# ANSYS 11 - Crank Step 8

Step 8: Postprocess the Results

Plot Deformed Shape

Main Menu > General Postproc > Plot Results > Deformed Shape

Select *Def + undef edge* and click *OK*.

This plots the deformed and undeformed shapes in the *Graphics* window. The maximum deformation *DMX* is 0.026148 inches as reported in the *Graphics* window. We should check that our results make sense. It appears that the boundary conditions have been satisfied as the tip of the shaft moves downward and the hole at the other end of the crank is held in place.
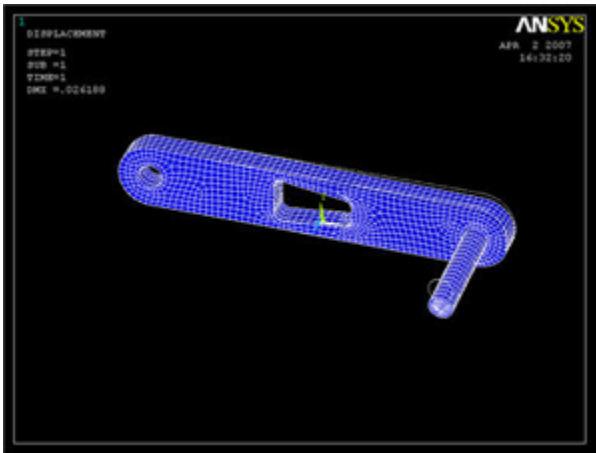
Animate the deformation

Utility Menu > PlotCtrls > Animate > Deformed Shape...

Select *Def + undeformed* and click *OK*. Select *Forward Only* in the *Animation Controller*. This is also a good way to check the boundary conditions have been applied correctly. Close the Animation Controller.

Plot Nodal Solution of von Mises Stress

For a quick refresher on von Mises stress, click Help. Search for von mises and click on the result *2.4 Combined Stresses and Strains* (this is lower down among the search results). Read through section 2.4.2.

Main Menu > General Postproc > Plot results > Contour Plot > Nodal Solu

Select *Nodal Solution > Stress > von Mises stress* and click *OK*. To change the range of stresses displayed, go to
Utility Menu > PlotCtrls > Style > Contours > Uniform Contours ...

and select *User specified*. Specify a range of minimum 0 and maximum 25000. We can now see more color variation in the model, and easily pick out the red areas.

When you plot the "Nodal Solution", ANSYS obtains a continuous distribution as follows:
1. It determines the average at each node of the values of all elements connected to the node.
2. Within each element, it linearly interpolates the average nodal values obtained in the previous step.
This procedure is in effect a smoothing operation.

The stress concentration located at the tip of the shaft can be ignored as the force is applied as a point load. Let's look at the results just for the crank by deselecting the elements within the pedal shaft volume. Go to

Utility menu > Select > Entities ...

Select *Volumes*, *By Num/Pick*, *From Full* and click *Apply*. Pick the crank volume and click *OK*. After we've selected a volume, we must next select all the elements in this volume. In the Select Entities window, select *Elements*, *Attached to*, *Volumes* and click *Apply*. Click *Replot* to display the new selection. Notice the deformation is exaggerated, revealing that deformation is primarily caused by torsion.
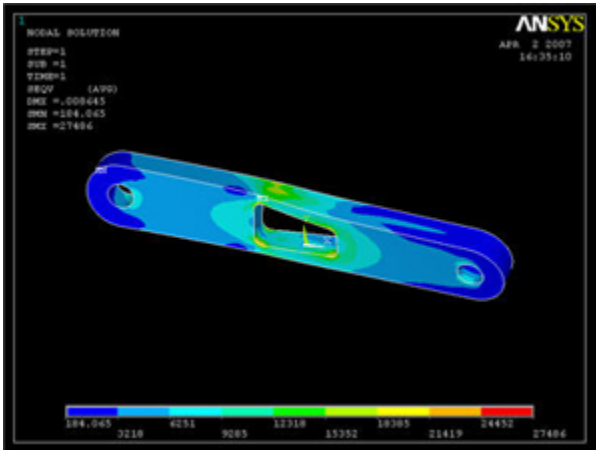
Higher Resolution Image

To select the whole model again, go to **Utility Menu > Select > Everything**.

