Examples of Heat Transfer Models

The following three heat transfer benchmark examples show how to model heat transfer using:

- Steady-state and transient analysis
- Temperature, heat flux, convective cooling, and radiation boundary conditions
- Thermal conductivity as a function of temperature

All examples are taken from a NAFEMS benchmark collection (Ref. 2).

1D Steady-State Heat Transfer with Radiation

The first example shows a 1D steady-state thermal analysis including radiation to a prescribed ambient temperature.

Model Definition

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

- The emissivity, ε, is 0.98.
- The Stefan-Boltzmann constant, σ, is 5.67×10^{-8} W/(m K^4).

In the domain, use the following material properties:

- The density, ρ, is 7850 kg/m^3.
- The heat capacity is 460 J/(kg °C).
- The thermal conductivity is 55.563 W/(m °C).

Results

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

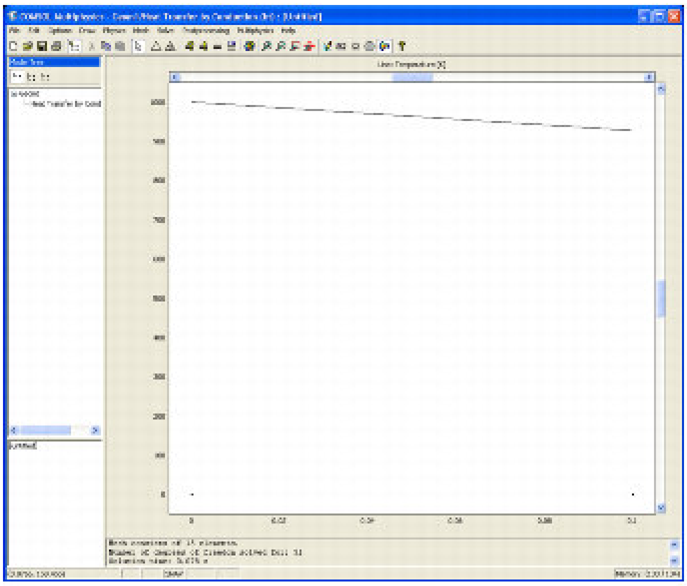


Figure 7-1: Temperature as a function of position.

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 as 926.97 K, which is the exact benchmark value to four significant digits.

Model Library Path: `COMSOL_Multiphysics/Heat_Transfer/heat_radiation_1D`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Go to the **Model Navigator** and select **1D** in the **Space dimension** list.

2 In the list of application modes, open the **COMSOL Multiphysics>Heat Transfer** folder and then the **Conduction** node.

3 Select **Steady-state analysis**.

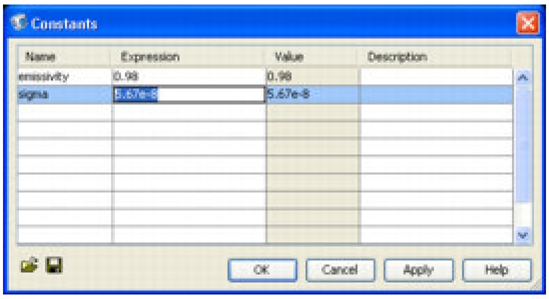
4 Click **OK**.

OPTIONS AND SETTINGS

1 Go to the **Options** menu and choose **Constants**.

2 Enter the following constants for the emissivity and the Stefan-Boltzmann constant:

NAME	EXPRESSION
`emissivity`	`0.98`
`sigma`	`5.67e-8`



GEOMETRY MODELING

1 Go to the **Draw** menu, point to **Specify Objects** and click **Line**.

2 In the **Line** dialog box, type `0 0.1` in the **x** edit field under **Coordinates**.

3 Click **OK**.

4 Click the **Zoom Extents** button.

PHYSICS SETTINGS

Boundary Conditions

1 Go to the **Physics** menu and choose **Boundary Settings**.

2 In the **Boundary Settings** dialog box select boundary 1.

[[Remove this text insert.]]

3 In the **Boundary condition** list select **Temperature**.

4 Type `1000` in the **Temperature** edit field.

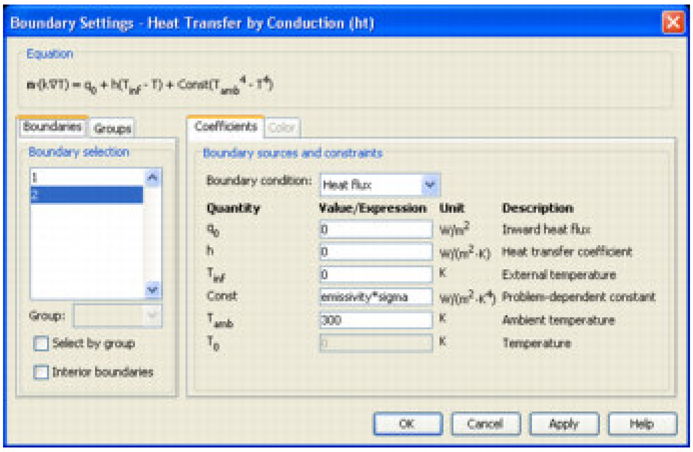
5 Select boundary 2.

