Modal Analysis of a Satellite - Pre-Analysis & Start-Up

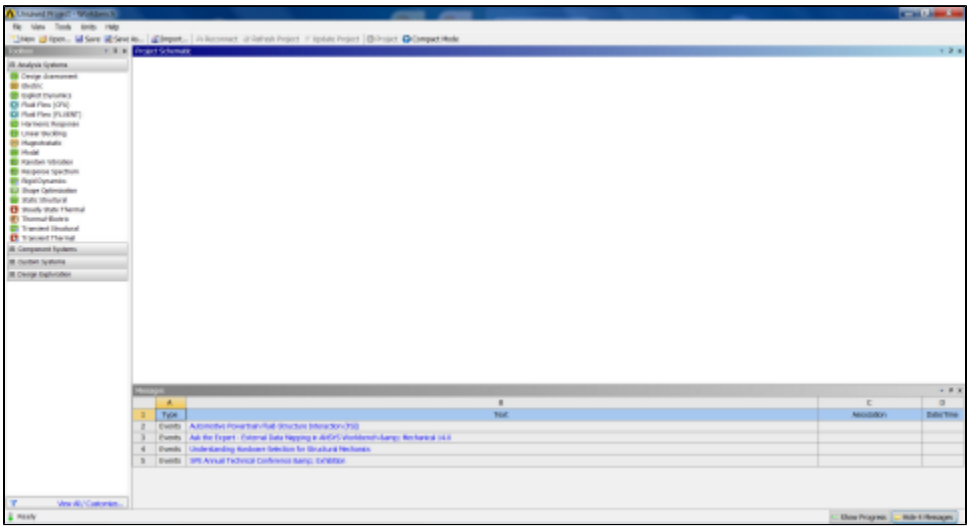
Author: Robert McBride, Cornell University

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

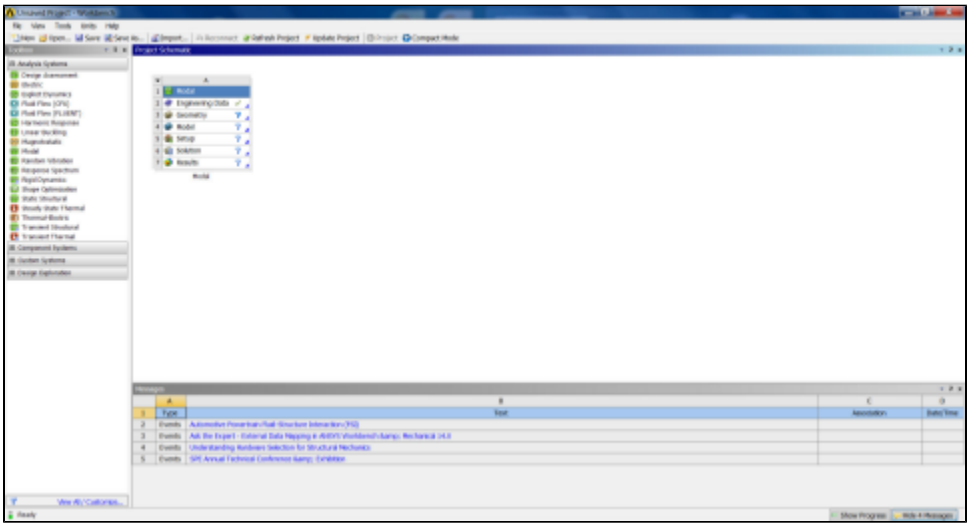
Pre-Analysis & Start-Up

Open ANSYS Workbench

Open ANSYS Workbench by going to Start > ANSYS > Workbench. This will open the start up screen seen as seen below.



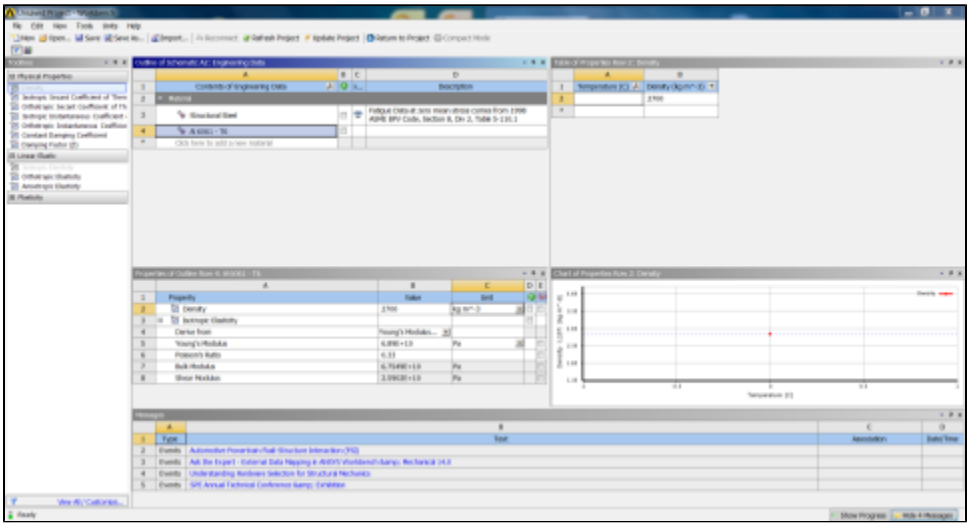
To begin, we need to tell ANSYS what kind of simulation we are doing. If you look to the left of the start up window, you will see the Toolbox Window. Take a look through the different selections. We are doing a modal analysis simulation. Load the ***Modal*** box by dragging and dropping it into the Project Schematic.



Name the project: `Modal Analysis of a Satellite`.

Engineering Properties

Now we need to specify what type of material we are working with. Double click ***Engineering Data*** and it will take you to the Engineering Data Menus.



If you look under the *Outline of Schematic A2: Engineering Data* Window, you will see that the default material is Structural Steel. The Problem Specification states we will be using Aluminum 6061-T6. To add a new material, click in an empty box labeled ***Click here to add a new material*** and give it a name. Our Material is `Al 6061-T6`. On the left hand side of the screen, in the *Toolbox* window, expand ***Linear Elastic*** and double click ***Isotropic Elasticity*** to specify the Elastic Modulus and Poisson's Ratio.

In the *Properties of: Al 6061-T6* window, Set the Elastic Modulus units to ***Pa***, set the magnitude as 68.9e9, and set the Poisson Ratio to `0.33`.

We need to define the density as well. Expand ***Physical Properties*** and double click ***density.*** In the *Properties of: Al 6061-T6* window, a density bar will have appeared. Define it as being 2700 kg/m^3.

Go to Step 2: Geometry

Go to all ANSYS Learning Modules