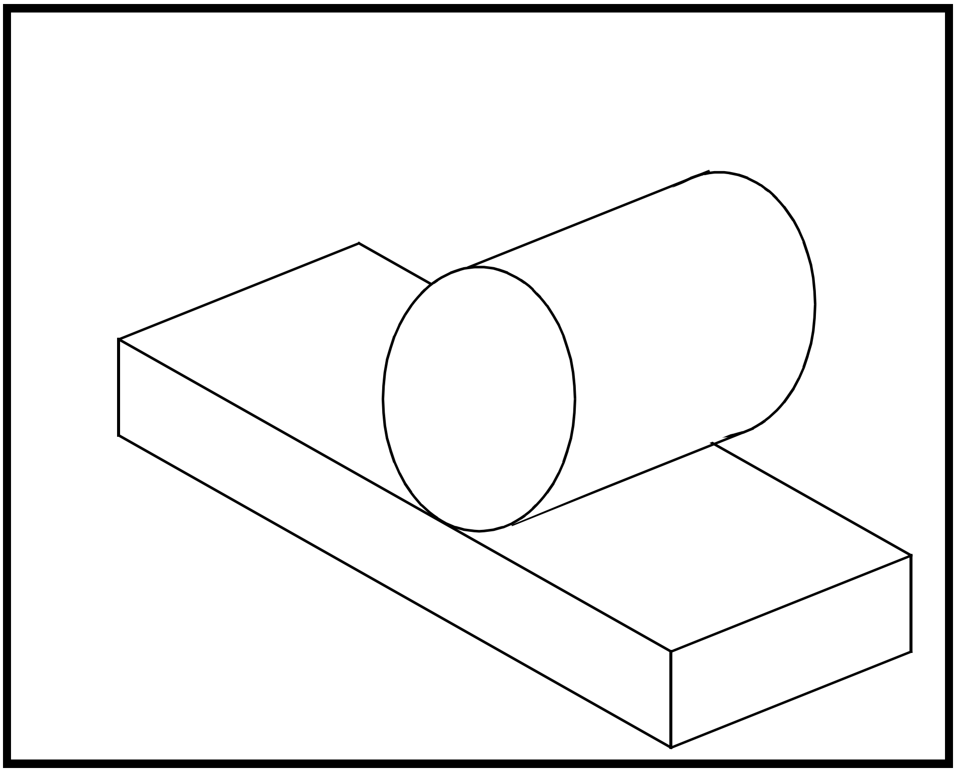
LESSON 9

Hertz Contact Problem



Objectives:

- Use of contact pair approach
- Multi-step analysis
- Post-processing of the results

Model Description:

In this example problem, a steel cylinder with a radius of 5” is pressed against a 2” deep aluminum base. The problem is linear except the contact condition at the base which is modeled using the contact pair approach. The problem is similar to the Hertz contact problem. (see Timoskenko and Goodier, 1951)

In this problem you will create the geometry. Then you will proceed to mesh, load and analyze the problem. The steel material properties have an Elastic Modulus of 30E6 and a Poisson’s ratio of 0.30 and the aluminum properties have an Elastic Modulus of 10E6 and a Poisson’s ratio of 0.33. Two steps will be used to model the contact and force application. The first step brings the cyclinder and base plate into contact. The second step applies a load of 100,000 lb/ inch depth to create the hertz contact condition.

Suggested Exercise Steps:

- Create a new database and name it **hertz.db**.
- Create the geometry for the cylinder and the base plate.
- Specify material and element properties.
- Specify boundary conditions for symmetry, base and contact confirmation. Also specify loadings for the cyclinder.
- Analyze the model and inspect the contact stresses.

Exercise Procedure:

1. Open a new database. Name it **hertz**.

File/New ...

Database Name: **hertz.db**

OK

Change the *New Analysis Preference* form to **MSC/ADVANCED_FEA**.

Analysis Code: **MSC/ADVANCED_FEA**

OK

2. Create the geometry for the cylinder and the base plate.

First create a group that contains the cylinder and name it **cylinder**

Group/Create...

New Group Name: **cylinder**

■ **Make Current**

Group Contents: **Add Entity Selection**

Apply

Cancel

◆ **Geometry**

Action: **Create**

Object: **Curve**

Method: **XYZ**

Vector Coordinates List: **<5, 0, 0>**

Origin Coordinates List: **[0, 0, 0]**

Apply

Action: **Create**

Object: **Surface**

Method: **Revolve**

■ **Patran 2 Convention**

Axis: **Coord 0.3**

Total Angle: **-180**

Surfaces per Curve: **2**

Curve List: **Curve 1**

Apply

Now create a group for the base plate.

Group/Create...

New Group Name: **plate**

■ **Make Current**

Group Contents: **Add Entity Selection**

Apply

Cancel

◆ **Geometry**

Action: **Create**

Object: **Surface**

Method: **XYZ**

Vector Coordinates List: **<16, -2, 0>**

Origin Coordinates List: **[-8, -5.1, 0]**

Apply

3. Post cylinder geometry.

Post only the geometry for the cylinder.

Group/Post...

Select Groups to Post: **cylinder**

Apply

Cancel

4. Create mesh seeds for the cylinder.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Method:	**One Way Bias**
Number:	**22**
L2/L1:	**-5**

