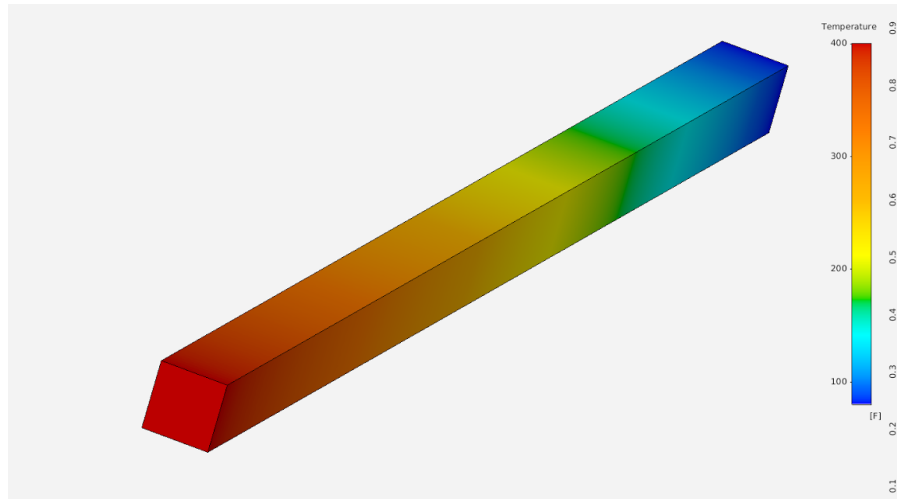


ANSYS AIM Tutorial

Heat Conduction in a Bar

Author(s): Sebastian Vecchi, ANSYS
Created using ANSYS AIM 18.1



[Problem Specification](#)

[Pre-Analysis & Start Up](#)

[Equations](#)

[Start-Up](#)

[Geometry](#)

[Make a square](#)

[Create Bar](#)

[Mesh](#)

[Set Mesh Size](#)

[Generate Mesh](#)

[Physics Set-Up](#)

[Specify Material](#)

[Add Solid Thermal Conditions](#)

[Solution/Results](#)

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.

Pre-Analysis & Start Up

Equations

The governing equation for heat transfer rate for a rectangular bar, as generalized by Fourier in 1807, is the following equation. In this equation, k is the proportionality factor as a function of material and temperature, A is the cross-sectional area and L is the length of the bar.

$$Q = kA \frac{(T_A - T_B)}{L} = -kA \frac{(T_A - T_B)}{L} = -kA \frac{dT}{dx}$$

The equation above can be written in terms of heat flux using the definition that heat flux is the amount of heat transfer per unit area. This one-dimensional form of Fourier's law of heat conduction is found below.

