

## INTRODUCTION TO COMSOL MULTIPHYSICS

In this chapter step-by-step instructions are provided for these one-dimensional problems. Many features of Comsol Multiphysics are not used or needed until you solve two- and three-dimensional problems, and these features are explained in Chapters 10-11 and Appendix D. For the boundary value problems treated here, we generally follow these steps:

1. Open Comsol Multiphysics and choose the dimensions of the problem (here 1) and the Physics modules we wish to solve.
2. Establish our preferences.
3. Create the geometry.
4. Insert the variables that define the problem and set the boundary conditions.
5. Create a mesh.
6. Solve the problem.
7. Examine the solution; here we limit ourselves to plots and integrals of the solution.

The icons shown in Figure 9.2 are ones that are used frequently in these examples. They are worth remembering.



**Figure 9.2. Icons used for setup: next, finish, show; geometry/mesh: build; plot: expression, plot; range of parameters.**

### EXAMPLE: HEAT TRANSFER IN A SLAB

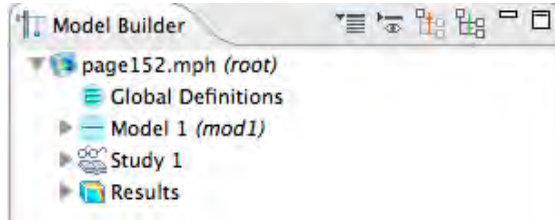
#### Solution using Comsol Multiphysics

Consider the problem of heat transfer in a slab with no internal heat generation but with a thermal conductivity that varies linearly with temperature – Eq. (9.1) with  $k = 1 + T$ .

$$\frac{d}{dx} \left[ (1 + T) \frac{dT}{dx} \right] = 0 \quad (9.20)$$
$$T(0) = 0, \quad T(1) = 1$$

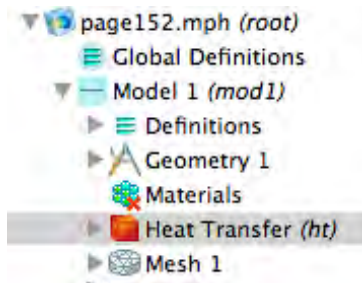
The boundary conditions are taken as  $T = 0$  at  $x = 0$  and  $T = 1$  at  $x = 1$ .

**Step 1 (Open and Choose Physics)** Open Comsol Multiphysics and choose 1D in the Model Wizard and click the next arrow. Then click on the arrow before ‘Heat Transfer’ to open that folder. Choose ‘Heat Transfer in Solids (ht)’ and click the next arrow. The Settings window will have a list of Preset options; select Stationary and then click the Finish flag. The nodes at the left of the screen are called the Model Builder and they provide a guide to the necessary steps (Figure 9.3).



**Step 2 (Set Preferences)** We will in turn use the Model 1, Study 1, and Results nodes. Note the icons at the top, right. The second one from the left is to 'show' additional things, and we choose 'Discretization' and 'Equation View'. In the root node we can choose a system of dimensions in the settings window; choose 'none'.

**Figure 9.3. Outline of Model Builder nodes**



**Step 3 (Geometry)** Open the Model 1 node by clicking on the arrow to the left to expand this node (Figure 9.4). Then right-click on the Geometry sub-node and choose Interval. The default interval is zero to one, which is what we want, so we accept it by clicking on the 'build' icon at the top of the settings window.

**Figure 9.4. Outline of Model 1 nodes**

**Step 4 (Define the Problem)** With the Model 1 node open, click on the arrow to the left of the Heat Transfer node and choose Heat Transfer in Solids. Then click on the arrow before it and in the Settings window set the thermal conductivity to 1.0 (for testing first). Right-click on Heat Transfer and choose Temperature. Choose the left node in the graphics window, use + in the settings window to add it to the box in the settings window. Leave the value at 0. Do this again for the right end and set the value to 1.

