

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/(m²·°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.

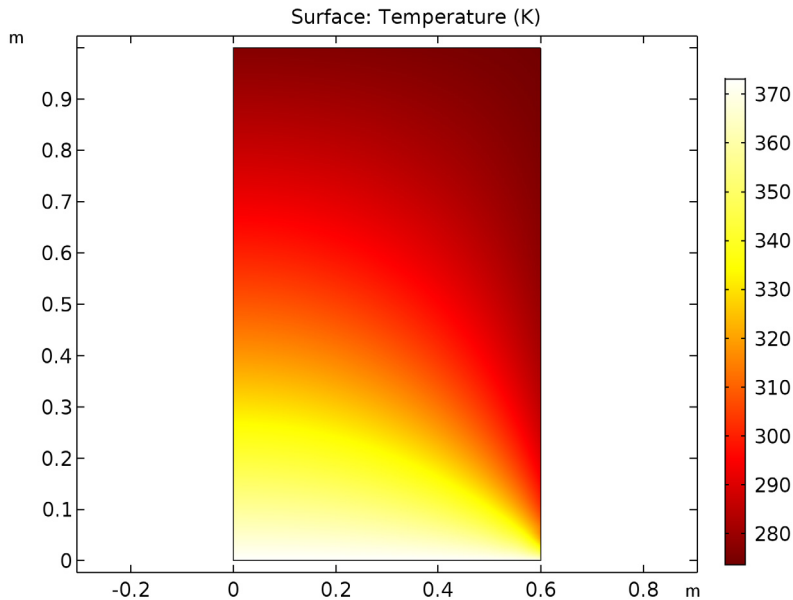


Figure 1: Temperature distribution resulting from convection to a prescribed external temperature.

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 **Preset Studies>Stationary**.
- 6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **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 On 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 On 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 Click the **Convective heat flux** button.
- 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, 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.

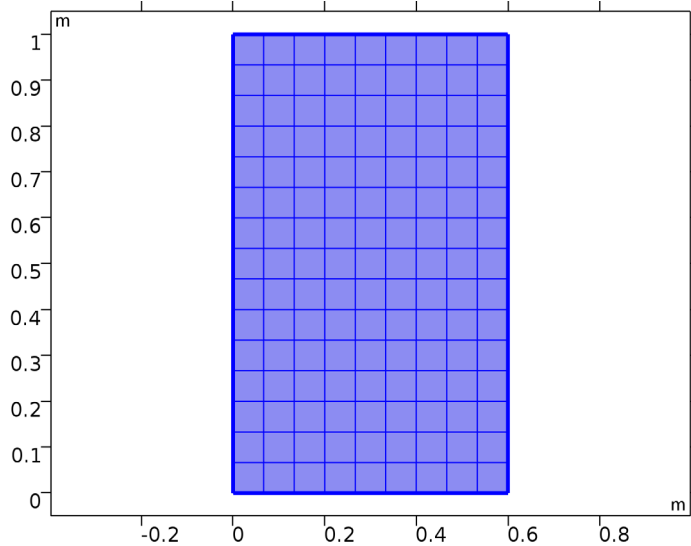
No other material properties enter into the domain equations for this stationary model.

MESH 1

Mapped 1

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

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



STUDY 1

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

RESULTS

Temperature (ht)

- 1 Click the **Zoom Extents** button on the **Graphics** toolbar.

The first default plot group shows the temperature field; compare with [Figure 1](#).

Data Sets

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 I

- 1 On 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 I

- 1 On the **Results** toolbar, click **Point Evaluation**.
- 2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.
- 3 From the **Data set** list, choose **Cut Point 2D I**.
- 4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
T	degC	Temperature

- 5 Click **Evaluate**.

TABLE

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

The result should be close to 18.265 °C.