Introduction to COMSOL

The following flowchart gives the overall procedure for setting up a problem into COMSOL Multiphysics, solving the problem and then analyzing the information.

1. Read the problem statement
2. Select Equation(s) and coordinate system that you will be using. (e.g. momentum balance, energy balance etc.)
3. Enter constants
4. Draw Geometry
5. Specify Boundary Conditions – for all boundaries
6. Create a mesh or grid by dividing the geometry into smaller elements.
7. Solve equations
8. Graph numerical results

The Navier-Stokes Equations

The Navier-Stokes are a special form of the momentum balance and are discussed in section 15.4 of de Nevers. The following assumptions are employed in their use:

1. The fluid has constant density
2. The flow is laminar throughout
3. The fluid is Newtonian
4. The 3-dimensional stresses in a flowing, constant-density Newtonian fljid have the same form as the 3-demensinoal stress in a solid body that obeys Hooke's law (perfectly elastic, isotropic solid)

The Incompressible Navier-Stokes application mode in COMSOL is somewhat more general than this and is able to account for arbitrary variations in viscosity and small variations in density; say through the Boussinesq approximation.

This section provides a summary of the participating equations and boundary conditions for one of the most commonly used physical descriptions used in mathematical modeling.

EQUATIONS

The equations in the Navier-Stokes application mode are defined by Equation 4-1 for a variable viscosity and constant density. The momentum balances and continuity equation form a nonlinear system of equations with three and four coupled equations in 2D and 3D, respectively.

$$\rho\frac{\partial \mathbf{u}}{\partial t} - \nabla\cdot\eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} + \nabla p = \mathbf{F}$$

$$\nabla\cdot\mathbf{u} = 0 \tag{4-1}$$

$$\begin{pmatrix} Accumulation\ of \\ momentum\ per\ unit\ volume \end{pmatrix} - Shear\ Stress + rate\ of momentum + pressure\ forces = gravity\ forces$$

where η denotes the dynamic viscosity (M L^{-1} T^{-1}) (e.g. Newtonian viscosity with units kg/(m s), **u** the velocity vector (L T^{-1}), ρ the density of the fluid (M L^{-3}), *p* the pressure (M L^{-1} T^{-2}) and **F** is a body force term (M L^{-2} T^{-2}) such as gravity. The first equation is the *momentum balance*, and the second is the *equation of continuity* for incompressible fluids.

Remember from your text that symbol,∇, is a mathematical operator. The gradient or "grad" of a scalar field such as pressure is defined as:

$$\nabla p = \mathbf{i}\frac{\partial p}{\partial x} + \mathbf{j}\frac{\partial p}{\partial y} + \mathbf{k}\frac{\partial p}{\partial z} \qquad \text{Equation (C.6)}$$

In addition the dot product of the symbol,∇, with another vector is called the divergence or "div" of a vector ***v***.

$$\nabla \cdot \mathbf{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} \qquad \text{Equation (C.5)}$$

For constant density fluids at steady-state the continuity equation which is a mass balance on a continuous fluid gives

$$\nabla \cdot \mathbf{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0 \qquad \text{Equation 15.7}$$

The Navier-Stokes Equation (constant density and viscosity) for rectangular coordinates from the de Nevers book is given as:

$$\rho\frac{D\mathbf{v}}{Dt} = \rho\frac{\partial \mathbf{v}}{\partial t} + \rho(\mathbf{v}\cdot\nabla\mathbf{v}) = -\nabla p + \rho\mathbf{g} + \mu\nabla^2\mathbf{v} \qquad \text{Equation 15.27}$$

Where the substantial time derivative is defined as

$$\rho\frac{D\mathbf{v}}{Dt} = \rho\frac{\partial \mathbf{v}}{\partial t} + \rho(\mathbf{v}\cdot\nabla\mathbf{v})$$

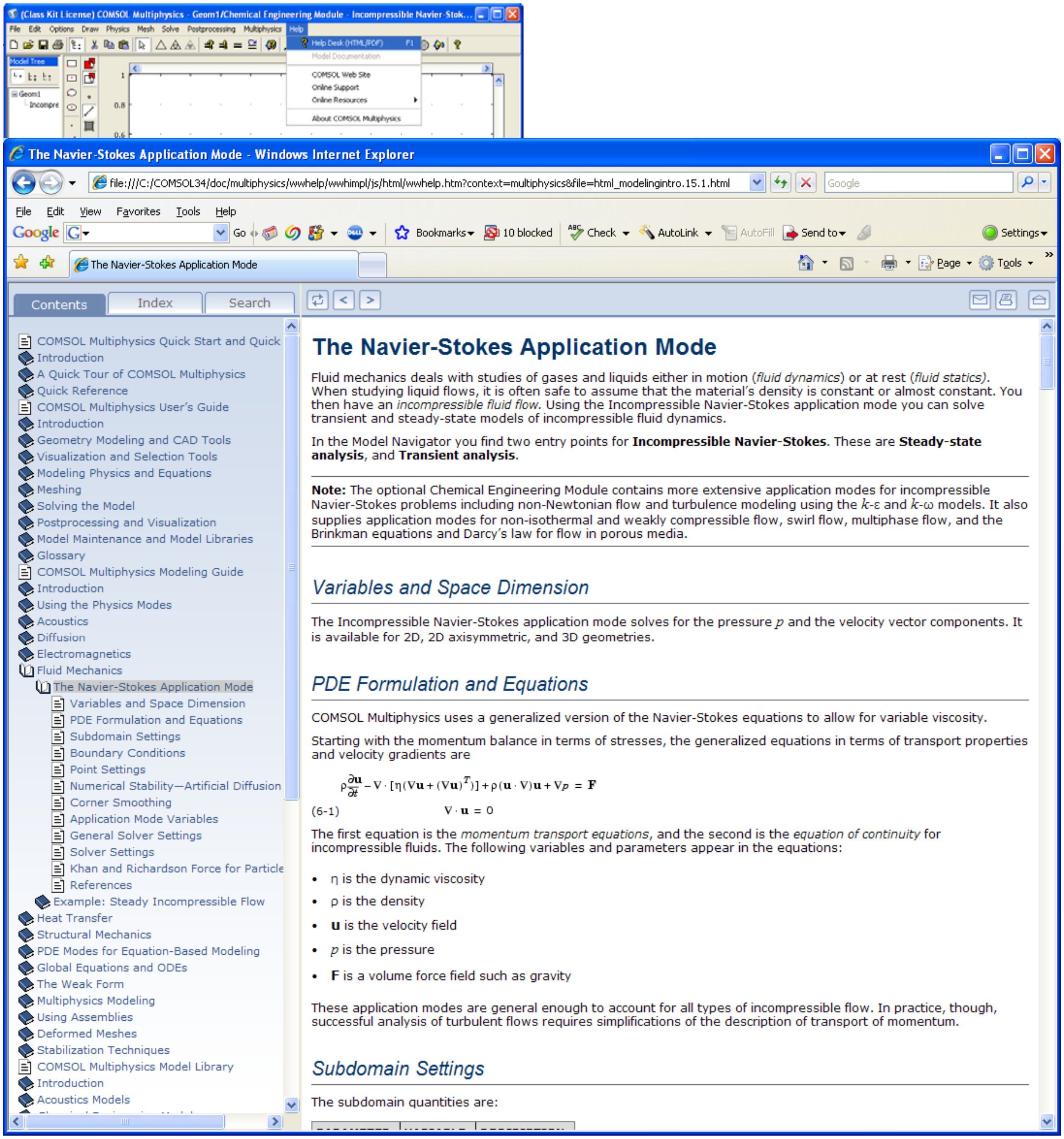
Boundary Conditions

The boundary conditions can be any of a number described below. A summary table is given in

