

S3 2D Bracket



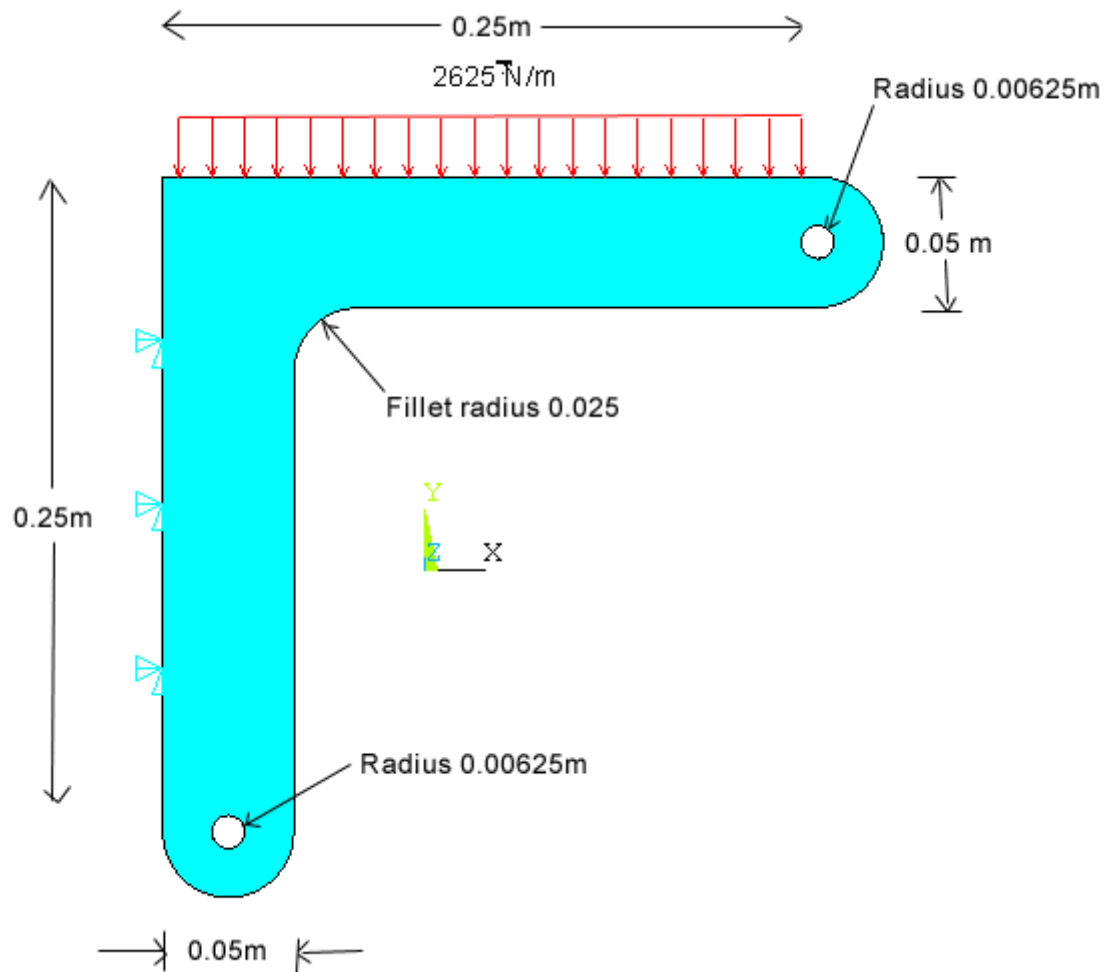
Structural #3: Analysis of a Steel Bracket

Introduction: In this example you will learn to use the Solid 8 Node element in ANSYS.

Physical Problem: Structural analysis of the Steel Bracket shown in the figure. This is a typical bracket used to support bookshelves.

Problem Description:

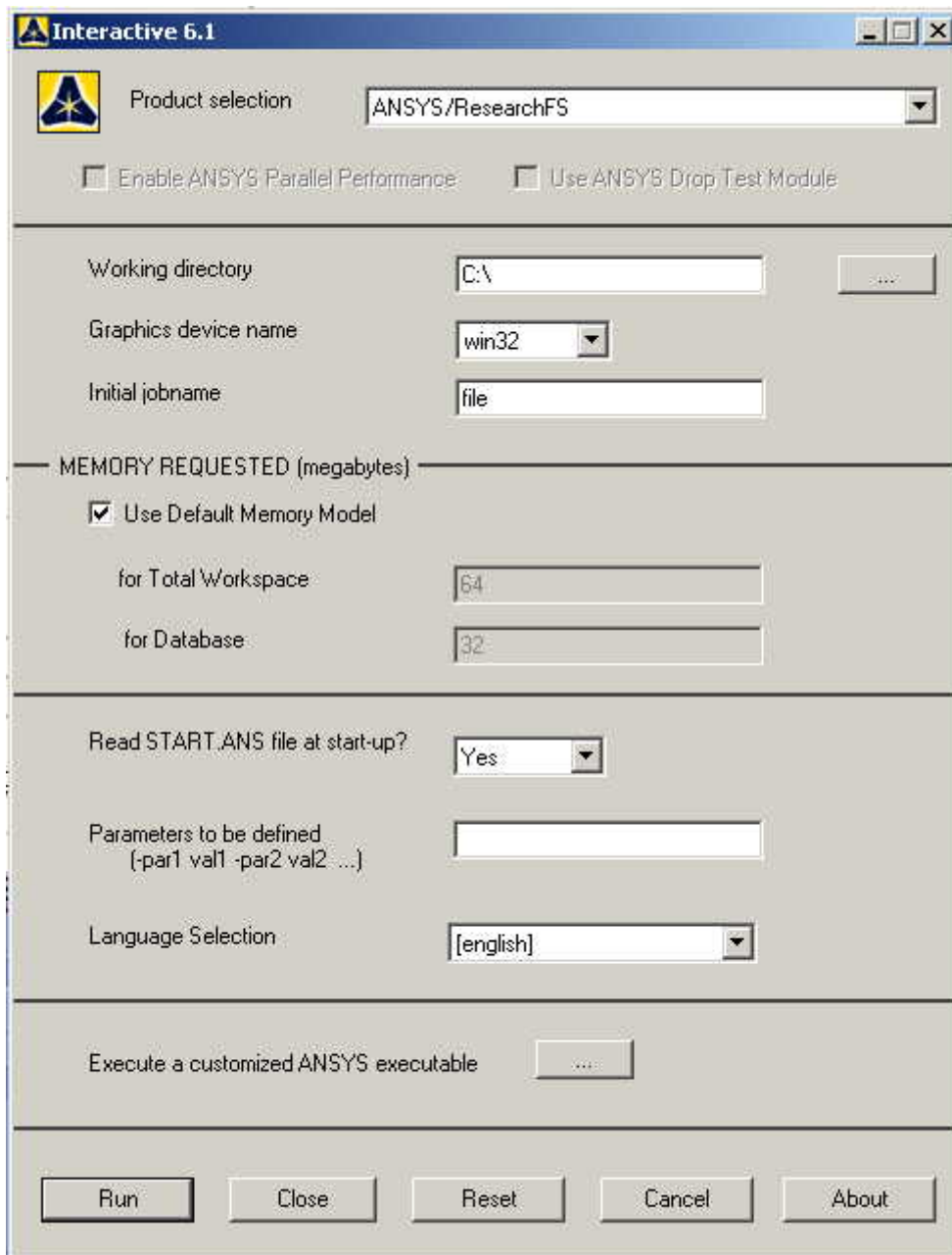
- We will model the bracket as a solid 8 node plane stress element. By a plane stress element we are assuming that there are no stresses in the thickness direction of the bracket.
- **Geometry:** The thickness of the bracket is 3.125 mm
- **Material:** Assume the structure is made of steel with modulus of elasticity $E=200$ GPa.
- **Boundary conditions:** The bracket is fixed at its left edge.
- **Loading:** The bracket is loaded uniformly along its top surface. The load is 2625 N/meter.
- **Objective:**
 - Plot deformed shape
 - Determine the principal stress and the von Mises stress. (Use the stress plots to determine these)
 - Remodel the bracket without the fillet at the corner, and see how principal stress and von Mises stress change.
- You are required to hand in print outs for the above.
- Figure:



