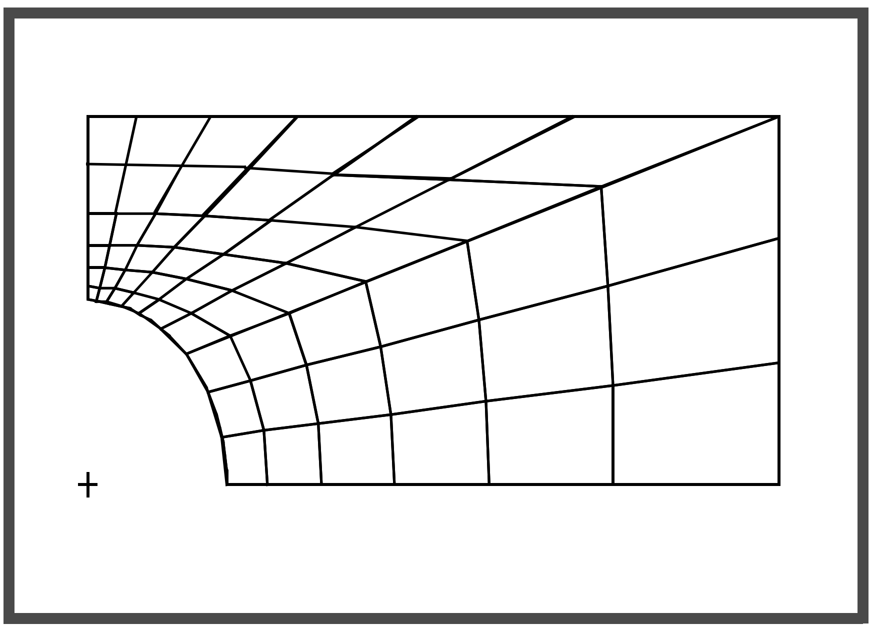
WORKSHOP 9

The Essence of Result Post-Processing

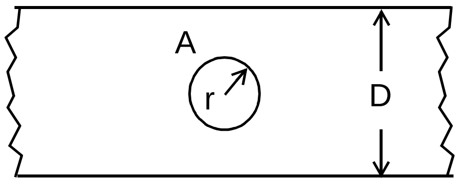


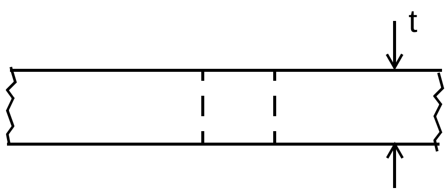
Objectives:

- Manually create the geometry for the tension coupon using the given dimensions then apply finite elements.
- Manually define material and element properties.
- Apply symmetric boundary constraints.
- Apply edge pressure to model.
- Run the analysis and read the results back into PATRAN.
- Generate a fringe plot of X stress tensor and compare results.

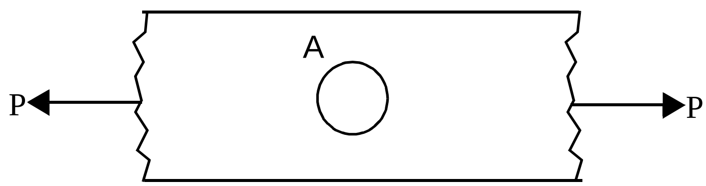
Theoretical Solution:

Central circular hole in a member of rectangular cross section





Elastic stress, axial tension



$$\sigma_{max} = \sigma_a = k\sigma_{\text{nom}}$$

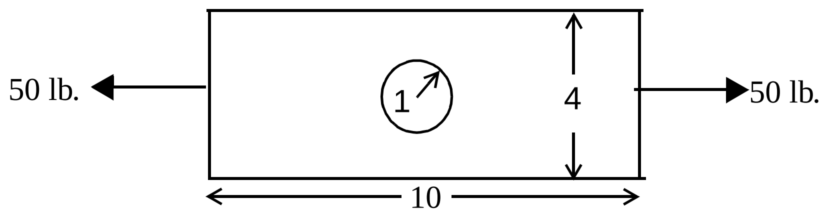
$$\text{where } \sigma_{\text{nom}} = \frac{P}{t(D-2r)}$$

$$k = 3.00 - 3.13\left(\frac{2r}{D}\right) + 3.66\left(\frac{2r}{D}\right)^2 - 1.53\left(\frac{2r}{D}\right)^3$$

Reference: Roark, Raymond and Young, Warren. Formulas for Stress and Strain. McGraw-Hill, Inc. 1982

Sample Calculation:

Dimensions are in inches. Figure is not drawn to scale.



Material: $E = 30\times10^6$

$\upsilon = 0.3$

Thickness: t = 0.125 in

Applied Load: P = 50 lb.

$$\sigma_{max} = \sigma_a = k\sigma_{\text{nom}}$$

$$\text{where } \sigma_{\text{nom}} = \frac{P}{t(D-2r)}$$

$$\sigma_{\text{nom}} = \frac{50}{0.125(4-2(1))} = 200$$

$$k = 3.00 - 3.13\left(\frac{2r}{D}\right) + 3.66\left(\frac{2r}{D}\right)^2 - 1.53\left(\frac{2r}{D}\right)^3$$

$$k = 3.00 - 3.13\left(\frac{2(1)}{4}\right) + 3.66\left(\frac{2(1)}{4}\right)^2 - 1.53\left(\frac{2(1)}{4}\right)^3$$

$$k = 2.159$$

$$\sigma_{max} = 2.159(200) = 431.8\text{psi}$$

Model Description:

Below is a schematic of a plate with a circular hole at the center (Fig. 9.1). The dimensions and loads are indicated. The finite element model will be simplified by creating the 1/4 tension coupon model with two simple surfaces as shown in Fig. 9.2. The plate is 0.125 inches thick; therefore it will be modeled with shell elements. The tension coupon will have the properties of Table 9.1. The resulting finite element model will have x- and y-symmetry constraints with an additional constraint for translation in the z-direction. The uniform edge pressure load of 100 psi will be applied to the right edge. Since the Patran edge pressure convention is in terms of lb/in, 100 psi must be converted to a running load by multiplying the pressure by the thickness of the plate (use the equation 100.*0.125).