Table 1: Summary of Common Boundary Conditions for Fluid Flow

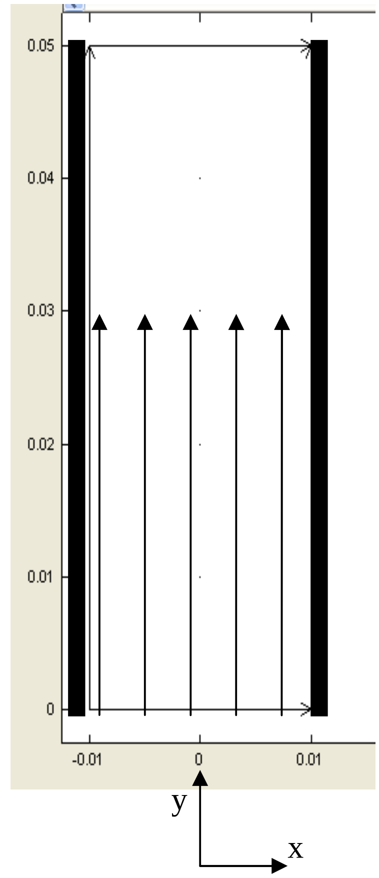
Boundary	Boundary Condition	Equation	Use
Wall			
	No-slip (default)	$\mathbf{u} = 0$	Used to specify that the velocity at a wall is zero.
	Sliding Wall	$\mathbf{u} = \mathbf{u}_w$	Used for a moving wall. You must specify the velocity of the wall.
Fluid Inlet			
	Inflow/Outflow boundary condition of velocity	$\mathbf{u} = -\mathbf{n}u_0$ or $\mathbf{u} = \mathbf{u}_0$	Used for a constant velocity with u=constant or a velocity specified by an equation such as $u = a\left(1 - \left(\frac{r}{R}\right)^2\right)$. The unit vector n is normal to the surface
	Pressure, No viscous Stress	$p = p_0$ and $\eta(\nabla \mathrm{u} + (\nabla \mathrm{u})^{\mathrm{T}})\mathbf{n}$	For this condition the inflow of fluid must be perpendicular to the boundary
Fluid Outlet			
	Velocity	$\mathbf{u} = -\mathbf{n}u_0$ or $\mathbf{u} = \mathbf{u}_0$	Becareful not to overspecify
	Pressure, No viscous Stress	$p = p_0$ and $\eta(\nabla \mathrm{u} + (\nabla \mathrm{u})^{\mathrm{T}})\mathbf{n}$	For this condition the outflow of fluid must be perpendicular to the boundary
Axis of Symmetry			
	Slip/Symmetry condition	$\mathbf{u} \cdot \mathbf{n} = 0$	This boundary condition states that there is no velocity perpendicular to an area or surface. This is used mainly in cases where there is symmetry. For example in pipe flow this is the boundary condition at r=0.
	Outflow/Pressure boundary condition	$p = p_0$	Used when pressure is known
	Normal flow/Pressure or “straight-out” boundary condition	$\mathbf{u} \cdot \mathbf{t} = \mathbf{0}$ and $p = p_0$	This is used for fully developed flow in which the flow is only perpendicular to the area.

More information about the Navier-Stokes Application Mode can be found in the Modeling Guide, Fluid Mechanics Chapter. To to Help, Help Desk (HTML/PDF). Choose Modeling Guide and then Fluid Mechanics.



The following is a simple example of use of the Incompressible Navier-Stokes application mode in the Chemical Engineering Module. It will acquaint you with the different menus and operations, as well as provide a study of developing laminar flow.

Steady-State Flow Between Parallel Plates

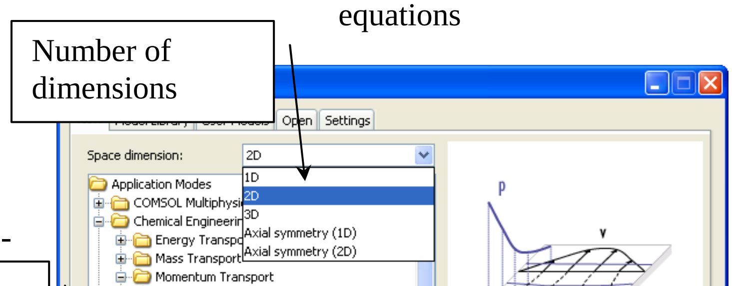


In this problem you will model the steady-state flow of a fluid that is being pumped through 2 vertical plates, with infinite width and having a spacing of 0.02 m as shown in the adjacent figure. The vertical direction coordinate is y and the horizontal direction coordinate is x. The fluid is not water and it has a uniform inlet velocity of 2×10^{-2} m/s, viscosity of 0.01 kg/(m s) and density of 1000 kg/m^3. The effect of gravity will be ignored for this problem. (**The usual practice with accounting for gravity in pump problems is to combine it into the pressure term**).

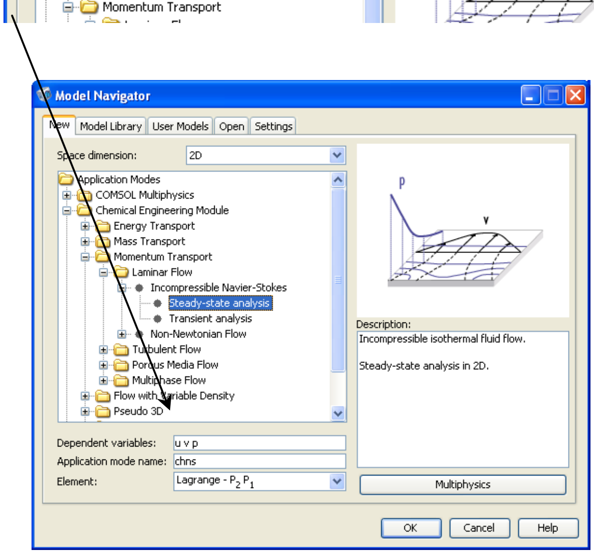
1. The initial part of this problem will be to specify a uniform velocity of 0.02 m/s at the entrance of the plates and determine a velocity profile. This problem will show a uniform velocity profile that transitions into fully developed flow which is a parabolic velocity profile. Formulate a momentum balance for this problem and obtain the analytical solution for the velocity profile.
2. In the second part of this problem you will specify a pressure drop and then calculate a velocity profile. In this case the velocity profile will be fully developed from the start of the geometry.

