



Model created in COMSOL Multiphysics 6.4

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 .

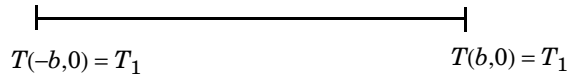


Figure 1: Modeling domain.

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

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

The model equation then becomes

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

with the associated initial condition

$$\tau = 0 \quad \Theta = 1$$

and boundary conditions

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

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 $f(\tau)$.

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

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 at $x = -1$ and $x = 1$. The temperature profiles shown in the graph are identical to the analytical solution given in Ref. 1.

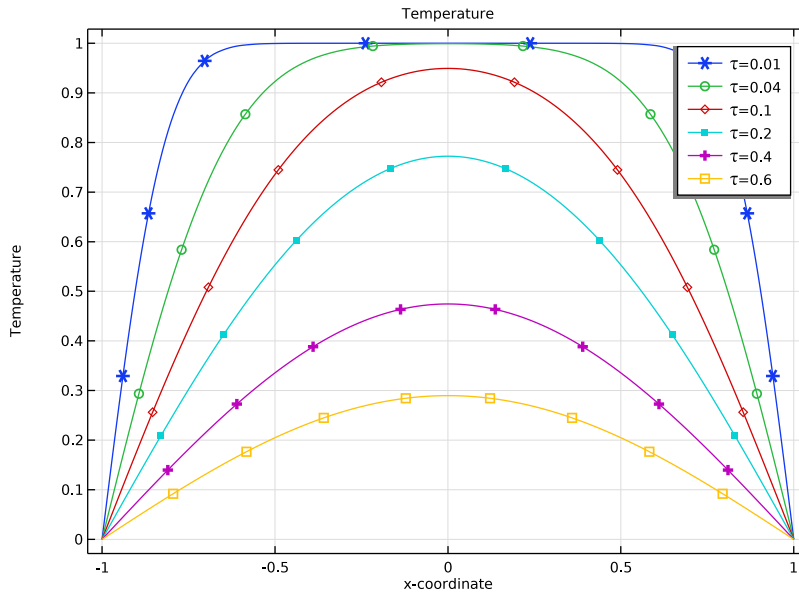


Figure 2: Temperature profiles.

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

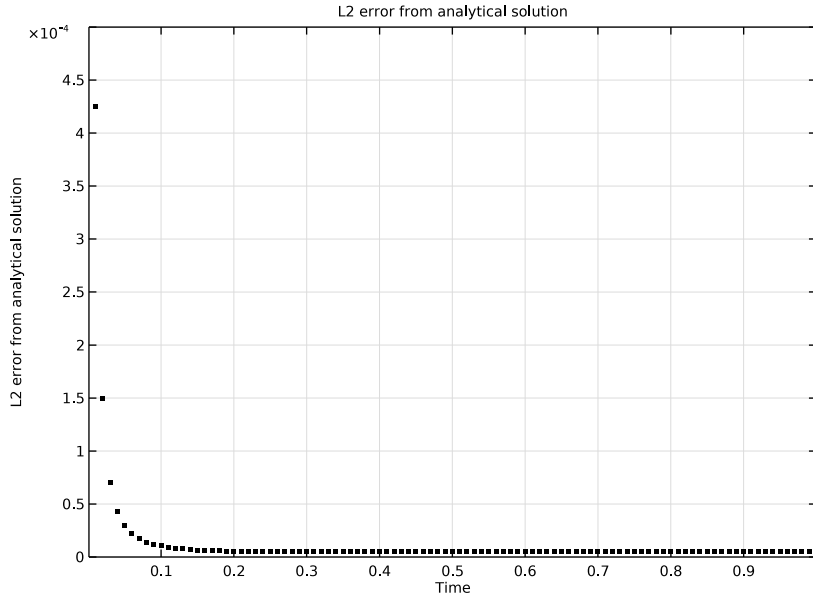


Figure 3: L^2 error between analytical and numerical solutions over time.

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  **ID**.
- 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 I

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 I (COMPI)

- 1 In the **Model Builder** window, click **Component I (comp1)**.
- 2 In the **Settings** window for **Component**, locate the **Units** section.
- 3 From the **Unit system** list, choose **None**.

GEOMETRY I

Interval 1 (i1)

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

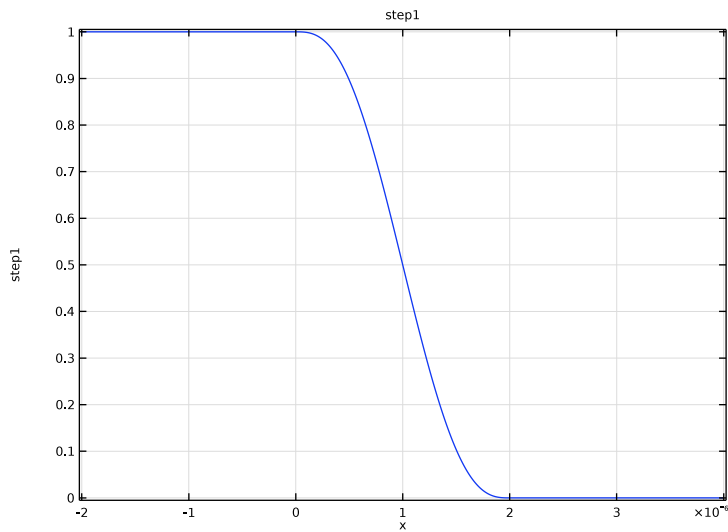
DEFINITIONS

Add a step function for use in the boundary conditions.

Step 1 (step1)


- 1 In the **Definitions** toolbar, click  **More Functions** and choose **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 I

- 1 In the **Model Builder** window, click **Initial Values I**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 In the T text field, type 1.

Temperature I

- 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 I

- 1 In the **Model Builder** window, under **Component I (comp1)** click **Mesh I**.
- 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 I (comp1) > Mesh I** click **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Finer**.

Size I

- 1 In the **Model Builder** window, right-click **Edge I** 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** checkbox. 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** checkbox. 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 **ID 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** checkbox.
- 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 **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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 $\sqrt{\text{intop1}((T-2*\text{sum}((-1)^n/((n+0.5)*\pi)*\exp(-(n+0.5)^2*\pi^2*t)*\cos((n+0.5)*\pi*x),n,0,1000))^2)}/\sqrt{\text{intop1}(T^2)}$.
- 5 Select the **Description** checkbox. 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 **ID Plot Group**, locate the **Axis** section.
- 3** Select the **Manual axis limits** checkbox.
- 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**.