ANSYS WB - Bike Crank - Pre-Analysis & Start-Up

Author: Benjamin Mullen, Cornell University

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

Pre-Analysis & Start-up

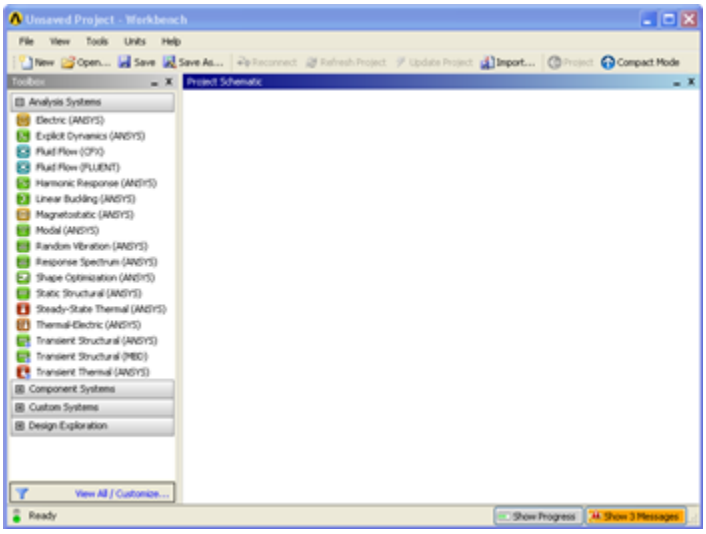
Hand Calculations

It is highly recommended that one perform some sort of hand calculation to predict expected results before attempting a simulation. We call this "Pre-Analysis". The hand calculation can later be used to check the results of the simulation.

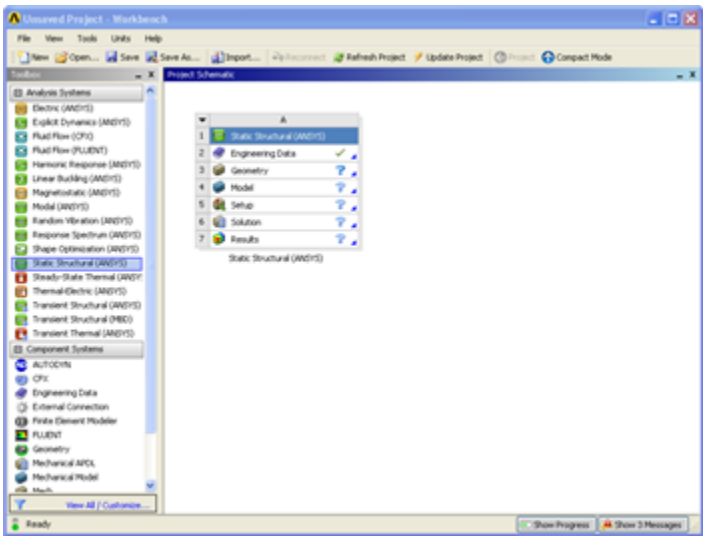
We use Euler-Bernoulli beam theory to calculate the expected strain values at the gage center. Click here to see the "Pre-Analysis" for the bike crank. Note that the hand calculation uses *tensor* shear strain while ANSYS reports *engineering* shear strain (which is two times the tensor shear strain).

Open ANSYS Workbench

Now that we have completed 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



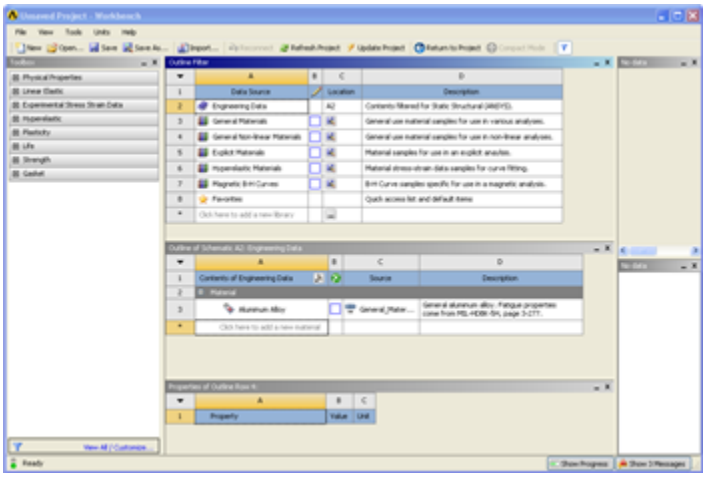
To begin, we need to tell ANSYS what kind of simulation we are doing. If you look to the left of the start up window, you will see the Toolbox Window. Take a look through the different selections. The bicycle crank is a static structural simulation. Load the ***Static Structural*** tool box by dragging and dropping it into the Project Schematic.



Name the Project `Bike Crank` .

Material

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 specifies the material's Modulus of Elasticity and Poisson ratio. 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 E and ν. In the *Properties of Outline Row 4: Al 6061-T6* window, Set the Elastic Modulus units to ***psi***, set the magnitude as `1E7`, and set the Poisson Ratio to `0.33`.

Now that the Material has been specified, we are ready to import the geometry into ANSYS.

Continue to Step 2: Geometry
Go to all ANSYS Learning Modules
See and rate the complete Learning Module