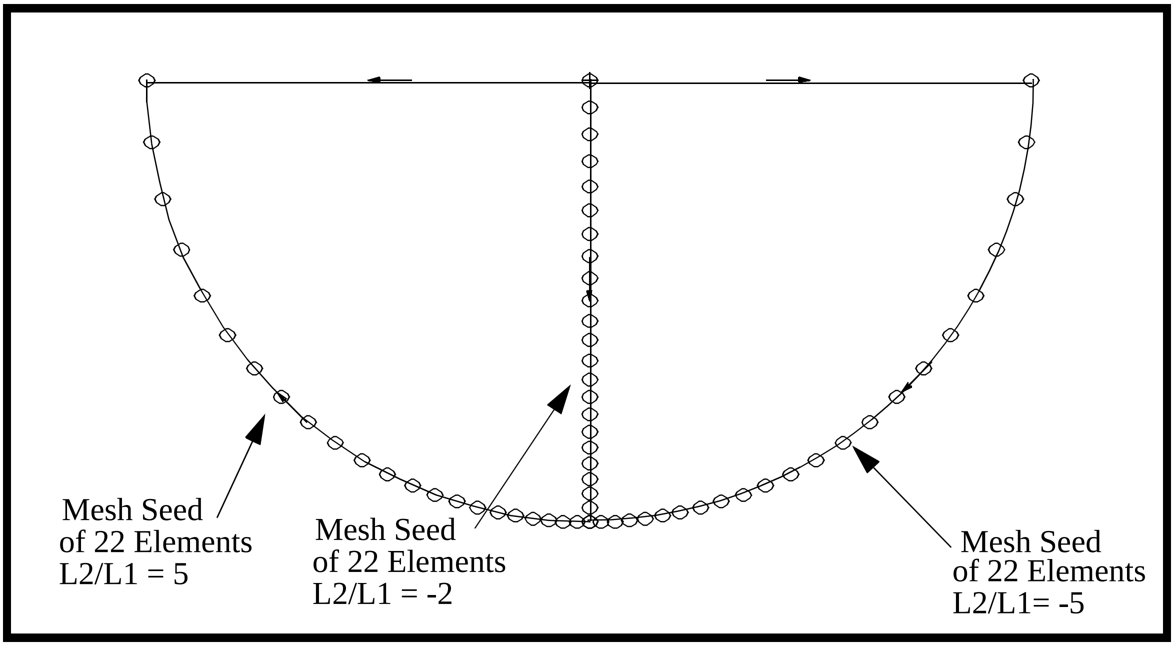
Select the curved edge on the bottom right of the cylinder

Curve List:	bottom right curve (see Figure 7.1)

Repeat the above steps with the bottom left curve setting **L2/L1 = 5** and Number = 22.

Repeat the above steps for the vertical centerline with **L2/L1 = -2** and Number = 22

Figure 7.1 - Mesh seed locations for cylinder



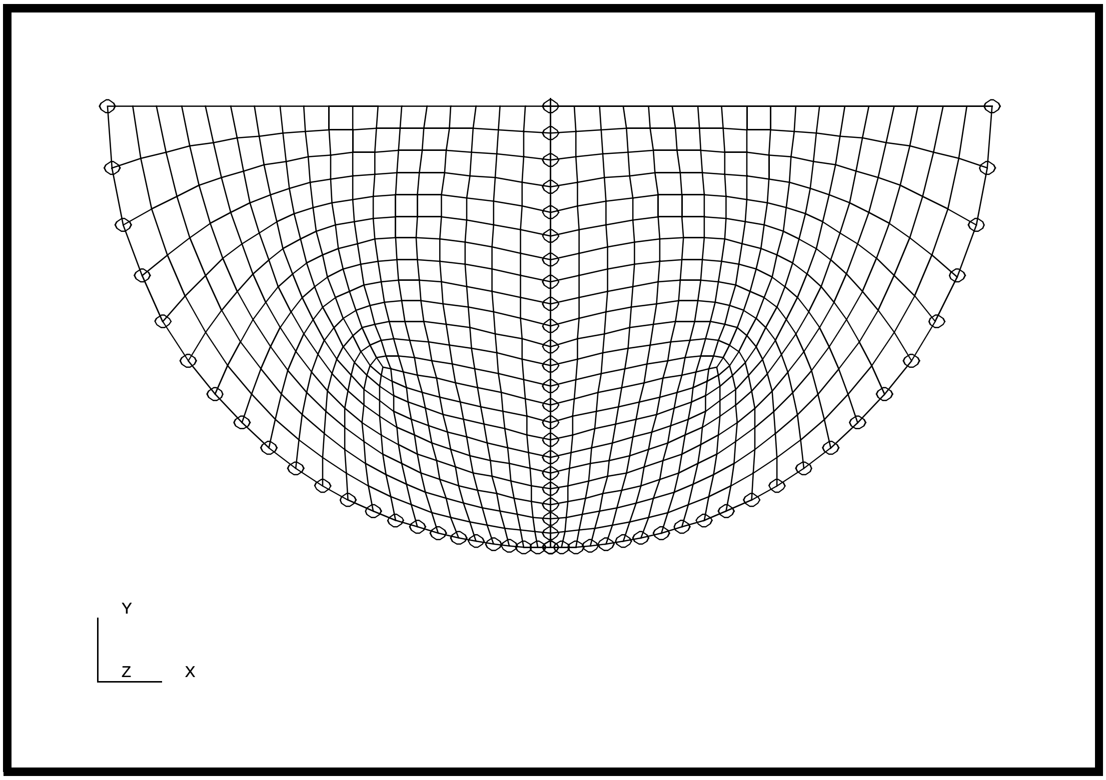
5. Mesh the surface with **Quad4** elements.

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Element Topology:	**Quad4**
Surface List:	all posted surfaces (Surface 1 & 2)

Apply

Your model should now look like the one shown in Figure 7.2:

Figure 7.2 - Meshed surface representing cylinder



6. Post only the base plate geometry.

Group/Post...

Select Groups to Post:	**plate**

Apply

Cancel

7. Create mesh seeds for the plate.

First seed the top edge of the plate surface.

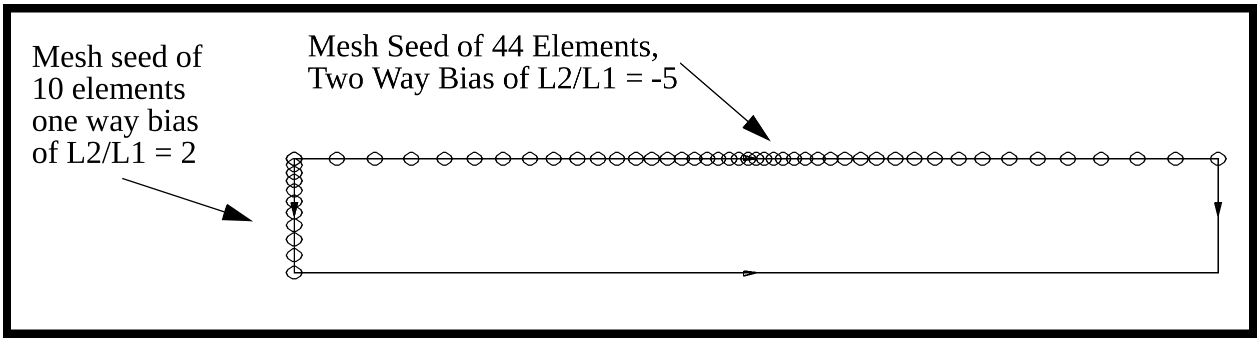
◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Method:	**Two Way Bias**
Number:	**44**
L2/L1:	**-5**
Curve List:	see Figure 7.3

Then seed the side of the plate.