IMPORTANT: Convert all dimensions and forces into SI units

STARTING ANSYS:

- Click on **ANSYS 6.1** in the programs menu.
- Select **Interactive**.
- The following menu that comes up. Enter the working directory. All your files will be stored in this directory. Also enter **64** for Total Workspace and **32** for Database.
- Click on **Run**.



MODELING THE STRUCTURE:

- Go to the ANSYS Utility Menu
 - Click **Workplane>WP Settings**
 - The following window comes up

WP Settings

☒ Cartesian
☐ Polar

☐ Grid and Triad
☒ Grid Only
☐ Triad Only

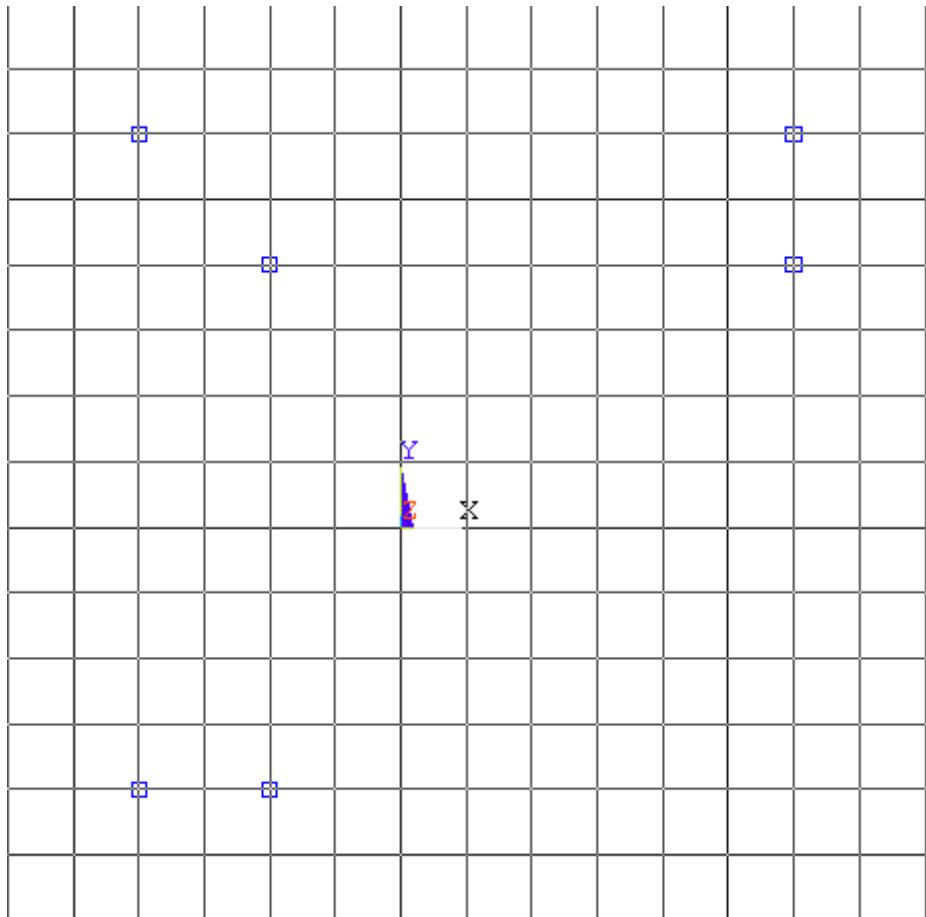
☒ Enable Snap

Snap Incr
Snap Ang

Spacing
Minimum
Maximum
Tolerance

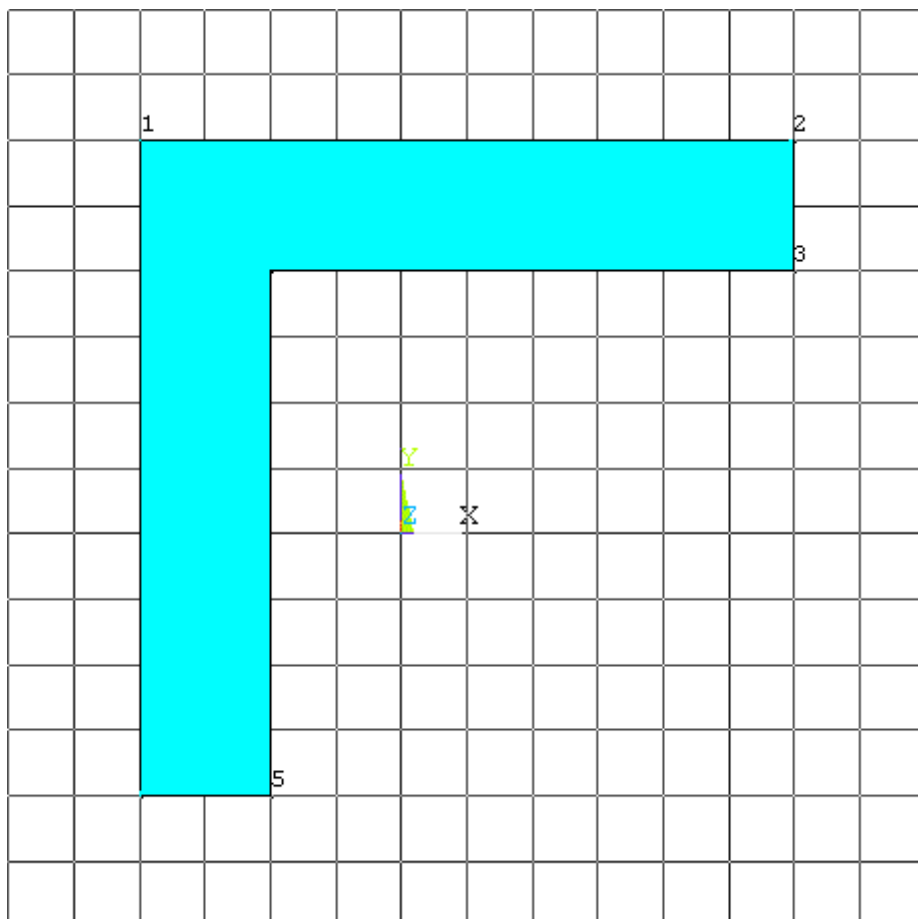
OK Apply
Reset Cancel
Help

- Check the **Cartesian** and **Grid Only** buttons
- Enter the values shown in the figure above.
- Go to the ANSYS Main Menu **Preprocessor>Modeling>Create>Keypoints>On Working Plane**
- Outline a part of the bracket as shown in the figure.

