

# ANSYS AIM - Heat Conduction in a Bar

Author(s): Sebastian Vecchi, ANSYS Inc

## [Problem Specification](#)

- [1. Pre-Analysis & Start-Up](#)
- [2. Geometry](#)
- [3. Mesh](#)
- [4. Physics Setup](#)
- [5. Results](#)

## Heat Conduction in a Bar

Created using ANSYS AIM 18.1

### Problem Specification

Consider the rectangular bar pictured below with a length of 120 inches and a height and width of 5 inches. One end of the bar is subjected to heating and can be assumed that said end is at 400 degrees Fahrenheit. The other end of the bar is exposed to the environment which is assumed to be at room temperature (80 degrees Fahrenheit).

In this tutorial, we will utilize ANSYS AIM to find the temperature throughout the bar and total heat flux.

[Go to Step 1: Pre-Analysis & Start-Up](#)

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