Transient Boundary Value Problem Solved in Comsol 4.1.
Copyright Bruce A. Finlayson, 2010
See also *Introduction to Chemical Engineering Computing*, Wiley (2006).

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.

Setup: Next: ⇨ ; Finish: 🏁 ; Show: 👁 ; **Create geometry/mesh: Build:** ▮ ;

Plot: Expression button: ▷ ; Plot: ✎ ; Range of parameters: ▭

Figure 9.2. Icons used for Setup, Creation of geometry and mesh, and Plotting

The next example is a transient heat transfer problem.

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}, \ \alpha = 2$$

$$\left. \frac{\partial T}{\partial x} \right|_{x=0} = 0, \ T(1,t) = 0, \ T(x,0) = 1$$

(9.35)

Step 1: choose the problem. To solve Eq. (9.35), open Comsol, choose 1D, next; Heat Transfer, then Heat Transfer in Solids, next; Time Dependent, finish.

Step 2: define the geometry. Right click on Geometry and choose Interval. The default is (0,1), which we keep. Choose Build All.

Step 3: define the problem. Then choose Heat Transfer; in the Settings panel, look at the equation which Comsol will solve. Several of the terms need to be set to zero: U-sub-trans, Q, Wsubp. Click on Heat Transfer in Solids, and change the parameters to User defined and k = 2, rho = 1, Cp = 1. Then the equation is the same as Eq. (9.35).

Step 4: set the boundary and initial conditions. To set the boundary conditions, right click on Heat Transfer and choose Temperature. Select the boundary point 2 at the right-hand side of the diagram and the + symbol to add it to the Selection. Change the value from 293.15 to 0. Click on the Thermal insulation node and see that boundary point 1 has thermal insulation (which we want) and boundary point 2 has been overridden. To set the initial condition, click on the Initial Value node and set T to 1.

Step 6: set the mesh. Select the mesh and choose Build All; notice that the nodal points are displayed in the graph.

Step 7: solve the problem. Right click on Study = to get the solution.

Step 8: examine the solution. Right click Data Sets and choose Parametric Extrusion 1D. The Settings window will list the times at which the solution will be plotted. Select the 1D Plot Group and change the Plot Settings to reflect what we have actually done, x, T, Solution to Problem 9.35 and click on the plot icon to get the new plot. To get the legends for the graph, click on the Line Graph subnode, choose Legends (near the bottom) and click on the Show legends box. Then replot to obtain Figure 9.13. Note that it isn't possible to have the temperature be both 0 (the boundary condition) and 1 (the initial condition) at the point $x = 1$, $t = 0$, and this introduces a small perturbation near that point. This perturpation quickly dampens out in time, though. It is really a manifestation of an improperly posed mathematical problem rather than an incorrect method, because such a discontinuity is impossible to create. This graph can be compared to graphs in your textbook to validate the computer solution.

Step 9: create a 3D plot. To see a colorful 3D plot, right click on Results, choose 2D Plot Group. Right click 2D Plot Group, choose Surface and make sure the data set chosen is Parametric Extrusion 1D. Right click again on Surface and choose Height Expression. A colorful 3D plot emerges, Figure 9.14.
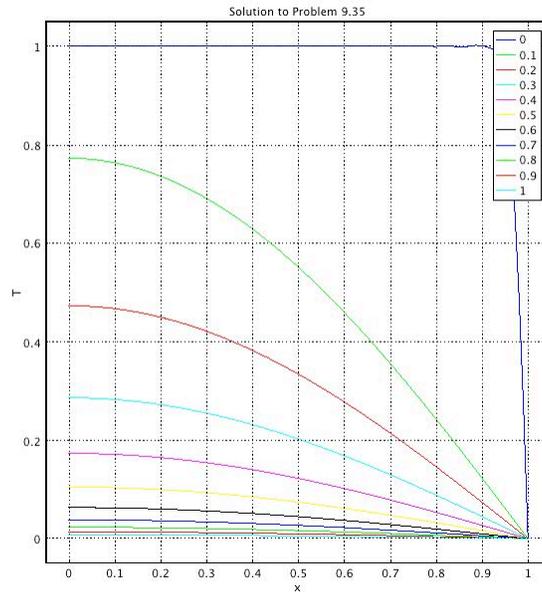

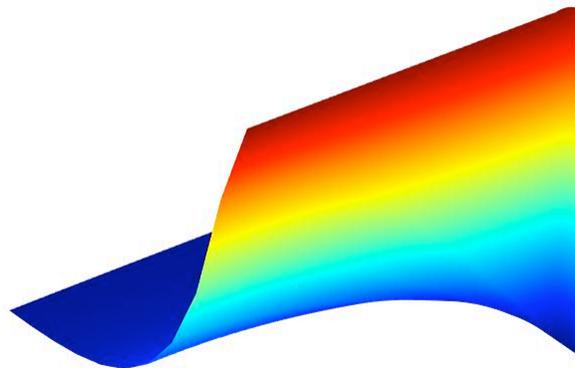**Figure 9-13. Solution of heat transfer problem, Eq. (9.34)**

Figure 9-14. Solution of heat transfer problem, Eq. (9.34) plotted in 3D