Created in COMSOL Multiphysics 6.2

Steady-State 2D Axisymmetric Heat Transfer with Conduction

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, the COMSOL Multiphysics simulation uses the kelvin temperature unit instead of degrees Celsius.

Model Definition

The modeling domain describes the cross section of a 3D solid as shown in Figure 1.

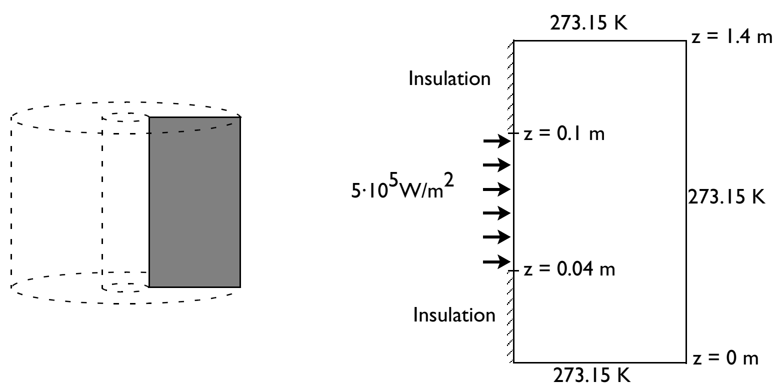


Figure 1: Model geometry and boundary conditions.

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 W/(m·K).

Results

The plot in Figure 2 shows the temperature distribution.

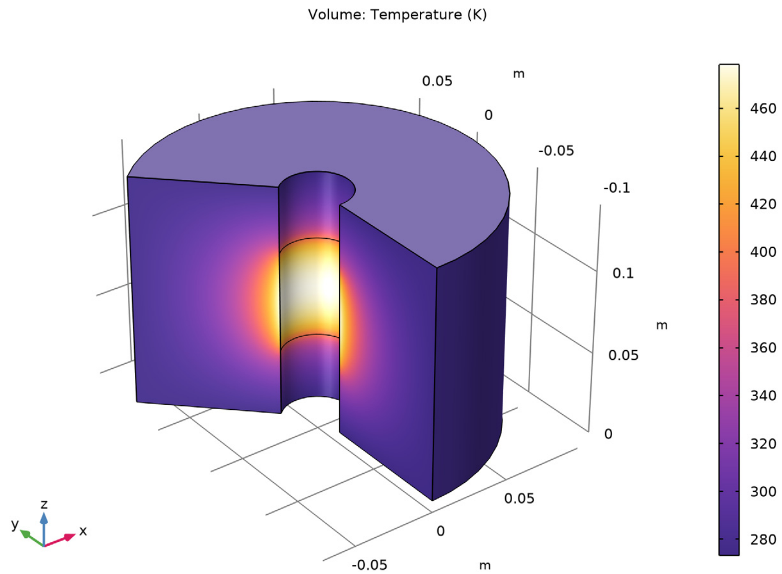


Figure 2: Temperature distribution.

The benchmark result for the target location (r = 0.04 m and z = 0.04 m) is a temperature of 59.82 °C (332.97 K). The COMSOL Multiphysics model, using a default mesh with about 540 elements, gives a temperature of 332.96 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, 1986.

Application Library path: `Heat_Transfer_Module/Tutorials,_Conduction/cylinder_conduction`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `0.08`.
4 In the **Height** text field, type `0.14`.
5 Locate the **Position** section. In the **r** text field, type `0.02`.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.
2 In the **Settings** window for **Point**, locate the **Point** section.
3 In the **r** text field, type `0.02 0.02`.
4 In the **z** text field, type `0.04 0.1`.
5 In the **Geometry** toolbar, click **Build All**.

HEAT TRANSFER IN SOLIDS (HT)

Solid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Solid 1**.
2 In the **Settings** window for **Solid**, locate the **Heat Conduction, Solid** section.

3 From the k list, choose **User defined**. In the associated text field, type `52`.

4 Locate the **Thermodynamics, Solid** section. From the C_p list, choose **User defined**. From the ρ list, choose **User defined**.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 In the **Settings** window for **Temperature**, locate the **Temperature** section.

3 In the T_0 text field, type `273.15[K]`.

4 Select Boundaries 2, 5, and 6 only.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

3 In the q_0 text field, type `5e5`.

4 Select Boundary 3 only.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

STUDY 1

In the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The default plot shows the temperature field on the 2D slice.

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

2 Go to the **Add Predefined Plot** window.

Add a predefined plot showing a 3D temperature distribution on a revolved surface; compare with Figure 2.

3 In the tree, select **Study 1/Solution 1 (sol1)>Heat Transfer in Solids>Temperature (ht)**.

4 Click **Add Plot** in the window toolbar.

RESULTS

Temperature 3D (ht)

1 In the **Settings** window for **3D Plot Group**, type `Temperature 3D (ht)` in the **Label** text field.

2 Click the **Zoom Extents** button in the **Graphics** toolbar.

Add another predefined plot showing isothermal contours in 2D section.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

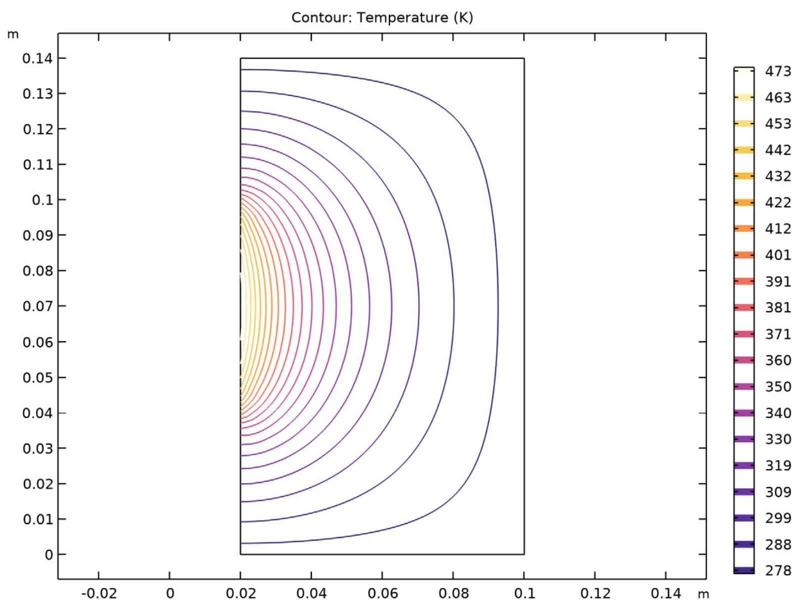
2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Heat Transfer in Solids> Isothermal Contours (ht)**.

4 Click **Add Plot** in the window toolbar.

RESULTS

Isothermal Contours (ht)



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

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

Cut Point 2D 1

1 In the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.

3 In the **R** text field, type `0.04`.

4 In the **Z** text field, type `0.04`.

Point Evaluation 1

1 In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Point 2D 1**.

4 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

The result is approximately 333 K.