results provided. This problem has the theoretical result. Users should not attempt these exercises unless they can construct geometry, create a finite element mesh, apply loads, material and element properties, run the analysis and post-process the results or if they are willing to make a few mistakes and take a few wrong turns while trying. MSC.Nastran for Windows is intended to be self-explanatory, so it is okay to explore and try different techniques to complete this exercise, especially with a qualified instructor to help when you get lost.

An experienced MSC.Nastran for Windows modeler should take no longer than 15 minutes to complete this problem, including the analysis.

Figure 14.1 - Load Conditions



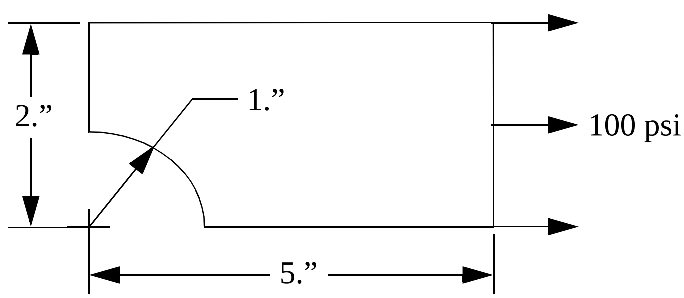
- Thickness =0.125 in.
- $E = 30,000,000$ psi
- $\nu = 0.3$

Answer (*Theory of Elasticity,* Timoshenko & Goodier, 3rd edition, page 95):

The answer given is the maximum at two points on the plate (Where?).

- $\frac{P}{A}$ Þ $K_f = 100.(4.3)$
- $\boldsymbol{\sigma_{xx} = 430}$ **psi**

Because of symmetry, the model simplifies into the model below.



Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0 and begin to create a new model.

   Double click on the icon labeled **MSC.Nastran for Windows V4.0.**

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

Open Model File:	**New Model**

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

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

   Model/Material...

Title:	**mat_1**
Youngs Modulus:	**30E6**
Poisson's Ratio:	**0.3**

   OK

   Cancel

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

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

   Model/Property...

Title:	**plate**

   To select the material, click on the list icon next to the databox and select **mat_1.**

Material:	**1..mat_1**
Thicknesses, Tavg or T1:	**0.125**

   OK

   Cancel

4. Create the geometry for the inner circle.

The model of this coupon is symmetrical in the x and y direction. Therefore, only a quarter of the geometryneeds to be created.

Tools/Advanced Geometry...

Geometry Engine: ● **Standard**

OK

First, create 2 arcs with radii of 1 using the **Geometry/Curve-Arc/Radius-Start-End** command.

Geometry/Curve-Arc/Radius-Start-End

CSys: **1..Basic Cylindrical**

Locate-Enter Location at Start of Arc.

R:	**1**	*T:*	**0**	*Z:*	**0**	**OK**

Locate-Enter Location at End of Arc.

R:	**1**	*T:*	**45**	*Z:*	**0**	**OK**

Radius: **1** **OK**

Repeat the above steps to create a second curve using the following data.

..Start..	*R:*	**1**	*T:*	**45**	*Z:*	**0**	**OK**
..End..	*R:*	**1**	*T:*	**90**	*Z:*	**0**	**OK**
Radius:		**1**		**OK**			