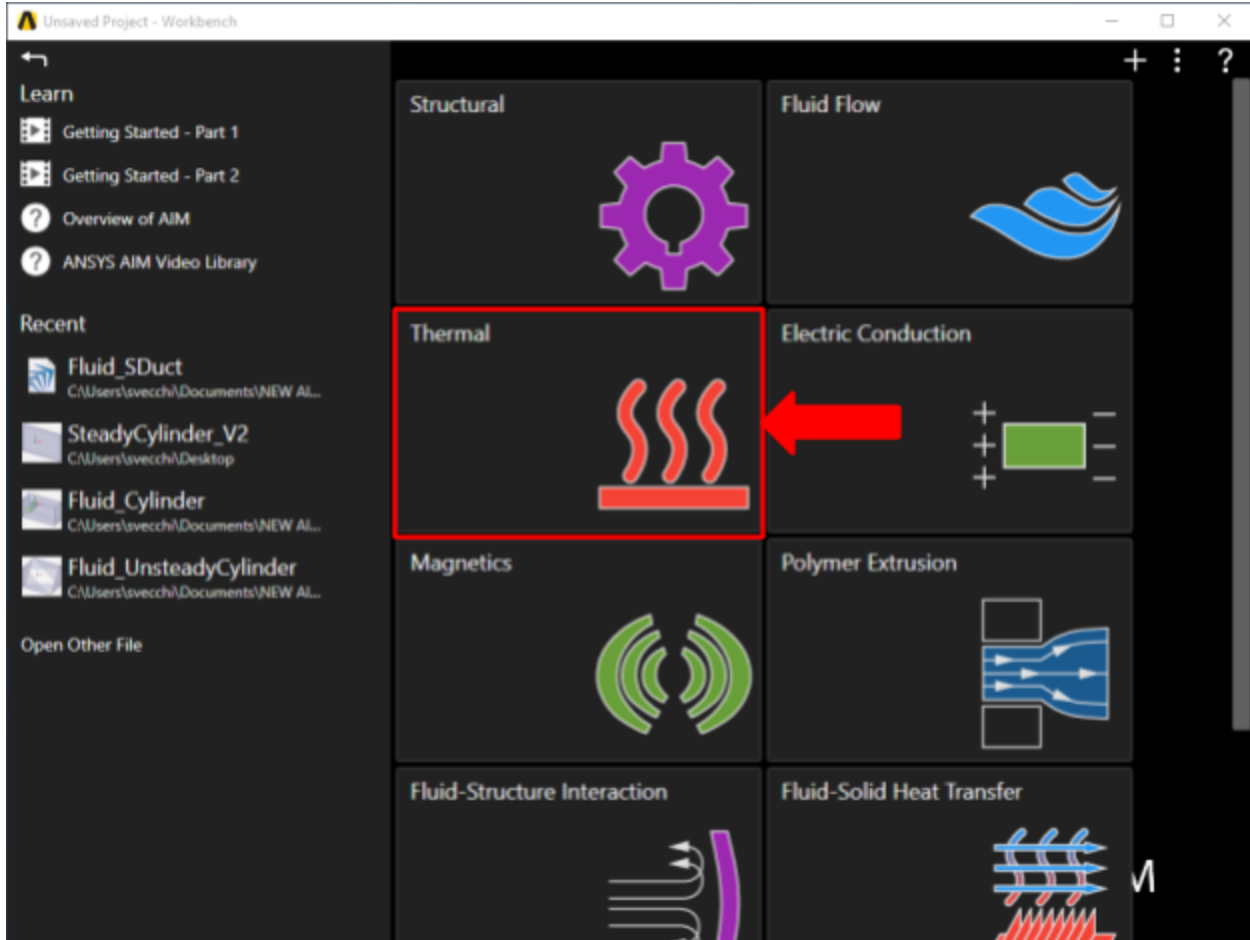
$$\dot{q} = -k \frac{dT}{dx}$$

A few words on the formatting on the following instructions:

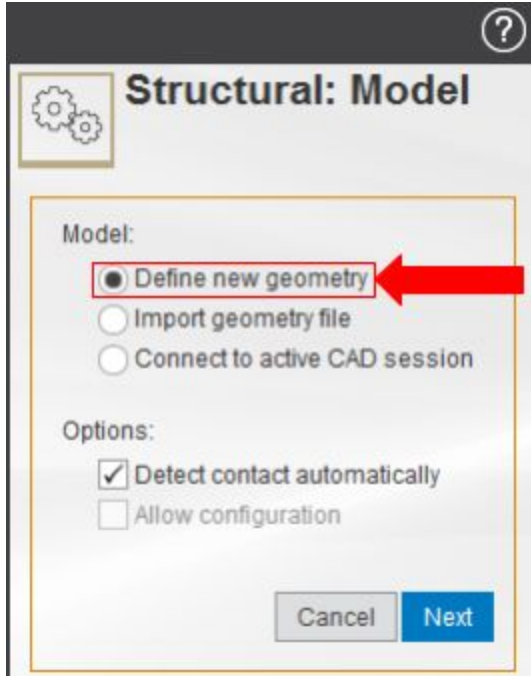
- 1) Notes that require you to perform an action are colored in blue
- 2) General information is colored in black, but does not require any action
- 3) Words that are **bolded** are labels for items found in ANSYS AIM
- 4) Most important notes are colored in red