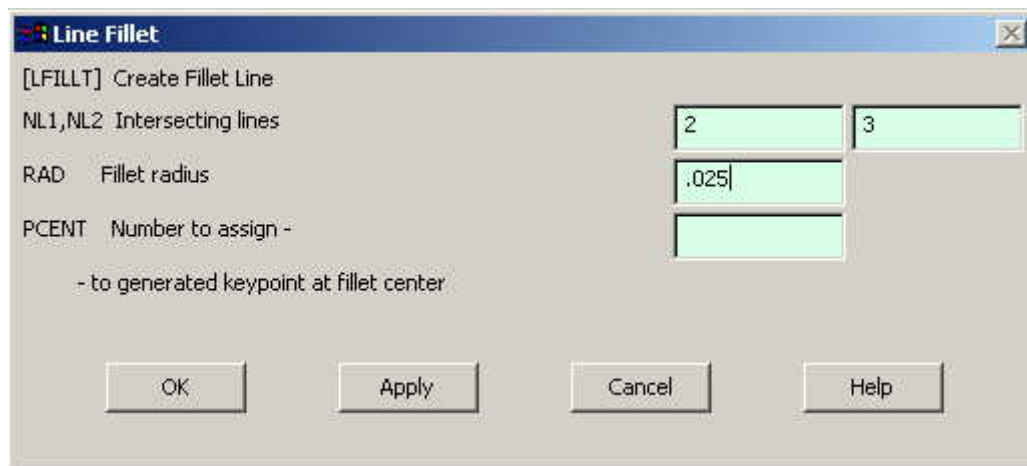


■ Note: To turn on the numbering: **ANSYS Utility Menu>Plot Controls>Numbering...**

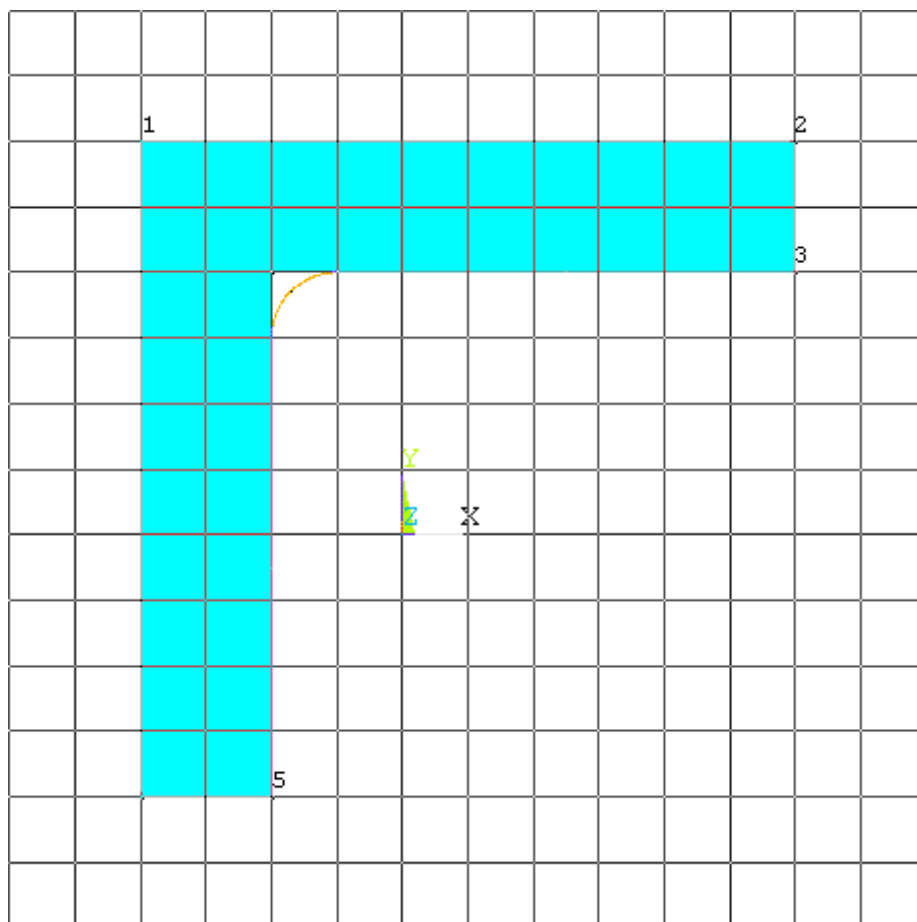
■ Now create lines between keypoints, then create area inside. Go to **Preprocessor>Modeling>Create>Areas>Arbitrary>By Lines.**



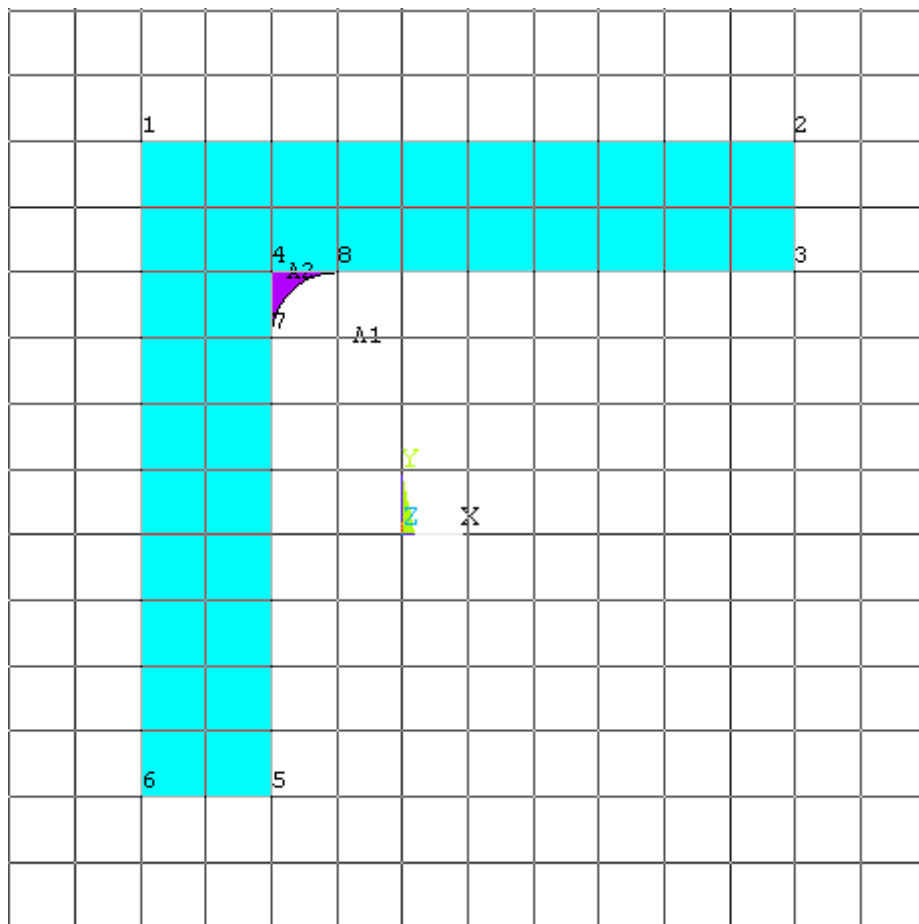
- Now go to **Preprocessor>Modeling>Create>Lines>Line Fillet**.
- The following window comes up. Select the two lines between which you want the fillet and click OK.