6 In the **Boundary condition** list select **Heat flux**.

7 Type `emissivity*sigma` in the **Problem-dependent constant** edit field.

8 Type `300` in the **Ambient temperature** edit field.

9 Click **OK**.

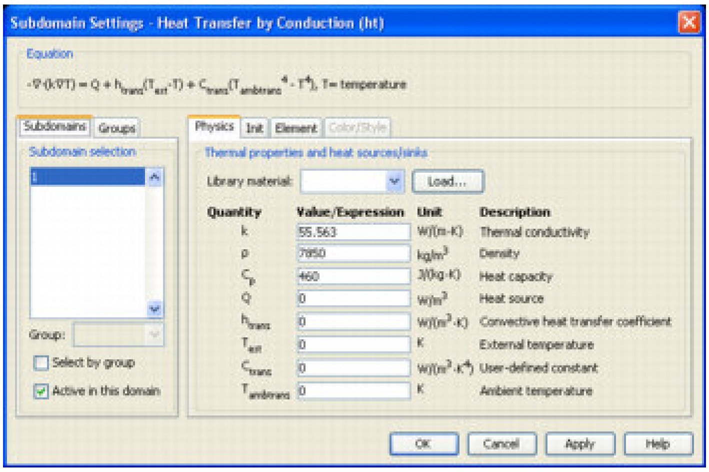


Subdomain Settings

1 Go to the **Physics** menu and choose **Subdomain Settings**.

2 In the **Subdomain Settings** dialog box enter the thermal properties in the domain according to the following table:

SUBDOMAIN	**1**
k (isotropic)	`55.563`
ρ	`7850`
C_p	`460`



Set the initial value to match the boundary condition. It serves as starting value for the nonlinear solver:

3 Click the **Init** tab.

4 Type `1000` as the initial value in the **Temperature** edit field.

5 Click **OK**.

MESH GENERATION

Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

Click the **Solve** toolbar button.

POSTPROCESSING AND VISUALIZATION

Figure 7-1 on page 174 shows the temperature distribution in the domain. Use the zoom tools to focus on the temperature at the right end.

2D Steady-State Heat Transfer with Convection

This example shows a 2D steady-state thermal analysis including convection to a prescribed external (ambient) temperature.

Model Definition

This model domain is 0.6×1.0 meters. 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 properties:

- The density, ρ, is 7850 kg/m^3.
- The heat capacity is 460 J/(kg °C).
- The thermal conductivity is 52 W/(m °C).

Results

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

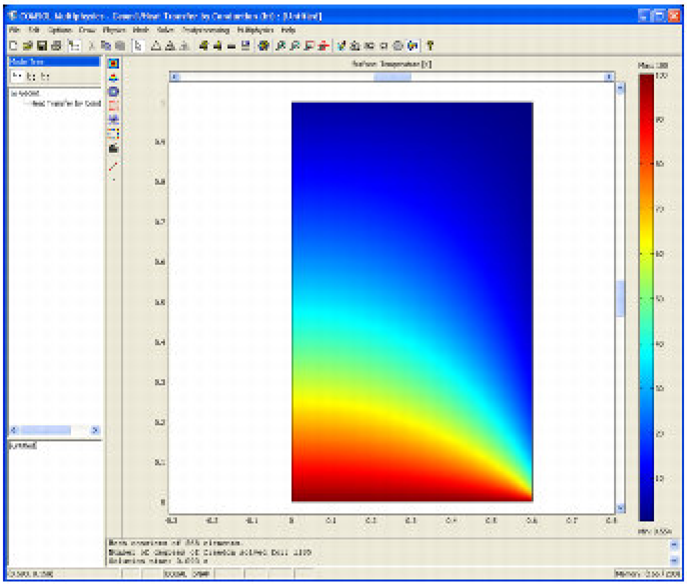


Figure 7-2: Temperature distribution resulting from convection to a prescribed external temperature.

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. Successive uniform refinements show temperatures of 18.26 °C and 18.25 °C, converging toward the benchmark result.

