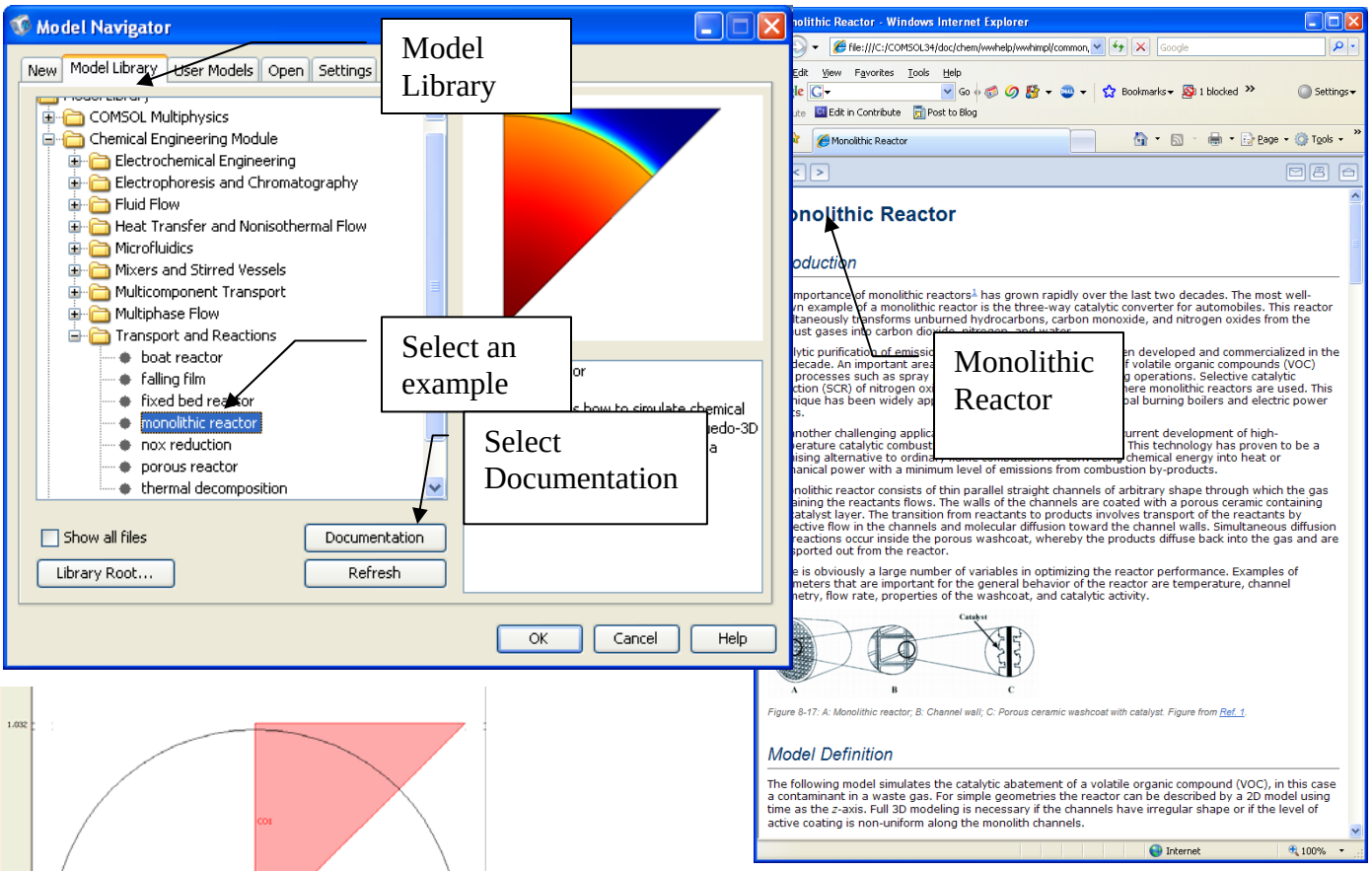
Open Comsol Multiphysics 3.2, Select the Model Library tab, Then select any model within the chemical engineering Module library as shown below. Then press the documentation button.

