

Heat Conduction in a Cylinder

Introduction

The following example illustrates how to build and solve a conductive heat transfer problem using the Heat Transfer interface. The model, taken from a NAFEMS benchmark collection, shows an axisymmetric steady-state thermal analysis. As opposed to the NAFEMS benchmark model, we use the temperature unit kelvin instead of degrees Celsius for this model.

Model Definition

The modeling domain describes the cross section of a 3D solid as shown in [Figure 1](#).

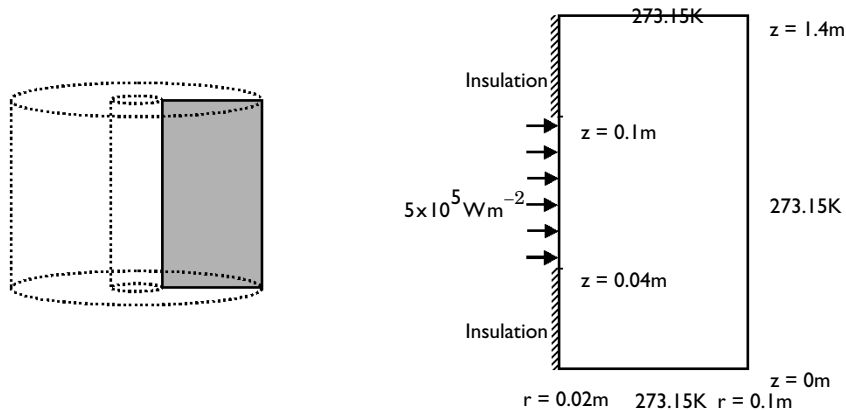


Figure 1: Model geometry and boundary conditions.

You set three types of boundary conditions:

- Prescribed heat flux
- Insulation/Symmetry
- Prescribed temperature

The governing equation for this problem is the steady-state heat equation for conduction with the volumetric heat source set to zero:

$$\nabla \cdot (-k \nabla T) = 0$$

The thermal conductivity k is $52 \text{ W}/(\text{m}\cdot\text{K})$.

Results

The plot in [Figure 2](#) shows the temperature distribution.

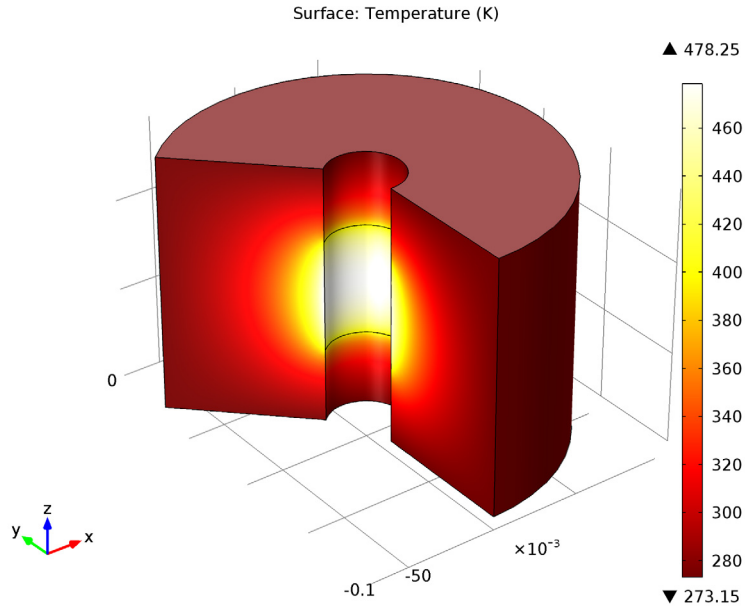


Figure 2: Temperature distribution.

The benchmark result for the target location ($r = 0.04 \text{ m}$ and $z = 0.04 \text{ m}$) is a temperature of 59.82°C , or $(59.82 + 273.15) \text{ K} = 332.97 \text{ K}$. The COMSOL Multiphysics model, using a default mesh with about 540 elements, gives a temperature of 332.957 K at the same location.

Reference

1. A.D. Cameron, J.A. Casey, and G.B. Simpson, *NAFEMS Benchmark Tests for Thermal Analysis (Summary)*, NAFEMS Ltd., 1986.

Model Library path: Heat_Transfer_Module/Tutorial_Models/
cylinder_conduction

Modeling Instructions

MODEL WIZARD

- 1** Go to the **Model Wizard** window.
- 2** Click the **2D axisymmetric** button.
- 3** Click **Next**.
- 4** In the **Add physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.
- 5** Click **Add Selected**.
- 6** Click **Next**.
- 7** Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.
- 8** Click **Finish**.

