General Introduction to Comsol 4.1.
Copyright Bruce A. Finlayson, 2010
See also  http://www.ChemEComp.Wordpress.com/

See the specific applications for step by step instructions. Remember:

**When in doubt right click.**

When you open Comsol Multiphysics, the screen shows five key windows.

Model Builder

Settings

Graphics

Messages

Progress

Results (where tabular values are kept)

On the left is the Model Builder Tree; that is where you define the problem you want to solve. As it expands, nodes (each item is called a node) are added making a vertical tree of nodes that collectively define the steps to define the class of problems. To the right of the Model Builder is the Settings window. This is where you will set values needed for your model, such as sizes, viscosities, etc. To the right of that is a blue Graphics window. As you define the geometry and mesh, it will be displayed here, as will the solution when you are done. Below the Graphics window are Messages and Progress and Results.

Stop now and look at the icons for each of those windows, since you can easily open them from a variety of screens by choosing the icon.

If your screen gets changed to something you don’t want, click on the screen refresh icon.
On the left is the Model Builder Tree, that is where you define the problem you want to solve. As it expands, nodes (each item is called a node) are added making a vertical tree of nodes that collectively define the steps to define the class of problems. To take action you must right click on a node (to see the possibilities) or select one of the items under the arrow.

The application is chosen at the very start, and other physics can be added later. You use the Model Wizard, answering the questions, clicking the next arrow, and clicking the flag for finish.

For example, if one chooses Laminar Flow, and the Stationary, when you open the Laminar Flow node, you can select Equations and see the Navier-Stokes equation in vector form. You need to adjust the Physical Model so that the Compressibility option is Incompressible flow. The turbulence option is not selected by default, so if you are solving for turbulent flow you would have to select it, too. The dependent variables are listed. Unfortunately, all possible dependent variables are listed (including those for turbulent flow) even when one has selected laminar flow.
Setting the material properties

When solving an equation that is in non-dimensional form, select Fluid Properties from the Laminar Flow node, and then look at Fluid Properties at the bottom of the Settings window. Change 'from material' to 'User defined' and set the density to a stand-in (frequently Reynolds number) and the viscosity to a stand-in (frequently 1.0). This will work for small Reynolds numbers. For larger Reynolds numbers (say over 100), one would put the density as 1.0 and the viscosity as 1/Reynolds number. See the page 201 for how to make the connection between the variables called density and viscosity in the program and the Reynolds number in your non-dimensional version. Of course, if you are solving for a specific fluid, then one can either insert the numerical values here, or set them in the Materials node. Notice that the Fluid Properties Settings also has an option to see the Equation. If the equations are not showing, choose the pull-down menu Options/Preferences. and click "Equation View".

Setting the boundary conditions

The boundary conditions are set by choosing the type of boundary condition (wall, inlet, outlet, etc.) and then selecting the boundaries on which that boundary condition is to hold. This procedure allows you to set a number of boundary conditions at once. To set a boundary condition, right-click on the Application (Laminar Flow, Heat Transfer, etc.) and choose the type of boundary condition (wall, inlet, temperature, etc.). Then select the boundaries that will use that boundary condition; then press the + sign to the right of the Domain Selection so that the numbers of the boundaries appear in the box.

Go back and redo this for another type of boundary condition, or the same type of boundary condition with different parameters. After right-clicking on the Application, you can also change the Domain Setting to All boundaries and the numbers of all boundaries will be placed into the box. Those boundaries already assigned will be indicated as (overridden).

Setting the mesh

If one clicks on the Mesh node, the Settings allows you to choose the density of elements through the Element size option; there are nine levels, from extremely course to extremely fine, with normal being the default value. Click on the Build All icon to cause the mesh to be created. (This icon represents a building.) There are a number of other options for creating meshes, and these can be accessed by using right click on the Mesh node. These are discussed elsewhere.

Solving the problem

Generally you right click Study and select =.
Results

Right click on Results and select the type of plot you would like, or the derived quantities you would like to see. These are explained in specific applications. The plot icon is .

To see some ready-made variables to plot, click on the triangles . To save a plot, click the camera icon.