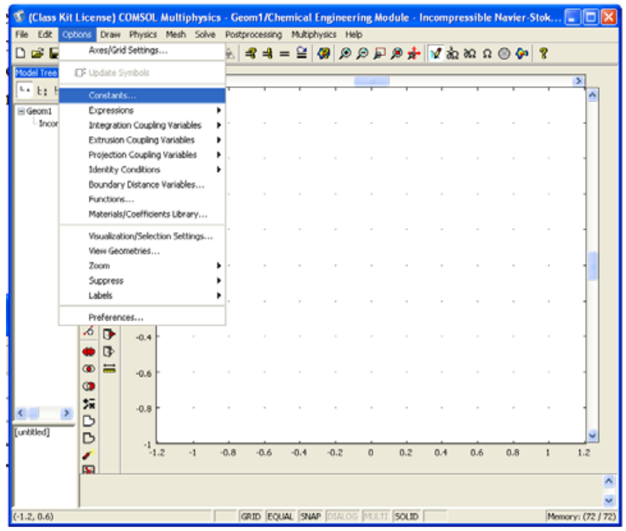
Modeling using the Graphical User Interface

1. Start COMSOL Multiphysics
2. In the **Model Navigator** , click the **New** page
3. Select the 2-dimension problem set of equations
4. Select **Chemical Engineering Module**, **Momentum Transport**, **Incompressible Navier-Stokes**, Steady-state analysis
5. Take note of the variables: u, v and ρ. The variable u is in the x direction and the variable v is in the y direction for this 2-diemsional problem.
6. Click **OK.**



Variables used u, v and ρ.



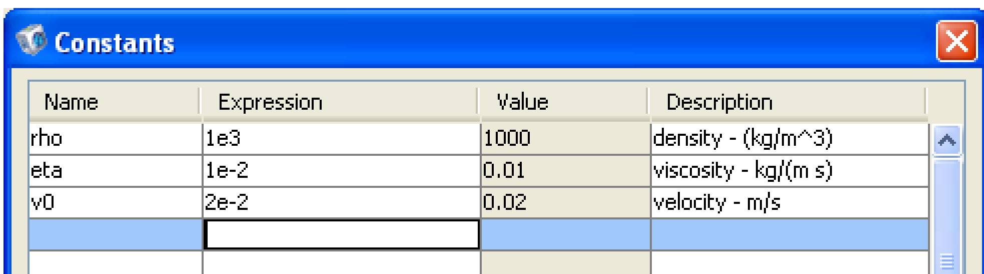


Options and Settings

The first step in the modeling process is to create a temporary database for the input data. The module suggests units in SI and expects you to use consistent set of units. In this case, all units are SI units. Define the following constants in the **Constants** dialog box in the **Option** menu. Note this is not water and it has a uniform inlet velocity of 2×10^{-2} m/s, viscosity of 0.01 kg/(m s) and density of 1000 kg/m^3.

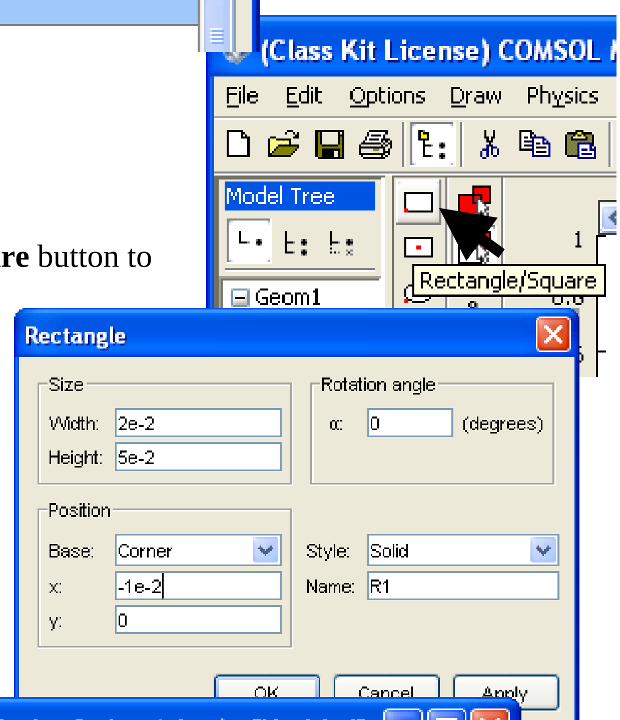
Table 2: Constants

NAME	EXPRESSION
rho	1e3
eta	1e-2
v0	2e-2



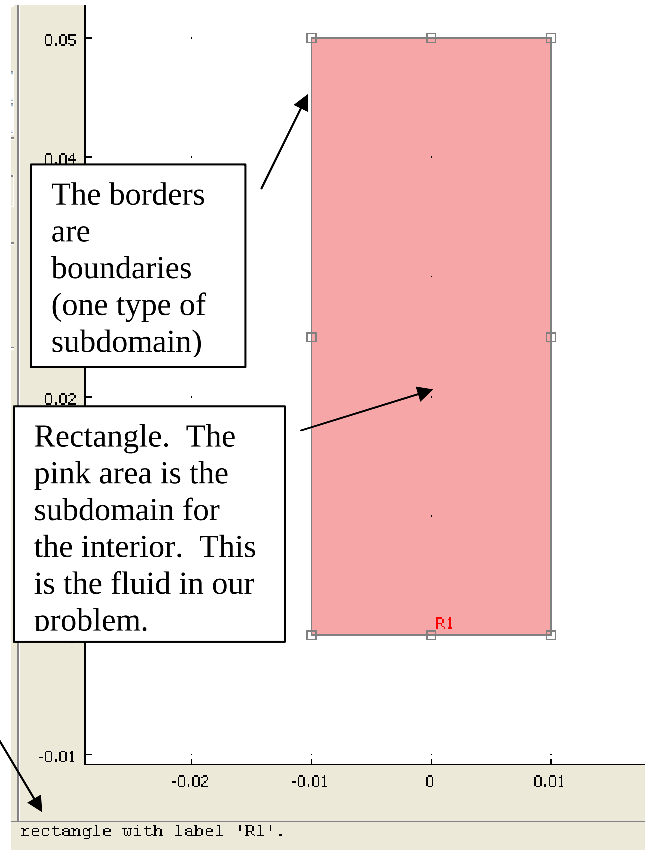
Geometry Modeling (Create Parallel Plates)

