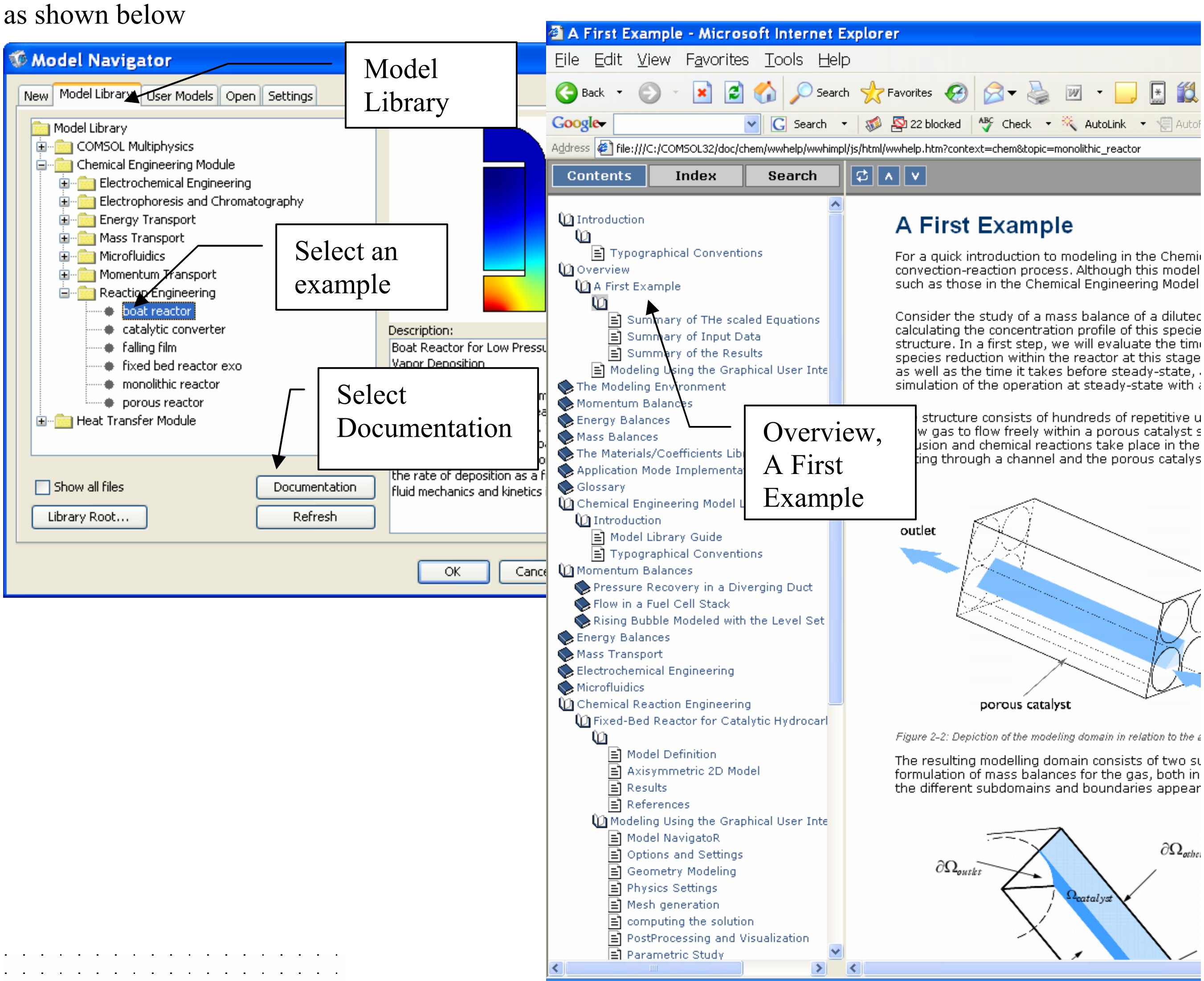
Open Comsol Multiphysics 3.2, Select the Model Library tab, Then select any model within the chemical engineering Module library as shown below. Then press the documentation button.

Then select Go to the Chemical Engineering Module
Overview
A First Example
as shown below



