Model created in COMSOL Multiphysics 6.4

Steady-State 2D Heat Transfer with Conduction

Introduction

This example shows a 2D steady-state thermal analysis including convection to a prescribed external (ambient) temperature. The example is taken from a NAFEMS benchmark collection (see Ref. 1).

Model Definition

This example considers 0.6 m-by-1.0 m domain. For the boundary conditions:

- The left boundary is insulated.
- The lower boundary is kept at 100°C.
- The upper and right boundaries are convecting to 0°C with a heat transfer coefficient of 750 $W/(m^2·°C)$.

In the domain use the following material property:

- The thermal conductivity is 52 W/(m·°C).

Results

The plot in Figure 1 shows the temperature field in the modeling domain.

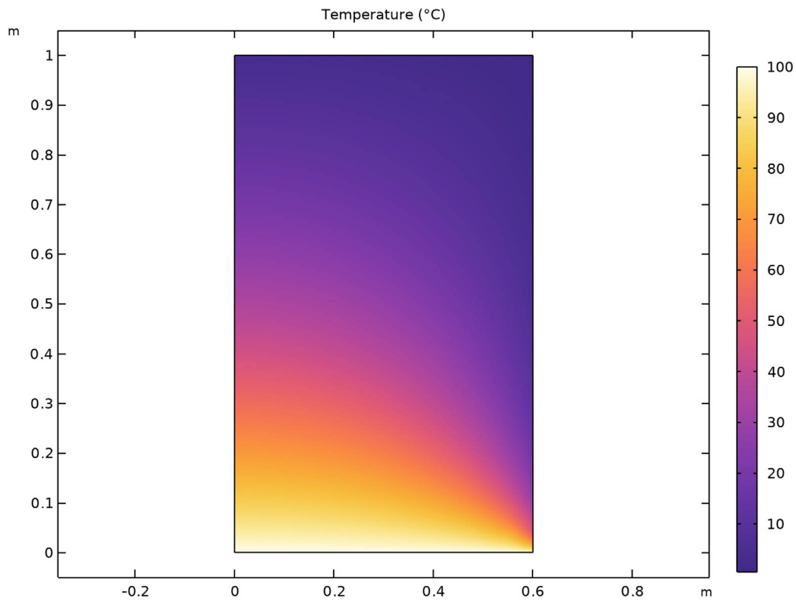


Figure 1: Temperature distribution resulting from convection to a prescribed external temperature.

The benchmark result for the target location (x = 0.6 m and y = 0.2 m) is a temperature of 18.25°C. The COMSOL Multiphysics model, using a mapped mesh with 9×15 quadratic elements, gives a temperature of 18.265°C.

Reference

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

Application Library path: `COMSOL_Multiphysics/Heat_Transfer/heat_convection_2d`

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**.
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 `0.6`.
4 Click **Build All Objects**.

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2 Select Boundary 2 only.
3 In the **Settings** window for **Temperature**, locate the **Temperature** section.
4 In the T_0 text field, type `100[degC]`.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2 Select Boundaries 3 and 4 only.
3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
4 From the **Flux type** list, choose **Convective heat flux**.
5 In the h text field, type `750`.

6 In the T_{ext} text field, type `0[degC]`.

Solid 1

1 In the **Model Builder** window, 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`.

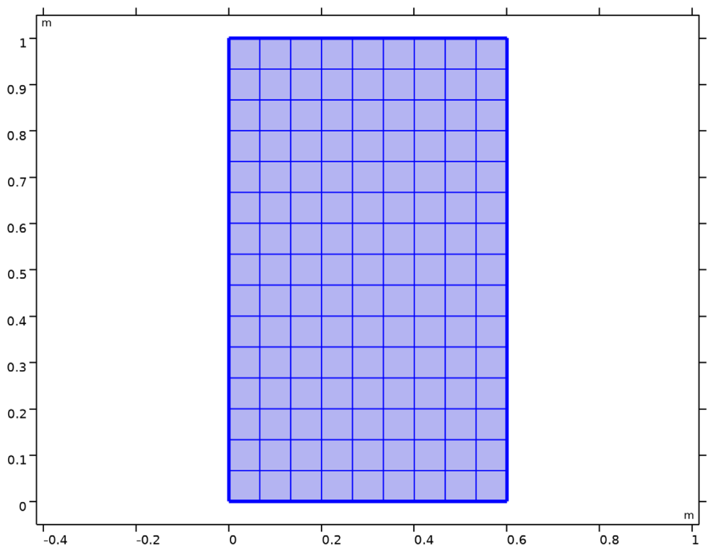
No other material properties enter into the domain equations for this stationary model.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.



STUDY 1

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

Change the unit of the temperature results to degrees Celsius.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **General** > **Temperature (K)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Temperature	K	°C

8 Click **Apply**.

The first default plot group shows the temperature field; compare with Figure 1.

Temperature (ht)

The benchmark value for the temperature at $x = 0.6$ m and $y = 0.2$ m is 18.25°C. To compare this value with that from the simulation, evaluate the temperature in this position.

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 **X** text field, type `0.6`.

4 In the **Y** text field, type `0.2`.

Point Evaluation 1

1 In the **Results** toolbar, click **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 should be close to 18.265°C.