# 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 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 $T_1$.

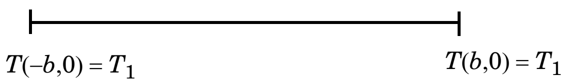



*Figure 1: Modeling domain.*

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

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

and boundary conditions

$$\eta = \pm 1 \qquad \Theta = 0$$

The analytical solution of this problem is (see Ref. 1, equation 12.1-31):

$$\Theta = 2\sum_{n=0}^{\infty}\frac{(-1)^n}{\left(n+\frac{1}{2}\right)\pi}\exp\left[-\left(n+\frac{1}{2}\right)^2\pi^2\tau\right]\cos\left(\left(n+\frac{1}{2}\right)\pi\eta\right)$$

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 Ref. 1.

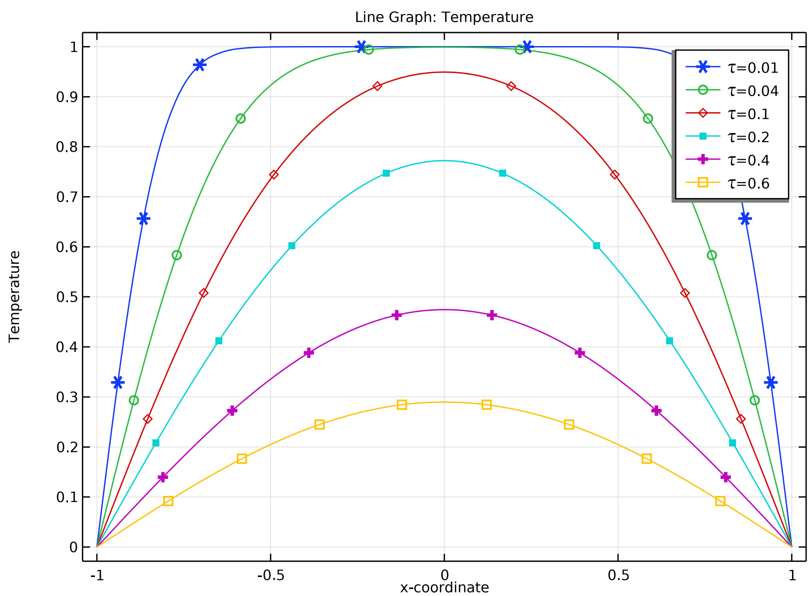



*Figure 2: Temperature profiles.*

The plot of the $L^2$ error between the analytical and numerical solutions over time (see Figure 3) confirms this conclusion.

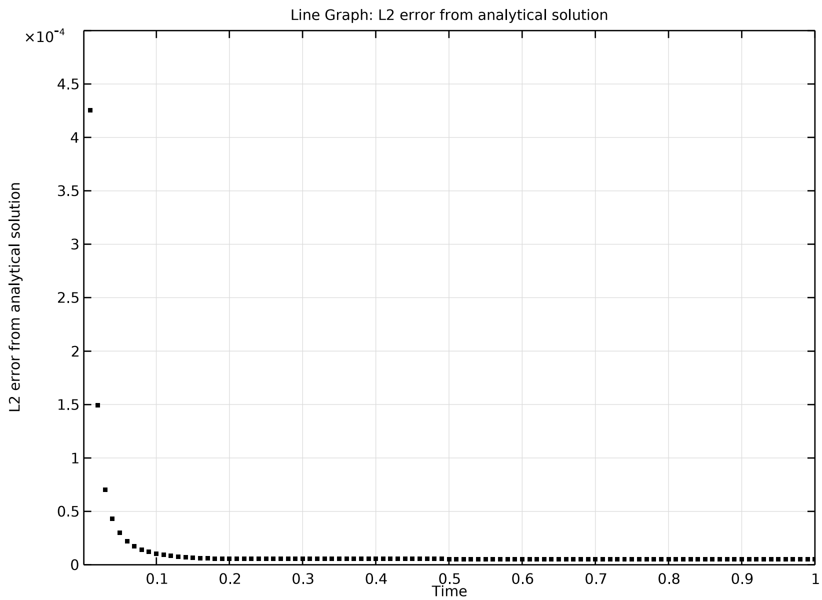



*Figure 3: L2 error between analytical and numerical solutions over time.*

## Reference


1. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, *Transport Phenomena*, 2nd ed., John Wiley & Sons, 2007.


**Application Library path:** `Heat_Transfer_Module/Tutorials,_Conduction/heat_conduction_in_slab`

## 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>Time Dependent**.

6 Click **Done**.

## GEOMETRY 1

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

## COMPONENT 1 (COMP1)

1 In the **Model Builder** window, click **Component 1 (comp1)**.

2 In the **Settings** window for **Component**, locate the **Units** section.

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

## 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 |
|---|
| -1 |

4 Click **Build All Objects**.

## DEFINITIONS

Add a step function for use in the boundary conditions.