Start-Up

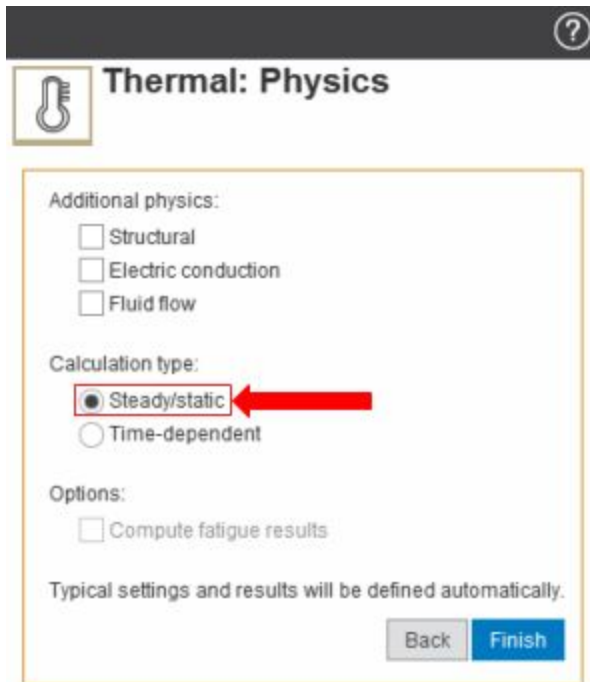
Now that we have the pre-calculations, we are ready begin simulating in ANSYS AIM. [Open ANSYS AIM by going to Start > All Apps > ANSYS 18.1 > ANSYS AIM 18.1](#). Once you are at the starting page of AIM, [select the Thermal template](#) as shown below.



You will be prompted by the **Thermal** template to either **Define new geometry**, **Import geometry file**, or **Connect to active CAD session**. Select **Define new geometry** and press **Next**.



For this problem, we will be using the default **Steady/static** calculation type. Press **Finish**. No additional physics are necessary.