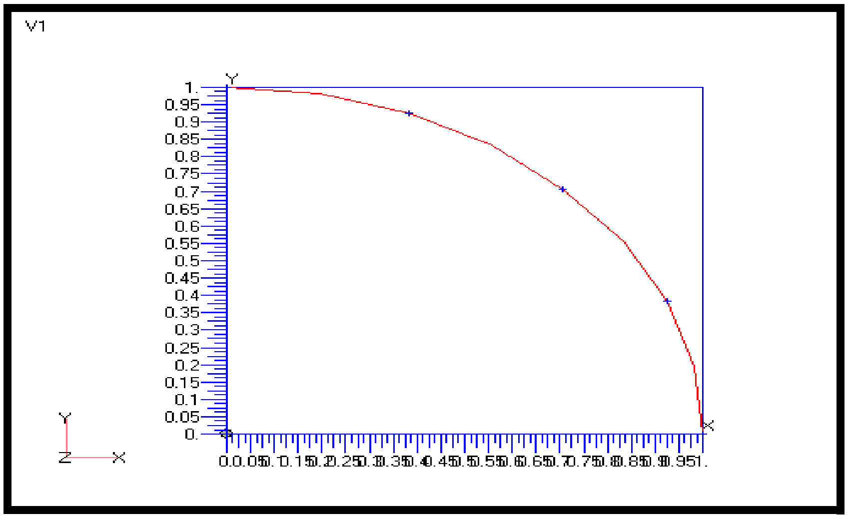
Cancel

To fit the display onto the screen, use the **Autoscale** feature.

View/Autoscale **<Ctrl+A>**

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

Figure 14.2



5. Create the geometry for the outer perimeter.

Create two opposite edges using the **Geometry/Curve-Line/Project Points...** command.

Geometry/Curve-Line/Project Points...

CSys: **0..Basic Rectangular**

...First Location...

X: **5** *Y:* **0** *Z:* **0** **OK**

...Second Location...

X: **5** *Y:* **2** *Z:* **0** **OK**

Repeat the above steps to create the second line using the following data.

...First...	*X:*	**5**	*Y:*	**2**	*Z:*	**0**	**OK**
...Second...	*X:*	**0**	*Y:*	**2**	*Z:*	**0**	**OK**

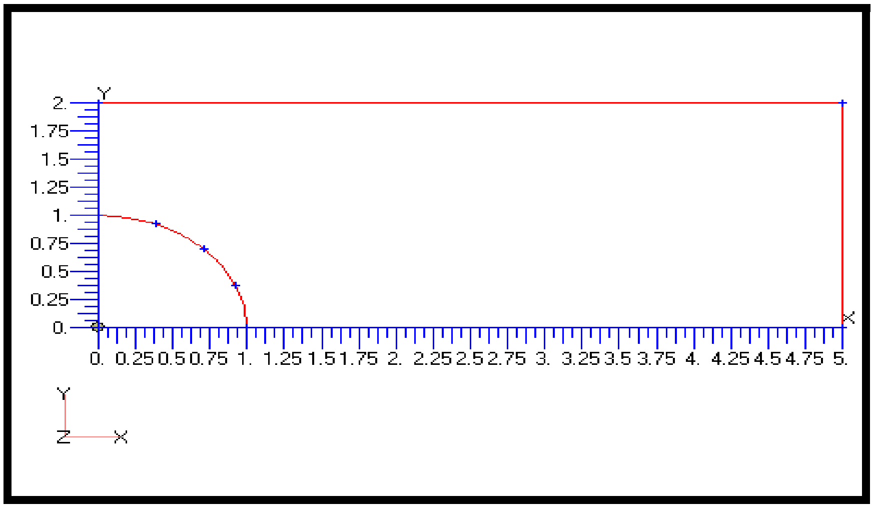
Cancel

To fit the display onto the screen, use the **Autoscale** feature.

View/Autoscale <Ctrl+A>

The viewport should appear as shown.

Figure 14.3



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

First, turn on the curve labels.

View/Options... <F6>

Options:	**Curve**
Label Mode:	**1..ID**
OK	

Create 2 surfaces using the **Geometry/Surface/Ruled** command.

Geometry/Surface/Ruled...

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

When we created the surfaces, two coincident curves were generated along the common edge.

Merge these curves.

Tools/Check/Coincident Curves...

Select All

OK

Options: ☒ **Merge Coincident Entities**

OK

Turn off the workplane.

Tools/Workplane...

☐ **Draw Workplane**

Done

View/Regenerate <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 curve and line. The direction will determine the bias ratio of the mesh seeds. In steps 4 through 6, the geometry of this model was created so that the direction of the lines, curves and surfaces were pointing in the direction shown in Figure 14.4.