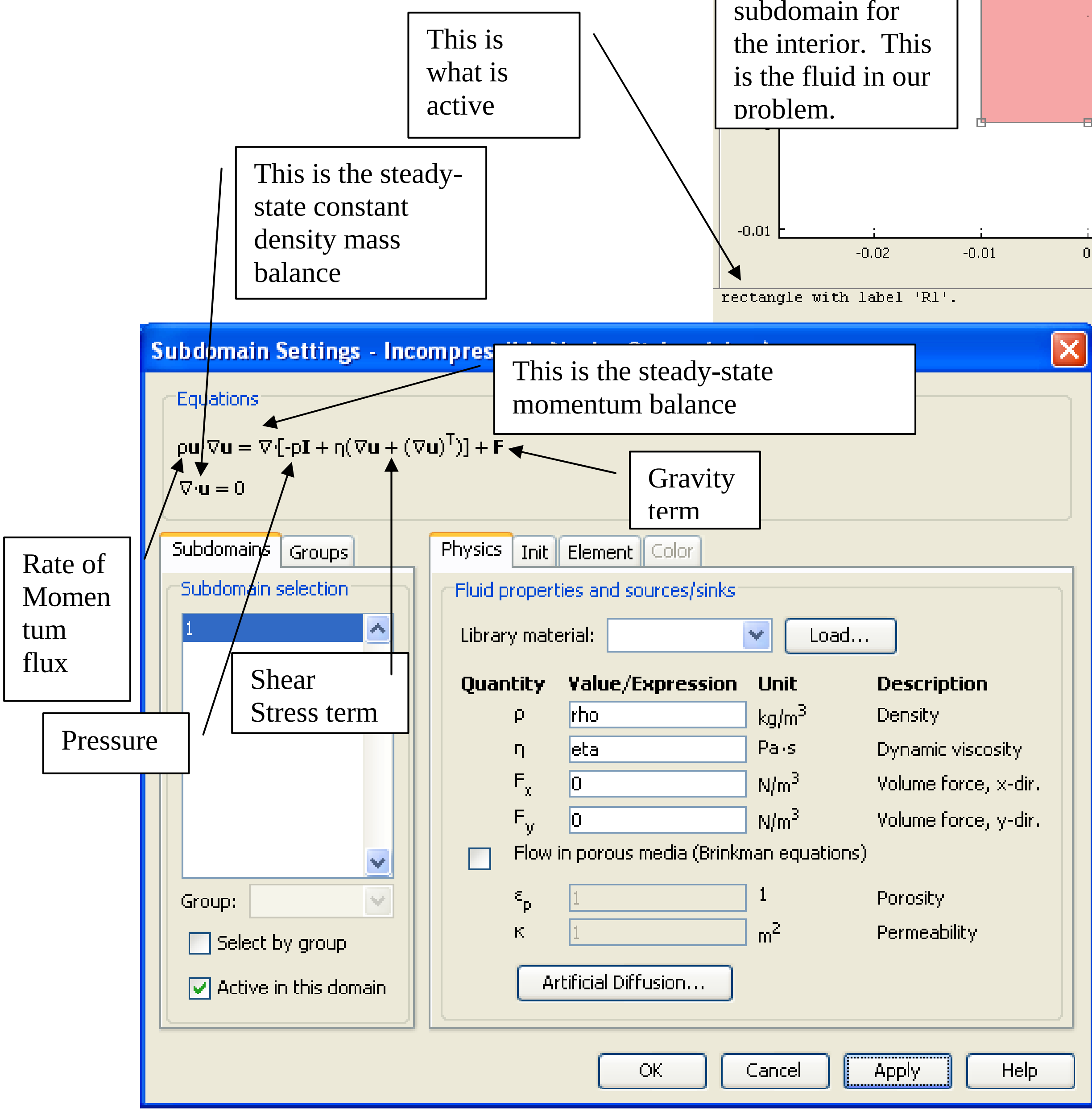
1. Press the **Shift** key and click the **Rectangle/Square** button to bring up the menu for the Rectangle/Square.
2. Type the values shown in the adjacent figure for the rectangle dimensions.
3. 2e-2	Click the **Zoom Extents** button in the Main toolbar.
4. Now you have created a domain that has 4 boundaries. Comsol labels as subdomains the 4 boundaries and the interior of the rectangle.



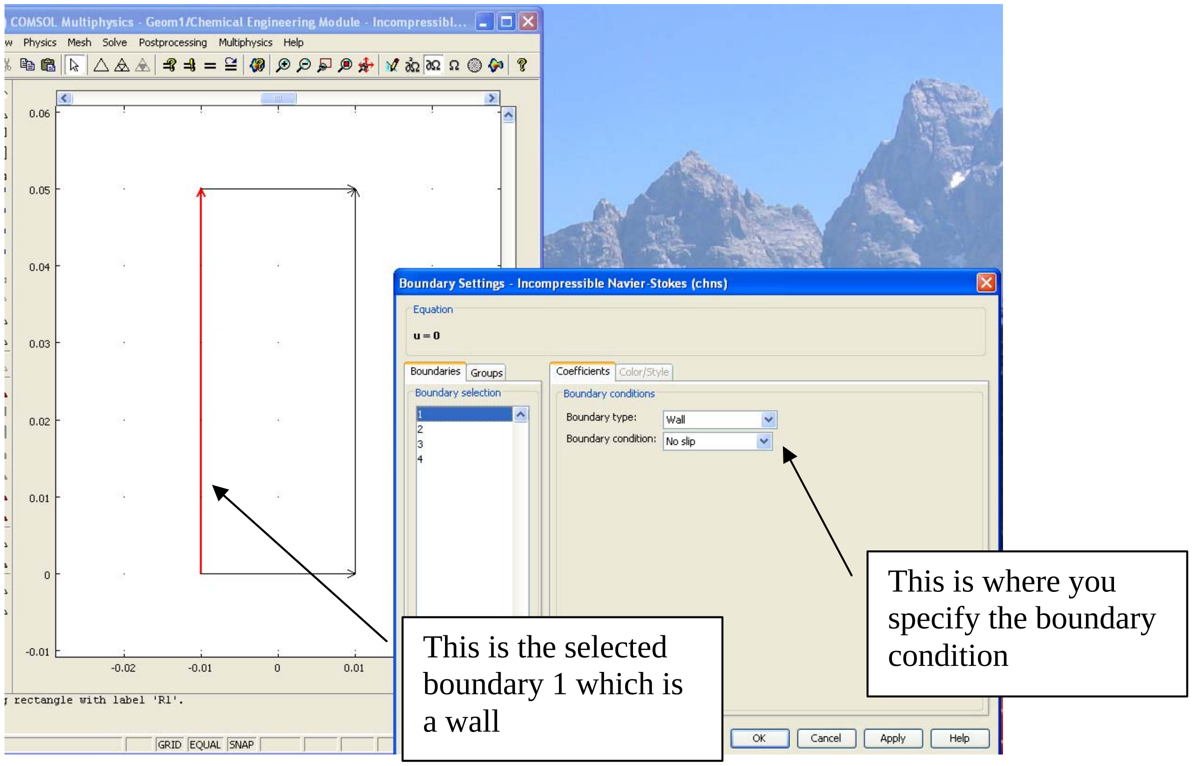
Subdomain Settings

This refers to inside the rectangle that should have a designation of R1. You will now specify the properties of the fluid inside the four boundaries or of the subdomain. Select **Subdomain Settings** from the **Physics** menu. Define the physical properties of the fluid, in this case viscosity and density, according to the image with heading Subdomain Settnigs – Incompressible Navier-Stokes (chns) given below. You must type in the same variable names as was given in Table 2.