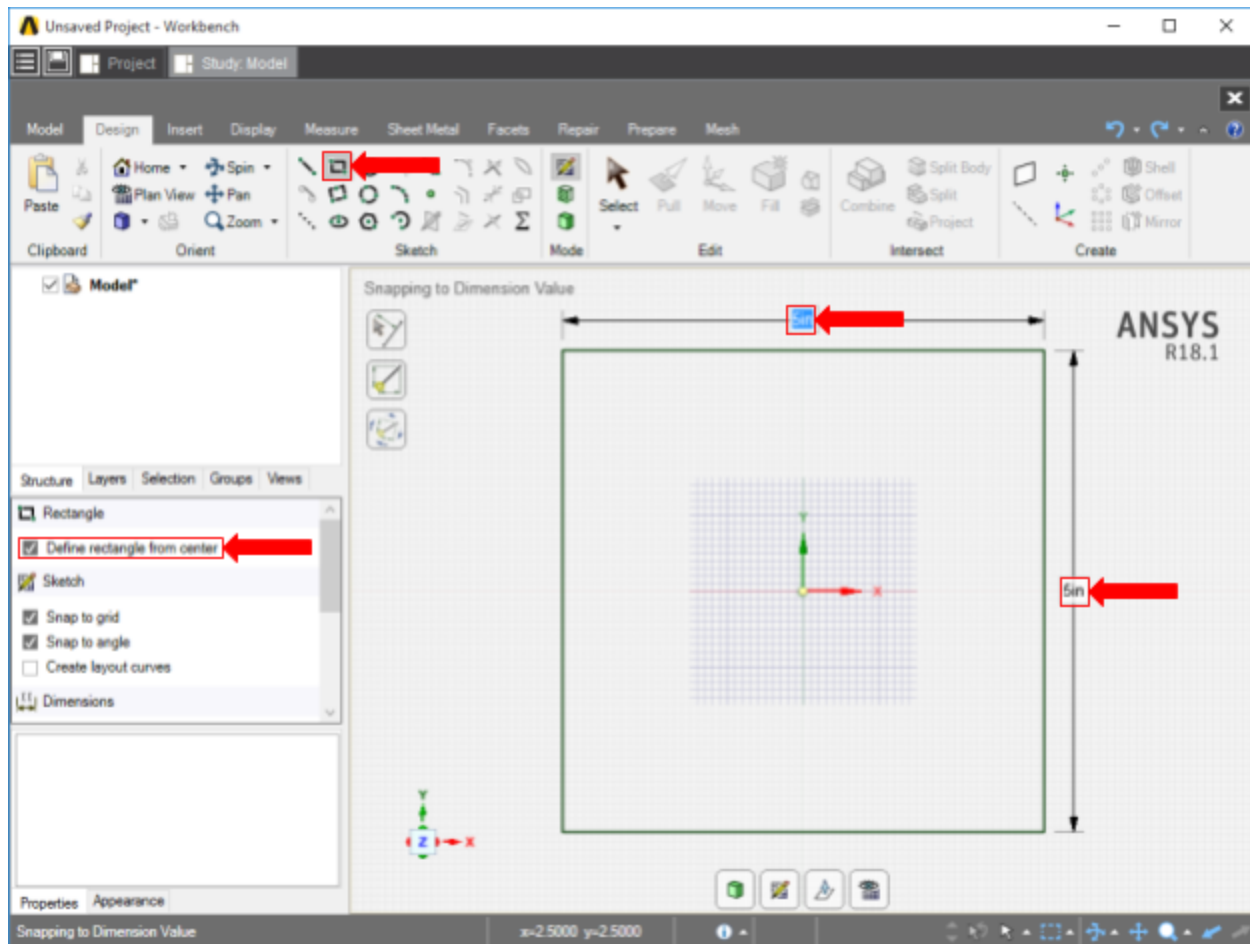


The **Model Editor** will launch automatically. In order to use the units given to us in the problem, press the **Project** button in the top left corner and select **Units > US Customary**.