Figure 9.1 - Plate with Circular Hole

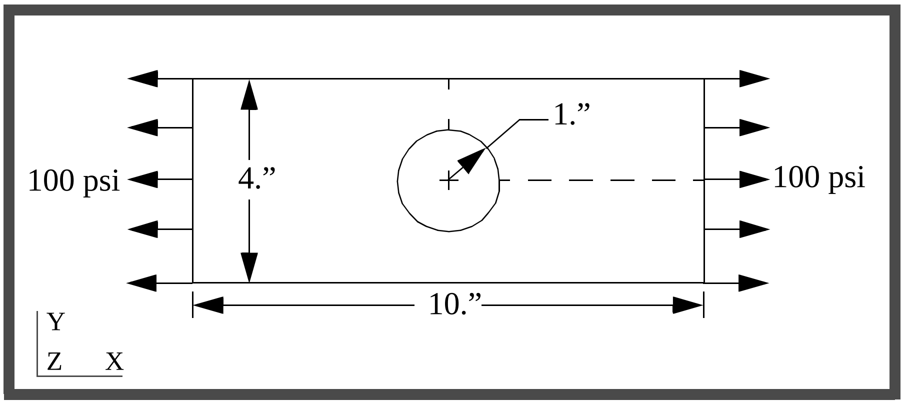
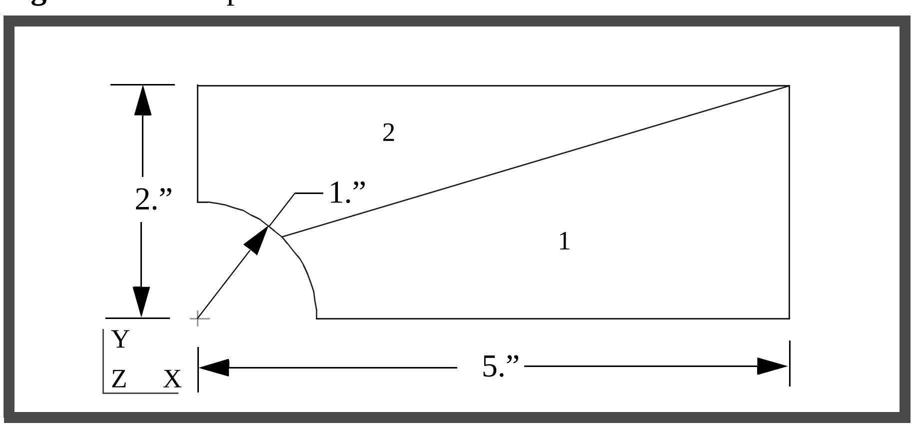


Table 9.1 - Tension Coupon Properties

Elastic Modulus:	**10E+06 psi**
Poisson Ratio	**0.3**
Plate Thickness:	**0.125 in**

Figure 9.2 - Simplified Model with Dimensions



Suggested Exercise Steps:

- Generate the necessary curves and surfaces of the tension coupon geometry in addition to the resulting finite elements using the dimensions and element connectivies.
- Define the symmetric boundary constraints (X-SYMMETRY, Y-SYMMETRY, RIGID_BODY) and apply the uniform pressure to one edge (TENSION_LOAD).
- Define material (MAT_1) and element properties (PLATE).
- Prepare the model for a Linear Static analysis.
- Generate and submit input file for MSC.Nastran.
- Generate a fringe plot of σ_{XX}.
- Review the results.

Exercise Procedure:

1. Open a new database called **prob9.db**.

◆ **File/New...**

New Database Name	**prob9**
OK	

In the New Model Preference form set the following:

Tolerance	◆ **Default**
Analysis code:	**MSC/NASTRAN**
OK	

In the next few steps you will create the necessary geometry for the plate model.

Whenever possible click ❑ **Auto Execute** (turn off).

2. First draw the arc of the model.

Create the bottom half.

◆ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**2D ArcAngles**
Radius	**1.0**
Start Angle	**0.0**
End Angle	**45.0**
Center Point List:	**[0,0,0]**
Apply	

Now the top half.

Radius	**1.0**
Start Angle	**45.0**

End Angle	**90.0**

Apply

3. Referring to the model dimensions (see Fig. 9.2), the outer edges of the geometry are created through the following steps:

First the top right.