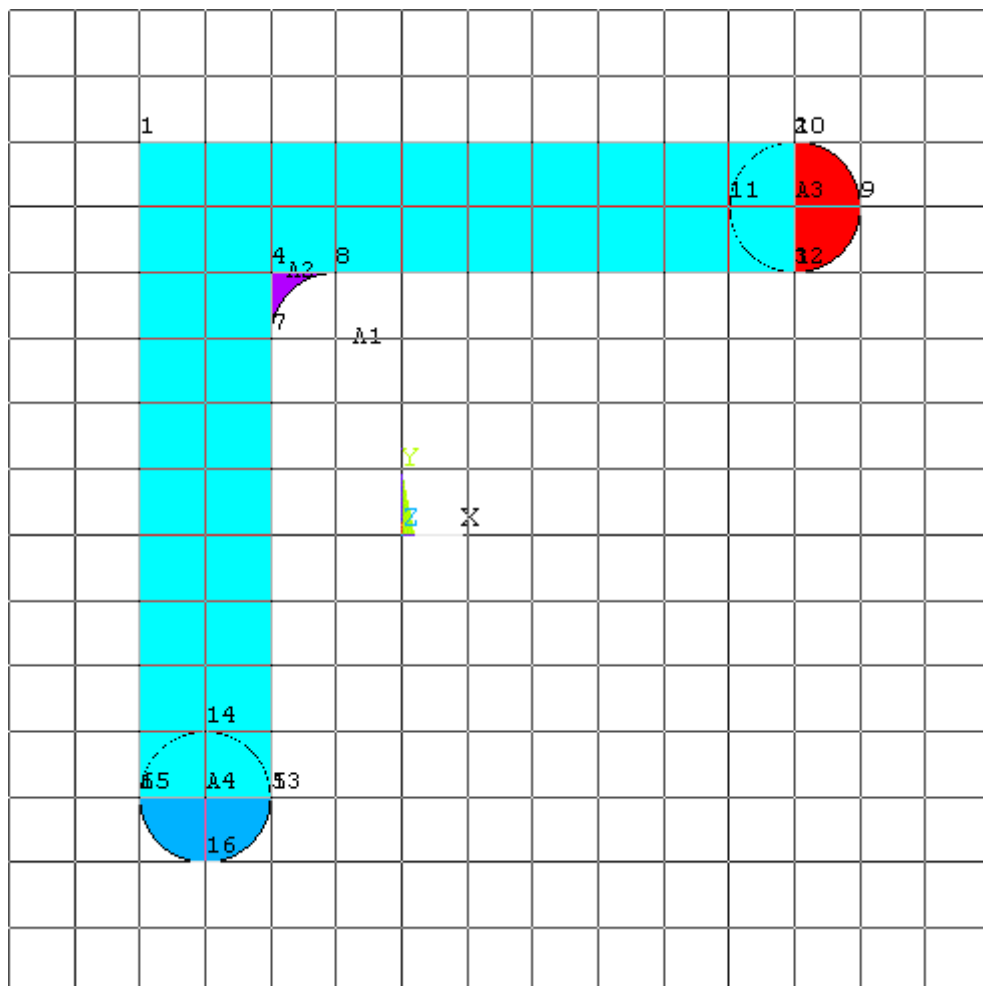
- In the box that comes up enter **0.025** for fillet radius and click OK.



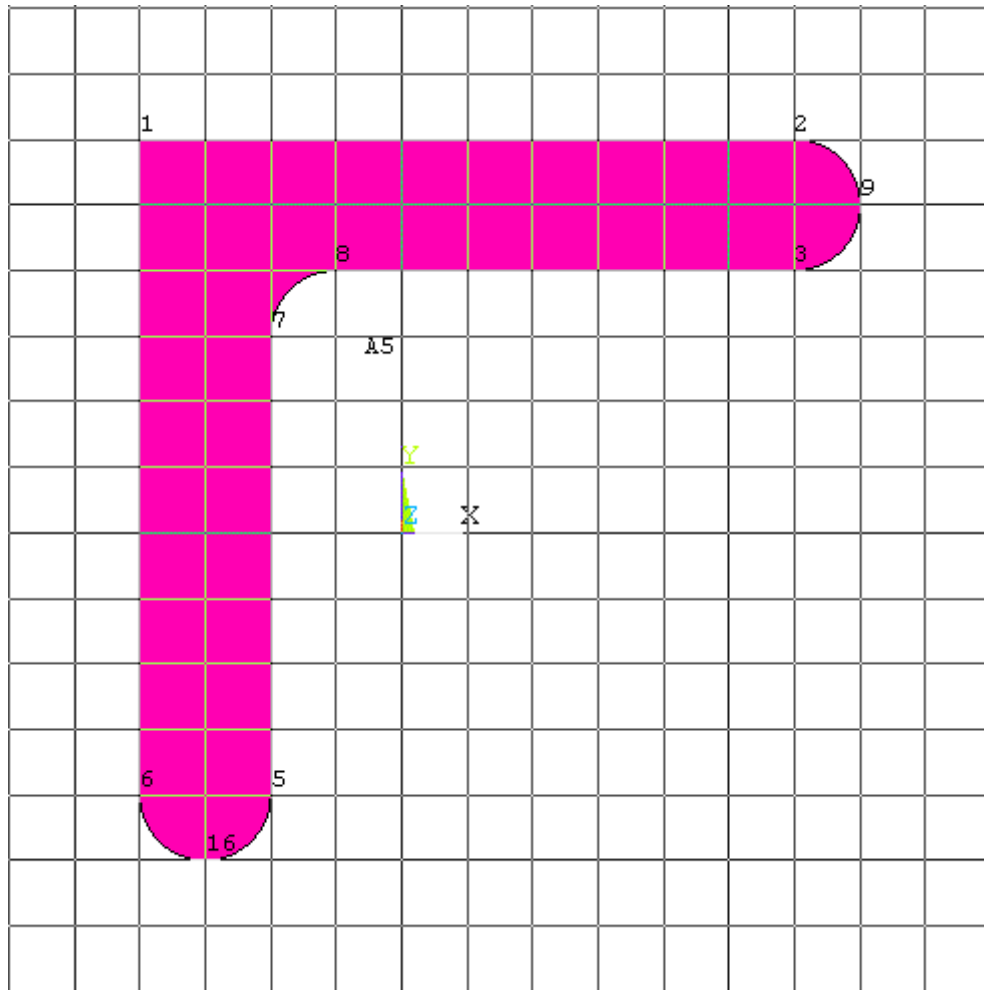
- Now go to **Preprocessor>Modeling>Create>Areas>Arbitrary>By Lines** to fill the fillet area.



