ANSYS WB - Plate With a Crack - Pre-Analysis & Start-Up

Author: Benjamin Mullen, Cornell University

Pre-Analysis and Statup

Pre-Calculations

Click here to view the Pre-Calculations pertaining to this geometry. Remember, it is very important to do some sort of pre-calculation before doing any sort of computational simulation; otherwise, you may get incorrect results from the simulation and you will have nothing to compare the results to.

Open ANSYS Workbench

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
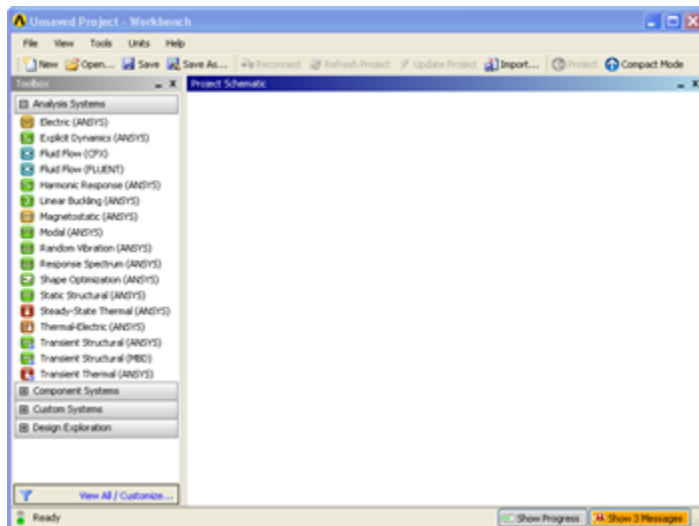
To begin, we need to tell ANSYS what kind of simulation we are doing. If you look to the left of the start up window, you will see the Toolbox Window. Take a look through the different selections. The plate with a hole is a static structural simulation. Load the static structural tool box by dragging and dropping it into the Project Schematic.
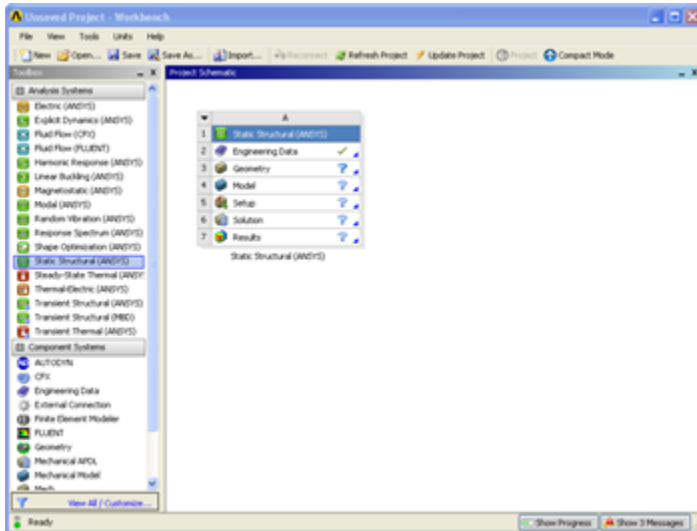
Name the Project "Plate with a Crack".

Continue to