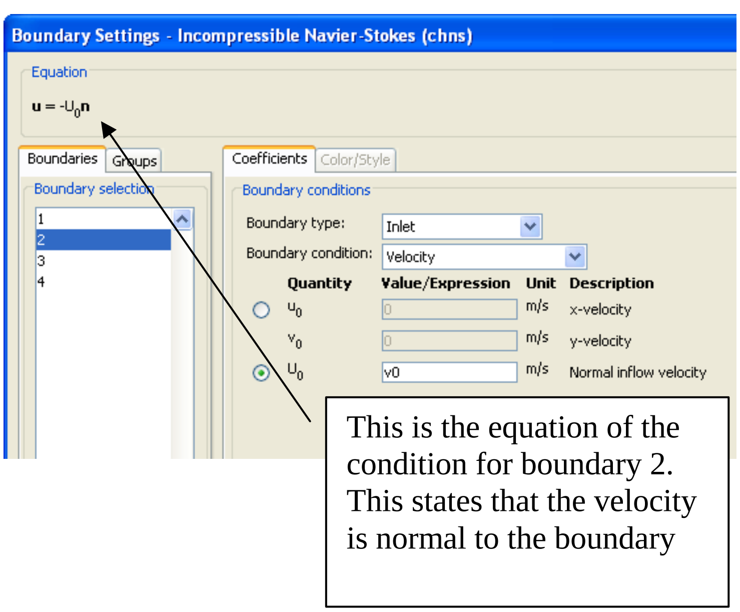


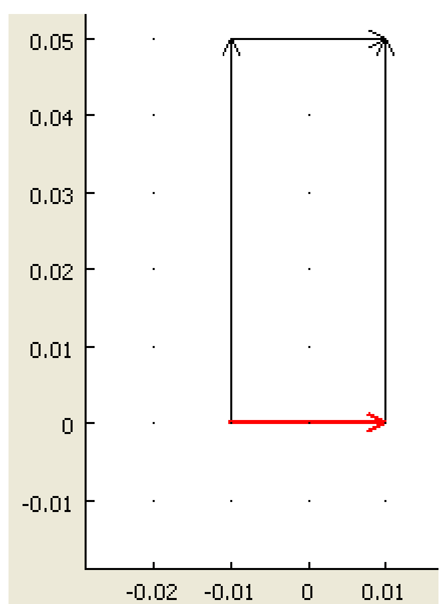
Boundary Conditions

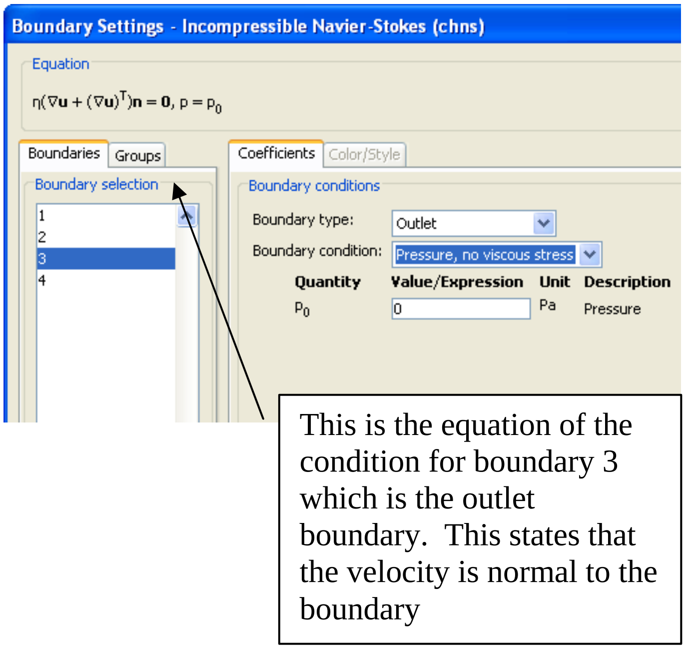


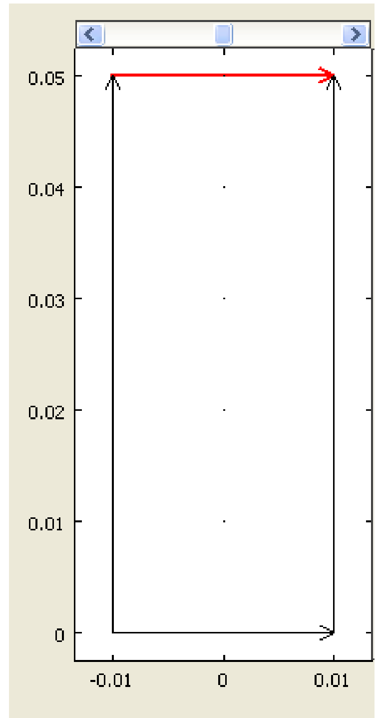
Select **Boundary Settings** from the **Physics** menu and enter the boundary conditions according to the following table:

BOUNDARY	1,4	2	3
Boundary condition	Wall No-slip	Inflow/Outflow velocity	Outlet Pressure, no viscous stress
U_0		v0	
P_0			0





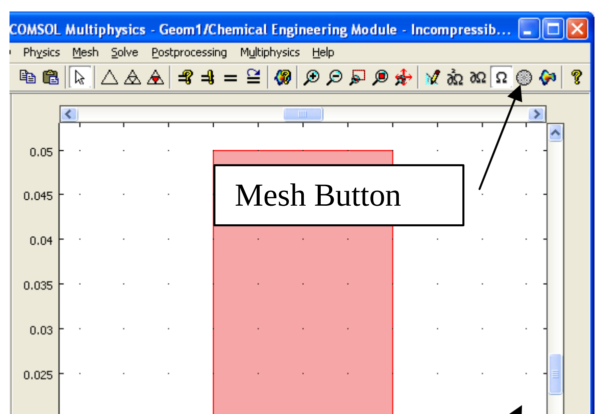


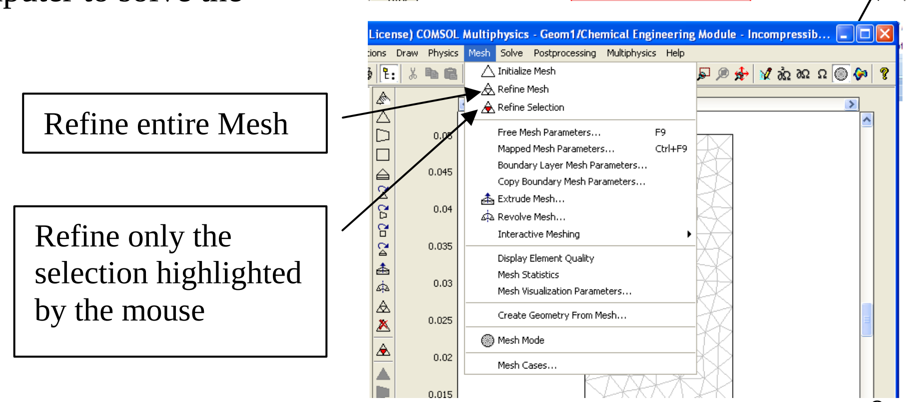


Why do you have to specify a boundary condition for the outlet? Remember that COMSOL uses a finite element method to obtain numerical results. This means that the solution is a set of numbers. When we integrated the equation we came up with another equation. In this regard the pressure derivative is an additional equation and requires an additional specification. This specification is to set the outlet pressure to 0 Pa gauge.

