

# AIM Heat Conduction in a Bar - Physics Set-Up

Author(s): Sebastian Vecchi, ANSYS Inc

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Results](#)

## Physics Set-Up

### Specify Material

Select the **Physics** task in the workflow. Press **Material Assignments** under **Physics Definition**. AIM should detect the body in question and highlight it. Change the **Material** to **Aluminum Alloy**. The entire object is now made of an aluminum alloy and we can add our boundary conditions.

### Physics Conditions

The temperatures for both sides can be input as **Solid Thermal Conditions**. Follow the highlighted blue **Next Step** button to add these temperatures via **Add > Solid Thermal Conditions > Temperature**. Another way of adding temperature is to press the **Physics** button in the workflow and press **Solid Thermal Conditions > Add > Temperature**. Select the appropriate faces and assign them the respective temperatures. Once you have selected a face, in order to assign it a temperature you must press the blue **+** button and then input the correlating temperature. The room temperature surface was said to be 80 degrees Fahrenheit and the heated surface was said to be 400 degrees Fahrenheit. Below is an example of what the bar will look like with the temperature conditions defined.

Press **Physics** in the workflow below to return to the main Physics display and press **Solver Options > Output Controls 1 > Output type > All**. Do this for both **Output Specification 1** and **Output Specification 2**.

Return to the **Physics** task and press the blue **Solve Physics** button in order to calculate the solution.

**Go to Step 5: Results**

[Go to all ANSYS AIM Learning Modules](#)