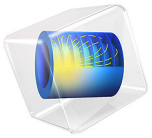

Steady-State 1D Heat Transfer with Radiation

Introduction

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

Model Definition

This 1D model has a domain of length 0.1 m. The left end is kept at 1000 K, and at the right end there is radiation to 300 K. The model uses the following material properties:

- For the radiation, the emissivity, ε, is 0.98.
- The thermal conductivity is 55.563 W/(m·K).

Results

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

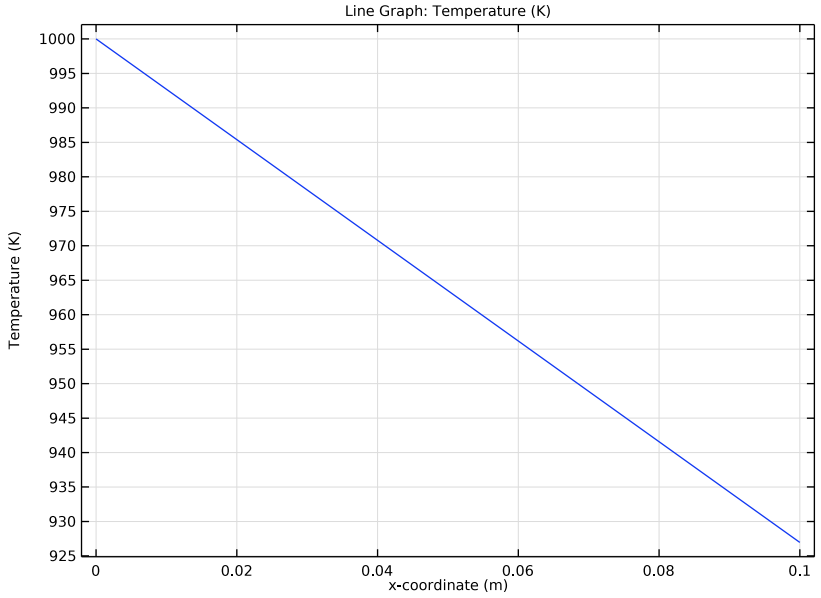

Figure 1: Temperature as a function of position.

The benchmark result for the right end is a temperature of 927.0 K. The COMSOL Multiphysics model, using a default mesh with 15 elements, gives a temperature at the end of 926.97 K, which is the exact benchmark value to four significant digits.

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_radiation_1d`

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 **1D**.

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

Interval 1 (i1)

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

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

3 In the table, enter the following settings:

Coordinates (m)
0
0.1

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 1 only.

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

4 In the T_0 text field, type 1000.

Surface-to-Ambient Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Surface-to-Ambient Radiation**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Surface-to-Ambient Radiation** section.

4 From the ε list, choose **User defined**. In the associated text field, type 0.98.

5 In the T_{amb} text field, type 300.

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 55.563.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type 1000.

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

The benchmark value for the temperature at the right end is 927.0 K. To compare the value from the simulation, evaluate the temperature at this position.

Point Evaluation 1

1. In the **Results** toolbar, click **Point Evaluation**.
2. Select Boundary 2 only.
3. In the **Settings** window for **Point Evaluation**, click **Evaluate**.