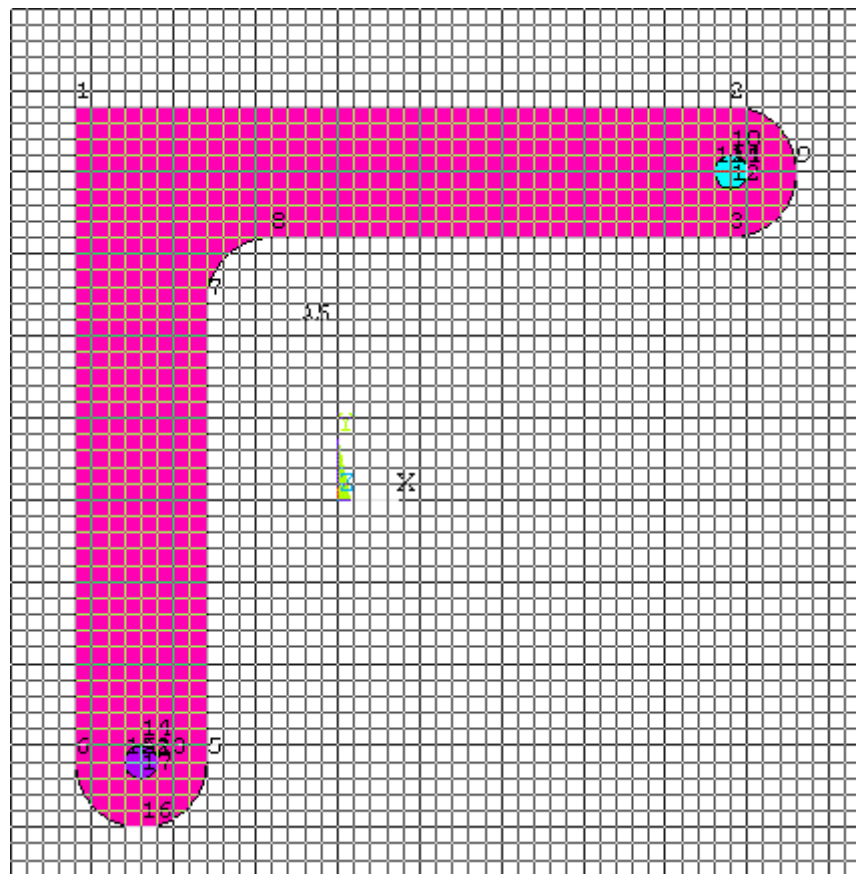
- Go to **Preprocessor>Modeling>Create>Areas>Circles>Solid Circle** and create the two circles with centre at the midpoint of the right edge and the bottom edge of the bracket and the diameter equal to the length of that edge.



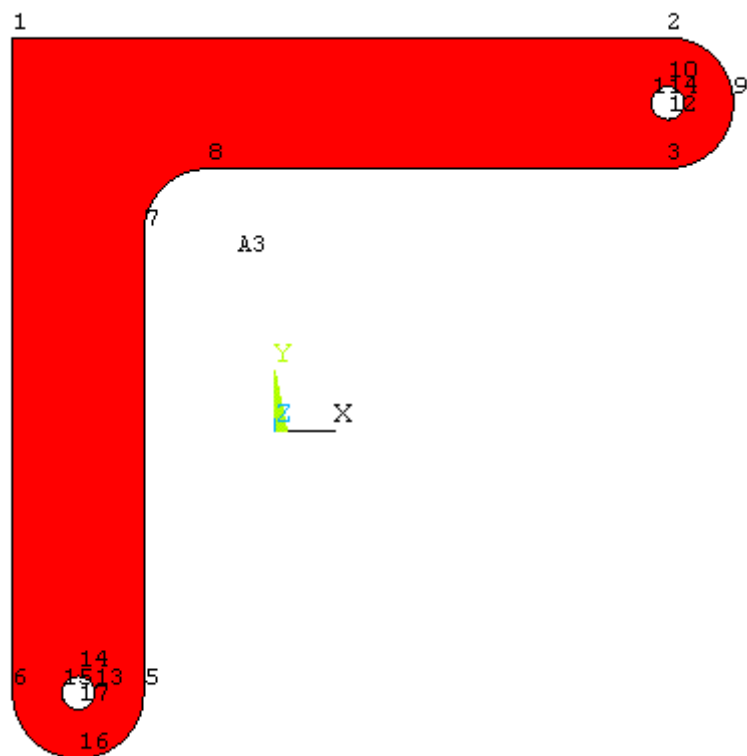
Now go to **Preprocessor>Modeling>Operate>Booleans>Add>Areas** and select all areas you have created to make a single area.



- Now go to **Workplane>WP Settings** and change the Snap Incr and grid settings to **0.00625**. We do this so that we can make the small inner circle whose radius is 0.00625 meter.
- Go to **Preprocessor>Modeling>Create>Areas>Circles>Solid Circle** and create the a circle with center at the midpoint of the right edge of the horizontal rectangle and the radius equals to **0.00625**. Do the same thing for the vertical rectangle.



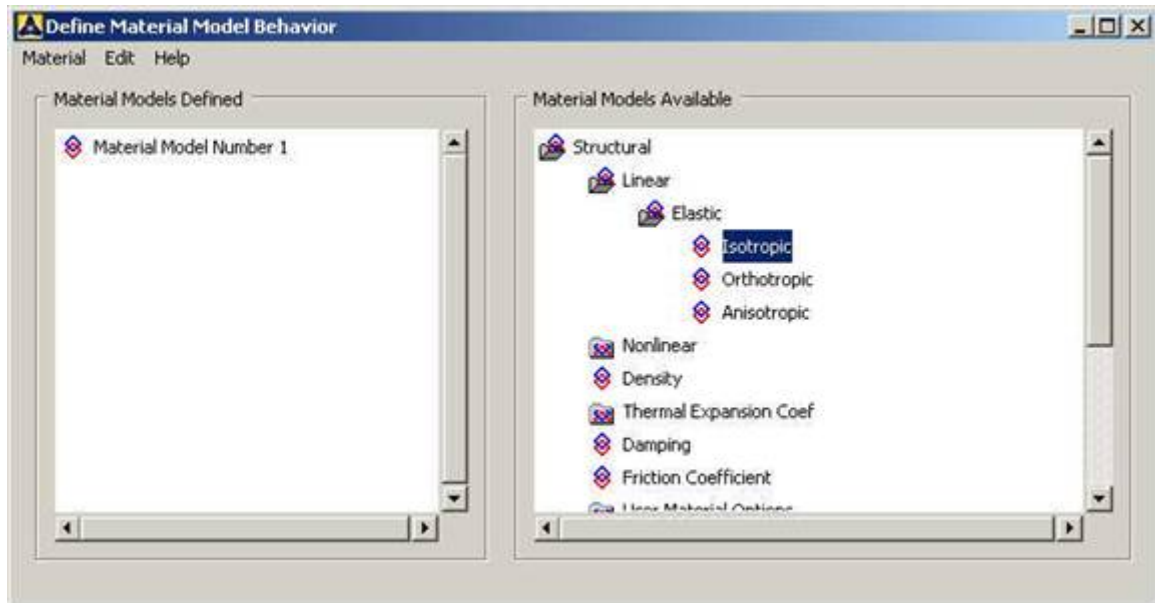
Now go to **Preprocessor>Modeling>Operate>Booleans>Subtract>Areas**. First select the base area from which the smaller area will be subtracted. Say OK. Now select the smaller circles and say OK. the smaller circles will now be subtracted and the figure will look like this:



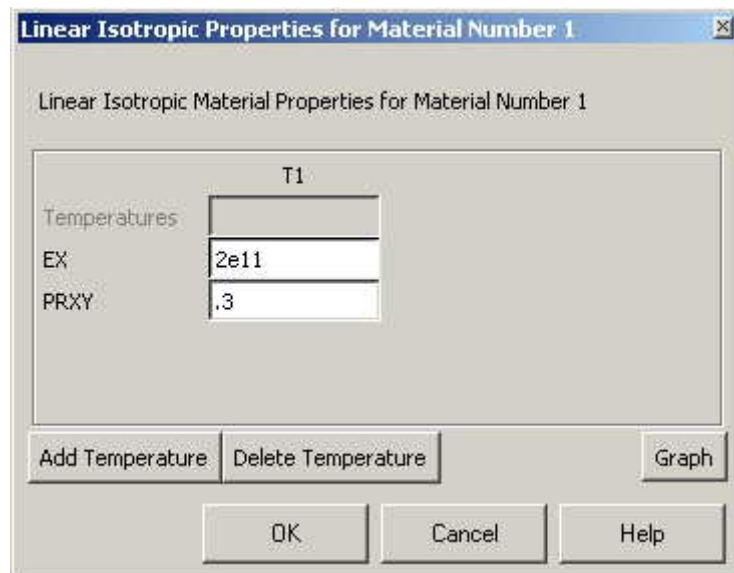
MATERIAL PROPERTIES:



Go to the ANSYS **Main Menu>Preprocessor>Material Props>Material Models**. From this window, select **Structural>Linear>Elastic>Isotropic**.



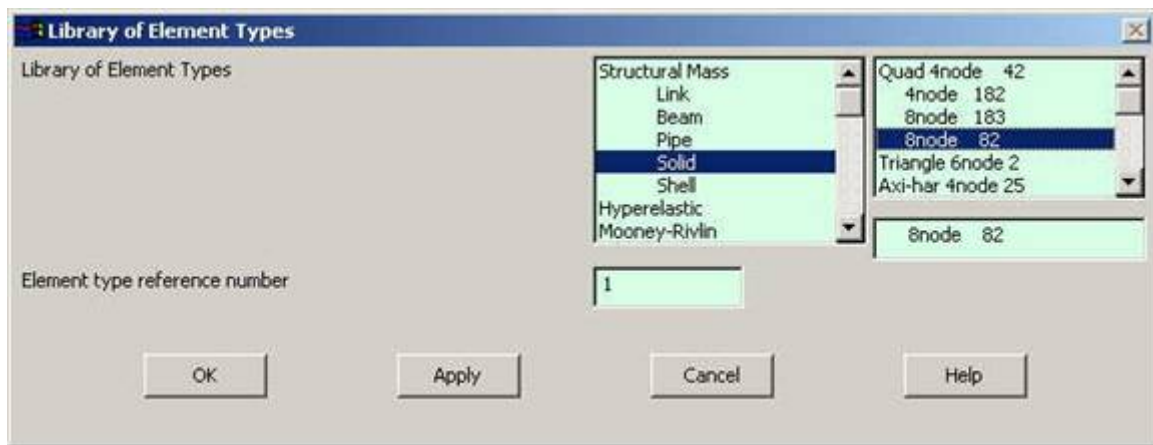
- Enter 1 for the Material Property Number and click OK. The following window comes up.



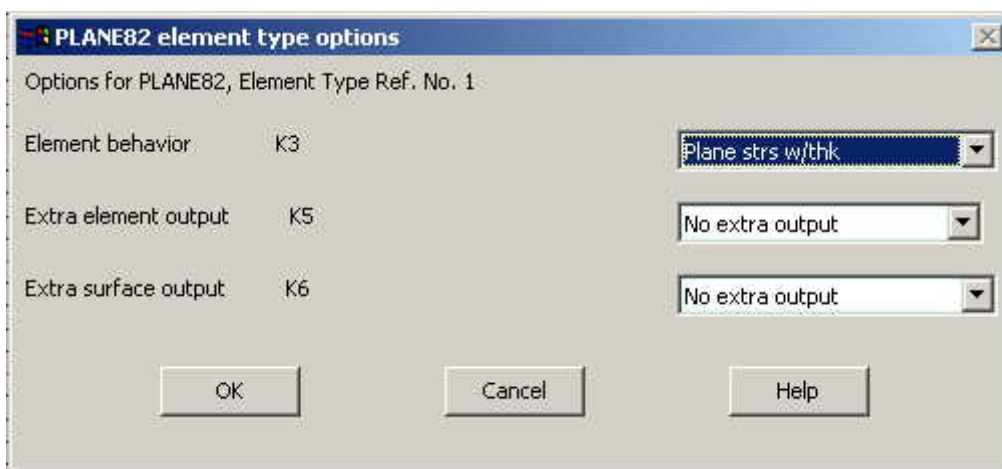
- Fill in **2e11** for the Young's modulus and **0.3** for minor Poisson's Ratio. Click OK
- Now the material 1 has the properties defined in the above table. We will use this material for the structure.

ELEMENT PROPERTIES:

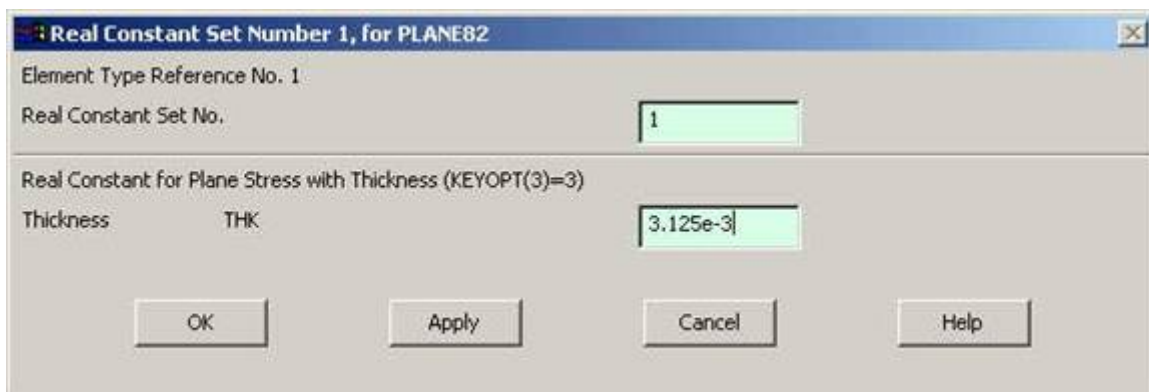
- **SELECTING ELEMENT TYPE:**
 - Click **Preprocessor>Element Type>Add/Edit/Delete...** In the 'Element Types' window that opens click on Add... The following window opens.