Action:	**Create**
Object:	**Mesh Seed**
Method:	**One Way Bias**
Number:	**10**
L2/L1:	**2**
Curve List:	see Figure 7.3

Figure 7.3 - Mesh seed locations for plate



8. Mesh the plate surface with **Quad4** elements.

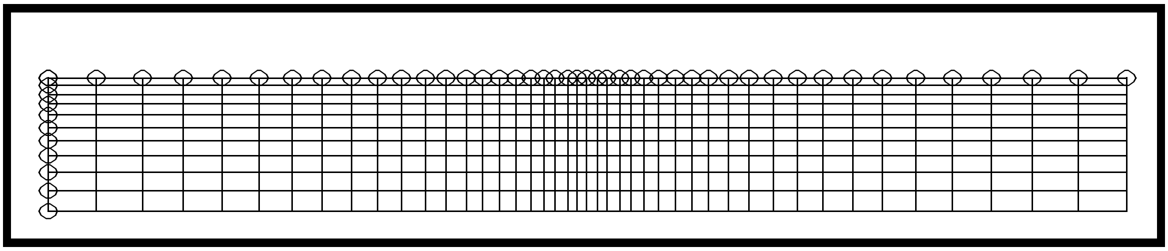
Action:	**Create**

Object:	**Mesh**
Type:	**Surface**
Element Topology:	**Quad4**
Surface List:	see Figure 7.3 (Surface 3)

Apply

Your model should now look like the one in Figure 7.4:

Figure 7.4 - Meshed surface representing plate



9. Create a new group **fem**.

This group will have all the finite elements for the entire model in it.

Group/Create...

New Group Name:	**fem**

■ **Make Current**

■ **Unpost All Other Groups**

Group Contents:	**Add All FEM**

Apply

Cancel

10. Equivalence the mesh.

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**
Equivalencing Tolerance:	**.005**

Apply

11. Verify element normals.

Since this is a 2-D plane strain model, the normals will need to be pointing in the positive **Z** direction. To assist you in reversing element normals, we will display the element normals right on the elements.

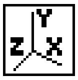
Action:	**Verify**
Object:	**Element**
Test:	**Normals**

◆ **Draw Normal Vectors**

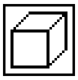
Apply

You will need to turn the model to see that the vectors are in the positive direction.

To quickly turn your model, select the following toolbar icon.

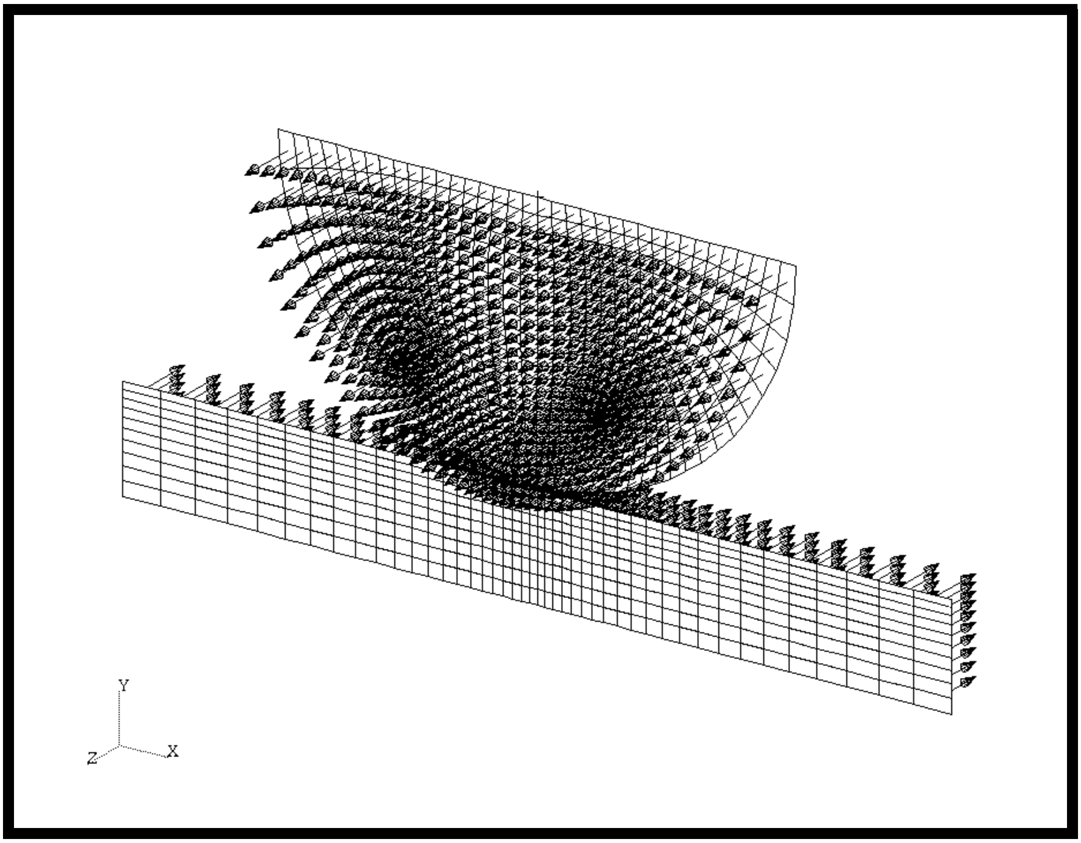

Iso 1 View

Also, it is helpful to change the model to **Hidden Line** render style, using the following toolbar icon.

