2D Heat Transfer with Convection

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 model domain is 0.6 m-by-1.0 m. 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

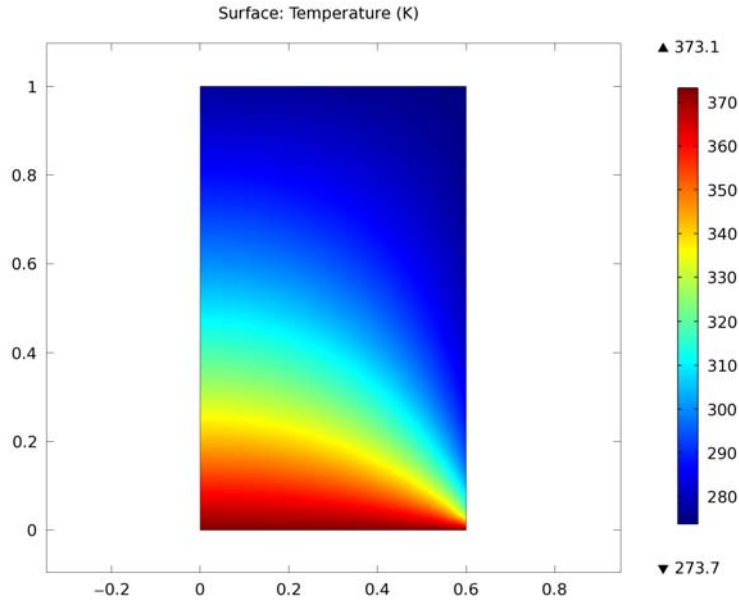The following plot shows the temperature as a function of position:

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 default mesh with 556 elements, gives a temperature of **18.28 °C**. Using an adaptive mesh refinement, the temperature becomes **18.25 °C**, which confirms that the result converges toward the benchmark result.

Reference

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

Model Library path: COMSOL_Multiphysics/Heat_Transfer/
heat_convection_2d

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

5 Click **Next**.

6 In the **Studies** tree, select **Preset Studies>Stationary**.

7 Click **Finish**.

GEOMETRY 1

Rectangle 1

1 In the **Model Builder** window, right-click **Model 1>Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type 0.6.

4 Click the **Build All** button.

HEAT TRANSFER

Temperature 1

1 In the **Model Builder** window, right-click **Model 1>Heat Transfer** and choose **Temperature**.

2 Select Boundary 2 only.

3 Go to the **Settings** window for Temperature.

4 Locate the **Temperature** section. In the T_0 edit field, type 100[degC].

Heat Flux 1

1 In the **Model Builder** window, right-click **Heat Transfer** and choose **Heat Flux**.

2 Select Boundaries 3 and 4 only.

3 Go to the **Settings** window for Heat Flux.

4 Locate the **Heat Flux** section. Click the **Inward heat flux** button.

5 In the h edit field, type 750.

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

Heat Transfer in Solids 1

1 In the **Model Builder** window, click **Heat Transfer in Solids 1**.

2 Go to the **Settings** window for Heat Transfer in Solids.

3 Locate the **Heat Conduction** section. From the k list, select **User defined**. In the associated edit field, type 52.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, right-click **Model 1>Mesh 1** and choose **Free Triangular**.

2 Right-click **Free Triangular 1** and choose **Build All**.

STUDY 1

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

RESULTS

Data Sets

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

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

2 Go to the **Settings** window for Cut Point 2D.

3 Locate the **Point Data** section. In the **x** edit field, type 0.6.

4 In the **y** edit field, type 0.2.

Derived Values

1 In the **Model Builder** window, right-click **Results>Derived Values** and choose **Point Evaluation**.

2 Go to the **Settings** window for Point Evaluation.

3 Locate the **Data** section. From the **Data set** list, select **Cut Point 2D 1**.

4 Locate the **Expression** section. From the **Unit** list, select **degC**.

5 Right-click **Point Evaluation 1** and choose **Evaluate**.