◆ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**
Vector Coordinates List	**< 0, 2, 0 >**
Origin Coordinates List	**[5, 0, 0]**

Apply

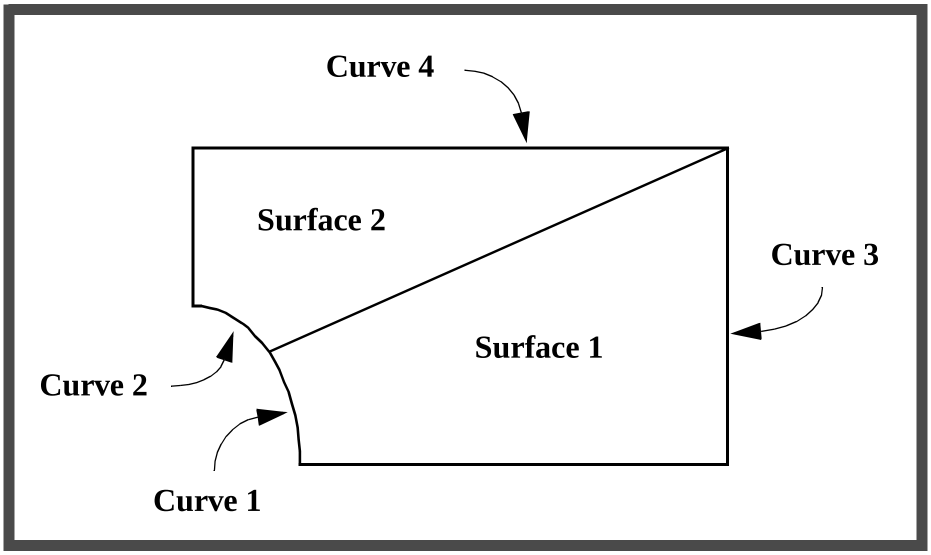
Now the top edge.

Vector Coordinates List	**< -5, 0, 0 >**
Origin Coordinates List	**[5, 2, 0]**

Apply

4. Now create two surfaces out of the curve model just drawn.

Figure 9.3: Surface and Curve Locations



◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**Curve**
Starting Curve List	**Curve 1**
Ending Curve List	**Curve 3**

Apply

Complete the surface model.

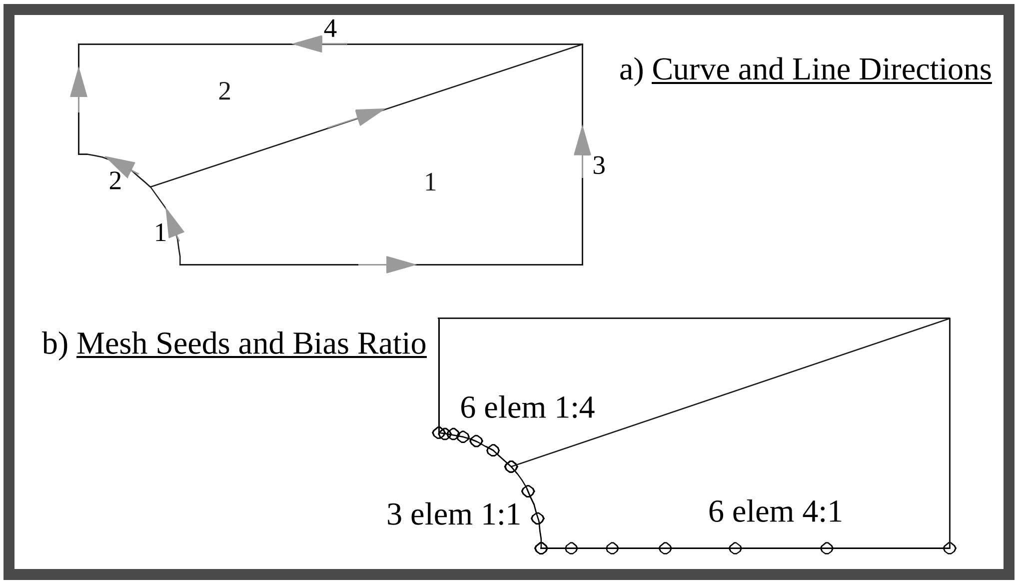
Starting Curve List	**Curve 2**
Ending Curve List	**Curve 4**

Apply

5. Create the Finite Element Model and mesh the surface.

It is essential to know the direction of each of the curves and lines. The direction will determine the bias ratio (L2/L1) of the mesh seeds. In steps 2 to 4, the geometry of this model was created so that the direction of the lines, curves and surfaces coincide with Fig. 9.4a. Simply following the next step will indicate the curves and lines directions.

Figure 9.4



First the elements with mesh seed ratios.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**One Way Bias**

◆ **Num Elems and L2/L1**

Number=	**6**
L2/L1=	**4**
Curve List	**Surface 1.1**

(Select the bottom edge)

Apply

Now change the bias ratio (L2/L1).

◆ **Num Elems and L2/L1**

Number=	**6**
L2/L1=	**0.25**

Curve List	**Curve 2** *(Select the top half of arc)*

Apply

Finally, the uniform mesh seed.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**

◆ **Number of Elements**

Number=	**3**
Curve List	**Curve 1** *(Select the bottom half of arc)*

Apply

Mesh the surface.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**

◆ **Isomesh**

Surface List	**Surface 1, 2**

Apply

Before continuing, Equivalence the entire model to delete any duplicate nodes created when meshing.

