

# Thermal Stresses in a Bar - Physics Setup

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](#)

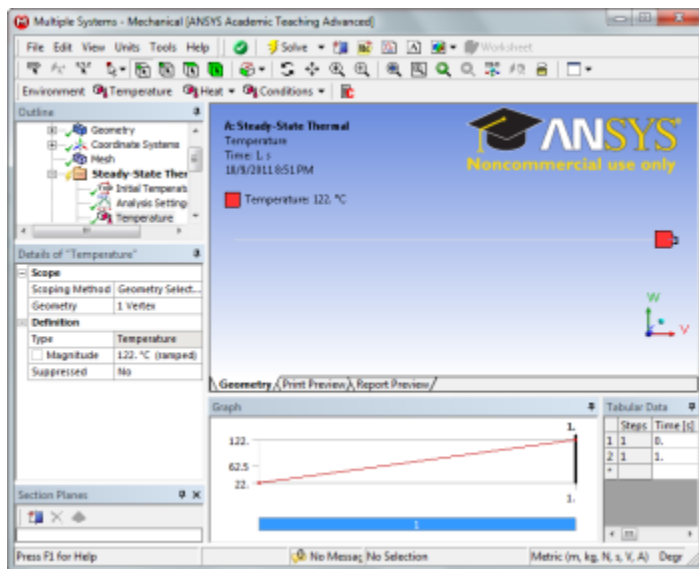
## Physics Setup

### Temperature Boundary Condition

First, under **Steady-State Thermal (A5)** click **Initial Temperature**. In the *Details* window, we can see that the initial temperature is 22 degrees Celsius. We want the temperature change to be 100 degrees Celsius. So, we need to specify a boundary condition of 122 degrees Celsius. Click on



 **Steady-State Thermal (A5)** to open the Thermal Environment menu in the menu bar. In the Thermal Environment menu, click on  **Temperature** to create a temperature essential boundary condition.

Next, we need to select the boundary for the boundary condition. Click the point selection filter  and select the right point of the line body. In the *Details* window, select **Geometry > Apply**. Now, specify the temperature **Magnitude** to 122.



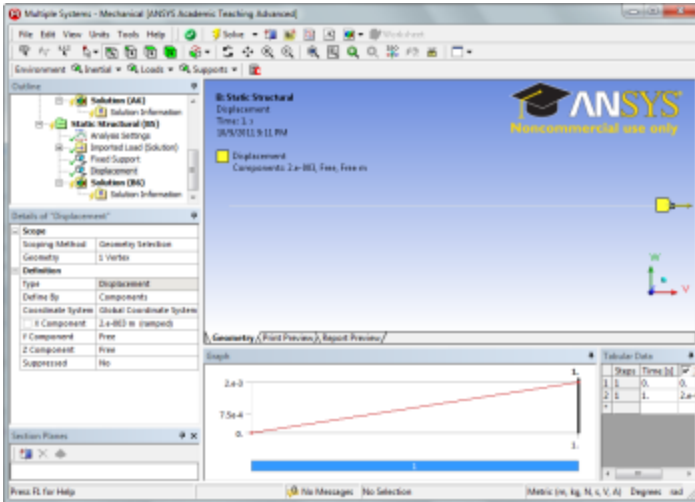
### Elasticity Boundary Conditions

Now, we have to specify the boundary conditions that will affect the elasticity solution to this simulation. In the *Outline* window, click on

 **Static Structural (B5)** to bring up the structural environment menu bar. In the structural environment, select **Supports > Fixed Support**. Ensuring that the point selection filter  is selected, select the left side of the bar, and in the details window select **Geometry > Apply**.

Details of "Fixed Support"	
[-] <b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	1 Vertex
[-] <b>Definition</b>	
Type	Fixed Support
Suppressed	No

Next, we need to specify a displacement constraint to represent the gap in the wall. In the Structural Environment menu, select **Supports > Displacement**. In the *Details* window, change the **X Component** from **Free** to **0.002 m**. Also, select the right point of the bar using the point selection filter, and select **Geometry > Apply**.



[Click here to enlarge](#)

There should now be 3 boundary conditions on the bar. The left end should be fixed, and the right end should have a temperature boundary condition and a displacement boundary condition.

**[Go to Step 5: Numerical Solution](#)**

[Go to all ANSYS Learning Modules](#)