Tensile Bar (Results-Interpretation) - Verification and Validation

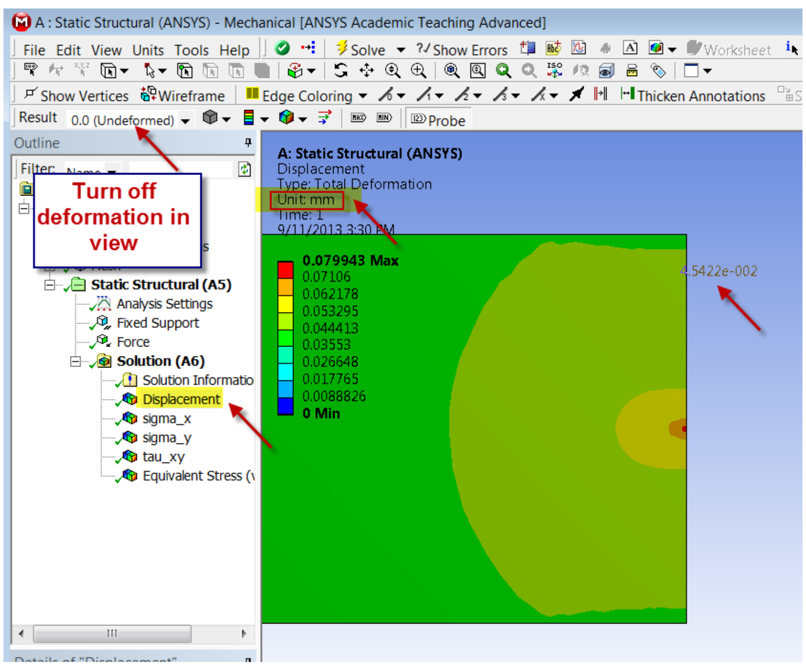
Author: Rajesh Bhaskaran, Cornell University

Verification and Validation

One can think of *Verification and Validation* as a formal process for checking results. Each of these terms has a specific meaning which we won't get into here. We have already done some checks on the ANSYS results by comparing them to the hand calculations and checking that the ANSYS solution agrees with the appropriate traction or displacement boundary condition at each boundary. Let's next check ANSYS's displacement value at the right boundary with the value in our hand calculations.

Check Displacement Value at the Right Boundary

- Bring up the *Displacement* result again by clicking on that object in the tree.
- I prefer to turn off the deformation in the view as per snapshot below.
- Zoom into the right end using the right mouse button.
- Click *Probe* and check the displacement values away from the point load.



I get a value around 0.045 mm at the right end away from the point load. This is about a 10% deviation from the hand calculation result of 0.05 mm we obtained in our *Pre-Analysis*. This is a reasonable agreement considering that the hand calculation ignores the high stress areas at the left and right ends. But these high stress areas (both tensile and compressive) affect relatively small areas of the model and so don't contribute a lot to the overall displacement.

Summary of Our Result Checks

1. The stress components agree well with hand calculations away from the right and left ends.
2. The displacement at the right end (away from the point force) is within about 10% from the hand calculation value.
3. The ANSYS solution agrees with the boundary conditions on traction as well as displacement.

Thus, we can be reasonably confident that the ANSYS model has been set-up correctly. We have however not checked that we have resolved the high stresses at the left and right ends correctly. So we cannot say anything about when the part would fail. Further mesh refinement may be needed. We also should get rid of the stress singularity at the point load (by distributing it over a region) and at the left corners (by filleting these corners).

Go to Exercises

Go to all ANSYS Learning Modules