

AIM Heat Conduction in Hollow Cylinder - Pre-Analysis

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results

Pre-Analysis & Start-Up

Pre-Analysis

The governing equation for axisymmetric radial heat flow for a homogeneous cylinder with inner radius r_i and outer radius r_o is displayed below. Note that the following equation assumes that the cylinder is long enough for end effects to be ignored.

In the above equation k is the thermal conductivity, A is the surface area, T is the temperature, r is the radial position and Q is the heat generation per unit area.


For the given problem there is no heat generation, thus the governing equation can be solved easily. The solution for temperature as a function of radial position is displayed below.


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](#) in the top right corner as shown below.



 **Thermal: Physics**


Additional physics:

☐ Structural

☐ Electric conduction

☐ Fluid flow

Calculation type:

☒ **Steady/static** 


☐ Time-dependent


Options:

☐ Compute fatigue results

Typical settings and results will be defined automatically.

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 a [Steady/static](#) calculation type so select it and press **Finish**. No addition physics are necessary. The Model Editor will launch automatically. In order to use the units given to us in the problem [press the Home button in the top left corner](#) and select [Units > U.S. Customary](#).



 **Thermal: Physics**


Additional physics:

☐ Structural

☐ Electric conduction

☐ Fluid flow

Calculation type:

☒ **Steady/static** 

☐ Time-dependent

Options:

☐ Compute fatigue results

Typical settings and results will be defined automatically.



Thermal: Physics

Additional physics:

- ☐ Structural
- ☐ Electric conduction
- ☐ Fluid flow

Calculation type:

- ☒ Steady/static
- ☐ Time-dependent

Options:

- ☐ Compute fatigue results

Typical settings and results will be defined automatically.

Back

Finish

[Go to Step 2: Geometry](#)

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