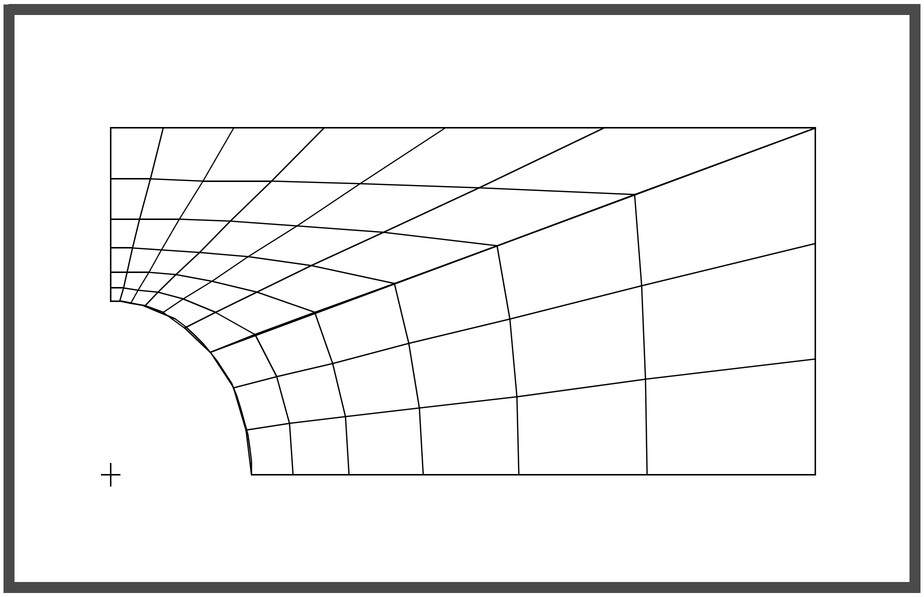
◆ **Finite Elements**

Action:	**Equivalence**
Object:	**All**
Type:	**Tolerance Cube**

Apply

Your model should appear as below.

Figure 9.5 - Completed Meshed Model



6. Now you will create the material properties for the plate.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**mat_1**

Input Properties...

Elastic Modulus=	**10E6**
Poisson Ratio=	**0.3**

OK

Apply

7. Give the plate a thickness using Properties.

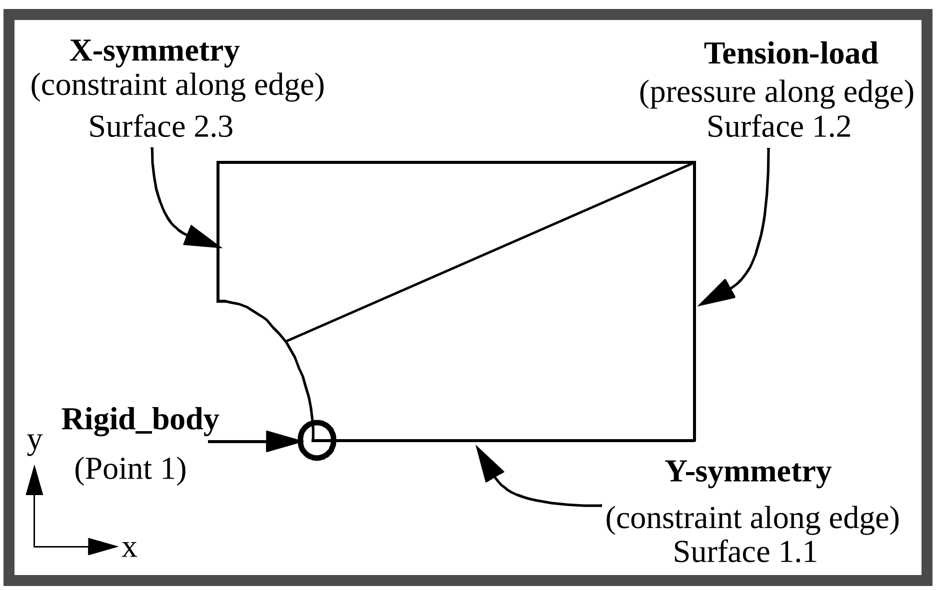
◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name	**plate**
Input Properties...	
Material Name	**m:mat_1**
Thickness	**0.125**
OK	
Select Members	**Surface 1, 2**
Add	
Apply	

8. Next you will apply the boundary conditions to the model using Loads/BCs.

This model will have three different contraints.

Figure 9.6 - Boundary Condition Placement



First, create the x-symmetry contraints placed on the left side of the model. Selecting geometry may be done by typing them in or clicking them as shown in Fig. 9.6.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name **x_symmetry**

Input Data...

Translations <T1 T2 T3> **< 0, , >**

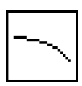
Rotations <R1 R2 R3> **< , 0, 0 >**

OK

Select Application Region...

Geometry Filter ◆ **Geometry**

The surface edge may be typed into the input box or by clicking on the edge itself. This is done by first clicking on the **Curve or Edge** icon from the Select Menu.



Curve or Edge

Select Geometry Entities **Surface 2.3**

(see Fig. 9.6)

Add

OK

Apply

Next, create the y-symmetry constraints.

New Set Name **y_symmetry**

Input Data...

Translations <T1 T2 T3> **< , 0, >**

Rotations <R1 R2 R3> **< 0, , 0 >**

OK

Select Application Region...

Geometry Filter ◆ **Geometry**

Select Geometry Entities **Surface 1.1**

(see Fig. 9.6)

Add

OK

Apply

Finally, assign one extra constraint set to remove rigid body motion in z-direction.

New Set Name **rigid_body**

Input Data...

Translations <T1 T2 T3> < , , 0 >

Rotations <R1 R2 R3> < , , >

OK

Select Application Region...

Geometry Filter ◆ **Geometry**

The point may be typed into the input box or by clicking on the edge itself. This is done by first clicking on the **Point or Vertex** icon from the Select Menu.