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



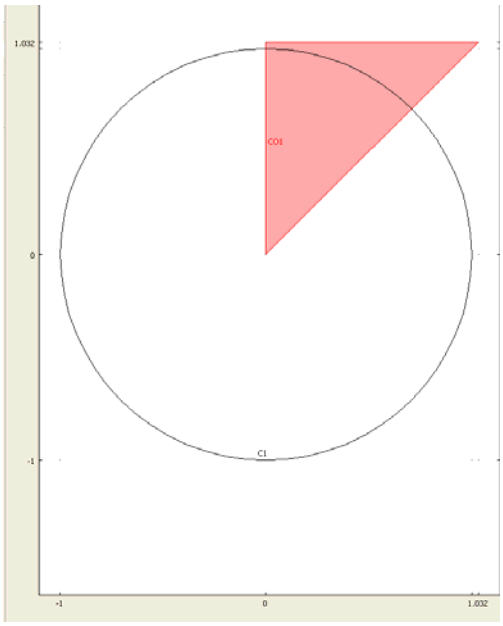


Figure 1: The first 2 shapes that you draw. C1 is the "elipse or E1" and CO1 is the triangle

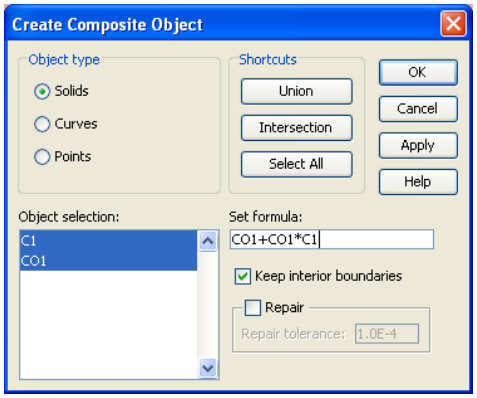


Figure 2: Correction in composite object formula from the Multiphysics 3.4 tutorial