View/Options... <F6>

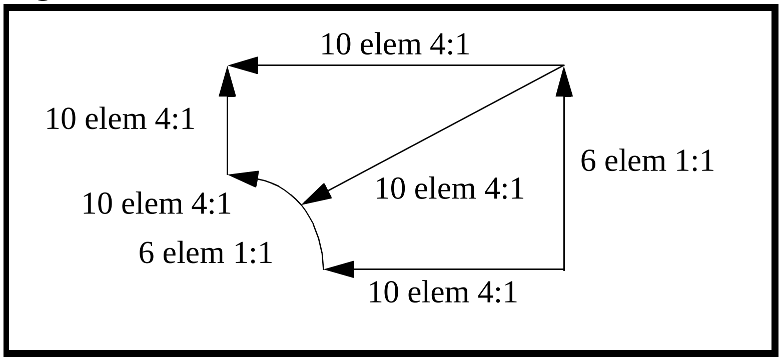
Category: ● **Tools and View Style**

Options: (highlight) **Curve and Surface Accuracy**

Parametric Directions: (highlight) **1..Show Arrows**

OK

Figure 14.4



Using the **Mesh/Mesh Control/Size Along Curve...** command, create the appropriate mesh seeds. The seeding varies depending on the *Number of Elements* and the density of the seeds set with the *Bias* inputs.

Mesh/Mesh Control/Size Along Curve...

<Select curve - ID #2, 4, 6, 8>

OK

● **Number of Elements** **10**

Node Spacing: ● **Biased**

Bias Factor: **4**

● **Small Elements at End**

OK

<Select curve - ID # **5**>

OK

● **Number of Elements** **10**

Node Spacing: ● **Biased**

Bias Factor: **4**

● **Small Elements at Start**

OK

<Select curve - ID # **1, 3**>

OK

● **Number of Elements** **6**

Node Spacing: ● **Equal**

OK

Cancel

8. Generate the finite elements.

Mesh the two surfaces using the **Mesh/Geometry/Surface...** command.

Mesh/Geometry/Surface...

<*select both surfaces* **+1, +2**>

OK

Property: **1..plate**

OK

9. Remove coincident grids from the model.

As you may have noticed on the screen, additional nodes were created while generating quad elements. To eliminate these coincident nodes, do the following:

Tools/Check/Coincident Nodes...

Select All

OK

When asked if it is OK to specify an additional range of nodes to merge, respond **No**.

No

Options: ☒ **Merge Coincident Entities**

OK

The message window should state that “11 Node(s) Merged”.

To better view the display and verify that duplicate nodes are deleted, do the following to remove the unnecessary labels.

View/Options... <F6>

Quick Options...

☐ **Node**

Labels Off

Done

Category:	● **Labels, Entities and Color**
Options: (highlight)	**Load-Force**
Label Mode:	**1..Load Value**
Options:	**Constraint**
Label Mode:	**1..Degree of Freedom**
Category:	● **Tools and View Style**
Options: (highlight)	**Curve and Surface Accuracy**
Parametric Direction:	**0..Off**

