Wind Turbine Blade - Pre-Analysis & Start-Up

Author: Ben Mullen, Cornell University

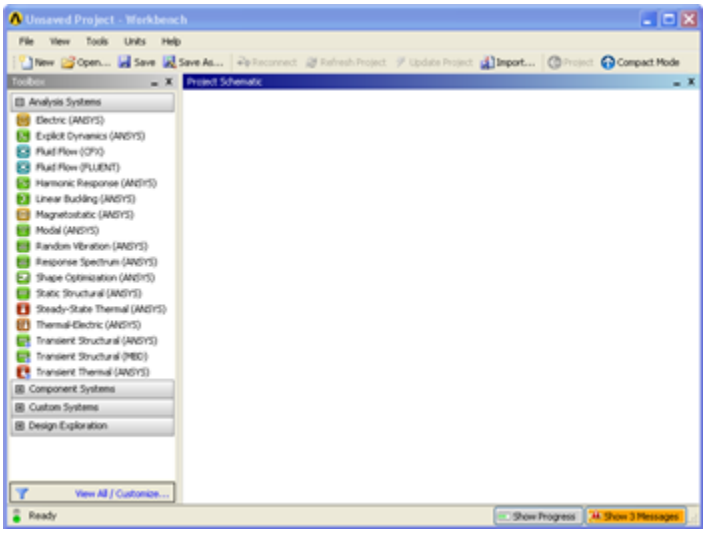
Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation
Exercises
Comments

Pre-Analysis and Start-Up

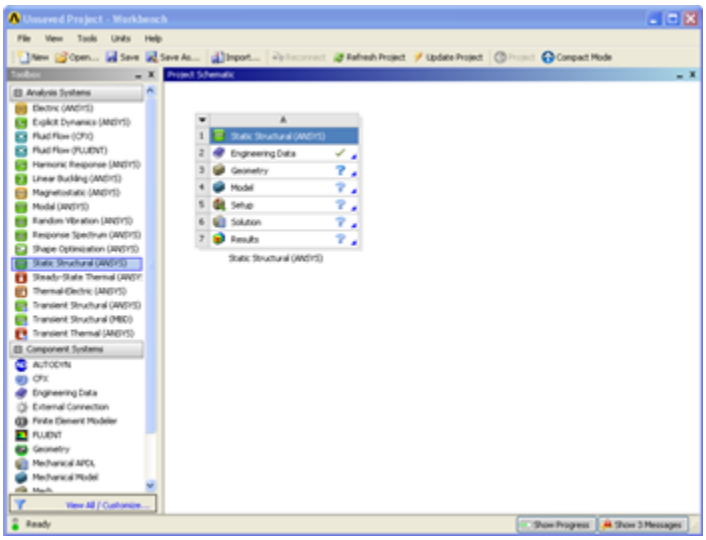
This tutorial is not being updated any more. We recommend that you follow this newer tutorial on fluid-structure analysis of a wind turbine blade. Thank you!

Open ANSYS Workbench

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. Because we are only doing a force loading, we will be doing a Static Structural simulation. Load the ***Static Structural*** tool box by dragging and dropping it into the Project Schematic.



Name the Project `Wind Turbine Blade` by doubling clicking *Static Structural (ANSYS)* underneath the project schematic.

Go to Step 2: Geometry

Go to all ANSYS Learning Modules