ANSYS - Plate with a Hole - 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 total reaction force in the x-direction is -7000 N.

Applied force = (pressure) x (projected distance in x-direction of the line along which the constant pressure acts) = (p) (r) = 7000 N in positive x-direction.

So the reaction cancels out the applied force in the x-direction. Similarly, you can check that this is true in the y-direction also.

Refine Mesh

Let's repeat the calculations on a mesh with overall element size level under *SmartSize* set to 4 instead of 5 and compare the results on the two meshes. Delete the current mesh:

Main Menu > Preprocessor > Meshing > Mesh Tool

Select ***Clear*** under ***Mesh***: and ***Pick All*** in the *pick* menu. The mesh is deleted.

Set the overall element size level under *SmartSize* to 4 by dragging the slider to the left. Click on ***Mesh*** and ***Pick All***.

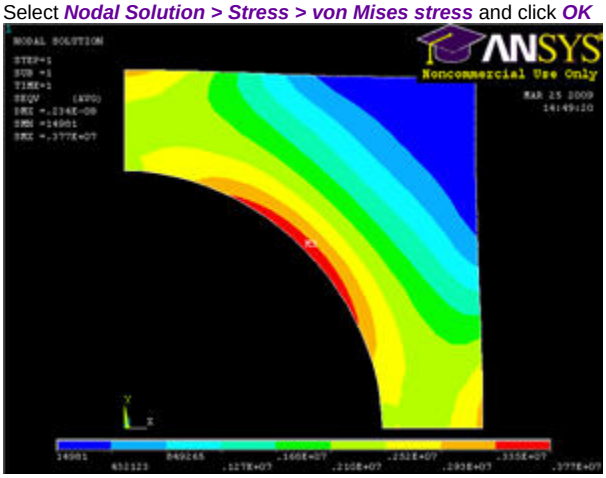
In the *Output* window, check how many elements are contained in this mesh? Your new mesh should have 320 quadrilateral elements.

Obtain a new solution: **Main Menu > Solution > Solve > Current LS**

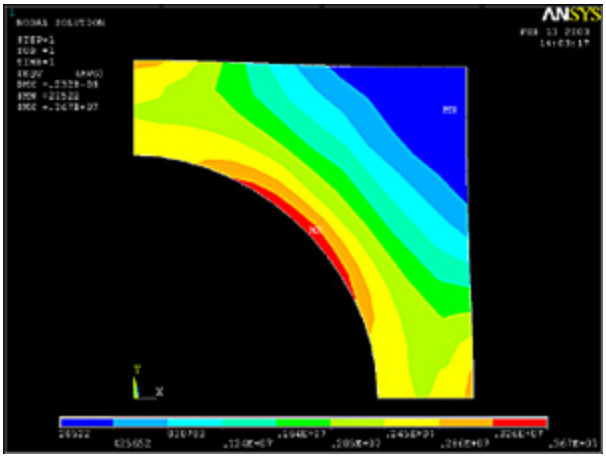
Plot nodal solution of the von Mises stress:

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

Select ***Nodal Solution > Stress > von Mises stress*** and click ***OK***



Compare this with the von Mises contours for the previous mesh:



The two results compare well with the finer mesh contours being smoother as expected. Compare the maximum stress and displacement values:

	Coarser Mesh	Finer Mesh
DMX	0.232e-8m	0.234e-8m
SMX	3.64MPa	3.77MPa

The maximum displacement value changes by less than 1% and the maximum von Mises stress value by less than 3%. This indicates that the meshes used provide adequate resolution.

Exit ANSYS

Utility Menu > File > Exit

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

Reference

Cook, R.D., Malkus, D.S., Plesha, M.E., and Witt, R.J., *Concepts and Applications of Finite Element Analysis*, Fourth Edition, John Wiley and Sons, Inc., 2002.

Go to Problem Set 1

See and rate the complete Learning Module

Go to all ANSYS Learning Modules