Geometry

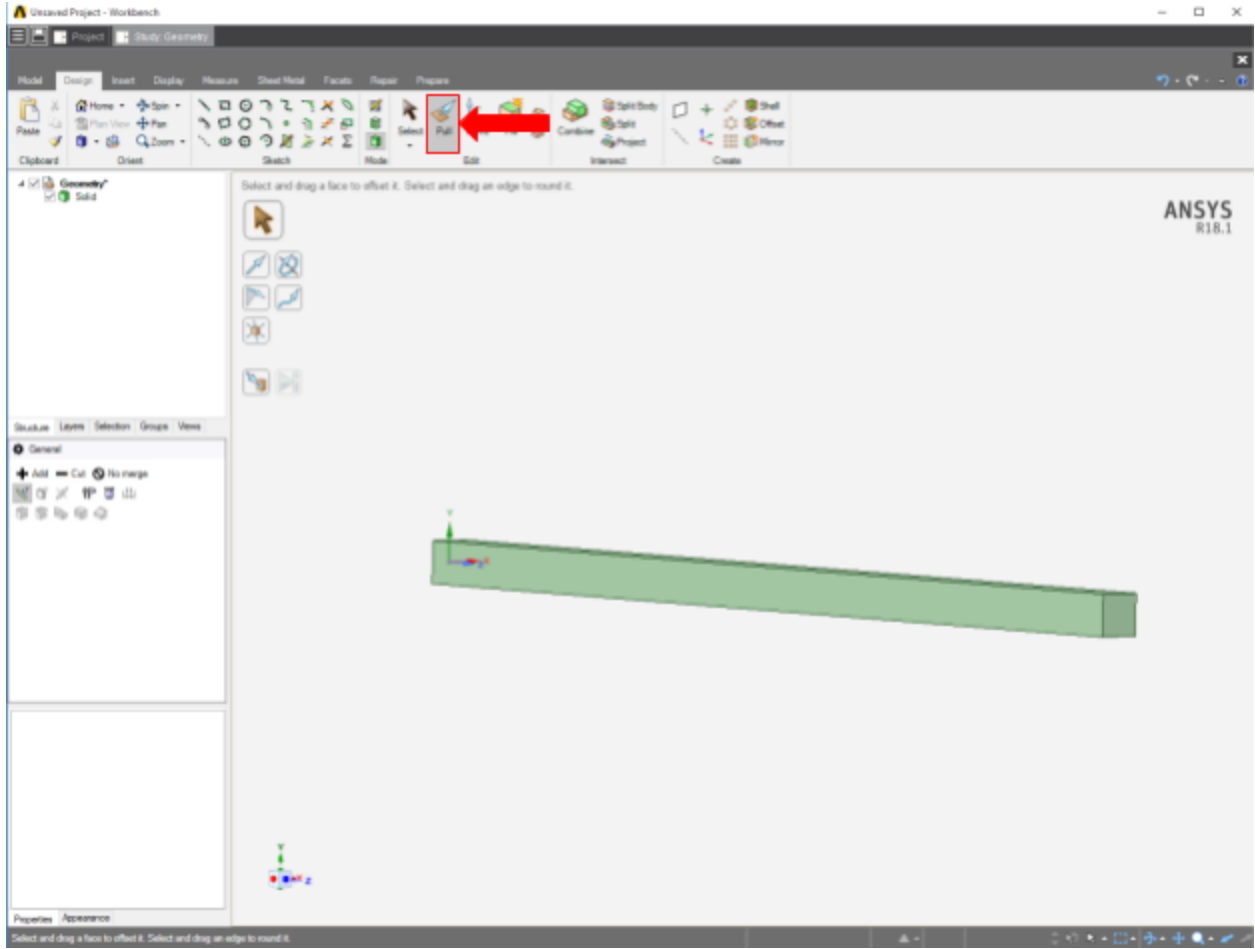
Make a square

Click on the **Z-axis** on the compass in the bottom left corner of the screen to look at only the XY-plane. Right click in the empty white space and choose **Select New Sketch Plane**, then left click on the grid that appears, so that the plane we are building on will be the XY-plane. Choose the **Rectangle** drawing tool in the **Sketch** subgroup of the **Design** Tab and click on **Define rectangle from center** in the options at the left. Select the origin and drag out the rectangle until it is the correct size, or dimension the rectangle using the highlighted boxes, pictured below, that appear as you are making the rectangle. **The height and width are 5 inches as defined by the problem.**



Create Bar

Use the **Pull** feature in the **Edit** subgroup and extrude the bar to 120 inches. The geometry is now complete and we can continue to meshing.



Mesh

Once you have exited the modeling window, initiate the meshing process by clicking on **Mesh** in the workflow.

