ANSYS 11 - Crank Step 9

Step 9: Validate the results

It is **very important** that you take the time to check the validity of your solution. This section leads you through some of the steps you can take to validate your solution.

Simple Checks

Does the deformed shape look reasonable and agree with the applied boundary conditions? We checked this in step 8.

Do the reactions at the supports balance the applied forces for static equilibrium? To check this, select

Main Menu > General Postproc > List Results > Reaction Solu

Select *All struc forc F* for Item to be listed and click *OK*. The forces in the X and Z directions are essentially zero and the total Y-reaction is 100.00 (lbf) as expected.

Refine Mesh

Let's repeat the solution on a finer mesh with more divisions in the z-direction. Repeat the mesh steps for the MESH200 element, but this time use smart size 3 and element size of 0.08. Repeat the mesh steps for the SOLID45 element and set the element edge length to 0.05 instead of 0.125. This will create 10 divisions through the thickness of the crank instead of 4. When warned that the picked volumes are already meshed, check *Yes* and click *OK* to remesh.

Obtain a new solution and plot the elemental solution of the von Mises stress:
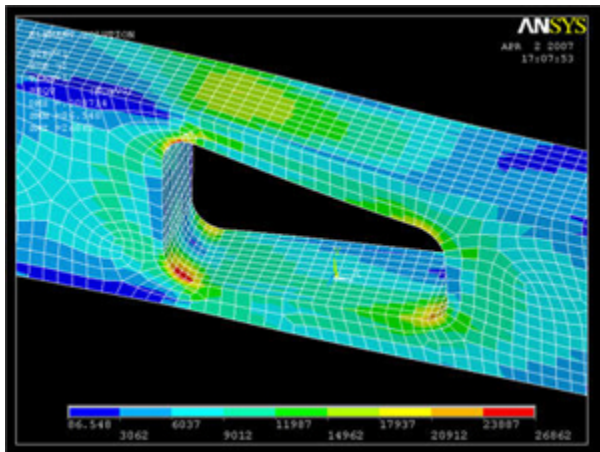
Higher Resolution Image

	Coarser Mesh	Finer Mesh
DMX	0.026148in	0.026651 in
SMX	25308psi	27942psi

The maximum displacement at the tip of shaft is 1.9% greater and the maximum stress is 10% greater. This indicates that the solution we have obtained is still dependent on the mesh. We would need to further refine the mesh. Do keep in mind that one would have to make more detailed comparisons between the solutions on the two meshes before we can make a definitive statement about the mesh independence of our results.

Exit ANSYS

Utility Menu > File > Exit

Select **Save Everything** and click **OK**.