# Heat Conduction in a Finite Slab

## *Introduction*

This simple example covers the heat conduction in a finite slab, modeling how the temperature varies with time. You first set up the problem in COMSOL Multiphysics and then compare it to the analytical solution given in Ref. 1.

In addition, this example model also shows how to avoid oscillations due to a jump between initial and boundary conditions by using a smoothed step function.

## *Model Definition*

The model domain is defined between $x = -b$ and $x = b$. The initial temperature is constant, equal to $T_0$, over the whole domain; see the figure below. At time $t = 0$, the temperature at both boundaries is lowered to $T_1$.

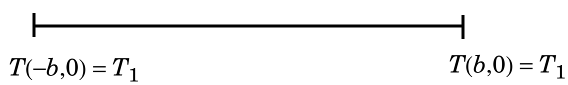



*Figure 1: Modeling domain.*

To compare the modeling results to the literature (Ref. 1), introduce new dimensionless variables according to the following definitions:

$$\Theta = \frac{T_1 - T}{T_1 - T_0} \qquad \eta = \frac{x}{b} \qquad \tau = \frac{\alpha t}{b^2}$$

The model equation then becomes

$$\frac{\partial\Theta}{\partial\tau} = \frac{\partial^2\Theta}{\partial\eta^2}$$

with the associated initial condition

$$\tau = 0 \qquad \Theta = 1$$

To model the temperature decrease at the boundaries use a smoothed step function of time $f(\tau)$.

$$\eta = \pm 1 \qquad \Theta = f(\tau)$$

This method is usually more realistic from a physical point of view than the sudden change in the temperature, and it is also better from a numerical point of view.

## Results and Discussion

Figure 2 shows the temperature as a function of position at the dimensionless times $\tau = 0.01$, $0.04$, $0.1$, $0.2$, $0.4$, and $0.6$. In this plot, the slab's center is situated at $x = 0$ with its end faces located a $x = -1$ and $x = 1$. The temperature profiles shown in the graph are identical to the analytical solution given in Carslaw and Jaeger (Ref. 1).

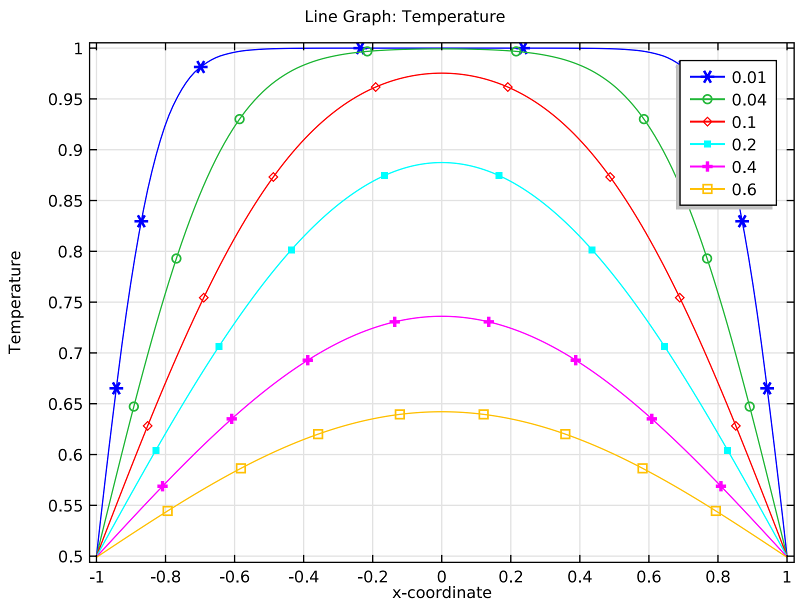



*Figure 2: Temperature profiles.*

## Reference

1. H.S. Carslaw and J.C. Jaeger, *Conduction of heat in Solids*, 2nd ed., Oxford University Press, p. 101, 1959.

**Model Library path:** `Heat_Transfer_Module/Tutorial_Models,_Conduction/heat_conduction_in_slab`

## *Modeling Instructions*

### MODEL WIZARD

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

2 Click the **1D** button.

3 Click **Next**.

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

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

7 Click **Finish**.

The **Heat Transfer in Solids** interface can be used for solving the dimensionless equations. You can switch off the dimensions using the following commands:

### MODEL 1

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

2 In the **Model** settings window, locate the **Model Settings** section.

3 From the **Unit system** list, choose **None**.

### GEOMETRY 1

*Interval 1*

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

2 In the **Interval** settings window, locate the **Interval** section.

3 In the **Left endpoint** edit field, type `-1`.

4 Click the **Build All** button.

5 Right-click **Model 1>Geometry 1>Interval 1** and choose **Rename**.

6 Go to the **Rename Interval** dialog box and type `Slab` in the **New name** edit field.

7 Click **OK**.

## DEFINITIONS

Add a step function for use in the boundary conditions.

