AIM Heat Conduction in a Bar - Pre-Analysis

Author(s): Sebastian Vecchi, ANSYS Inc

Pre-Analysis

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.

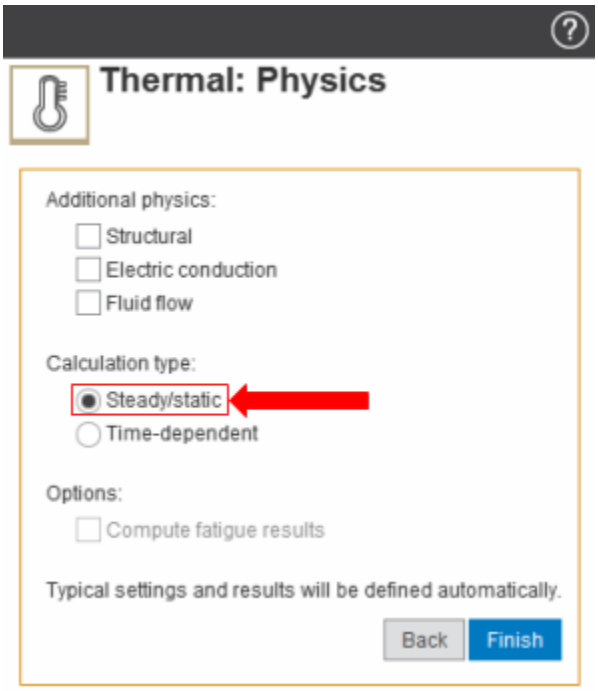
$$q'' = -k\frac{dT}{dx}$$

A few words on the formatting on the following instructions:

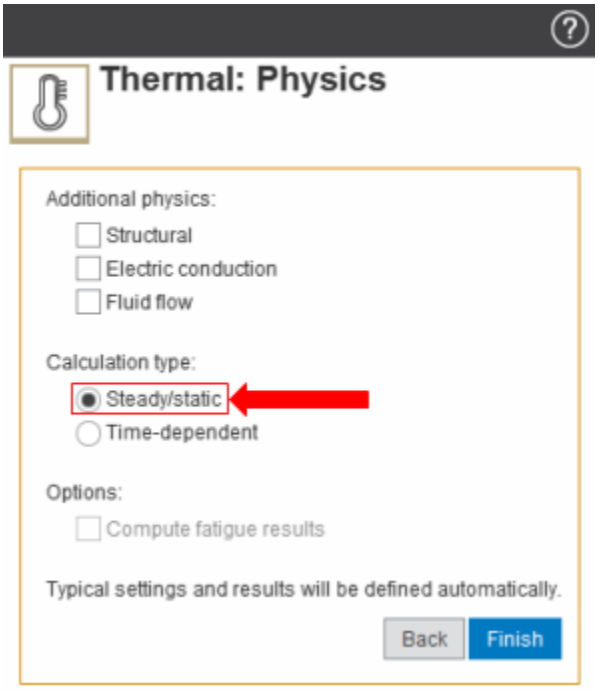
1. Notes that require you to perform an action are colored in blue
2. General information will be colored in black, but do not require any action
3. Words that are **bolded** are labels for items found in ANSYS AIM
4. Most important notes will be colored in red

Start-Up

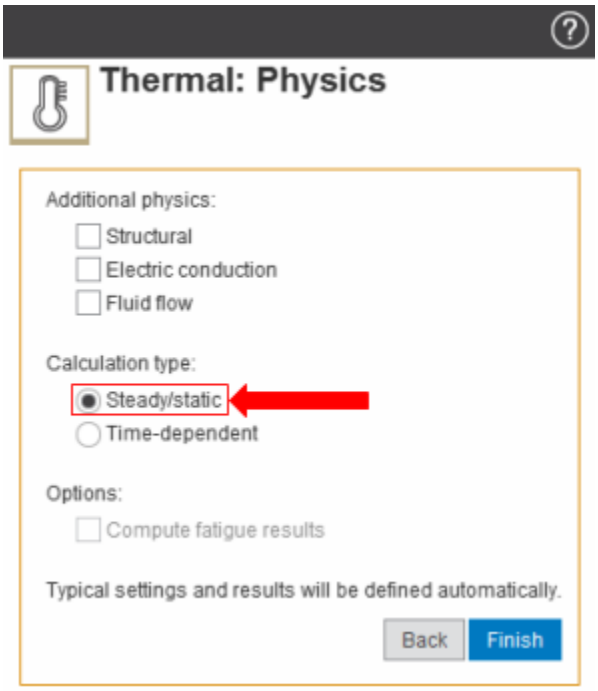
Now that we have the pre-calculations, we are ready begin simulating in ANSYS AIM. Open ANSYS AIM by going to **Start** > **All Apps** > **ANSYS 18.1** > **ANSYS AIM 18.1**. Once you are at the starting page of AIM, select the **Thermal** template as shown below.



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**.

Go to Step 2: Geometry

Go to all ANSYS AIM Learning Modules