Thermal Stresses in a Bar - Pre-Analysis & Start-Up

Author: Ben Mullen, Cornell University

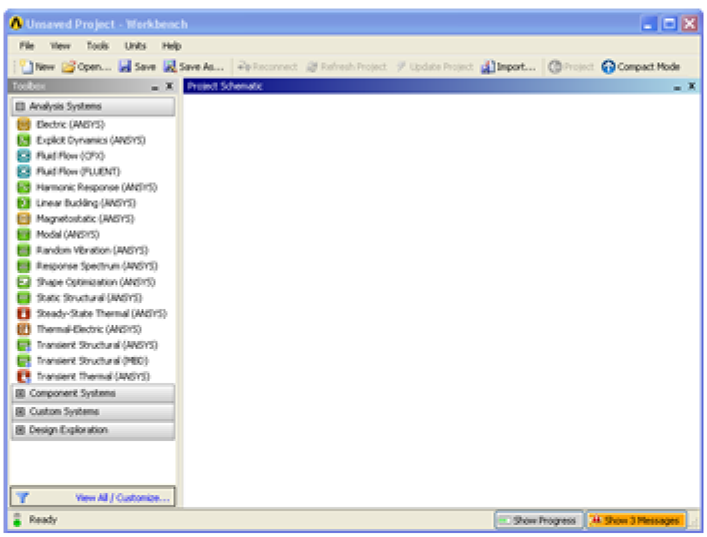
Pre-Analysis & Start-Up

Pre-Analysis

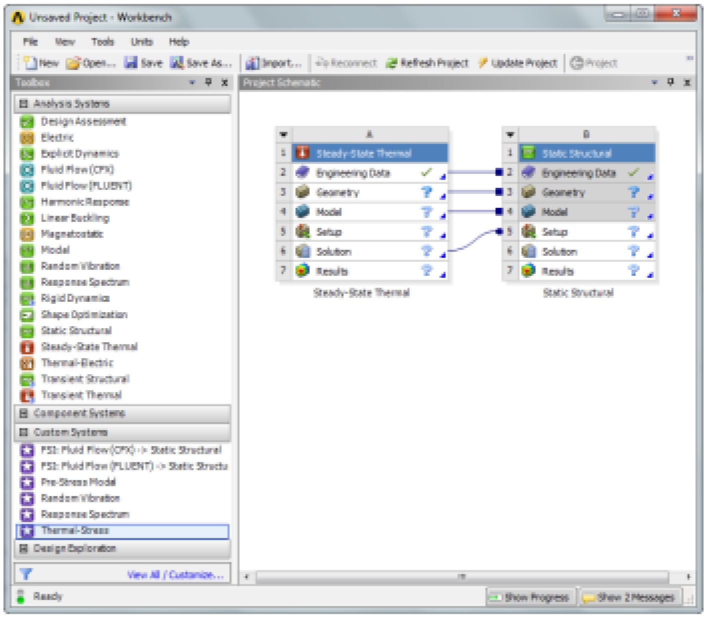
Click here to see the pre-analysis for this problem Remember, it is important to perform some sort of analysis to compare your simulation results to in order to verify your results.

Start-Up

Now that we have the pre-calculations, we are ready to do a simulation in ANSYS Workbench! Open ANSYS Workbench by going to Start > ANSYS > Workbench. This will open the start up screen seen as seen below



To begin, we need to tell ANSYS what kind of simulation we are doing. If you look to the left of the start up window, you will see the Toolbox Window. Take a look through the different selections. Expand the ***Custom Systems*** and find ***Thermal Stress***. Load the ***Thermal Stress*** scenario by double clicking it.



The default material is structural steel, which we will use to make the geometry in ANSYS.

Go to Step 2: Geometry

Go to all ANSYS Learning Modules