



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.

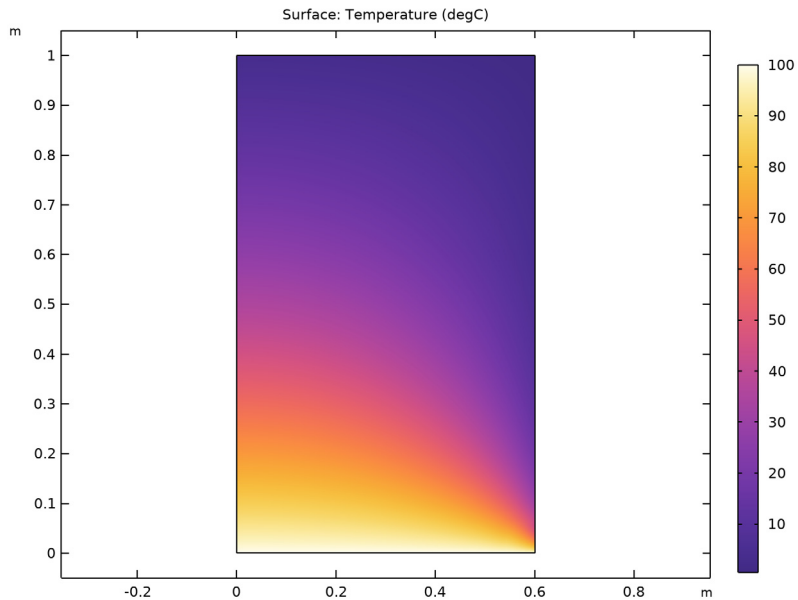


Figure 1: Temperature distribution resulting from convection to a prescribed external temperature.

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

GEOMETRY I

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 **Model Builder** window, under **Component 1 (comp1)** right-click **Heat Transfer in Solids (ht)** 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

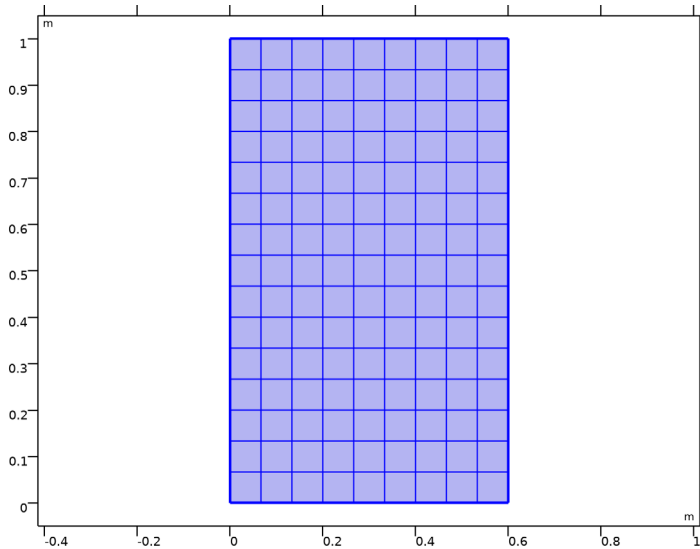
- 1 In the **Model Builder** window, click **Solid 1**.
- 2 In the **Settings** window for **Solid**, locate the **Heat Conduction, Solid** section.
- 3 From the k list, choose **User defined**. In the associated text field, type 52.

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 **Home** toolbar, click  **Compute**.


RESULTS

Surface

- 1 In the **Model Builder** window, expand the **Results>Temperature (ht)** node, then click **Surface**.

- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **degC**.


Temperature (ht)

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


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

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 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 I

- 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 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 .