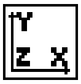

Hidden Line

Your model should look like the figure shown in Figure 7.5:

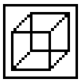
Figure 7.5 - Directions of elements’ normals



You will notice that the plate elements are pointing in the **negative Z** direction. You will have to reverse them. First return to the original view by selecting the front view toolbar icon and change the model back to wireframe by picking the wireframe icon:



Front View


Wire Frame

Now post only the elements on the plate.

Group/Post ...

Group: **plate**

Apply

Cancel

◆ **Finite Elements**

Action: **Modify**

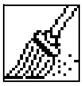
Object: **Element**

Test: **Reverse**

Element List: Select all of the plate elements

Apply

Reset the graphics display by selecting the Reset Graphics icon.

 **Reset Graphics**

12. Create the steel and aluminum materials. The Steel material will have a Young's 30E6, and Poisson's Ratio 0.30. The Aluminum material will have a Young's 10E6, and Poisson's Ratio 0.33

◆ **Materials**

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Material Name: **steel**

Input Properties...

Constitutive Model: **Elastic**

Elastic Modulus: **30E6**

Poisson's Ratio: **0.30**

Apply

Cancel

Repeat the above step for to create an aluminum material. Change the *Material Name* to be **aluminum**. Change the *Elastic Modulus* to be **10.E6** and the *Poisson's Ratio* to be **0.33**.

13. Create a new group named **all**.

This group will contain all the geometries and the finite elements for the entire model in it.

Group/Create...

New Group Name: **all**

■ **Make Current**

■ **Unpost All Other Groups**

Group Contents: **Add All Entities**

Apply

Cancel

14. Create the element properties **cylinder** and **plate**.

◆ **Properties**

Action: **Create**

Dimension: **2D**

Type: **2D Solid**

Property Set Name: **cylinder**

Options:

Plain Strain

Standard Formulation

Input Properties...

Material Name: **steel**

Thickness: **1.0**

OK

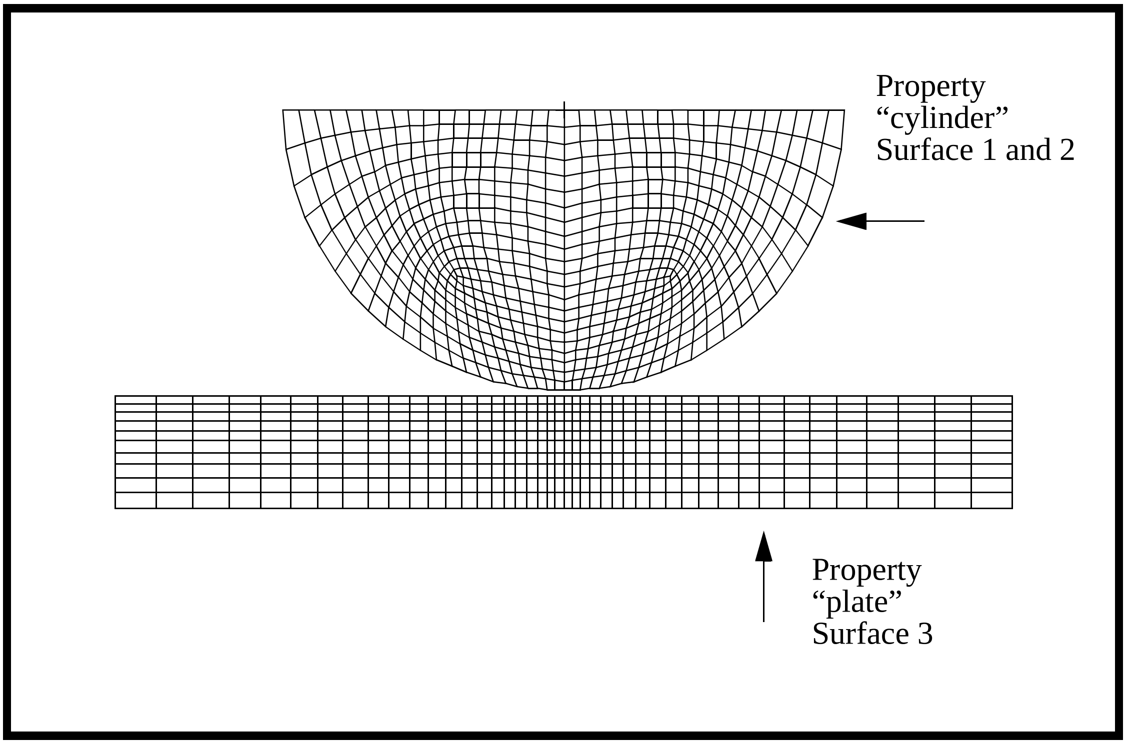
Select Members: see Figure 7.6

Add

Apply

Repeat the same process with a new *Property Set Name* of **plate**. The *Input Properties* are the same except the material will be **aluminum**. Enter **Surface 3** (shown in Figure 7.6) in the *Select Members* list box.

Figure 7.6 - Elements to be assigned properties



15. Fix the base in the X-direction.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**fixed_base_1**
Input Data...	
Translations <T1,T2,T3>:	**< 0, , >**
Rotations <R1,R2,R3>:	**< , , >**
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**

Select Nodes:	Select any node along the vertical middle of the base

Add

OK

Apply

16. Fix the base in the Y direction.

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**fixed_base_2**

Input Data...

Translations <T1,T2,T3>:	**< , 0, >**
Rotations <R1,R2,R3>:	**< , , >**

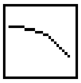
OK

Select Application Region...