*Step 1*

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Functions>Step**.

2 In the **Step** settings window, locate the **Parameters** section.

3 In the **Location** edit field, type `5e-4`.

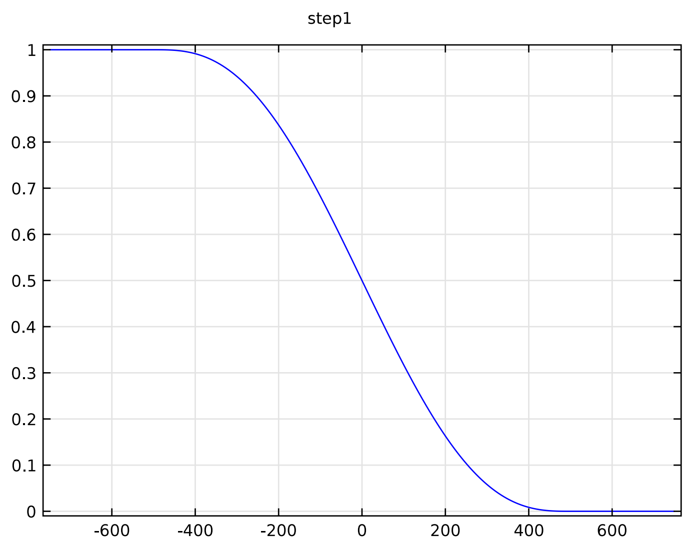
4 In the **From** edit field, type `1`.

5 In the **To** edit field, type `0`.

6 Click to expand the **Smoothing** section. In the **Size of transition zone** edit field, type `1e3`.

Optionally, you can inspect the shape of the step function.

7 Click the **Plot** button.




## HEAT TRANSFER IN SOLIDS

*Heat Transfer in Solids 1*

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

2 In the **Heat Transfer in Solids** settings window, locate the **Heat Conduction, Solid** section.

3 From the $k$ list, choose **User defined**. In the associated edit field, type `1`.

4 Locate the **Thermodynamics, Solid** section. From the ρ list, choose **User defined**. In the associated edit field, type `1`.

5 From the $C_p$ list, choose **User defined**. In the associated edit field, type `1`.

### *Initial Values 1*

1 In the **Model Builder** window, under **Model 1>Heat Transfer in Solids** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the $T$ edit field, type `1`.

### *Temperature 1*

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

2 Select Boundaries 1 and 2 only.

3 In the **Temperature** settings window, locate the **Temperature** section.

4 In the $T_0$ edit field, type `step1(t)`.

## MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Finer**.

4 Click the **Build All** button.

## STUDY 1

### *Step 1: Time Dependent*

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Time Dependent** settings window, locate the **Study Settings** section.

3 In the **Times** edit field, type `range(0,0.01,1)`.

To make sure that the transition of the boundary temperature from 1 to zero is represented correctly by the transient solver, use the initial time step that is smaller than the transition zone of the step function.

*Solver 1*

1 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solver 1** node, then click **Time-Dependent Solver 1**.

3 In the **Time-Dependent Solver** settings window, click to expand the **Time Stepping** section.

4 Select the **Initial step** check box.

5 In the associated edit field, type `0.00010`.

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

## RESULTS

*Temperature (ht)*

The default plot shows the temperature distribution along the slab for all time steps. You can compare the COMSOL Multiphysics solution to that of Ref. 1 by plotting the temperature for a given set of output times, as in Figure 2.

1 In the **1D Plot Group** settings window, locate the **Data** section.

2 From the **Time selection** list, choose **From list**.

3 In the **Times** list, choose **0.01**, **0.04**, **0.1**, **0.2**, **0.4**, and **0.6**.

4 Click the **Plot** button.

5 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Line graph**.

6 In the **Line Graph** settings window, click to expand the **Legends** section.

7 Select the **Show legends** check box.

8 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.