Comparing the $_{xx}$ Stress with von Mises Stress

To verify that the bending stress in the crank is relatively insignificant, we can compare the element $_{xx}$ solution with the elemental von Mises solution.

General Postproc > Plot Results > Contour Plot > Element Solu

Click on Stress, then X-Component of stress , then Apply.

If grey areas are appearing in your contour plots, you should go to **Utility Menu > PlotCtrls > Style > Contours > Uniform Contours ...**, select *Auto calculated*, and click *OK*.

Notice that the top-left and bottom-right corners of the cutout area are now blue, and that the scale has been readjusted to show that blue is now a large negative stress value. If this were a case of pure bending, we would expect the top of the crank to be in tension, not compression!

To find out information about specific points on the model, go to

General Postproc > Query Results > Subgrid Solu

Select *Stress*, *X-direction SX*, and click *OK*. The picking window will appear, and you can click on any point in the model. Click *OK* when finished.
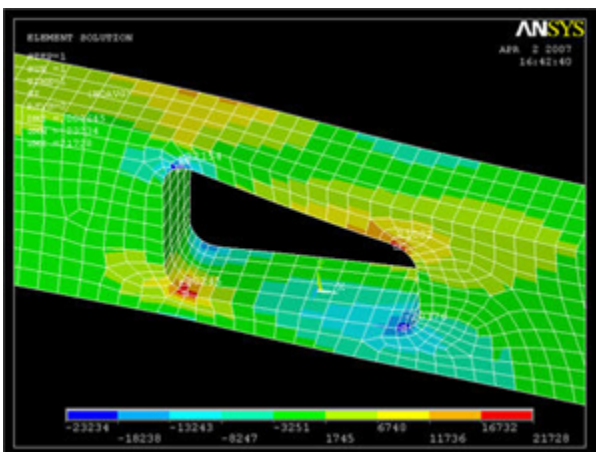
Compare the stress values with the von Mises stress. (Click on von Mises stress, then *OK*)

Investigate the Stress Concentration

Let's zoom in on the red area. Use the mouse wheel to zoom in and out in the view area. Some other viewing functions: Holding down the Ctrl key and the left mouse button allows you to pan the view, while holding the Ctrl key and the right mouse button allows you to rotate the view. Hold down the right mouse button and draw a rectangle to zoom in on a specific region.

Recall that the nodal solution shows smoothed stress values. Let's compare the nodal solution with the element, i.e. non-smoothed, solution.

**General Postproc > Plot Results > Contour Plot > Element Solu**

Click on *Stress*, then *von Mises Stress*, then the *OK* button. In the vicinity of the cut-out corners, there are fairly significant discontinuities in the von Mises stress across adjacent elements. This suggests that we need to refine the mesh at least in this region. This is done in the next step.
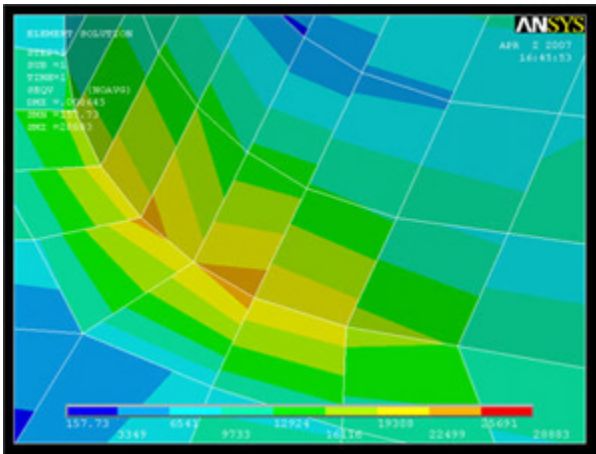
Calculate Average Strain in Specified Area

In a perfect world, we would be able to validate the ANSYS results by comparing them with strain gage measurements at selected locations. We unfortunately don't have strain gage measurements for this particular geometry but will anyway show you the process by which you can calculate the average strain over an area where a strain gage would be placed. This will help prepare you to compare your ANSYS results with strain gage measurements for a different geometry for which you may have experimental data.