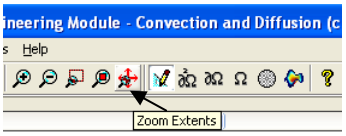


Figure 3: Zoom Extents button

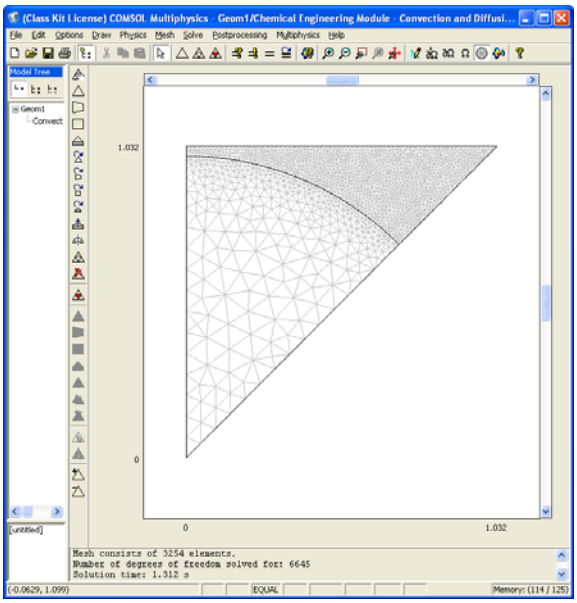


Figure 4: Meshing result

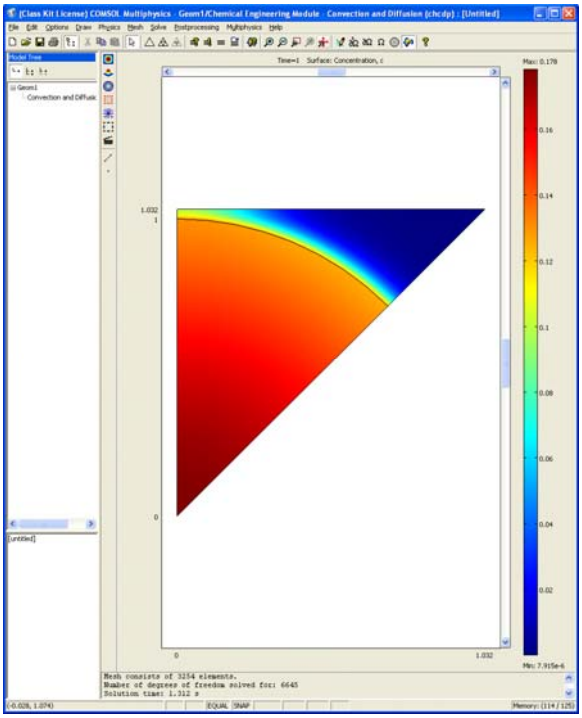


Figure 5: Default Surface Plot after solution solved

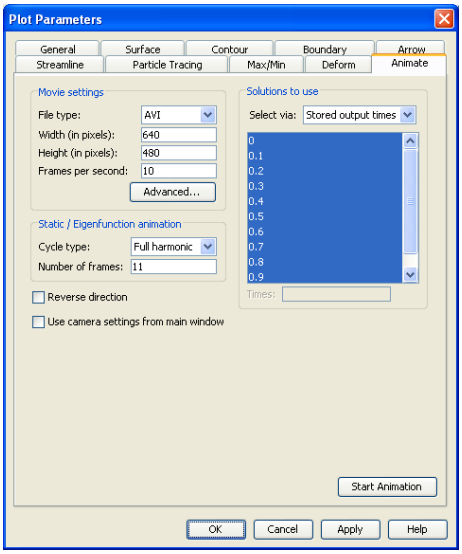


Figure 6: Plot Parameters, Animate Tab, press the Start Animation button

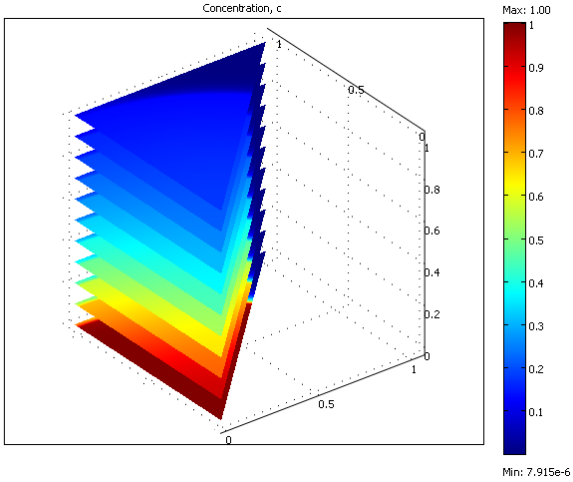


Figure 7: Result of Plotting both Domains 1 and 2 using the Domain Plot Parameters - Surface Plot (You can rotate this figure by clicking on it with your mouse and dragging it.)

Compute the average concentration at the inlet and outlet boundaries. The average concentration is given by

$\bar{c} = \frac{\iint (c * u_{dl}) dxdy}{\iint u_{dl}\, dxdy}$. To do this first calculate the average velocity and then calculate the integral of $c*u_{dl}$. Dividing these two numbers gives you the average velocity at a surface.

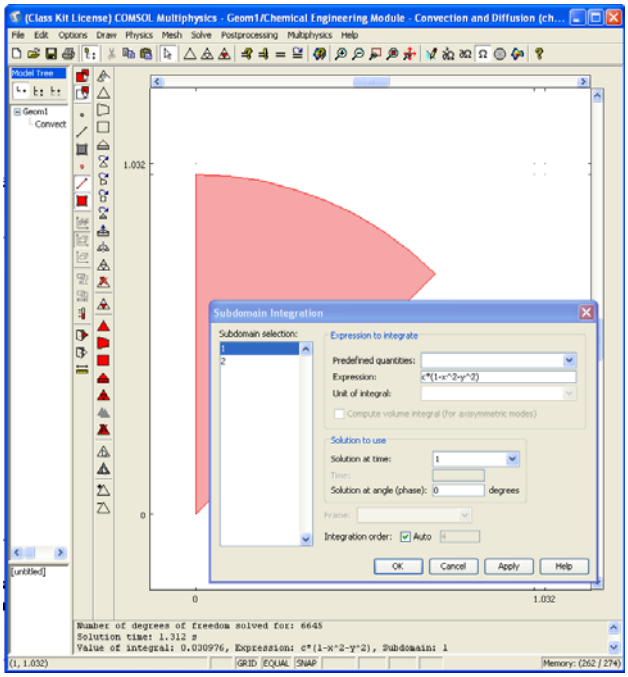


Figure 8: Subdomain Integration of the gas phase.

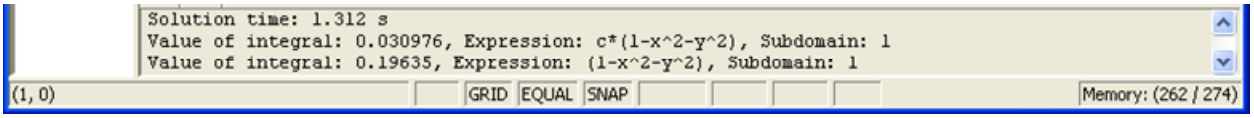


Figure 9: After the second integration

$$\bar{c} = \frac{\iint (c * u_{dl}) \mathrm{d}x\mathrm{d}y}{\iint u_{dl}\ \mathrm{d}x\mathrm{d}y} = \frac{0.030976}{0.19635} = 0.158$$