Model Library Path: `COMSOL_Multiphysics/Heat_Transfer/heat_convection_2D`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Go to the **Model Navigator** and select **2D** in the **Space dimension** list.

2 In the list of application modes, open the **COMSOL Multiphysics>Heat Transfer** folder and then the **Conduction** node.

3 Select **Steady-state analysis**.

4 Click **OK**.

GEOMETRY MODELING

1 On the **Draw** menu point to **Specify Objects** and click **Rectangle**.

2 In the **Rectangle** dialog box, find the **Size** area and type `0.6` in the **Width** edit field, then type `1` in the **Height** edit field.

3 Click **OK**.

4 Click the **Zoom Extents** button.

PHYSICS SETTINGS

Boundary Conditions

The default boundary condition is thermal insulation, so you must set boundary conditions for only three of the boundaries.

1 Go to the **Physics** menu and choose **Boundary Settings**.

2 In the **Boundary Settings** dialog box select boundary 2.

3 In the **Boundary condition** list select **Temperature**.

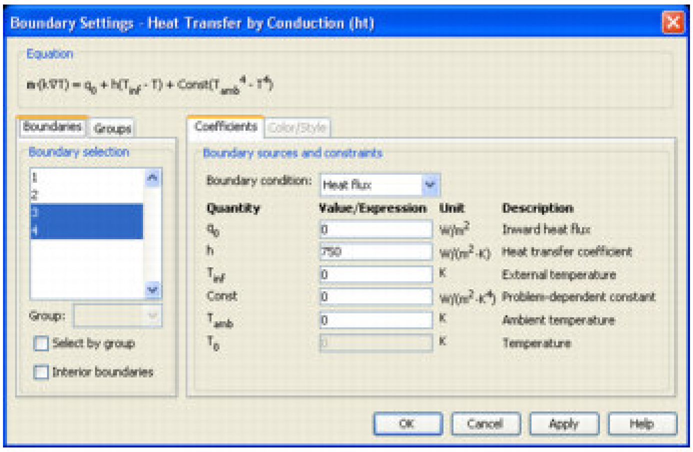
4 Type `100` in the **Temperature** edit field.

5 Select boundaries 3 and 4.

6 In the **Boundary condition** list select **Heat flux**.

7 Type `750` in the **Heat transfer coefficient** edit field.

8 Click **OK**.



Subdomain Settings

1 Go to the **Physics** menu and choose **Subdomain Settings**.

2 In the **Subdomain Settings** dialog box enter the thermal properties in the domain according to the following table:

SUBDOMAIN	1
k (isotropic)	52
ρ	7850
	460

C_p	

3 Click **OK**.

MESH GENERATION

Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

Figure 7-2 on page 178 shows the temperature distribution in the domain. To get a plot showing the numerical value at the reference point, use a cross-section plot:

1 Go to the **Postprocessing** menu and choose **Cross-Section Plot Parameters**.

2 In the **Cross-Section Plot Parameters** dialog box click the **Point** tab.

3 In the **Coordinates** area enter `0.6` in the **x** edit field and `0.2` in the **y** edit field.

4 Click **Apply**.

2D Axisymmetric Transient Heat Transfer

This example shows an axisymmetric transient thermal analysis with a step change to 1000 °C at time 0.

Model Definition

This model domain is 0.3×0.4 meters. For the boundary conditions:

- The left boundary is the symmetry axis.
- The other boundaries have a temperature of 1000 °C. The entire domain is at 0 °C at the start, which represents a step change in temperature at the boundaries.

In the domain use the following material properties:

- The density, ρ, is 7850 kg/m^3.
- The heat capacity is 460 J/(kg °C).
- The thermal conductivity is 52 W/(m °C).

Results

The following plot shows temperature as a function of position after 190 seconds:

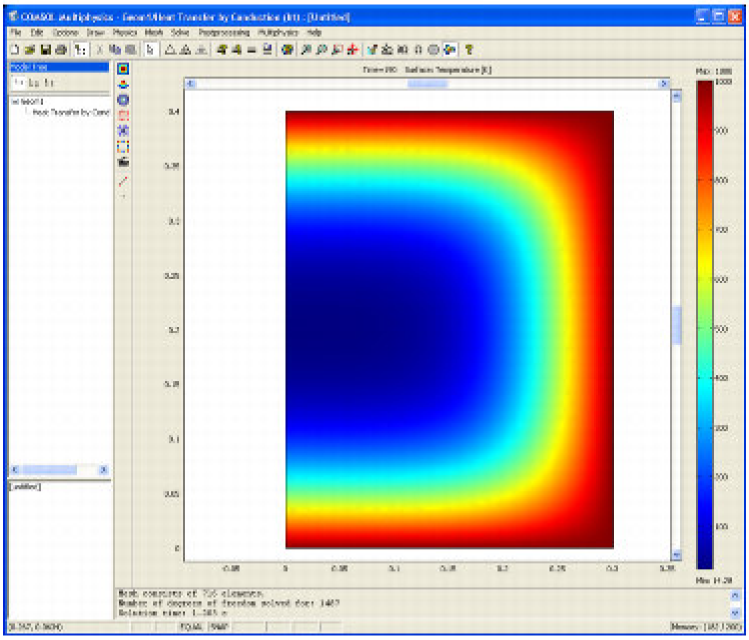


Figure 7-3: Temperature distribution after 190 seconds.

The benchmark result for the target location (r = 0.1 m and z = 0.3 m) is a temperature of 186.5 °C. The COMSOL Multiphysics model, using a default mesh with 712 elements, gives a temperature of 186.52 °C.

Model Library Path: `COMSOL_Multiphysics/Heat_Transfer/heat_transient_axi`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Go the **Model Navigator** and select **Axial symmetry (2D)** in the **Space dimension** list.

2 In the list of application mode, select **COMSOL Multiphysics> Heat Transfer>Conduction**.

3 Select **Transient analysis**.

4 Click **OK**.

GEOMETRY MODELING

1 Go to the **Draw** menu, point to **Specify Objects** and click **Rectangle**.

2 In the **Rectangle** dialog box go to the **Size** area and enter `0.3` in the **Width** edit field and `0.4` in the **Height** edit field.

3 Click **OK**.

4 Click the **Zoom Extents** button.

PHYSICS SETTINGS

Boundary Conditions

The default boundary condition is thermal insulation, so you need to set boundary conditions only for three of the boundaries.

1 Go to the **Physics** menu and choose **Boundary Settings**.

2 In the **Boundary Settings** dialog box select boundary 1.

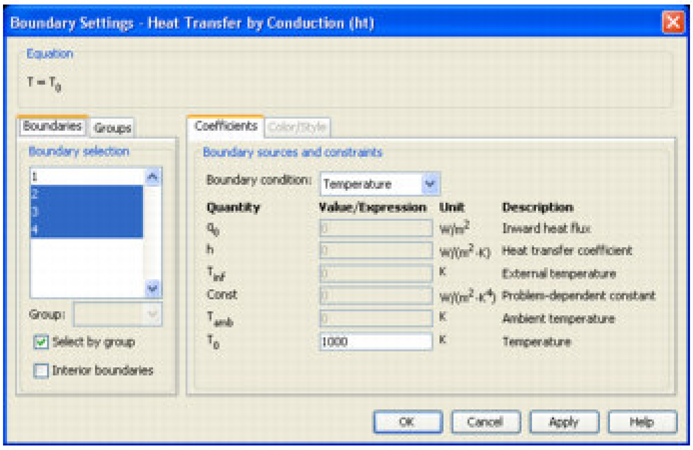
3 In the **Boundary condition** list select **Axial symmetry**.

4 Select the **Select by group** check box and choose boundaries 2, 3, and 4 by selecting one of them.

5 Select **Temperature** in the **Boundary condition** list.

6 Type `1000` in the **Temperature** edit field.

7 Click **OK**.



Subdomain Settings

1 Go to the **Physics** menu and choose **Subdomain Settings**.

2 In the **Subdomain Settings** dialog box enter the thermal properties in the domain according to the following table:

SUBDOMAIN	**1**
k (isotropic)	52
ρ	7850
C_p	460

3 Click **OK**.

MESH GENERATION

Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

1 Go to the **Solve** menu and choose **Solver Parameters**.

2 In the **Time stepping** area in the **Solver Parameters** dialog box enter `0:10:190` in the **Times** edit field.

3 Click **OK**.

4 Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

Figure 7-3 on page 182 shows the temperature distribution in the domain. To get a plot showing the numerical value at the reference point, use a cross-section plot:

1 Go to the **Postprocessing** menu and choose **Cross-Section Plot Parameters**.

2 In the **Cross-Section Plot Parameters** dialog box click the **Point** tab.

3 Select the **Point plot** button.

4 Under **Coordinates** enter `0.1` in the **r** edit field and `0.3` in the **z** edit field.

5 Click **Apply**.

You can also click in the temperature plot at (0.1, 0.3) to display the temperature at that point in the Message Log.

References

1. Frank P. Incropera and David P. DeWitt, *Fundamentals of Heat and Mass Transfer*, 4th edition, John Wiley & Sons, New York, 1996.

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