Let's assume that the strain gage is placed on the front face of the crank (z=0.5") roughly halfway between the left hole and the cutout as shown below.
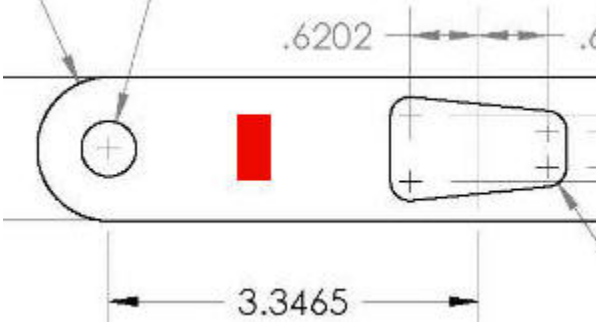
In the coordinate system in our model, let's say this area is given by -1.77" x -1.57" and 0.45" y 0.85". To find the average strain in this area, we will select all nodes that lie in this area and then list the strain values at these nodes. You can copy these values over to Excel or MATLAB to find the average value.

ANSYS provides extensive capabilities, referred to as "select logic", for selecting a subset of the full model using various criteria. We'll use select logic to select the nodes on the front face of the crank. We'll first select the area corresponding to this face.
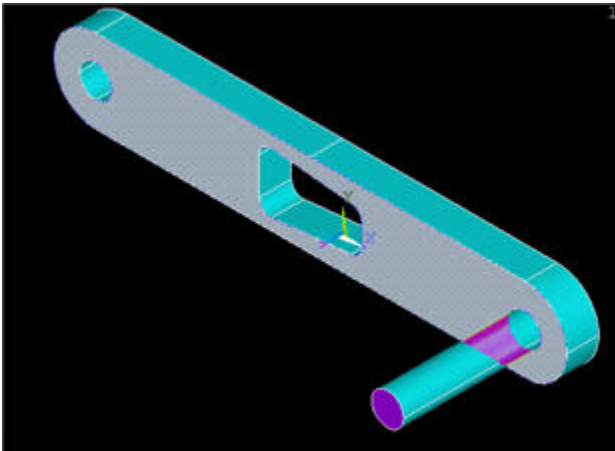
Utility Menu > Plot > Areas

Utility Menu > Select > Entities

Select *Areas* from the pull-down menu at the top. Make sure *By Num/Pick* is selected below that. Click *Apply*.

Hold down the left mouse button until the front face is picked. Click **OK** in the pick menu.

Only the area corresponding to this face is selected currently. Verify this by clicking **Replot** in the *Select Entities* menu (this replots areas).

One key thing to remember about ANSYS' "select logic" is that the various entity types (areas, volumes, nodes, elements, etc) are selected independently. So all nodes are still "selected", not just the ones that are located on the front face of the crank. Verify this: **Utility Menu > Plot > Nodes**.

We next select the nodes attached to the previously selected area. In the *Select Entities* menu, select **Nodes** from the pull-down menu at the top and **Attac hed** to below that. Select **Areas, All** below that. Click **Apply**.
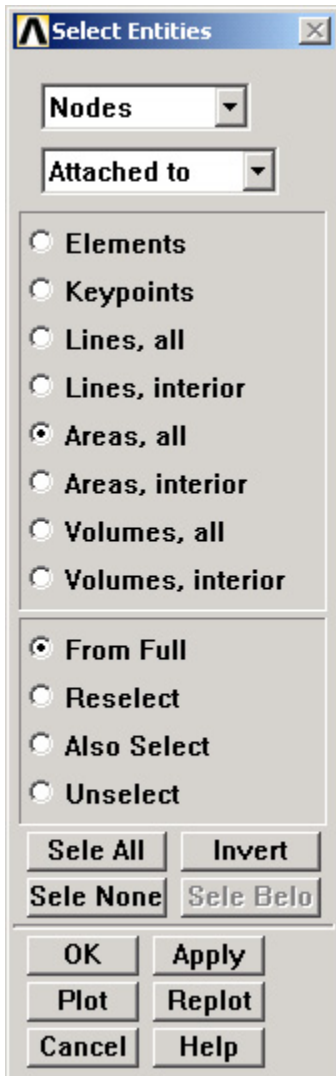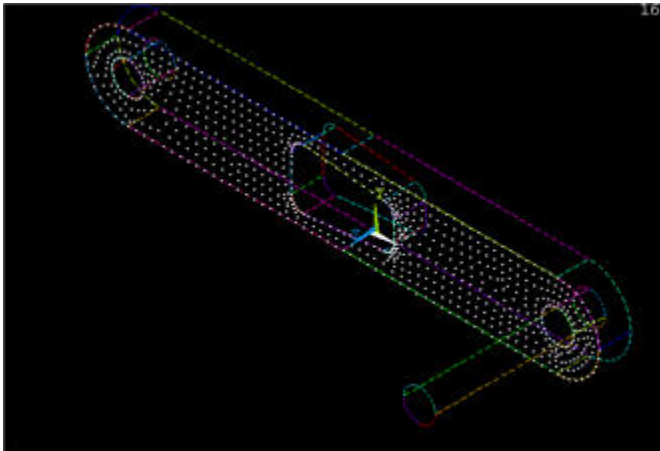
Check that only nodes attached to the front face are currently selected by clicking **Replot** in the *Select Entities* menu (this replots nodes).