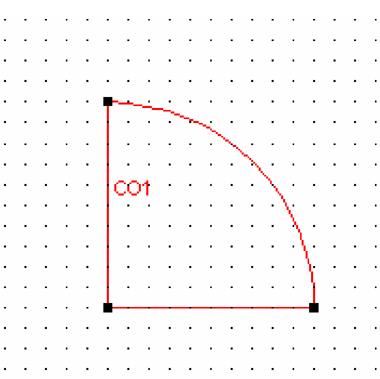


Figure 1: The first shape that you draw

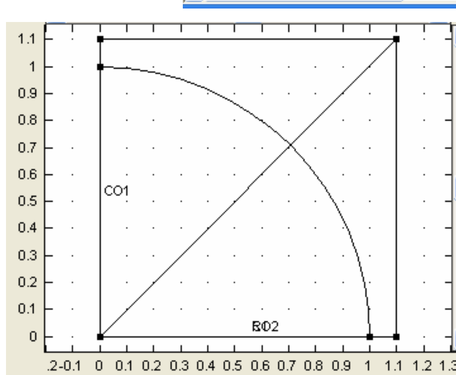
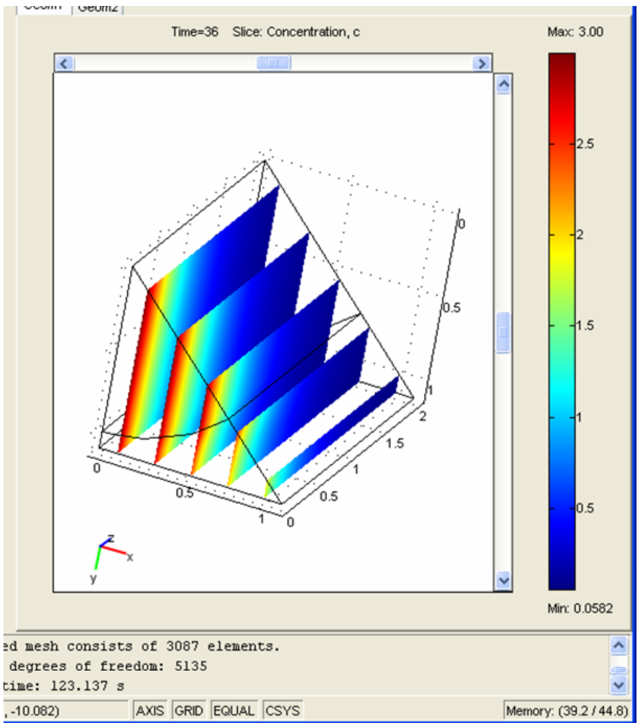
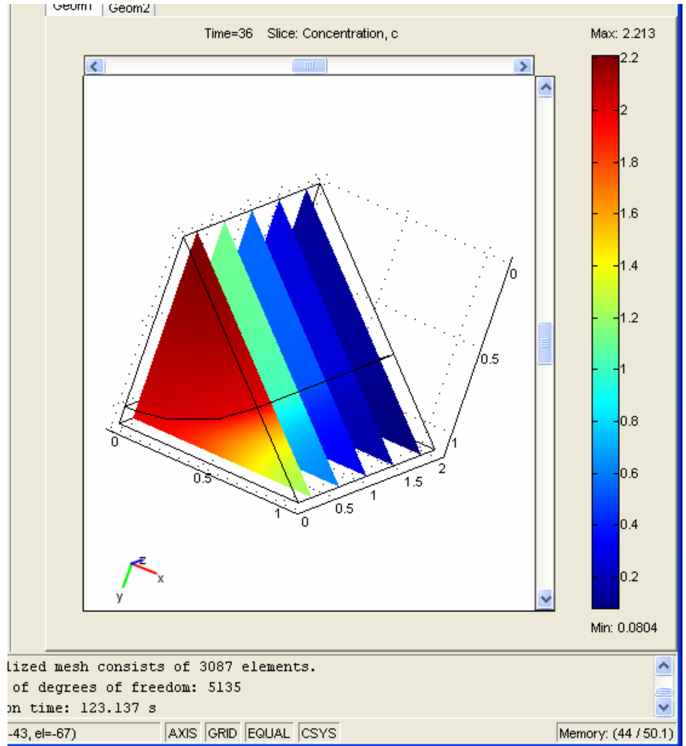


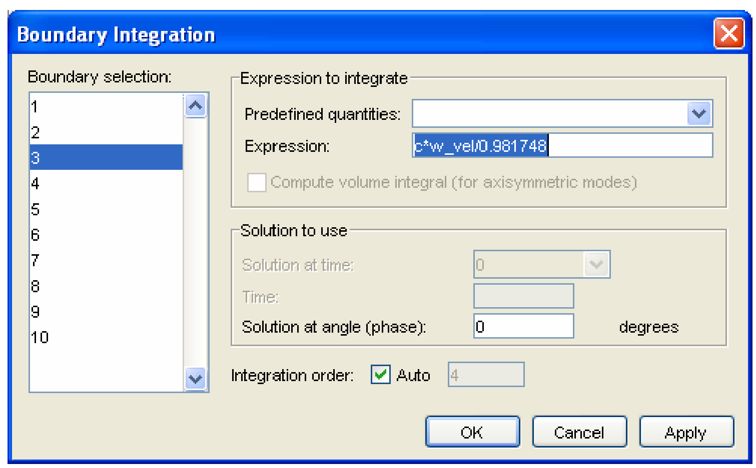
Figure 2: All 3 shapes together





Submit at the end of the period:

1. Turn in 3-D slice plot with 5 z-level slices
2. Compute the average concentration at the inlet and outlet boundaries. The average concentration is given by $\bar{c} = \dfrac{\iint (c * w\_vel)\, dxdy}{\iint w\_vel\, dxdy}$. To do this first calculate the average velocity and then calculate the integral of c*w_vel. Dividing these two numbers gives you the average velocity at a surface. Try the inlet surface first to make sure that you get cin=3 mol/m^3. Use Postprocessing, boundary integration. Surface 3 should be your inlet and 4 should be your outlet. (e.g. for the inlet I get the average velocity as 0.098175m/s and the average concentration c*w_vel/0.098175=2.9999) From these values of average concentration you will be able to calculate a conversion (assume dilute gas).



3. Increase the flowrate by a factor of 2 and 10 times the initial flowrate. Describe what happens giving the minimum outlet concentration, average outlet concentration, and conversion for each of the three cases.
4. Compute the average concentration
5. Vary the reaction rate constant, k, as suggested in the tutorial. Use values of the base case, 1 and 10 s^{-1}. Explain what happens to the outlet concentration.