Point or Vertex

Select Geometry Entities **Point 1**

(see Fig. 9.6)

Add

OK

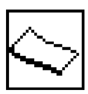
Apply

Now the pressure load may be added to the model.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Pressure**
Type:	**Element Uniform**
New Set Name	**tension_load**
Target Element Type:	**2D**
Input Data...	
Edge Pressure	**-12.5**
OK	
Select Application Region...	
Geometry Filter	◆ **Geometry**

You can either type in the selected edge in the input box or click on the right edge of the model. This is done by first clicking on the **Edge** icon from the Select Menu..

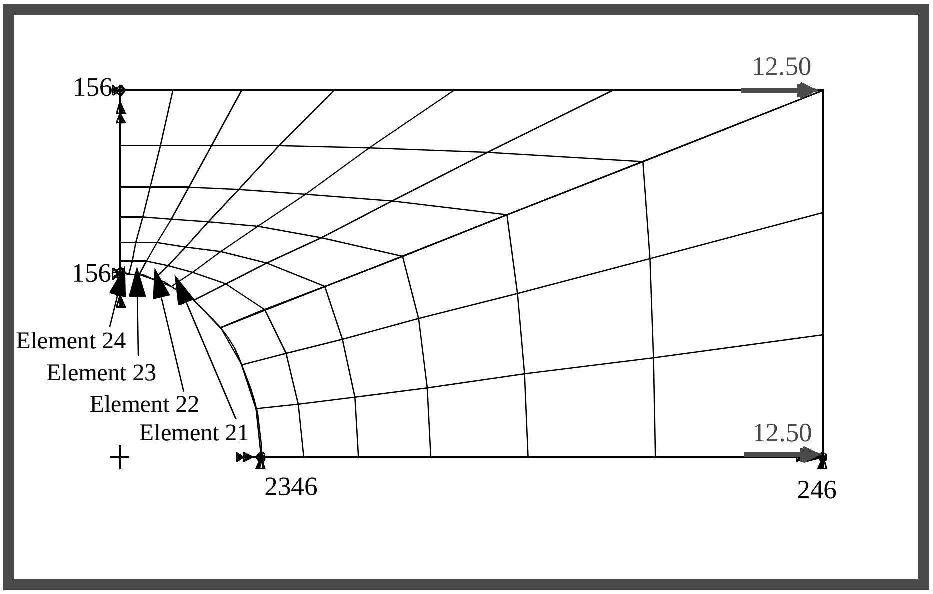

Edge

Select 2D Elements or Edge	**Surface 1.2** *(see Fig. 9.6)*
Add	
OK	
Apply	

NOTE: Positive pressure is always going into the surface/edge according to Patran convention. Therefore, the tensile load must be applied as -12.5.

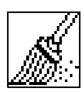
Your finished model should look like Fig. 9.7.

Figure 9.7: Finished Model and Critical Elements



NOTE: The individual locations of critical elements as shown in Fig. 9.7 may vary from model to model due to different methods in meshing.

Click on the **Reset Graphics** icon to clean up the display.


Reset Graphics

9. Now we are ready to submit the file for analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name	**prob9**

Solution Type...

Solution Type:	◆ **LINEAR STATIC**

OK

Apply

An MSC.Nastran input file called **prob9.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green.

Submitting the Input File for Analysis:

10. Submit the input file to MSC.Nastran for analysis.

    10a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob9.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

11. When the run is completed, edit the **prob9.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

Comparison of Results:

12. Compare the results obtained in the **.f06** file with the results on the following pages.

Table 9.2 - Stress Values of Critical Elements in Tension Coupon

S T R E S S E S   I N   Q U A D R I L A T E R A L   E L E M E N T S   (Q U A D 4)   OPTION = BILIN

ELEMENT ID	GRID-ID	FIBRE DISTANCE	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
21	CEN/4	-6.250000E-02	2.527787E+02	3.541285E+01	-7.969472E+01	-18.1259	2.788668E+02	9.324759E+00	2.743233E+02
		6.250000E-02	2.527787E+02	3.541285E+01	-7.969472E+01	-18.1259	2.788668E+02	9.324759E+00	2.743233E+02
	31	-6.250000E-02	3.329183E+02	4.498705E+01	-7.969472E+01	-14.4837	3.535047E+02	2.440072E+01	3.419579E+02
		6.250000E-02	3.329183E+02	4.498705E+01	-7.969472E+01	-14.4837	3.535047E+02	2.440072E+01	3.419579E+02
	32	-6.250000E-02	2.960301E+02	5.534706E+01	-7.969472E+01	-16.7570	3.200261E+02	3.135102E+01	3.055592E+02
		6.250000E-02	2.960301E+02	5.534706E+01	-7.969472E+01	-16.7570	3.200261E+02	3.135102E+01	3.055592E+02
	39	-6.250000E-02	1.717414E+02	2.410294E+01	-7.969472E+01	-23.5959	2.065524E+02	-1.070805E+01	2.121092E+02
		6.250000E-02	1.717414E+02	2.410294E+01	-7.969472E+01	-23.5959	2.065524E+02	-1.070805E+01	2.121092E+02
	38	-6.250000E-02	2.174817E+02	1.974807E+01	-7.969472E+01	-19.4358	2.456027E+02	-8.372890E+00	2.498943E+02
		6.250000E-02	2.174817E+02	1.974807E+01	-7.969472E+01	-19.4358	2.456027E+02	-8.372890E+00	2.498943E+02
22	CEN/4	-6.250000E-02	3.155676E+02	3.340709E+01	-7.550673E+01	-14.0779	3.345027E+02	1.447201E+01	3.275066E+02
		6.250000E-02	3.155676E+02	3.340709E+01	-7.550673E+01	-14.0779	3.345027E+02	1.447201E+01	3.275066E+02
	32	-6.250000E-02	3.959351E+02	5.548793E+01	-7.550673E+01	-11.9604	4.119300E+02	3.949300E+01	3.936720E+02
		6.250000E-02	3.959351E+02	5.548793E+01	-7.550673E+01	-11.9604	4.119300E+02	3.949300E+01	3.936720E+02
	33	-6.250000E-02	3.589660E+02	4.756311E+01	-7.550673E+01	-12.9354	3.763085E+02	3.022061E+01	3.621452E+02
		6.250000E-02	3.589660E+02	4.756311E+01	-7.550673E+01	-12.9354	3.763085E+02	3.022061E+01	3.621452E+02
	40	-6.250000E-02	2.404000E+02	1.252301E+01	-7.550673E+01	-16.7662	2.631482E+02	-1.022517E+01	2.684069E+02
		6.250000E-02	2.404000E+02	1.252301E+01	-7.550673E+01	-16.7662	2.631482E+02	-1.022517E+01	2.684069E+02
	39	-6.250000E-02	2.771927E+02	2.111808E+01	-7.550673E+01	-15.2644	2.977986E+02	5.121256E-01	2.975429E+02
		6.250000E-02	2.771927E+02	2.111808E+01	-7.550673E+01	-15.2644	2.977986E+02	5.121256E-01	2.975429E+02
23	CEN/4	-6.250000E-02	3.562378E+02	2.631285E+01	-5.701716E+01	-9.5335	3.658135E+02	1.673714E+01	3.577387E+02
		6.250000E-02	3.562378E+02	2.631285E+01	-5.701716E+01	-9.5335	3.658135E+02	1.673714E+01	3.577387E+02
	33	-6.250000E-02	4.212818E+02	4.596736E+01	-5.701716E+01	-8.4504	4.297526E+02	3.749659E+01	4.122851E+02
		6.250000E-02	4.212818E+02	4.596736E+01	-5.701716E+01	-8.4504	4.297526E+02	3.749659E+01	4.122851E+02
	34	-6.250000E-02	3.992823E+02	3.912743E+01	-5.701716E+01	-8.7846	4.080933E+02	3.031643E+01	3.938112E+02
		6.250000E-02	3.992823E+02	3.912743E+01	-5.701716E+01	-8.7846	4.080933E+02	3.031643E+01	3.938112E+02
	41	-6.250000E-02	2.978658E+02	8.693192E+00	-5.701716E+01	-10.7608	3.087020E+02	-2.143013E+00	3.097790E+02
		6.250000E-02	2.978658E+02	8.693192E+00	-5.701716E+01	-10.7608	3.087020E+02	-2.143013E+00	3.097790E+02
	40	-6.250000E-02	3.171970E+02	1.466790E+01	-5.701716E+01	-10.3266	3.275861E+02	4.278739E+00	3.254679E+02
		6.250000E-02	3.171970E+02	1.466790E+01	-5.701716E+01	-10.3266	3.275861E+02	4.278739E+00	3.254679E+02
24	CEN/4	-6.250000E-02	3.759466E+02	1.756750E+01	-2.366564E+01	-3.7618	3.775026E+02	1.601149E+01	3.697570E+02
		6.250000E-02	3.759466E+02	1.756750E+01	-2.366564E+01	-3.7618	3.775026E+02	1.601149E+01	3.697570E+02
	34	-6.250000E-02	4.274517E+02	3.222290E+01	-2.366564E+01	-3.4145	4.288637E+02	3.081088E+01	4.143184E+02

Column groups: FIBRE DISTANCE; STRESSES IN ELEMENT COORD SYSTEM (NORMAL-X, NORMAL-Y, SHEAR-XY); PRINCIPAL STRESSES (ZERO SHEAR) (ANGLE, MAJOR, MINOR, VON MISES)

ELEMENT ID	GRID-ID	FIBRE DISTANCE	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
		6.250000E-02	4.274517E+02	3.222290E+01	-2.366564E+01	-3.4145	4.288637E+02	3.081088E+01	4.143184E+02

35	-6.250000E-02	4.213328E+02	3.189201E+01	-2.366564E+01	-3.4648	4.227657E+02	3.045916E+01	4.083889E+02
	6.250000E-02	4.213328E+02	3.189201E+01	-2.366564E+01	-3.4648	4.227657E+02	3.045916E+01	4.083889E+02
42	-6.250000E-02	3.300369E+02	4.373753E+00	-2.366564E+01	-4.1347	3.317477E+02	2.662980E+00	3.304243E+02
	6.250000E-02	3.300369E+02	4.373753E+00	-2.366564E+01	-4.1347	3.317477E+02	2.662980E+00	3.304243E+02
41	-6.250000E-02	3.351263E+02	4.824241E+00	-2.366564E+01	-4.0774	3.368133E+02	3.137250E+00	3.352557E+02
	6.250000E-02	3.351263E+02	4.824241E+00	-2.366564E+01	-4.0774	3.368133E+02	3.137250E+00	3.352557E+02

13. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

14. Proceed with the Reverse Translation process, that is attaching the **prob9.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

Action: **Attach XDB**

Object: **Result Entities**

Method: **Local**

Select Results File...