GEOMETRY 1

Rectangle 1

- 1** In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Rectangle**.
- 2** In the **Rectangle** settings window, locate the **Size** section.
- 3** In the **Width** edit field, type 0.08.
- 4** In the **Height** edit field, type 0.14.
- 5** Locate the **Position** section. In the **r** edit field, type 0.02.

Point 1

- 1** In the **Model Builder** window, right-click **Geometry 1** and choose **Point**.
- 2** In the **Point** settings window, locate the **Point** section.
- 3** In the **r** edit field, type 0.02.
- 4** In the **z** edit field, type 0.04.

Point 2

- 1** Right-click **Geometry 1** and choose **Point**.
- 2** In the **Point** settings window, locate the **Point** section.
- 3** In the **r** edit field, type 0.02.
- 4** In the **z** edit field, type 0.1.

Form Union

In the **Model Builder** window, under **Model 1 > Geometry 1** right-click **Form Union** and choose **Build Selected**.

HEAT TRANSFER IN SOLIDS*Heat Transfer in Solids 1*

- 1 In the **Model Builder** window, under **Model 1 > Heat Transfer in Solids** click **Heat Transfer in Solids 1**.
- 2 In the **Heat Transfer in Solids** settings window, locate the **Heat Conduction** section.
- 3 From the k list, choose **User defined**. In the associated edit field, type 52.
- 4 Locate the **Thermodynamics** section. From the C_p list, choose **User defined**. From the ρ list, choose **User defined**.

Temperature 1

- 1 In the **Model Builder** window, right-click **Heat Transfer in Solids** and choose **Temperature**.
- 2 In the **Temperature** settings window, locate the **Temperature** section.
- 3 In the T_0 edit field, type 273.15[K].
- 4 Select Boundaries 2, 5, and 6 only.

Heat Flux 1

- 1 Right-click **Heat Transfer in Solids** and choose **Heat Flux**.
- 2 In the **Heat Flux** settings window, locate the **Heat Flux** section.
- 3 In the q_0 edit field, type 5e5.
- 4 Select Boundary 3 only.

MESH 1

In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Build All**.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS*Temperature, 3D*

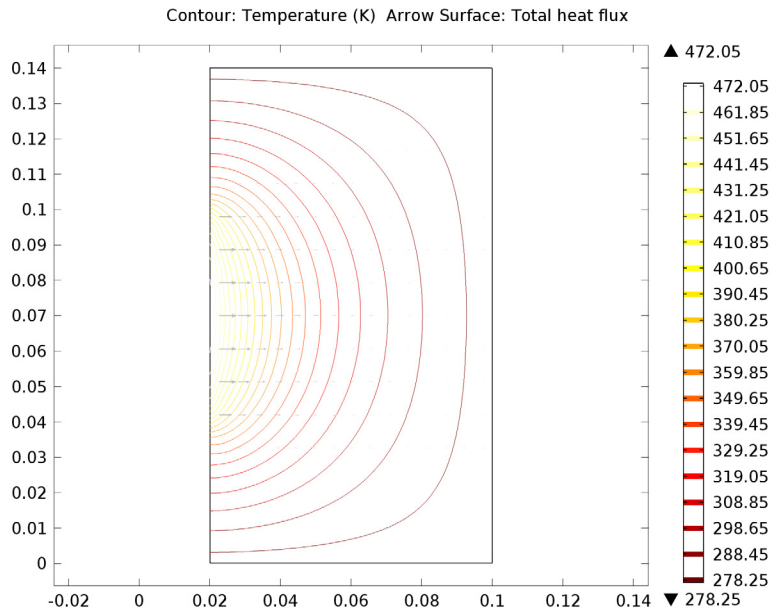
The first default plot is a revolved 3D plot visualizing the temperature field on the surface; compare with [Figure 2](#).

- 1 Click the **Zoom Extents** button on the Graphics toolbar.
- 2 In the **Model Builder** window, expand the **Results>Temperature, 3D** node.

Isothermal contours

- 1 In the **Model Builder** window, expand the **Results>Isothermal contours** node.
- 2 Right-click **Isothermal contours** and choose **Plot**.
- 3 Click the **Zoom Extents** button on the Graphics toolbar.

The second default plot is a combined contour and arrow plot of the temperature field and the total heat flux.



To obtain the temperature value at any point, just click at that point in the Graphics window; The result appears in the Results window at the bottom of the COMSOL Desktop.

Alternatively, you can create a Cut Point data set and Point Evaluation feature as follows.

Data Sets

- 1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Cut Point 2D**.
- 2 In the **Cut Point 2D** settings window, locate the **Point Data** section.

- 3 In the **r** edit field, type 0.04.
- 4 In the **z** edit field, type 0.04.

Derived Values

- 1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Point Evaluation**.
- 2 In the **Point Evaluation** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Cut Point 2D 1**.
- 4 Click the **Evaluate** button.
The result is approximately 333 K.