

Thermal Stresses in a Bar - Geometry

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

Geometry



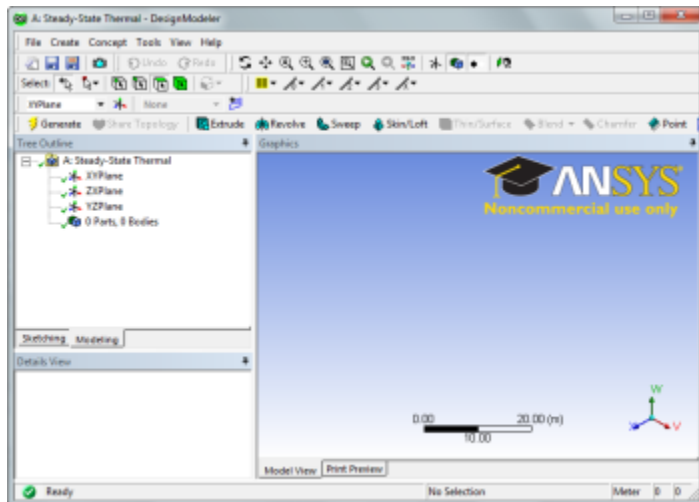
For users of ANSYS 15.0, please check [this link](#) for procedures for turning on the Auto Constraint feature before creating sketches in DesignModeler.

Change the Geometry Properties

In the **Steady-State Thermal** box, right click **Geometry** and select **Properties** . We need to allow ANSYS to recognize line bodies as valid geometries. We accomplish this by checking the box marked **Line Bodies** .

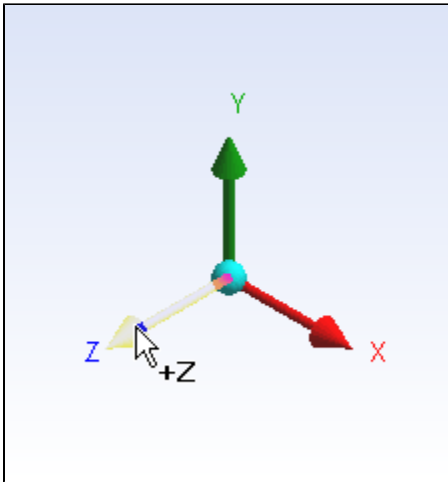
Open the Design Modeler

We are now ready to create the geometry in ANSYS. We will be creating a one dimensional line body to represent the steel bar. To open the design modeler, double click **Geometry** . After the design modeler is launched, you will be prompted on the default units. Select **Meters** and press **OK** .

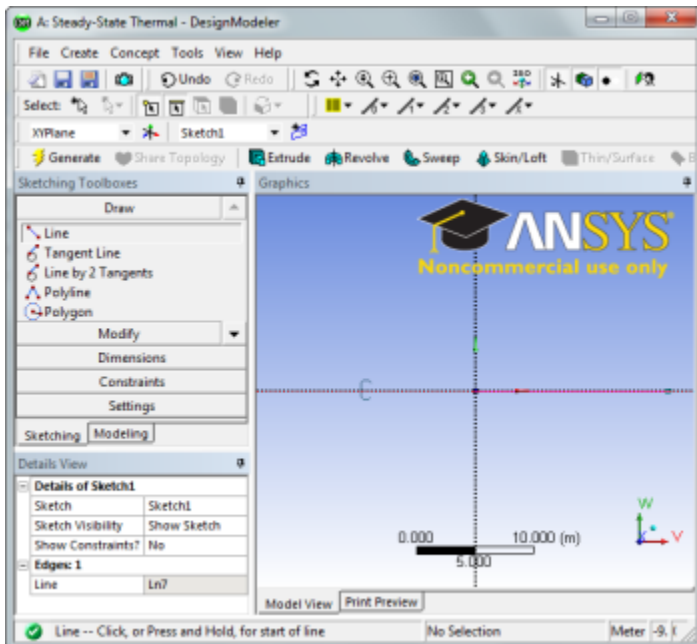


Draw a Line

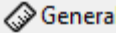
Next, we need to draw a line to represent the length of the bar. To begin sketching, we need to look at a plane to sketch on. Click on the Z-axis of the compass in the bottom right hand corner of the screen to look at the x-y plane.

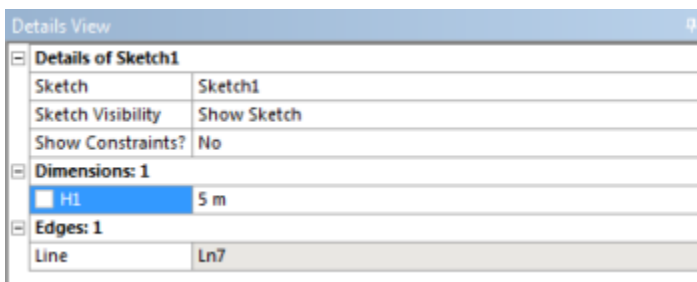


Next, click the **Sketching** tab in the *Outline* window to bring up the sketching menu. Next, select  **Line**. To draw a line, first click the origin, followed by a point on the x-axis.



Dimension the line

Next, we need to assign the length to the line. In the **Sketching Toolbox** click the **Dimensions** tab, and select  **General**. Click on the line to create a dimension. In the *Details* window, change **H1** to 5 meters.

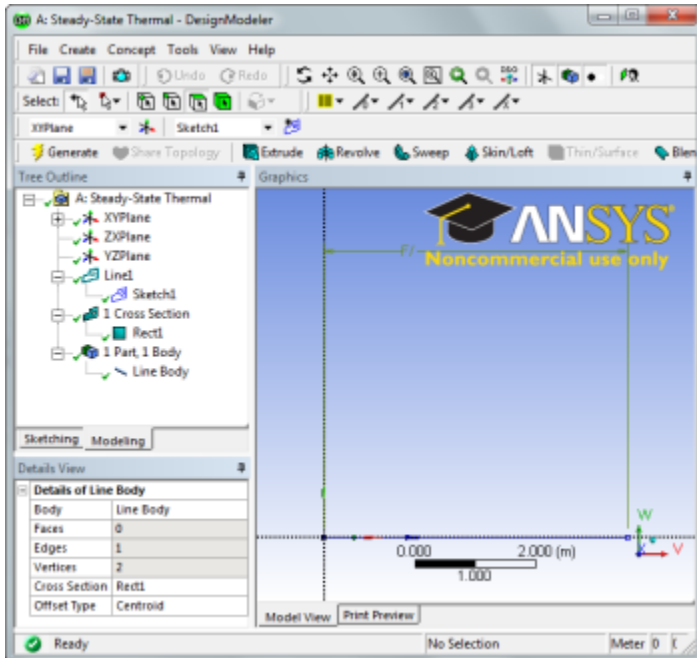


Concept - Line from Sketches

Next, we need to create the line body from the sketch. In the menu bar, click **Concept > Lines from Sketches**. Next, select the line we drew in the *Graphic* window, and in the *Details* window select **Apply**. Finally, press  **Generate** to create the line body.

Specify Cross Section

In the menu bar, go to **Concept > Cross Section > Rectangular** to create a cross section. In the *Details* menu, specify both **B** and **H** to 0.1. After the cross section dimensions are specified, we need to set the cross section to the line body. Expand the **1 Part, 1 Body** and select the **Line Body**. In the *Details* window, change the **Cross Section** to **Rect1**.



You may now close the design modeler.

[Go to Step 3: Mesh](#)

[Go to all ANSYS Learning Modules](#)