Geometry Filter:	◆ **Geometry**
Select Geometric Entities:	see Figure 7.7

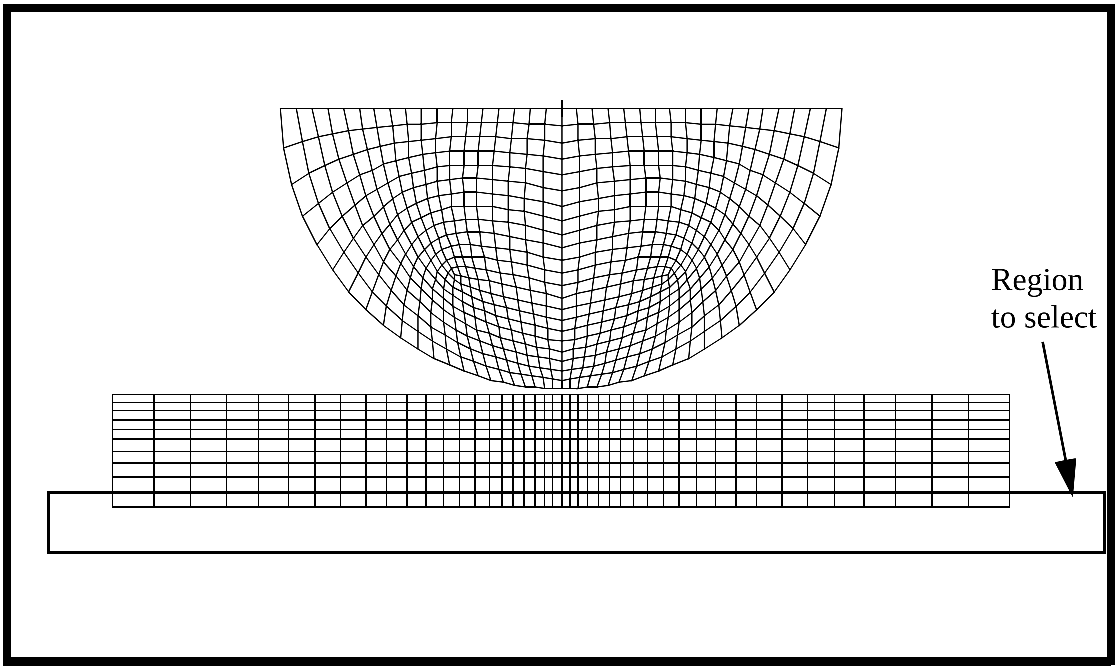
Click in the *Select Geometric Entities* databox and select the curve on the bottom edge of model as shown in Figure 7.7:

First, you will want to click on the curve icon:



Curve

Figure 7.7 - Fixed base of plate.



Add

OK

Apply

17. In this step, you are going to create a displacement set that will be used to close the gap between the cylinder and plate and insure contact. Apply a displacement of -0.105 in the Y direction along the top surface. Note that since the contact area between the cylinder and the base is small, it is assumed that the vertical displacements along the diameters of the steel ball is uniform.

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**close_gap**
Input Data...	
Translations <T1,T2,T3>:	**< 0, -0.105, >**
Rotations <R1,R2,R3>:	**< , , >**

OK

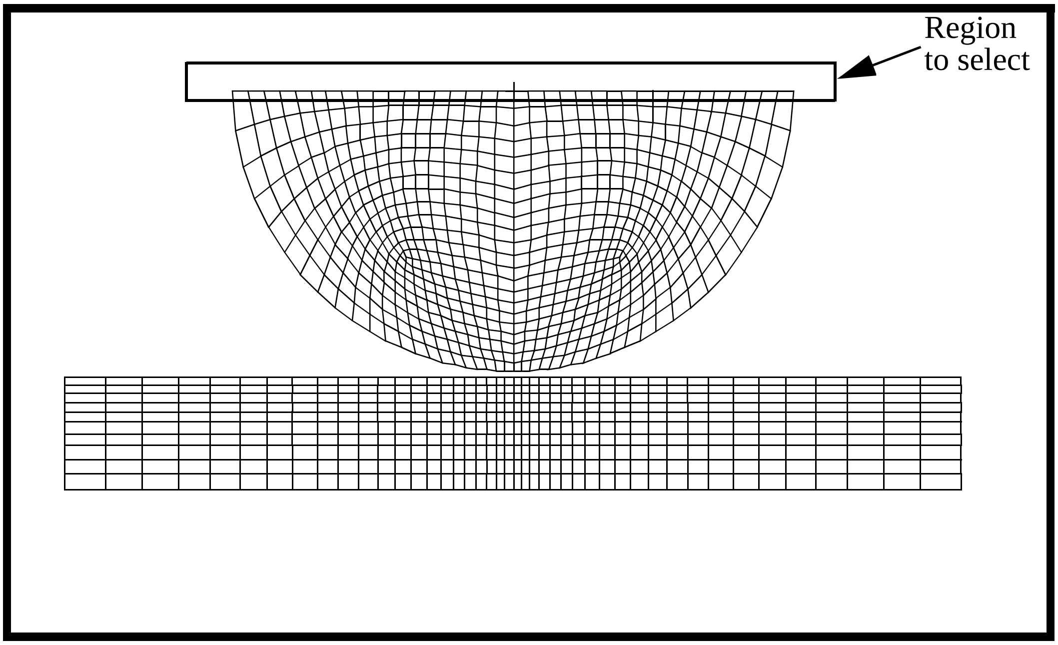
Select Application Region...

Geometry Filter: ◆ **FEM**

Select Nodes: see Figure 7.8

Click in the *Select Nodes* databox and select the nodes on the top edge of the model as shown in Figure 7.8:

Figure 7.8 - Selected region to apply close_gap to



Add

OK

Apply

18. Create contact conditions for the contact pair cylinder and base plate.

Action: **Create**

Object: **Contact**

Type:	**Element Uniform**
Option:	**Deform-Deform**
Analysis Type:	**Structural**
New Set Name:	**cylinder_plate_contact**

