Solution of Heat Transfer in a Sphere using Comsol Multiphysics.

Reference: Bruce A. Finlayson, *Introduction to Chemical Engineering* Computing, 2nd ed., Wiley (2012). See also www.ChemE.Comp.com and ChemEComp.Wordpress.com

Open and Set Preferences: SI system of units.

Define the Problem: right click the Untitled.mph; choose 3D, Heat Transfer in Solids, Transient.

Define the Geometry: right click Model 1/Geometry, choose a sphere, set the radius to 0.01 (in m), click Build All.

Set the Physical Parameters: right click Materials, choose Material, then built in; select Water, liquid, right click it and choose Add Material to Model.

Check the model, Heat Transfer in Solids; open the node and select Heat Transfer in Solids; check that k, rho and Cp are all "From material". Keep the node open.

Set the Boundary Conditions and Initial Conditions: right click Heat Transfer in Solids and choose Temperature; set it to 290 K; then change Manual to All boundaries; select Initial Values and set it to 310 K.

Create a Mesh: click on Mesh and then Build All for a default mesh.

Solve the Problem: before doing that, save the model; then open the Study node. Set the time range to 0 to 100, with steps (for saving the solution) of 10 seconds; right click on Study and choose Compute. The time range was determined by trial and error.

Examine the Solution: The picture that appears is for the boundary temperature; since it is always 290, it doesn't change even if you look at it at different times. (This is one disadvantage of solving this problem in 3D when you don't need the full 3D equations.) Open the Results node (it should be open);right click on Data Sets and select Cut Plane. In the window choose the x-y plane at z = 0. To get a 2D picture of that plane, right click Results and choose 2D Plot Group. In the window choose Cut Plane if it isn't selected. Then right click 2D Plot Group and choose Surface. The Expression is already T, the temperature. Change Data set to Cut Plane. Click on the plot symbol to see the plot. The Time can be adjusted, and there will be pictures of the temperature at times 0, 10, 20,…100. You can then see the colder temperature moving in from the boundary.

Change to a Heat Transfer BC: To back to the Heat Transfer in Solids sub-node and right click; choose Temperature, right click it and delete it. Then right click Heat Transfer in Solids and choose Heat Flux. Then change to Inward heat flux, set the heat transfer coefficient to 100, change Manual to All boundaries. Right click Study and choose Compute.

Select the 2D Plot Group under Results and then plot. You can change the time and you can see the two solutions are different. You can make movies of the solution as well as make many other plots.

Finite element information: If you want to see what the finite elements are like, choose the eye in Model Builder and select Discretization. Then when you create a model you can look at Discretization. For the problem given below, the finite elements are quadratic, but you can change them if you like.