### *Step 1 (step1)*

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Step**.

2 In the **Settings** window for **Step**, locate the **Parameters** section.

3 In the **Location** text field, type `1e-6`.

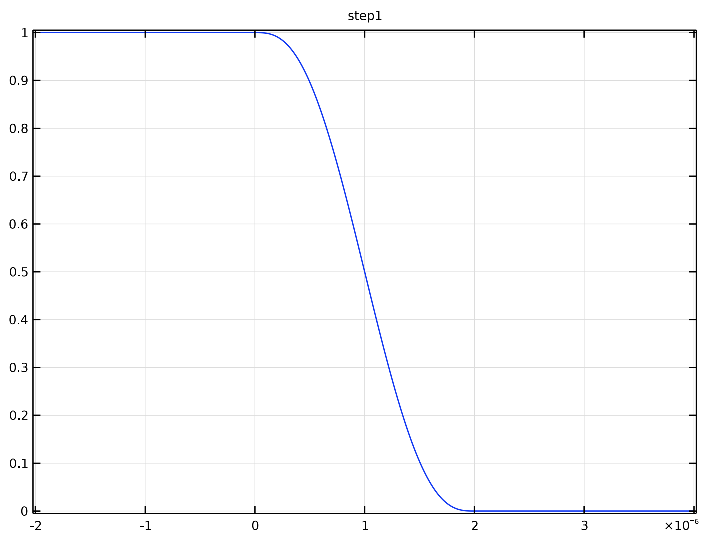
4 In the **From** text field, type `1`.

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

6 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `2e-6`.

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

7 Click **Plot**.




Add a nonlocal integration coupling for the computation of the relative $L^2$ error between numerical and analytical solutions.

*Integration 1 (intop1)*

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Selection** list, choose **All domains**.

## HEAT TRANSFER IN SOLIDS (HT)

*Solid 1*

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** 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 `1`.

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

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

### *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 `1`.

### *Temperature 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Click in the **Graphics** window and then press Ctrl+A to select both boundaries.

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

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

## MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

### *Size*

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Finer**.

### *Size 1*

1 In the **Model Builder** window, right-click **Edge 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All boundaries**.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `1e-4`.

8 Click **Build All**.

## STUDY 1

### *Step 1: Time Dependent*

1. In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3. In the **Output times** text 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 an initial time step that is smaller than the transition zone of the step function.

### *Solution 1 (sol1)*

1. In the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
3. In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
4. Select the **Initial step** check box. In the associated text field, type `2e-7`.
5. From the **Maximum step constraint** list, choose **Constant**.
6. In the **Maximum step** text field, type `1e-3`.
7. In the **Study** toolbar, click **Compute**.

## RESULTS

### *Temperature (ht)*

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

1. In the **Settings** window for **1D Plot Group**, locate the **Data** section.
2. From the **Time selection** list, choose **From list**.
3. In the **Times (s)** list, choose **0.01**, **0.04**, **0.1**, **0.2**, **0.4**, and **0.6**.
4. In the **Temperature (ht)** toolbar, click **Plot**.

### *Line Graph 1*

1. In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Line Graph 1**.
2. In the **Settings** window for **Line Graph**, click to expand the **Coloring and Style** section.
3. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

4 Click to expand the **Legends** section. Select the **Show legends** check box.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

| Legends |
|---|
| `\tau=0.01` |
| `\tau=0.04` |
| `\tau=0.1` |
| `\tau=0.2` |
| `\tau=0.4` |
| `\tau=0.6` |

7 In the **Temperature (ht)** toolbar, click **Plot**.

Next plot the relative $L^2$ error between the numerical and analytical solutions over time.

### *Relative L2 Error*

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Relative L2 Error` in the **Label** text field.

### *Line Graph 1*

1 In the **Relative L2 Error** toolbar, click **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `sqrt(intop1((T-2*sum((-1)^n/((n+0.5)*pi)*exp(-(n+0.5)^2*pi^2*t)*cos((n+0.5)*pi*x),n,0,1000))^2))/sqrt(intop1(T^2))`.

5 Select the **Description** check box. In the associated text field, type `L2 error from analytical solution`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `t`.

8 Locate the **Coloring and Style** section. From the **Color** list, choose **From theme**.

9 In the **Relative L2 Error** toolbar, click **Plot**.

As the analytical solution shows oscillations at initial time, change the settings of the graph for a better readability, to get the plot of Figure 3.

*Relative L2 Error*

1 In the **Model Builder** window, click **Relative L2 Error**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `1e-3`.

5 In the **x maximum** text field, type `1`.

6 In the **y minimum** text field, type `0`.

7 In the **y maximum** text field, type `5e-4`.

8 In the **Relative L2 Error** toolbar, click **Plot**.