Thermal Stresses in a Bar - Mesh
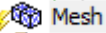
Author: Ben Mullen, Cornell University
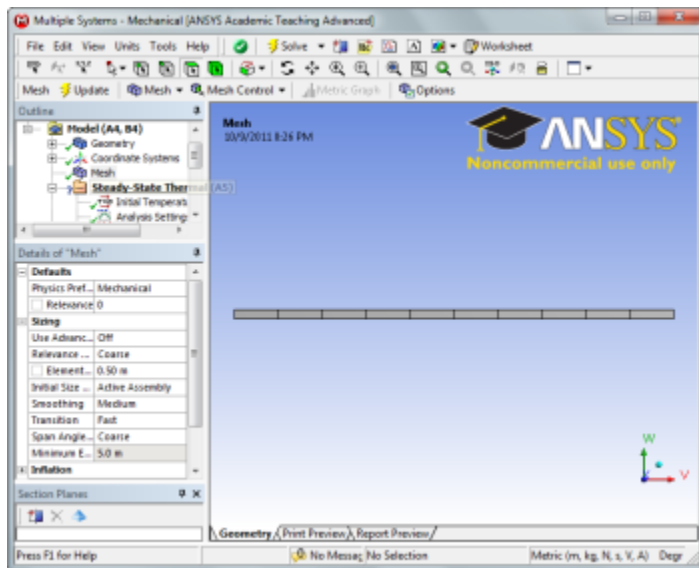
Mesh

Open the Solver

Open the Solver by double clicking `4` **Model**   under the *Steady State Thermal* project.

Set the mesh size

When the solver opens, click on   **Mesh**   in the *Outline* window to open the Mesh menu in the menu bar. In the *Details* window, expand *Sizing*, and change the *Element Size* from `Default` to `0.5 m` (Make sure the units are set to meters by selecting **Units > Metric (m, kg, N, s, V, A)** in the menu bar)

Generate the Mesh

When the sizing has been specified, press **Mesh > Generate Mesh** to create the mesh.

Go to Step 4: Physics Setup

Go to all ANSYS Learning Modules