

AIM Heat Conduction in a Bar - Results

Author(s): Sebastian Vecchi, ANSYS Inc

[Problem Specification](#)

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

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Results](#)

Results

Press the **Results** button in the [Workflow](#) to extract information from the simulation that was done. In order to find information that can be readily used, first press **Evaluate Results**. Once the evaluation is complete, AIM will automatically output a contour in the Results section under **Objects**. This contour is pictured below and shows the temperature of the bar in a steady state. We can adjust the coloring by going to **Appearance > Color distribution** and in the drop down menu selecting **Logarithmic**. Try it yourself and compare the differences.

In order to get the total heat flux, [return to the Results task and press Add > Contour](#). A new panel will appear and you will be asked to input a variable. In the drop down menu, find **Thermal > Heat Flux Magnitude**, then press **Evaluate**. A contour will generate much like the one below. Notice that the minimum and maximum values are essentially the same, indicating a uniform heat flux along the length of the bar. The color variations indicate very small fluctuations in the numerical results.

In order to calculate the sum heat flow reaction of the simulation, [return to the Results task, click on Add next to Results, select Heat Flow Reaction, and press Evaluate](#). Automatically, a calculated value will be created with the corresponding value displayed in the panel and graphics window.

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