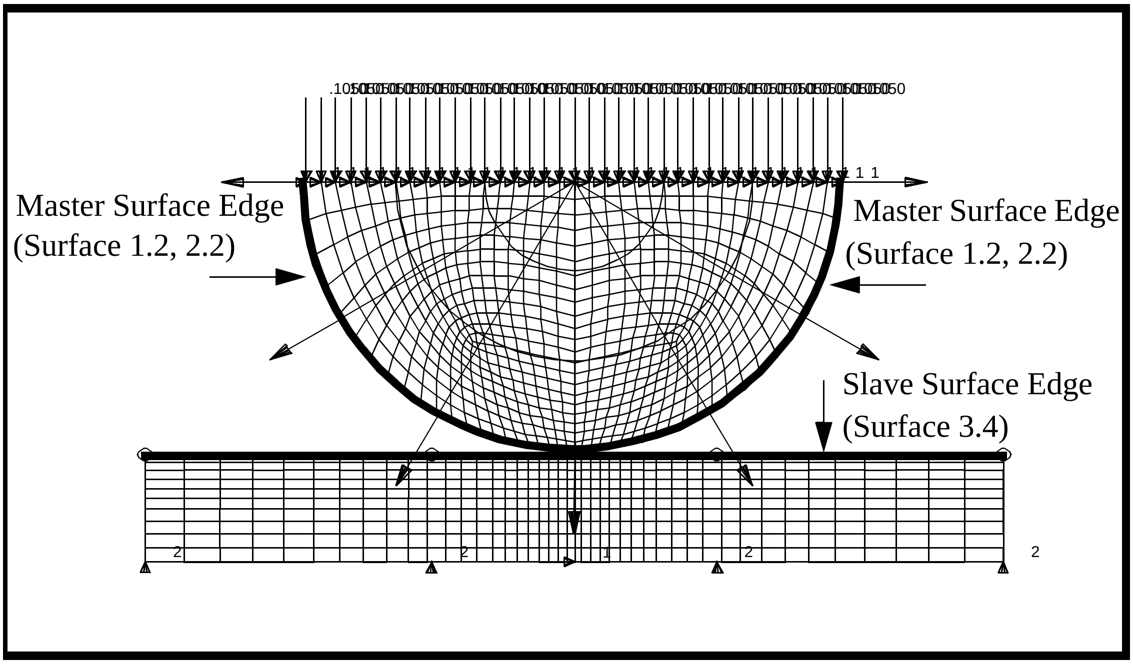
Input Data...

Contact Type:	**General**
Sliding:	**Small**
Surface Behavior:	**Hard**
Friction Formulation:	**Penalty**
Viscous Damping:	**OFF**
Penetration Type:	**One Sided**

OK

Select Application Region...

Figure 7.9 - Contact surface definition



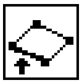
Geometry Filter:	◆ **Geometry**
Master Surface:	**2D Solid Edge**

Slave Surface:	**2D Solid Edge**
Active Region:	**Master**
Select Surface Edges:	see Figure 7.9
Add	
Active Region:	**Slave**
Select Surface Edges:	see Figure 7.9
Add	
OK	
Apply	

19. The final Load and Boundary Condition will be the applied pressure at the top of the cyclinder. We will simulate a load of 100,000 lb per unit depth across the entire cyclinder. The applied edge pressure will be simply the load per unit depth divided by the diameter, or 100,000/10.0 = 10,000 psi pressure.

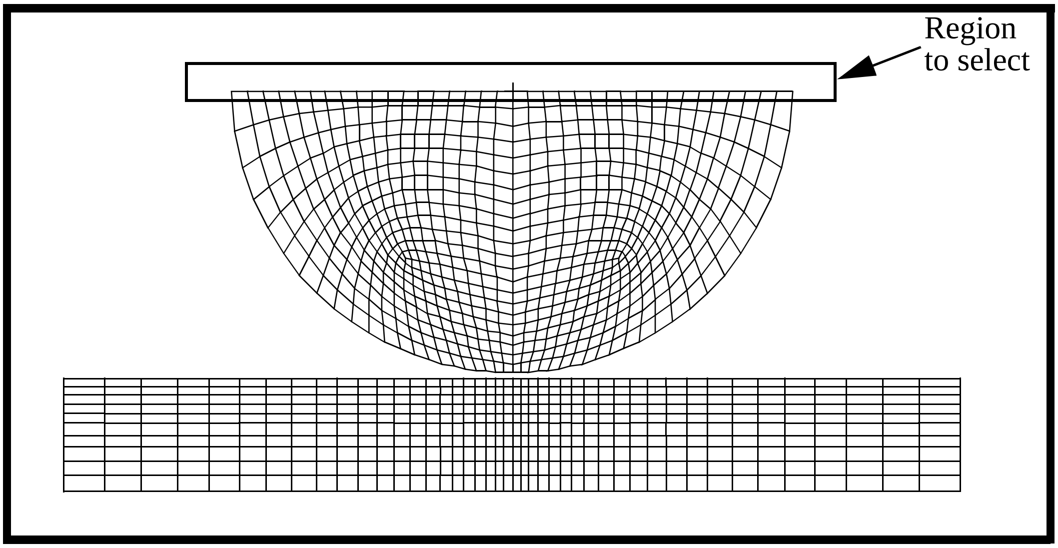
Action:	**Create**
Object:	**Pressure**
Type:	**Element Uniform**
New Set Name:	**hertz_load**
Target Element Type:	**2D**
Input Data...	
Edge Pressure:	**10000**
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**
Select 2D Element or Edge:	

Click on the select element edge icon.Then,

 **Edge of Element**

select the top edge of the model as shown in Figure 7.10:

Figure 7.10 - Region to apply pressure to



Add

OK

Apply

20. You will now create the two load cases for analysis.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**contact_step_1**
Assign/Prioritize Loads/BCs	
Select LBCs to Add to Spreadsheet:	**Conta_cylinder_plate** **Displ_close_gap** **Displ_fixed_base_1** **Displ_fixed_base_2**

OK

Apply

Change the name in the *Load Case Name* databox

Load Case Name:	**hertz_load_step_2**
Assign/Prioritize Loads/BCs	
Select LBCs to Add to Spreadsheet:	**Press_hertz_load**
(select rows containing)	**Displ_close_gap**
Remove Selected Rows	
OK	
Apply	

To review, you should have 2 Load Cases with the following Loads and Boundary Conditions:

Load Case #	Load/BC's
contact_step_1	Conta_cylinder_plate Displ_fixed_base_1 Displ_fixed_base_2 Displ_close_gap
hertz_load_step_2	Conta_cylinder_plate Displ_fixed_base_1 Displ_fixed_base_2 Press_hertz_load

21. Now create the analysis steps. Both steps will use the default analysis parameters and output requests.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	**hertz**
Step Creation...	
Job Step Name:	**step_1_hertz_contact**
Solution Type:	**Nonlinear Static**
Select Load Cases...	
Available Load Cases:	**contact_step_1**
OK	
Apply	

Create a second step to perform the Hertz contact analysis.

Job Step Name:	**step_2_hertz_contact**
Solution Type:	**Nonlinear Static**
Select Load Cases...	

