WORKSHOP 1

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.

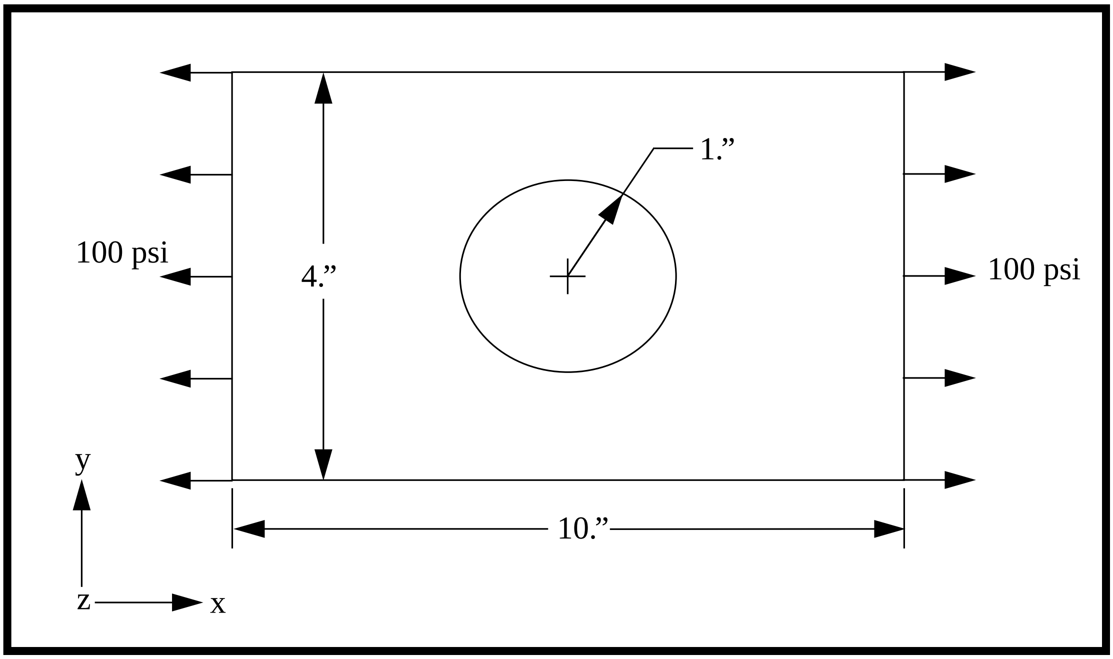
Model Description:

The following exercises are provided for the advanced user who would like to practice on more practical engineering problems. The descriptions are intentionally brief with minimal step-by-step description provided.

For the following problem, the user is expected to run the complete analysis and verify the results provided. This problem has a 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 1.1 - Load Conditions



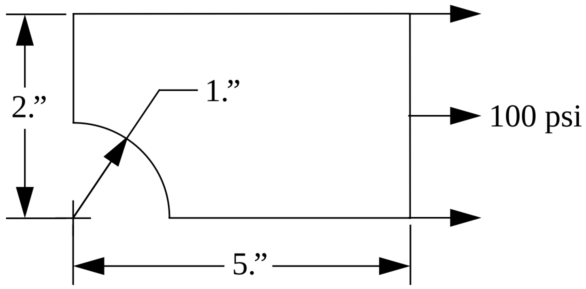
- Thickness =0.125 in.
- $E = 30,000,000$ psi
- $\nu = 0.3$

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

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

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

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



Exercise Procedure:

1. Start up MSC/NASTRAN for Windows V3.0 and begin to create a new model.

Double click on the icon labeled **MSC/NASTRAN for Windows V3.0**

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**
Young's 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 NASTRAN geometry for the inner circle.

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

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

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

5. Create the NASTRAN 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

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

First, turn on the curve labels.

View/Options...

Options: **Curve**

Label Mode: **1..ID**

OK

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

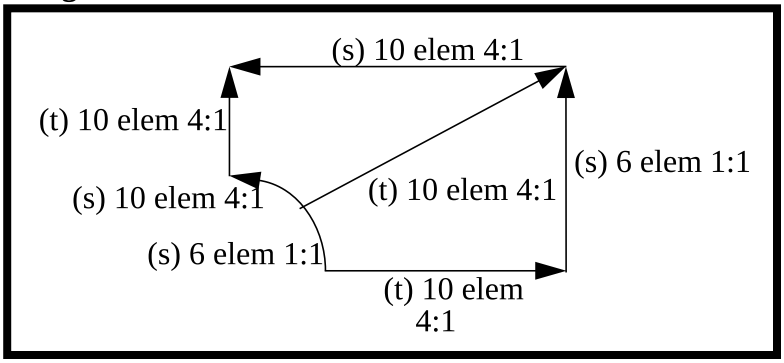
Geometry/Surface/Ruled...

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

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 1.2. To verify the direction of your model, use the **List** options.

Figure 1.2



Using the **Mesh/Mesh Control/Mapped Divisions on Surface** command, create the appropriate mesh seeds. The seeding varies depending on the *Number of Elements*, listed below, and the density of the seeds set with the *Bias* inputs. Both values constitute the number of mesh seeds in either the *s* or *t* parametric directions labeled in Figure 1.2, above.