Mesh Generation

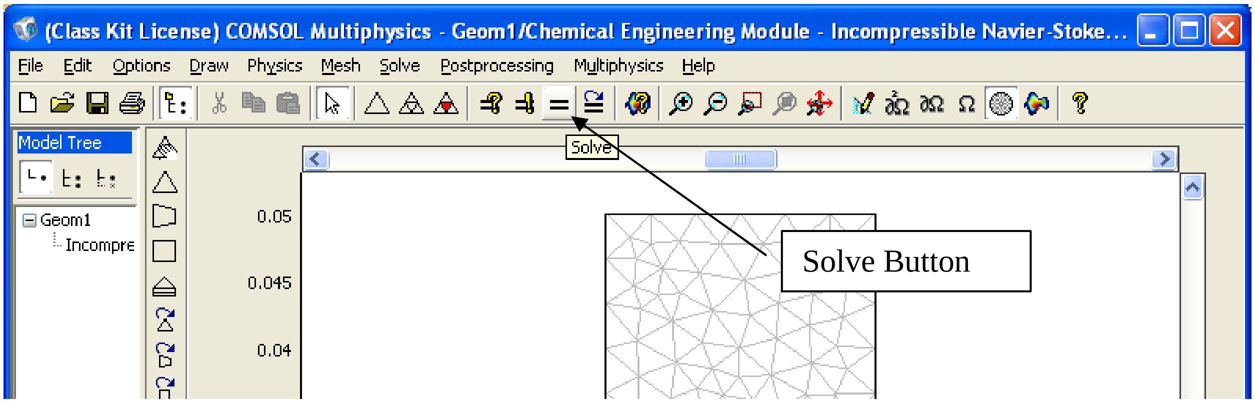
Select the mesh generation button or select Mesh and then Initialize Mesh. Later you can return to this to either make the mesh size smaller (Refine Mesh) for the entire subdomain (rectangle) or select a small portion of the subdomain where the velocity is changing rapidly and make that mesh size smaller. Remember the more elements the longer it will take the computer to solve the problem.





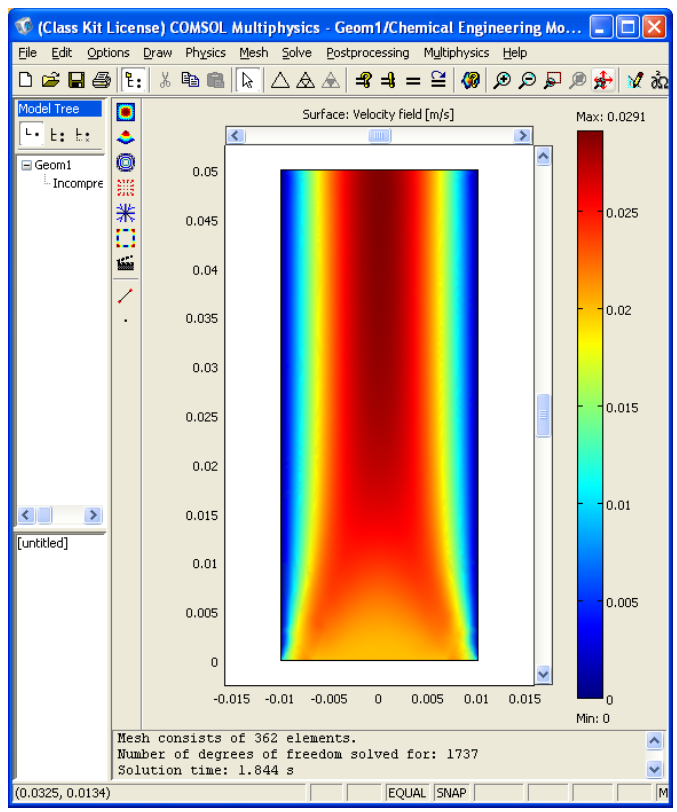
Computing the Solution

Click the **Solve** button in the Main toolbar.



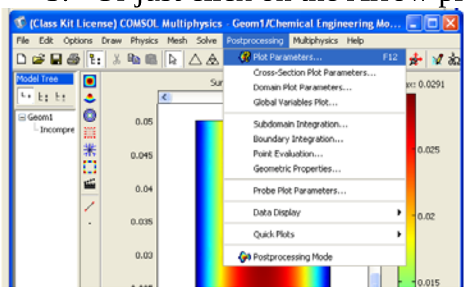
Postprocessing and Visualization

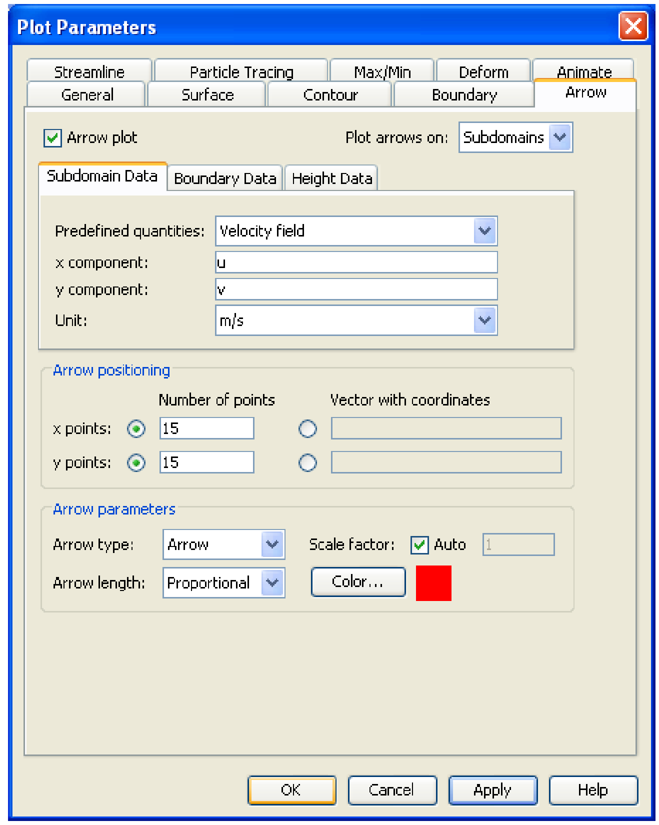
The default figure shows the velocity field, which is the absolute value of the velocity vector. In this graph notice that the flow transitions from a uniform velocity at y=0 and a full developed velocity profile. Determine at what y distance the velocity profile can be considered fully developed. Explain your answer.

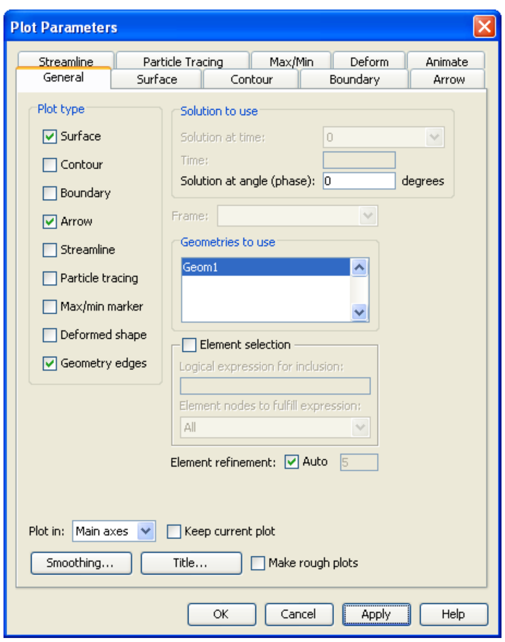


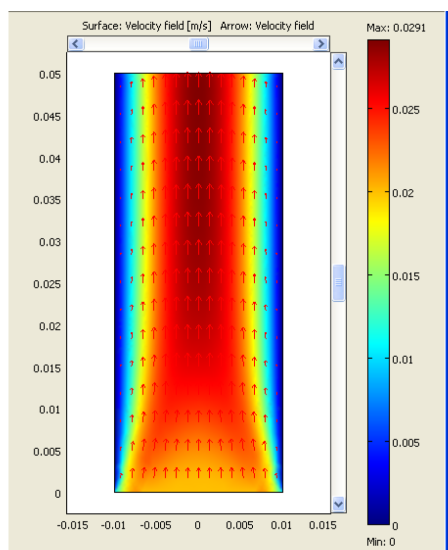
Add arrows showing the magnitude and direction of the velocity. Do the following