OK

10. Create the model constraints.

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

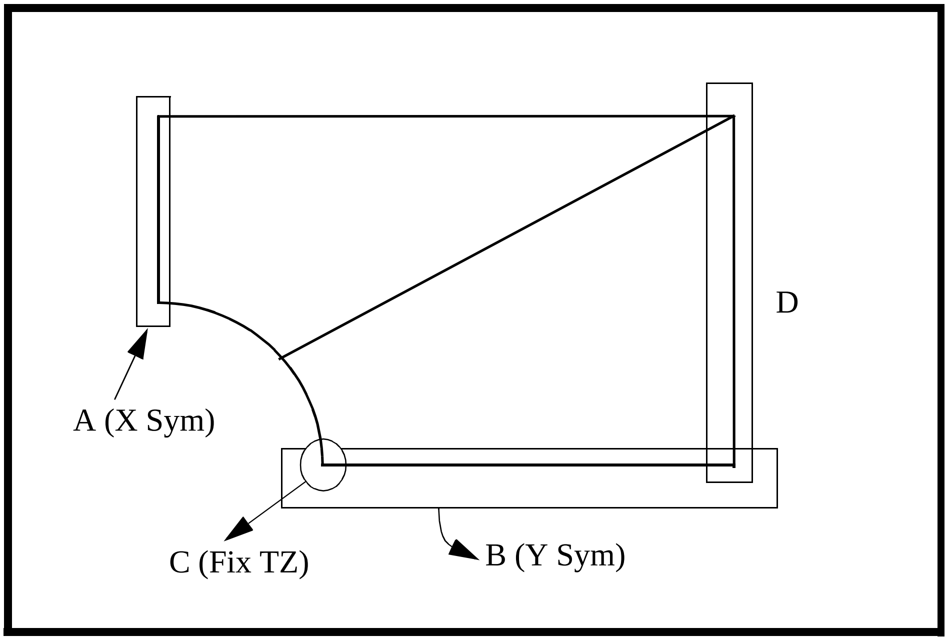
Model/Constraint/Set...

Title:	**constraint**

OK

This constraint set will have 3 different constraints.

Figure 14.5



Now define the first relevant constraint for the model. Use Figure 14.5 to identify appropriate curve selections.

Model/Constraint/Nodal...

Method^	**on Curve**

<Select Curve 5 on the left edge.   Box "A" in Figure 14.5>

OK

X Symmetry

On the *DOF* box, TX, RY, and RZ should now have check marks.

☒ **TX**

☒ **RY** ☒ **RZ**

OK

Next, define the second relevant constraint for the model.

Method^	**on Curve**

<Select Curve 8 on the bottom edge.   Box "B" in Figure 14.5>

OK

Y Symmetry

On the *DOF* box, TY, RX, and RZ should now have check marks.

☒ **TY**

☒ **RX** ☒ **RZ**

OK

Finally, define the third relevant constraint for the model.

<Select node 122 on the bottom left corner. Circle “C” in Figure14.5>

OK

On the *DOF* box, choose TZ to restrain movement in the Z direction.

☒ **TZ**

OK

A warning message will appear: “Selected concurrents already exist. OK to Overwrite (No = Combine)?” Select **NO** to combine.

No

Cancel

NOTE: This constraint is necessary to constrain the rigid body motion in and out of plane.

11. Create the loading conditions.

Before creating the appropriate loading a load set needs to be created. Do so by performing the following:

Model/Load/Set...

Title: **tension**

OK

Since pressure is not an available option, the pressure must be converted into nodal forces and applied to the model.

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

Note that when N4W writes to the .dat file, the 2 nodes on each corner will have 2.083 lbs applied to them, while the 5 inner nodes will have 4.167 lbs applied to them.

Model/Load/On Curve...

<Select Curve 3 on the right edge. Box "D" in Figure 14.5>

OK

(highlight) — **Force Per Length**

FX ☒ — **12.5**

OK

Cancel

To visualize nodal forces

Model/Load/Expand...

OK

View/Regenerate <Ctrl+G>

12. Create the input file for analysis.

File/Export/Analysis Model...

Analysis Format/Type: **1..Static**

OK

Change the directory to **C: \temp**.

File Name: **tension**

Write

☒ **Run Analysis**

OK

When asked OK to Save Model Now? Respond **Yes**.

Yes

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.

Continue

13. View the results of the analysis.

View the X Normal Stress Fringe Plot.

View/Select... <F5>

Contour Style: ● **Contour**

Deformed and Contour Data...

Output Vectors/Contour: **7020..Plate Top X Normal Stress**

OK

OK

From the Stress Scale, what is the maximum stress?

Maximum Stress = _______________

Compare this value to the theoretical value.

This concludes the exercise.

File/Save

File/Exit

Max Average Nodal Stress	431.3