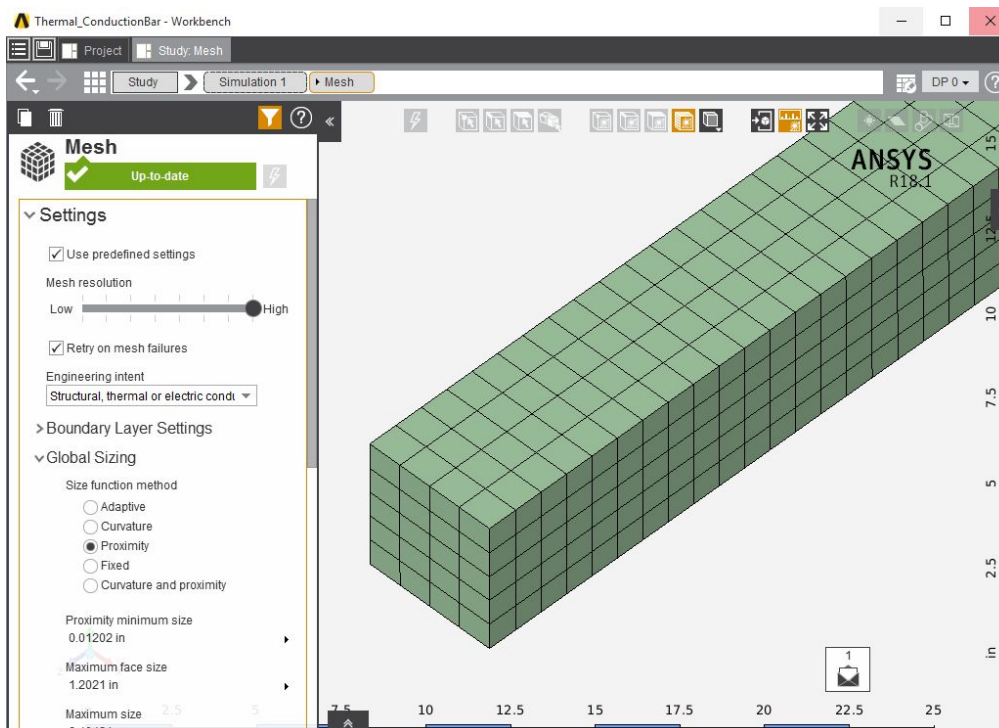


Set Mesh Size

Using the **Mesh resolution** slider, we can edit how precise our mesh is for our calculations. Move the slider all the way to **High** so that we can get the best possible data. Under **Global Sizing**, select the **Proximity** option for the **Size function method**.

Generate Mesh

Click **Generate Mesh** under **Output** or at the top of the screen by the status window for **Mesh**. AIM should detect you are ready to generate the mesh and highlight the buttons in blue.



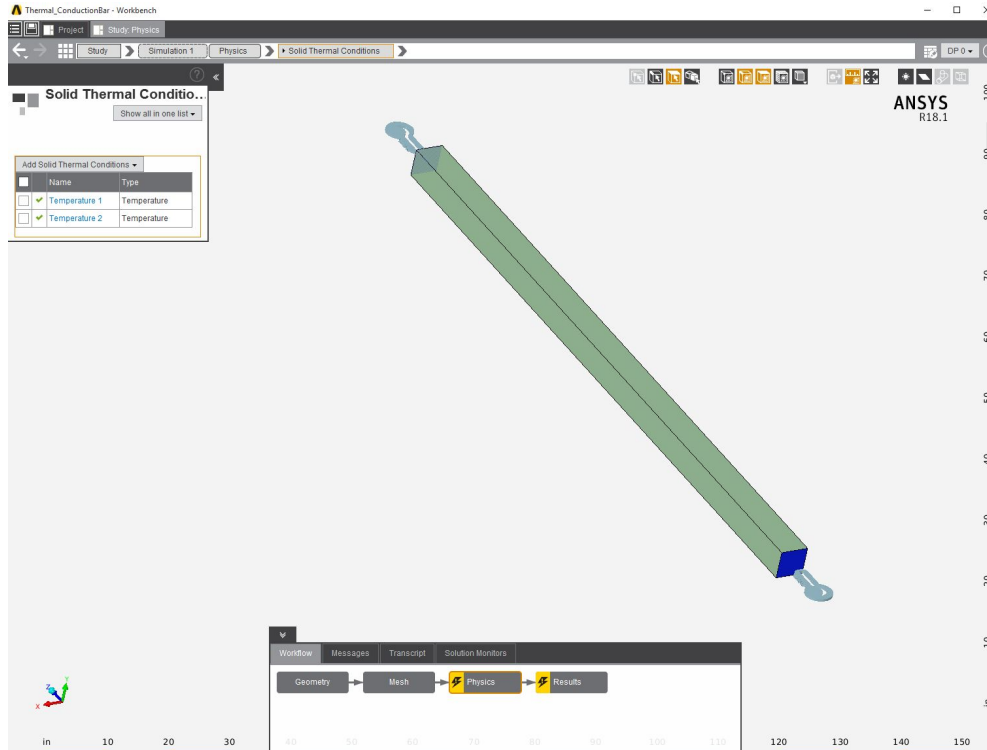
Physics Set-Up

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.