Mesh/Mesh Control/Mapped Divisions on Surface...

<Select the upper surface - ID # 1 > **OK**

	s	**t**
Number of Elements:	**10**	**10**

Bias	**0.25**	**4**	**OK**

Notice that for Curve 2 and 4 (“s” 10 element 4:1), we want the mesh seeds to be more compact at the left end of the curve. Since curve 4 is in the reverse direction, the bias ratio of 4 is inverted to get 0.25.

<Select the upper surface - ID # 2 > **OK**

	s	**t**	
Number of Elements:	**6**	**10**	
Bias:	**1**	**4**	**OK**

Cancel

8. Generate the finite elements.

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

Mesh/Geometry/Surface...

<select both surfaces>

OK

Property: **1..plate**

OK

Note that some of the quad elements may not look uniform.

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

As the message window states, no nodes have been merged because the coincident nodes have been auto-merged.

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

View/Options...

Quick Options...

☐ **Node**

Labels Off

Done

Options:	**Load-Force**
Label Mode:	**1..Load Value**
Options:	**Constraint**
Label Mode:	**1..Degree of Freedom**

OK

10. Create the model constraints.

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

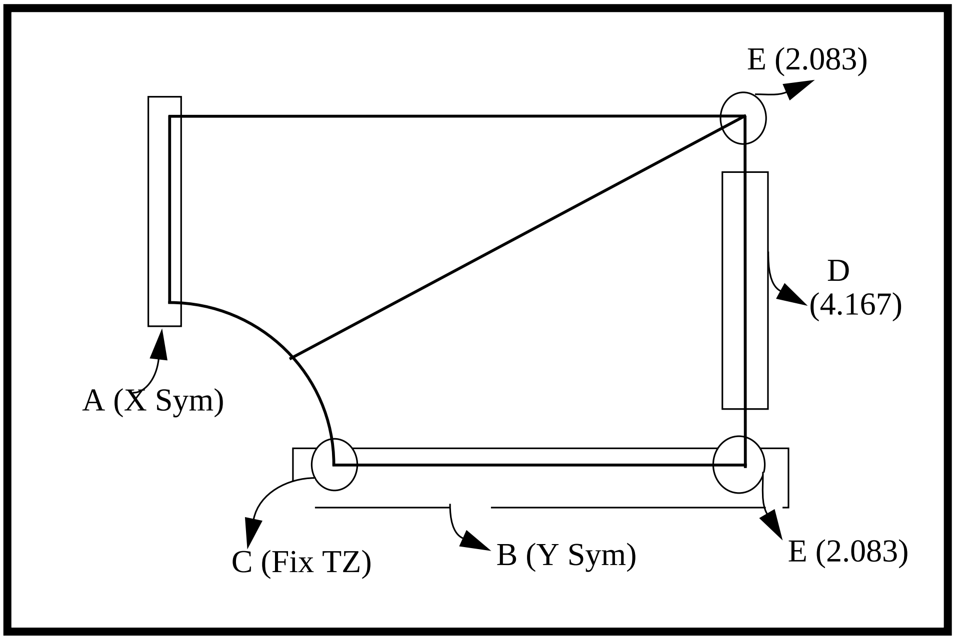
Model/Constraint/Set...

Title: **constraint**

OK

This constraint set will have 3 different constraints.

Figure 1.3



Now define the first relevant constraint for the model. When selecting the appropriate boxes shown above, use the **Shift+Mouse Click** function.

Model/Constraint/Nodal

<select all nodes on the left edge. Box "A" in Figure 1.3>

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.

<select all nodes on the bottom edge. Box "B" in Figure 1.3>

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 the node on the bottom left corner. Circle "C" in Figure1.3>

OK

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

☒ **TZ**

OK

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

NO

Cancel

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 .25 in^2 (2 in x 0.125 in) can be converted to a total equivalent nodal force of 25 lbs. After distributing the 25 lbs over the 7 nodes that are on the right edge, 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/Nodal...

<select the inner 5 nodes on the right edge. Box "D" in Figure 1.3>

OK

Highlight **Force**.

Force

FX ☒ **4.167**

OK

<select the 2 corner nodes on the right edge. Circles "E" in Figure 1.3>

OK

Highlight **Force**.

Force

FX ☒ **2.083**

OK

Cancel

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

Yes

File Name: **tension**

Save

When the MSC/NASTRAN manager is through running, MSC/NASTRAN 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 smoothly, we will not bother with the details this time.

Continue

13. View the results of the analysis.

To view the VonMises Stress Fringe Plot, select the following:

View/Options...

Quick Options...

☐ **Load - Force**
☐ **Constraint**

Done

OK

View/Select...

Contour Style: ● **Contour**

Deform and Contour Data

Output Vectors/Contour: **7033..Plate Top VonMises Stress**

Contour Vector

☒ **2D Component** — **7033..Plate Top VonMises Stress**

☒ **CSys** — **0..Basic Rectangular**

OK

OK

OK

From the Stress Scale, what is the maximum stress?

Maximum Stress = _______________

Compare this value to the theoretical value.

This concludes the exercise.

Maximum Stress:	**~ 398.5 psi**

Analysis of a Tension Coupon