- Type **1** in the Element type reference number.
- Click on **Structural Solid** and select **Quad 8 node 82**. Click OK. Close the 'Element types' window.
- Click **Preprocessor>Element Type>Add/Edit/Delete...** In the 'Element Types' window that opens click on Options... The following window opens.



- Select **Plane strs w/thk** for K3 and click OK.
- So now we have selected Element type 1 to be a Structural Solid 8 node element. The bracket will now be modeled as elements of this type.
- Now we need to define the thickness for this element.
- Go to **Preprocessor>Real Constants**
- In the "Real Constants" dialog box that comes up click on Add
- In the "Element Type for Real Constants" that comes up click OK. The following window comes up.



- Fill in the relevant values and click on OK.
- We have now defined the thickness of the element.

MESHING:

- DIVIDING THE BRACKET INTO ELEMENTS:
 - Go to **Preprocessor>Meshing>Size Controls>Manual Size>Lines>Picked Lines**. Pick all the lines on the outer boundary of the figure and click OK.

- In the menu that comes up type **0.0125** in the field for 'Element edge length'.

- Click on OK.
- Repeat the process to divide the lines forming the small inner circle. In this case enter **0.001** in the field for 'Element edge length'.
- Now go to **Preprocessor>Meshing>Mesh>Areas>Free**.
- Select the area and click on OK in the "Mesh Areas" dialog box.
- Now the bracket is divided into Solid 8 node elements.

BOUNDARY CONDITIONS AND CONSTRAINTS:

- APPLYING BOUNDARY CONDITIONS
 - The bracket is fixed at the left edge.
 - Go to Main Menu **Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Lines**.
 - Select the line on the left edge and click OK. The following window comes up:

- Select All DOF and click OK.

■ APPLYING FORCES

- Go to Main Menu **Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Line**.
- Select the top line.
- Click on OK in the 'Apply PRES on lines' window. The following window will appear:

Apply PRES on lines

[SFL] Apply PRES on lines as a Constant value

If Constant value then:

VALUE Load PRES value 2625

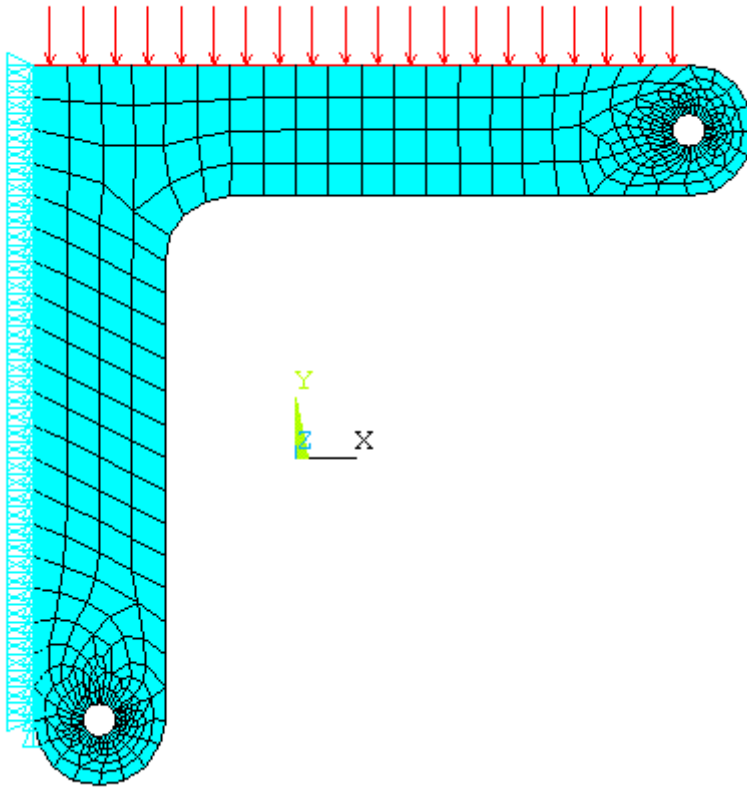
If Constant value then:

Optional PRES values at end J of line
(leave blank for uniform PRES)

Value

OK Apply Cancel Help

- Enter the value of the pressure as shown above.
- Click OK.
- The model should look like the one below.



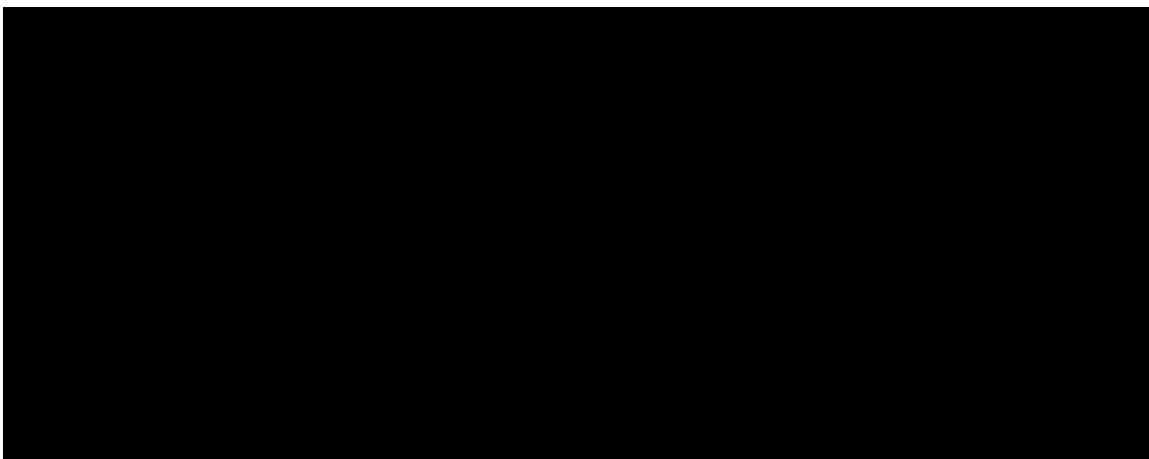
- Now the Modeling of the problem is done.