We can get a better idea of where these nodes are located by plotting nodes as well as lines.

Utility Menu > PlotCtrls > Multi-Plot Controls ... **> OK**

Select **Lines** and **Nodes** and click **OK**.

Utility Menu > Plot > Multi-Plots

From these currently selected nodes, we next select nodes that satisfy the following criterion: -1.77" x -1.57". In the *Select Entities* menu, retain **Nodes** at the top. Select **By Location** and **X coordinates** below that. Enter **Min,Max** values as per the snapshot below. Since we want the nodes to be selected from the current set rather than the full set, choose the **Reselect** radio button. Click **Apply** and then **Replot**.

You should see that only the nodes that are in the desired *x*-coordinate range are selected.
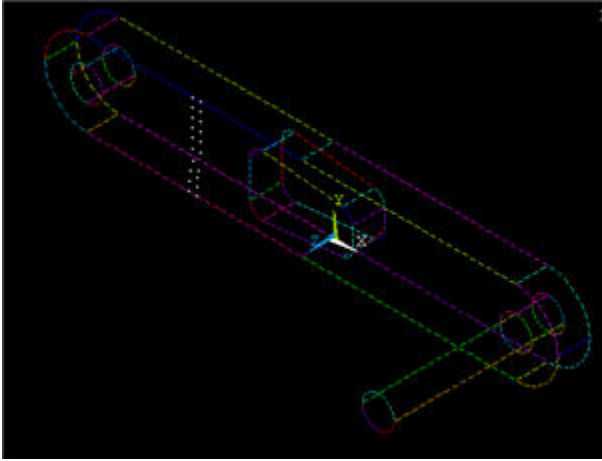
Save your work:**Toolbar > SAVE_DB**

Next, from the current set of nodes, select nodes that satisfy the following criterion: 0.45" y 0.85" by appropriately modifying the previous select action step. The snapshot below shows what I get: four nodes remain in the selected set. If you mess up, resume from your *.db* file.
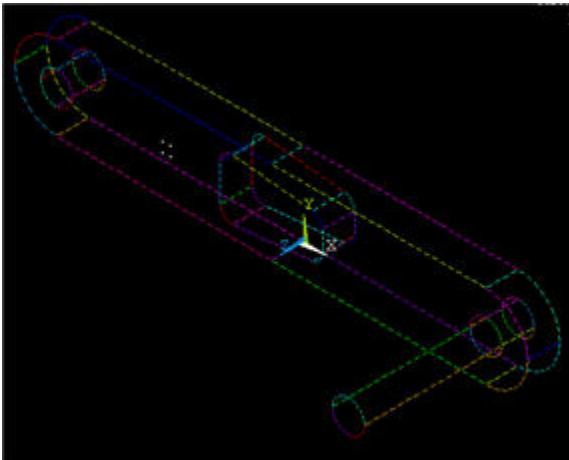
Now all actions on nodes will be performed only on the four nodes that are currently selected. For instance, to list the strain values at these nodes, choose

Main Menu > General Postproc > List results > Nodal Solution > Elastic Strain > X-Component of elastic strain

You can save these values to a text file using **File > Save as**. You can then read in the values from the text file into Excel or MATLAB for further processing such as finding the average.

Once you are done with the select operations, exit the *Select Entities* menu and choose

Utility Menu > Select > Everything

Utility Menu > Plot > Nodes

Save your work:**Toolbar > SAVE_DB**

Go to Step 9: Validate the results
See and rate the complete Learning Module

Go to all ANSYS Learning Modules