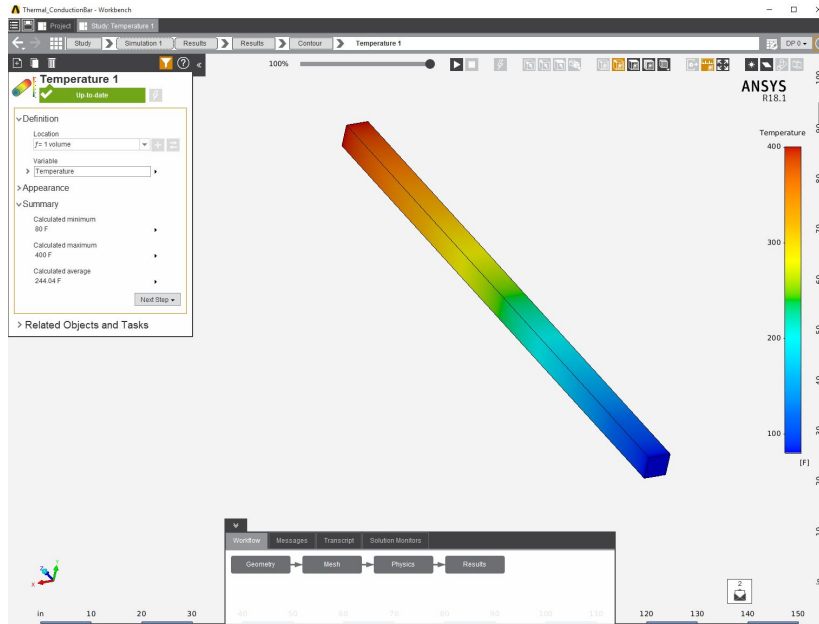


Press **Physics** in the workflow below to return to the main Physics display and **press Solver Options > Output Controls 1 > Output type > All**. Do this for both **Output Specification 1** and **Output Specification 2**.

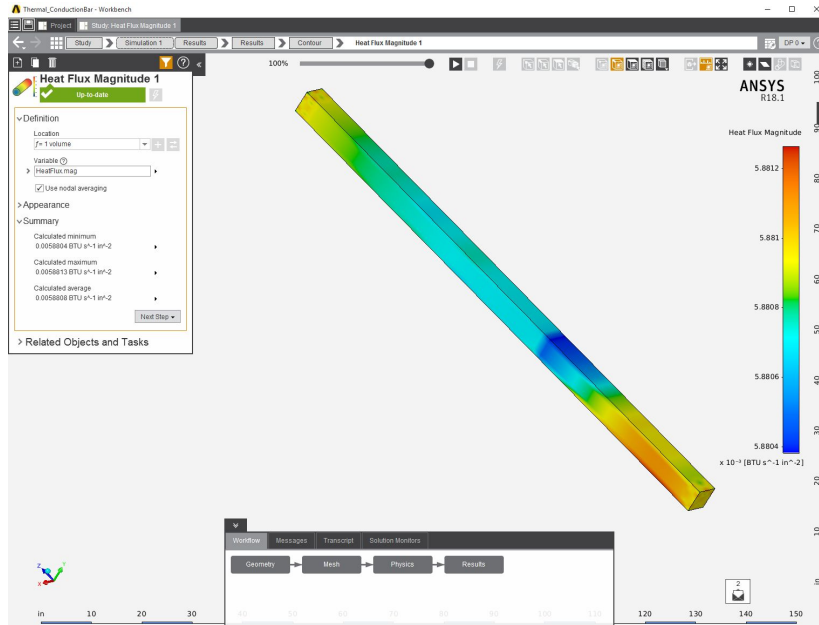
Return to the **Physics** task and press the blue **Solve Physics** button in order to calculate the solution.

Solution/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.