SOLUTION:

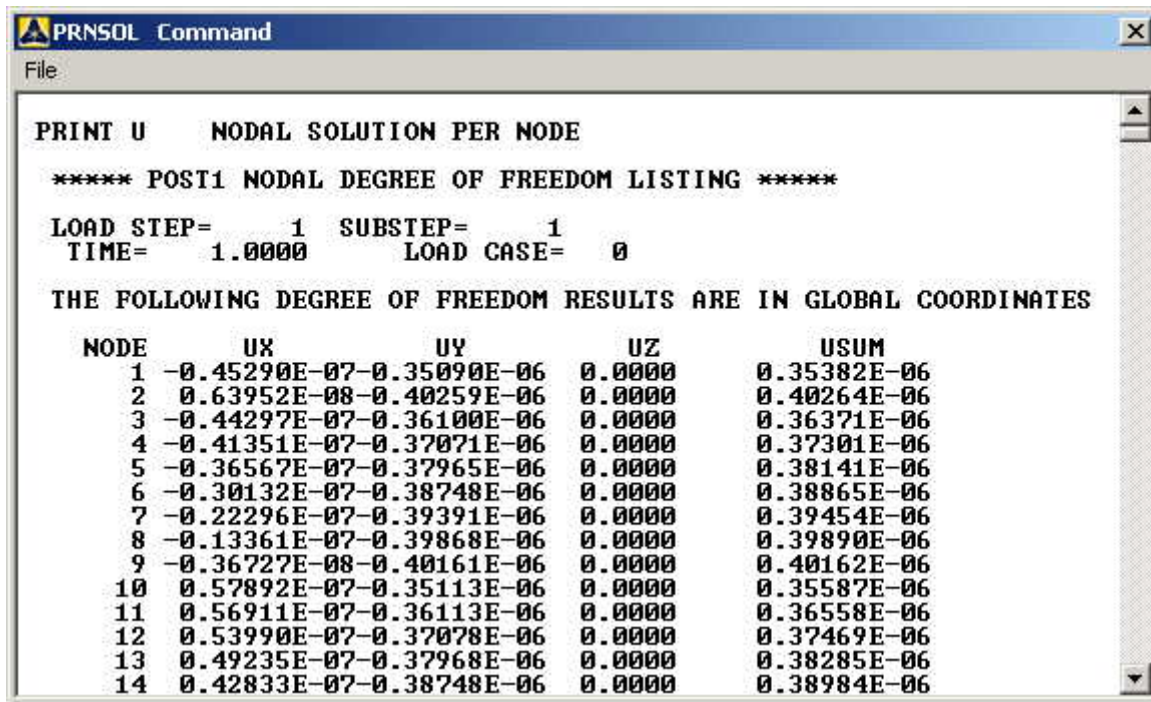
- Go to ANSYS **Main Menu>Solution>Analysis Type>New Analysis**.
- Select static and click on OK.
- Go to **Solution>Solve>Current LS**.
- Wait for ANSYS to solve the problem.
- Click on OK and close the 'Information' window.

POST-PROCESSING:

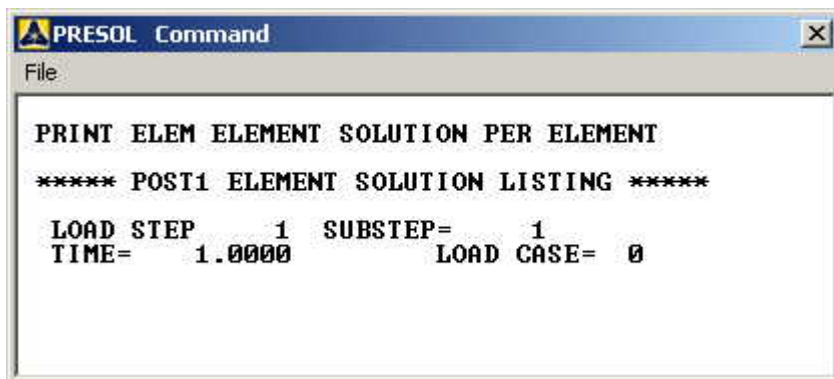
- Listing the results.
 - Go to ANSYS Main Menu
 - Click on **General Postprocessing>List Results>Nodal Solution**. The following window will come up.



- Select DOF solution and All U's. Click on OK. The nodal displacements will be listed as follows.

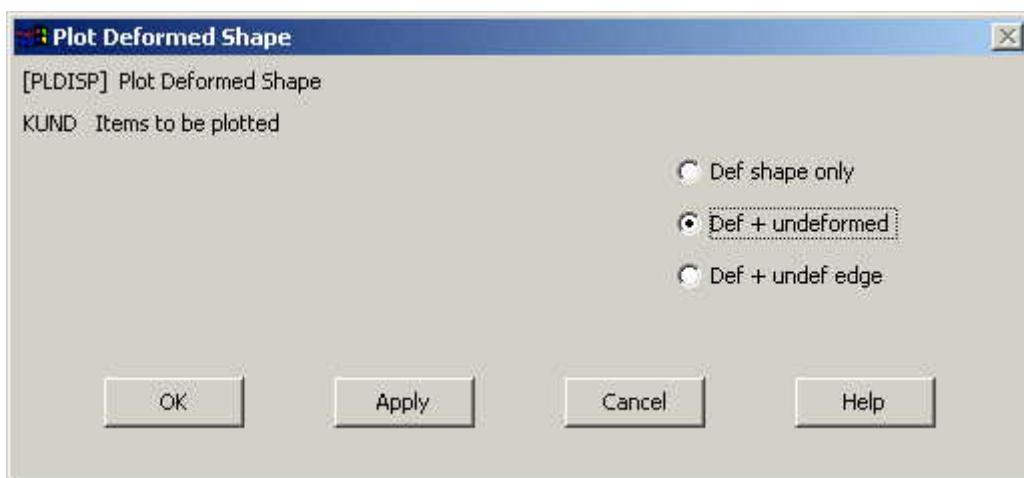


- Similarly you can list the stresses for each element by clicking [General Postprocessing>List Results>Element Solution](#). Now select **LineElem Results**. The following table will be listed.

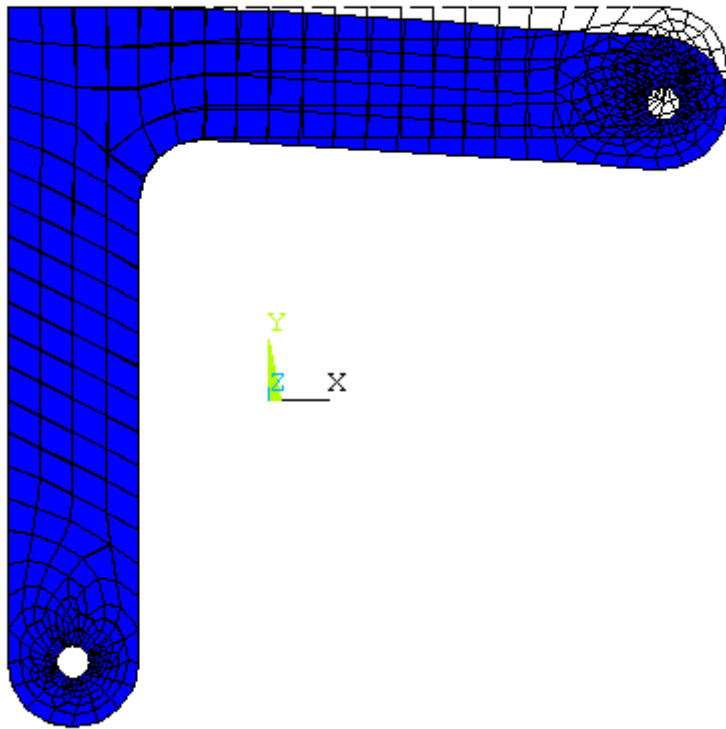


MODIFICATIONS:

- You can also plot the displacements and stress.
- Go to [General Postprocessing>Plot Results>Deformed shape](#). The following window comes up.



Select **Def+undeformed** and click OK. The output will be like the figure below.



Select a stress (**SEQV**) to be plotted and click OK. The output will be like this.