Available Load Cases: **hertz_load_step_2**

OK

Apply

Cancel

22. Select the steps and submit the job for analysis. Make sure that the Default Static Step is not selected.

Step Selection...

Selected Job Steps: **step_1_hertz_contact**
step_2_hertz_contact

Apply

Apply

23. Monitor the analysis by monitoring the **.sta** and **.msg** files. These files can be viewed during or after a job has completed using **more** or **tail** commands. Once the job is finished, read the results.

◆ **Analysis**

Action: **Read Results**

Object: **Result Entities**

Method: **Translate**

Select Results File...

Selected Results File: **hertz.fil**

OK

Apply

When PATRAN opens again the results will be incorporated into the database.

24. Post the group **fem**.

Group/Post...

Select Groups to Post: **fem**

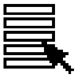
Apply

Cancel

25. Make fringe plots of Von Mises, σyy and σxy.

◆ **Results**

Click on **Select Results**

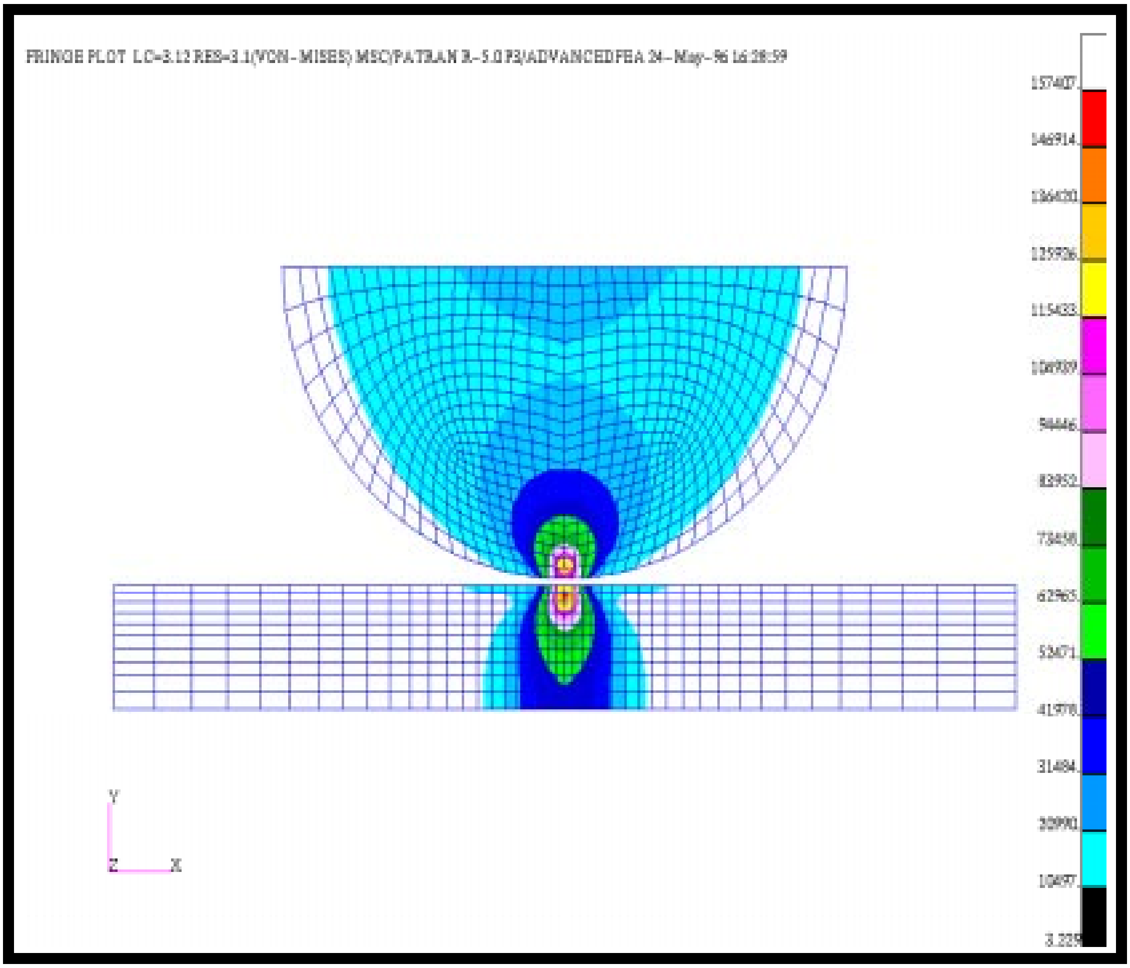


Action:	**Create**
Object:	**Quick Plot**
Select Results Cases:	select the last load case
Select Fringe Result:	**Stress,Components**
Result Quantity:	**Von Mises**

Apply

Your model should appear as shown in Figure 7.11:

Figure 7.11 - Stress plot of results



Clear the viewport using the following icon in the top menu bar.

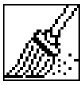
Reset Graphics

Now create the σyy plot.

Result Quantity: **Y-Component**

Apply

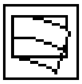
Clear the viewport using the following icon in the top menu bar.

 **Reset Graphics**

Finally, use the above procedure to create a σ_{xy} plot to see the shear stresses. Make sure clean up the viewport using the Reset Graphics Icon.

26. Make a deformed plot with Von Mises stresses plotted.

Before we make the deformed plot, we need to change the scale factor of the result to 1.0 so that the plot will represent the actual deformation In the Results form select the **Deform Attributes** icon..