Select Results File **prob9.xdb**

OK

Apply

When the translation is complete bring up the Results form.

15. We can now begin result post processing.

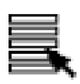
Now we will generate the fringe plot of the X stress tensor.

◆ **Results**

Action: **Create**

Object: **Fringe**

Now click on the **Select Results** icon.



Select Results

Select Result Case(s) **Default, A1:Static Subcase**

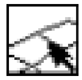
Select Fringe Result **Stress Tensor**

Position...(At Z1)

Close

Quantity: **X Component**

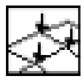
Next click on the **Target Entities** icon.

 **Target Entities**

Target Entity:	**Current Viewport**

NOTE: This feature allows you to view fringe plots of specific elements of your choice.

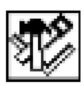
Click on the **Display Attributes** icon.

 **Display Attributes**

Style:	**Discrete/Smooth**
Display:	**Element Edges**
Title	**Tension Coupon, Stress Tensor, - X Component, At Z1**

NOTE: Display Attributes form allows you the ability to change the displayed graphics of fringe plots.

Now click on the **Plot Options** icon.

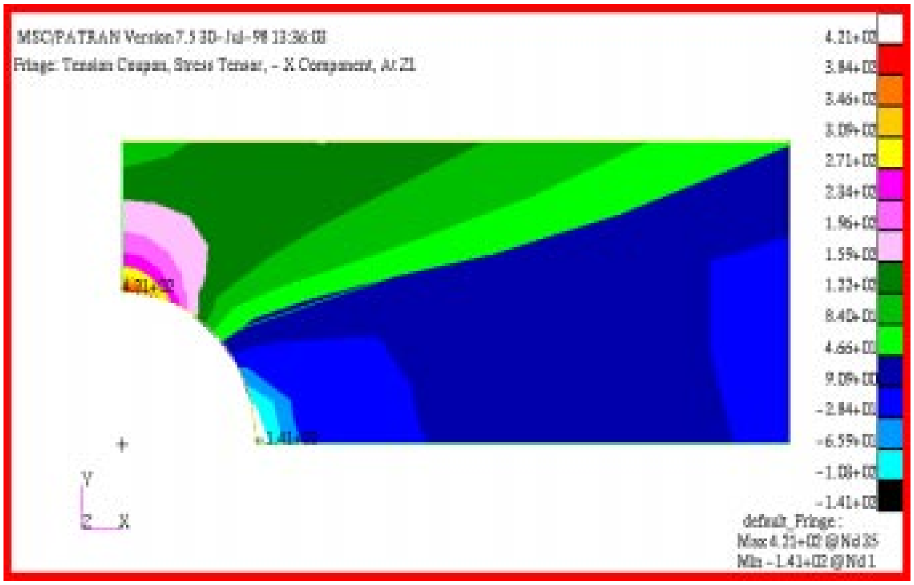
 **Plot Options**

Coordinate Transformation:	**None**
Filter Values:	**None**
Domain:	**All Entities**
Method:	**Derive/Average**
Extrapolation:	**Shape Fn.**

Apply

See Fig. 9.8 and review results.

Figure 9.8



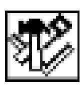
16. Let's examine the results.

Question 1: What coordinate is your X component in reference to? Is it referring to global, element, material,...?

Question 2: Does your plot match the Nastran results listed in Table 9.2?

17. Now create a different fringe plot and review the results.

Click on the **Plot Options** icon.

 **Plot Options**

First, let's define our X component in the global coordinate system.

Coordinate Transformation:	**Global**
Filter Values:	**None**

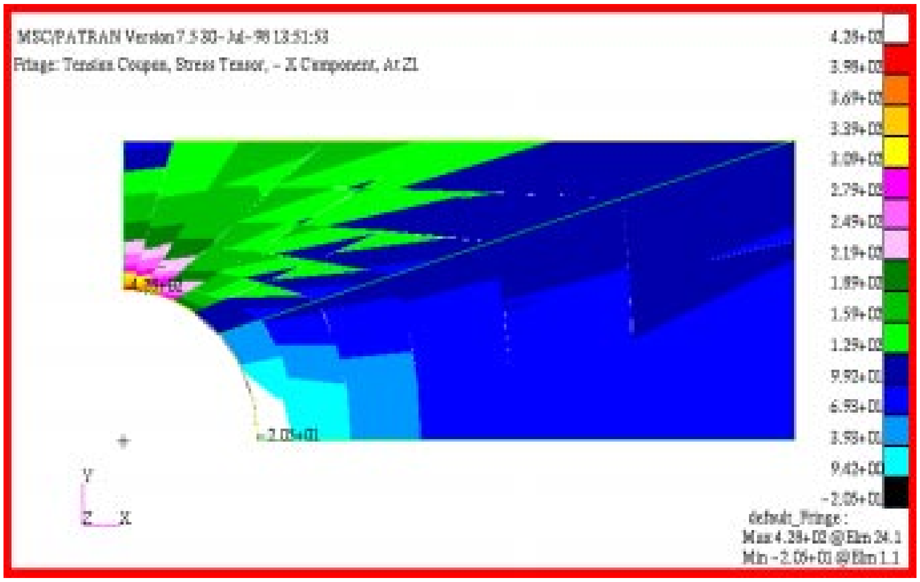
Turn off the averaging domain since Nastran results are reported at each element.

Domain:	**None**
Method:	**Derive/Average**
Extrapolation:	**Shape Fn.**

Apply

See Fig. 9.9 and review results.