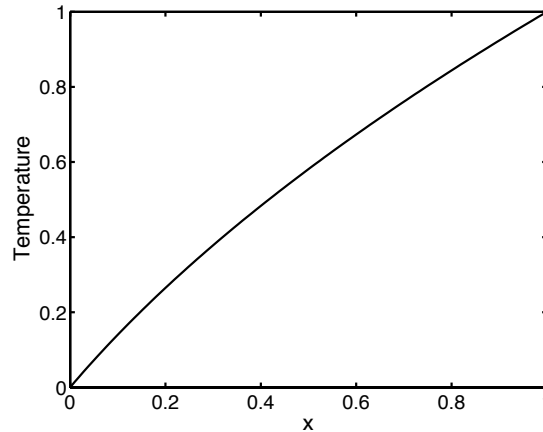
**Step 5 (Create a Mesh)** Choose Mesh and the Settings window for Size will appear. Accept them by clicking the Build icon. Sixteen nodes will be identified in the Graphics window. If you forget this step, it will be performed automatically for you when you solve the problem.

**Step 6 (Solve the Problem)** Choose Study and right click =. A plot of  $T(x)$  should appear in the Graphics Window.

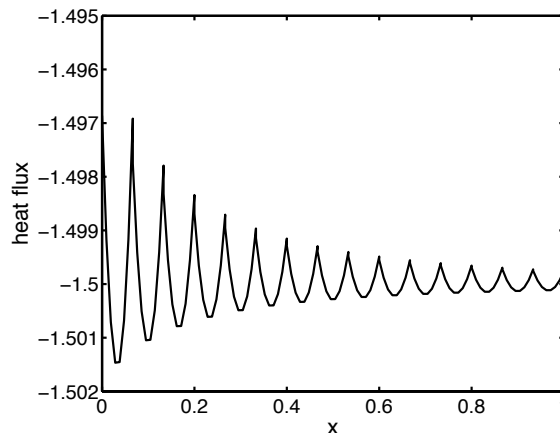
**Step 7 (Examine the Solution)** If it doesn't, or to create another graph, right-click Results and choose 1D Plot Group. Right-click 1D Plot Group and choose Line Graph. A plot of  $T(x)$  should appear in the Graphics Window. The solution is a straight line from (0,0) to (1,1), as it should be. The heat flux is plotted by creating another Line Graph, but this time changing the expression for the y-Axis Data. Choose the expression button, select Heat Transfer, then Conductive Heat Flux, then the xx-component of heat flux (there is only an x-component in this problem though). The heat flux is a constant  $-1$ . If a plot does not appear, click the plot icon.

**Step 8 (Solve the Real Problem)** Go back to the Heat Transfer in Solids sub-node and change the thermal conductivity to  $1+T$ . Choose Study and right click =. Examine the solution in the same way. The temperature is shown in Figure 9.5 and the heat flux is shown in Figure 9.6. The temperature departs slightly from a straight line. The heat flux has some weird looking

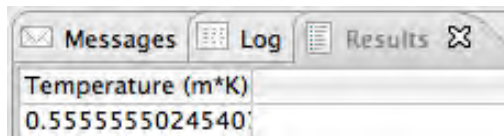
oscillations between -1.5015 and -1.497, but these are an artifact of the numerical method. This is because the temperature is approximated using quadratic functions on finite elements, but the derivative (proportional to heat flux) is not continuous across element boundaries. If you click on 1D Plot Group, then Axis in the Settings window, you can check the Manual box and set the axis limits for the y to  $-2 \rightarrow 0$ ; on that scale the oscillations are scarcely noticeable.



**Figure 9.5. Solution to Eq. (9.20)**



**Figure 9.6. Heat flux in problem, Eq. (9.20)**



**Figure 9.7. Messages/Log/Results**

**Step 9 (Calculate Averages)** You can calculate the average temperature. Open the arrow the left of Results right-click Derived Values; right click Derived Values and choose Integration and then Line Integration. In the Settings window, note

that the variable being integrated is  $T$ . Click = and look at the Results icon (Figure 9.7, below the graph) to see the value: 0.55556. This is the value of the average temperature. You can check that this is the average by using Integration instead of Average and integrate  $T$  and 1 separately.

$$\int_0^1 T dx \bigg/ \int_0^1 dx = 0.55556 \quad (9.21)$$

**Step 10 (Plot the Thermal Conductivity)** You can plot the thermal conductivity by right-clicking on Results; choose 1D Plot Group. Then right-click on it and choose Line Graph. In the Settings window, click on the expression button and choose heat transfer/thermal conductivity to plot. Remove the units from the Settings window if they appear. The plot looks like Figure 9.5 except with the y-axis going from 1 to 2 ( $1+T$ ).