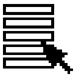


Scale Factor:	**1.0**

◆ **True Scale**

❒ **Show Undeformed Entities**

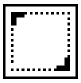
Go back to the **Selects Results** form



Select Result Cases:	select last load case
Select Fringe Result:	**Stress, Components**
Result Type:	**Von Mises**
Select Deformation Result:	**Deformation, Displacement**

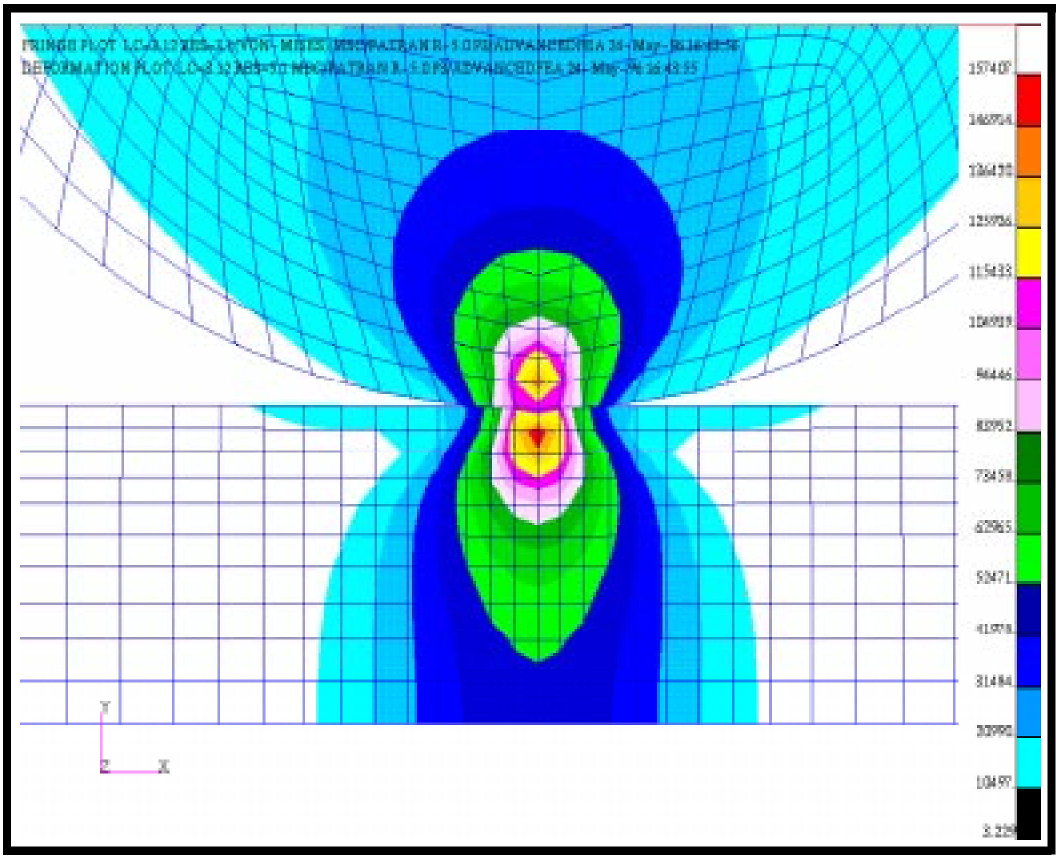
Apply

Notice that the two bodies are in contact, and that there is even a slight deformation of both the plate and cylinder. To get a better view, zoom in on the contact zone by using the **View Corners** icon.

 **View Corners**

Your model should now appear as shown in Figure 7.12:

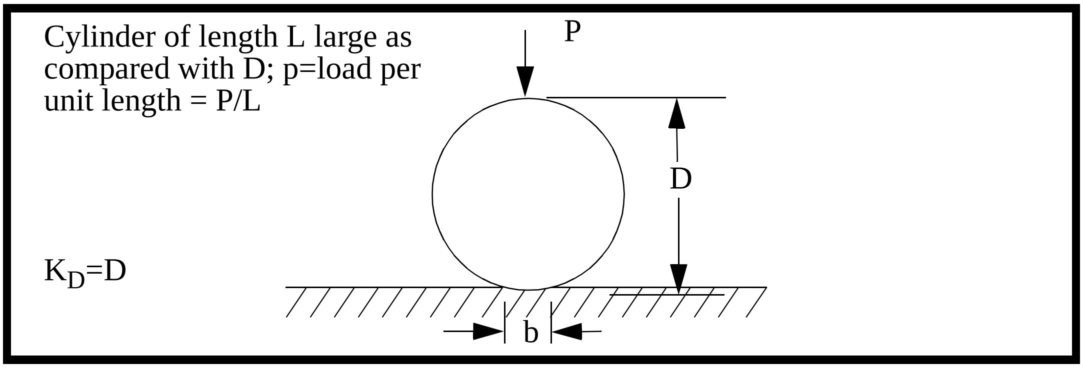
Figure 7.12 - Zoomed in plot of stress and contact verification



27. Theoretical Comparison

From the 6th Edition of Roarks's Formulas for Stress and Strain (by Warren C. Young, 1989, pg 651) we can derive the contact patch, b, and the maximum compressive stress

Figure 7.13 - Theoretical representation of loading



The contact width for a full model is given by

$$b = 1.60\sqrt{pK_DC_E}$$

where

$$C_E = \frac{1-\upsilon^2}{E_1} + \frac{1-\upsilon^2}{E_2}$$

and the maximum stress is given by

$$Max\sigma_c = 0.798\sqrt{\frac{p}{K_DC_E}}$$

For our solution, C_E, is equal to

$$C_E = \frac{1-(0.3)^2}{30\times10^6} + \frac{1-(0.33)^2}{10\times10^6} = 1.1944\times10^{-7}$$

The contact area for a half model is equal to

$$\frac{b}{2} = 0.80\sqrt{100,000(10)1.1944\times10^{-7}} = 0.276$$

The Maximum compressive stress is given by

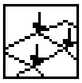
$$Max\sigma_c = 0.798\sqrt{\frac{100,000}{(10)1.1944\times10^{-7}}} = 230,900$$

We can compare the maximum Y compressive stress values to a summary listing of the maximum stresses output for the cylinder and base plate from text report.

28. Generate a text report of the Y compressive stresses..

Action:	**Create**
Object:	**Report**
Method:	**Preview**
Select Results Cases:	**select last increment of Step2, Time = 2.0**
Select Report Result:	**Stress,Components**

Select the **Display Attributes** icon:.



Sorting Options ...

Sort Order:	◆ **Ascending**
Sort Data By:	◆ **Y Component**

OK

Apply

The results will be printed to the UNIX shell that you started P3 from. If you scroll to the top of the list output by the Text Reporter, you will see the most severely loaded element. Use that information to fill out the table below.

Table 1: Maximum Compressive Stress

	Theoretical	AFEA	% Diff
Max σ_c	**230,900**		

Close the database and quit PATRAN.

This concludes this exercise.