Figure 9.9



As you can see, the new results are significantly different from our previous plot. However, this plot correctly illustrates the individual element stresses in global X-axis. At last, we finally have a plot that is technically correct.

Question: Does this plot match the results listed in Table 9.2? Why not?

18. Create another fringe plot and review the results.

Click on the **Plot Options** icon.

Plot Options

This time define the X component by changing the previous global coordinate system to none.

Coordinate Transformation:	**None**
Filter Values:	**None**

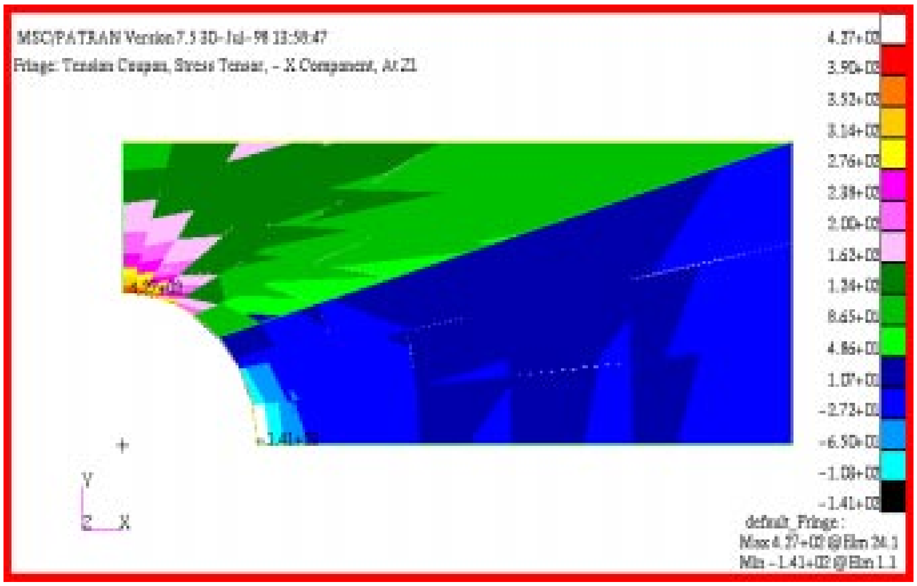
Turn off the averaging domain since Nastran results are reported at each element.

Domain:	**None**
Method:	**Derive/Average**
Extrapolation:	**Shape Fn.**

Apply

See Fig. 9.10 and review results.

Figure 9.10



Cross reference this plot with the our previous results table. Be certain that they are the same.

19. Create a new coordinate system.

◆ **Geometry**

Action:	**Create**
Object:	**Coord**
Method:	**3Point**
Origin	**[0, 0, 0]**
Point on Axis 3	**[0, 0, 1]**

Point on Plane 1-3	**[1, 1, 0]**

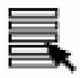
Apply

Generate the X stress plot again, but use CID option.

◆ **Results**

Action:	**Create**
Object:	**Fringe**

Click on the Select Results icon.



Select Results

Select Result Case(s)	**Default, A1:Static Subcase**
Select Fringe Result	**Stress Tensor**

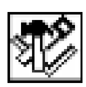
Position...(At Z1)

Close

Quantity:	**X Component**

The previous form selections are still valid and will not be changed, therefore, we will skip the Target Entities and Display Attributes icons and continue on to the Plot Options form.

Now click on the **Plot Options** icon.



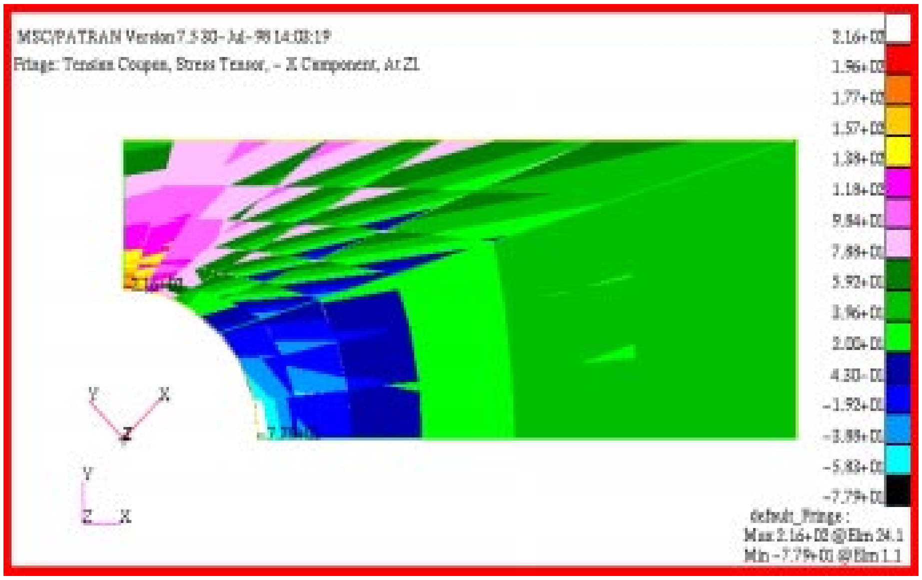
Plot Options

Coordinate Transformation:	**CID**
Select Coordinate Frame	**Coord 1**
Filter Values:	**None**
Domain:	**None**
Method:	**Derive/Average**
Extrapolation:	**Shape Fn.**

Apply

See Fig. 9.11 and review results.

Figure 9.11



Quit MSC.Patran when you are finished with this exercise.