

Thermal Stresses in a Bar - Mesh

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](#)

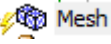
[Comments](#)

Mesh

Open the Solver

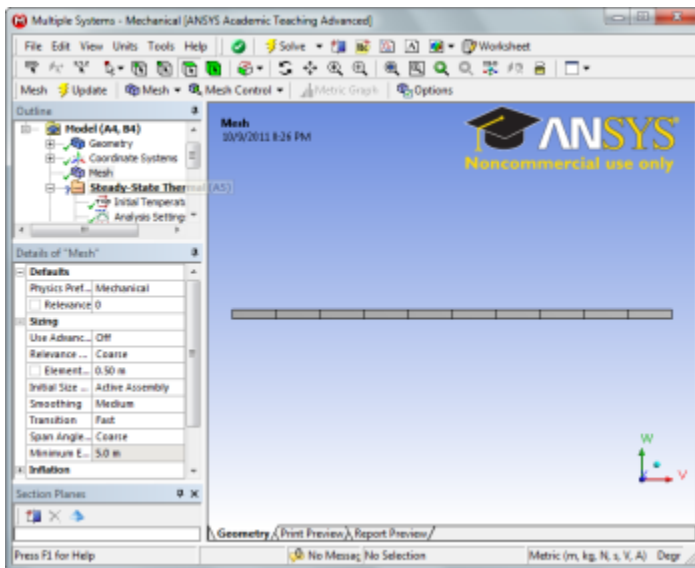
Open the Solver by double clicking  Model  under the *Steady State Thermal* project.

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](#)