1. Select Postprocessing
2. Plot Parameters
3. Choose the Arrow tab
4. Click on Arrow Plot.
5. Or just click on the Arrow plot box

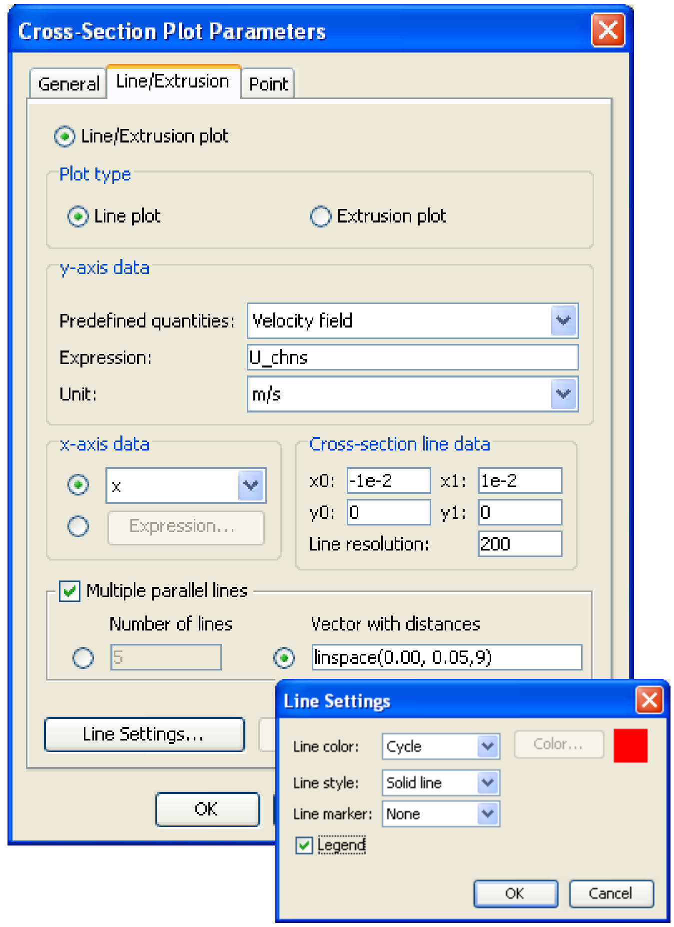






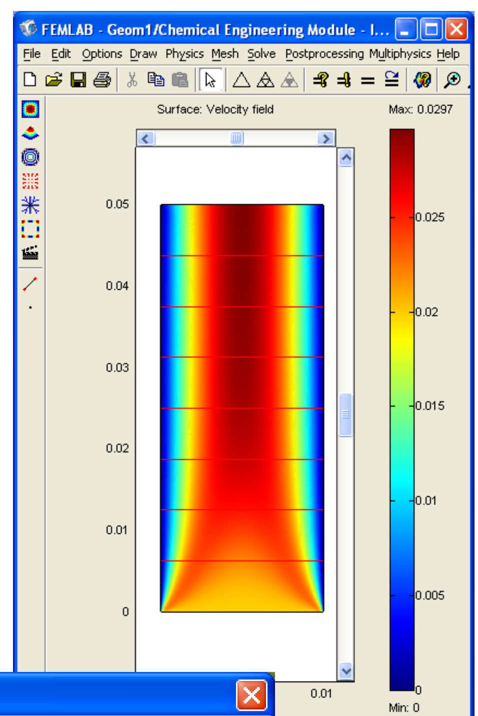


It is also possible to create cross-sectional plots of the velocity along horizontal lines at different positions along the channel.

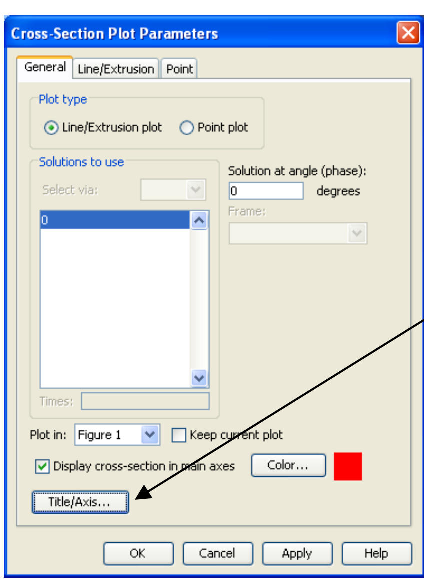


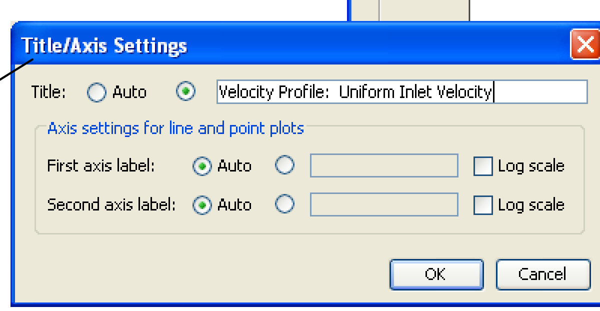
1. Select **Postprocessing**
2. Select **Cross-Section Plot Parameters**
3. Select **Line/Extrusion plot** as **Plot type**.
4. Click the **Line/Extrusion** tab.
5. The **y-axis data** is by default set to **Velocity field.**
6. Select x as the x-axis data from the drop down menu.
7. Set the **Cross-section line data** according to the table below:

PROPERTY	VALUE
x0	-1e-2
x1	1e-2
y0	0
y1	0



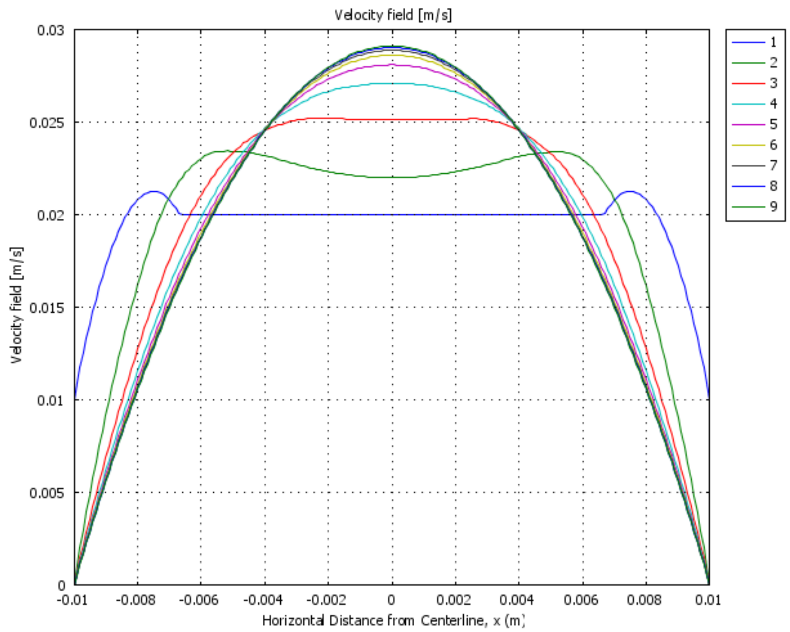
8. Select the **Multiple parallel lines** check box.
9. Select the **Vector with distances** button and type `linspace(0.00,0.05,9)` in the **Vector with distances** edit field. The linspace command will create an equidistant vector from 0.00 m to 0.05 m with 9 steps.
10. Select Line Settings and click on the Legend box
11. Go back to the General tab
12. Click on Title/Axis ButtonAdd a title: Velocity Profile: Uniform Inlet Velocity
13. Click **OK**.



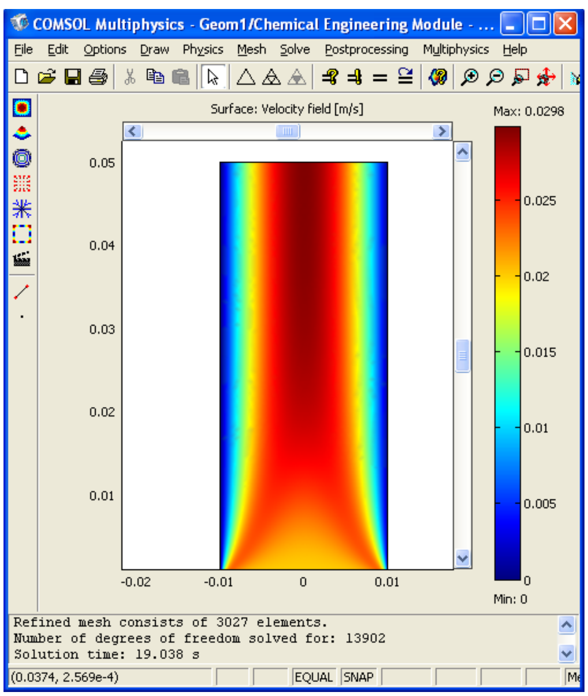


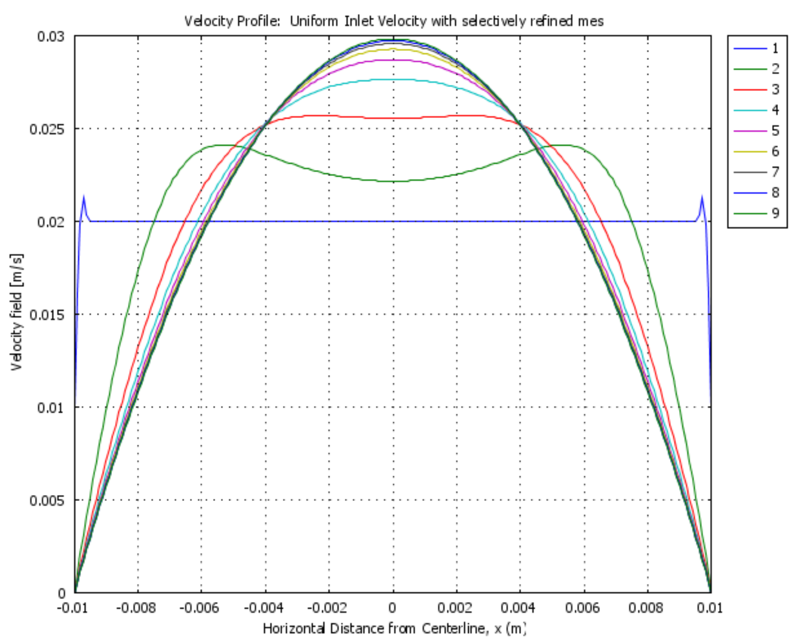
This gives a plot in which velocity is on the y axis and the width of the plate is on the x-axis. Each curve starting with the blue line corresponds to a given distance in the flow direction (y direction). Notice the appearance of the horizontal red lines on the original post process contour solution plot shown above. These lines are the 9 cross sections that were requested.

The resulting plot shows how the velocity profile develops along the main direction of the flow. At the outlet, we can see that the flow appears nearly to be fully developed and is a parabolic velocity profile. To refine the solution at the initial boundary, you can refine the mesh and then solve again.



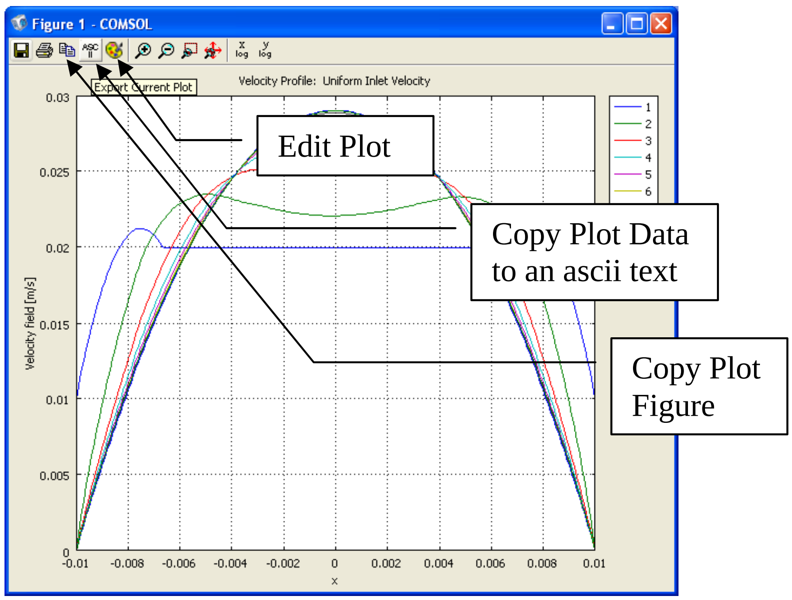
The plot below was produced by selecting the button refine selection and then dragging your mouse over the mesh in the entrance region of the slot. Also if you wish to visually compare plots you need to Notice how the problem with the discontinuity reduces.

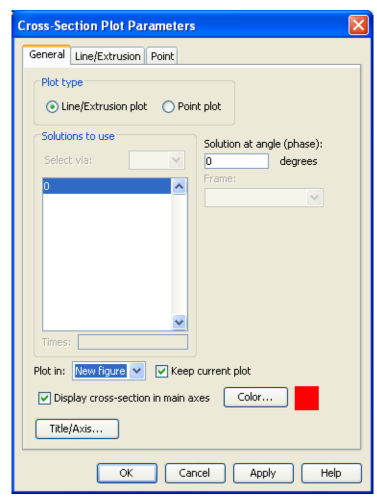




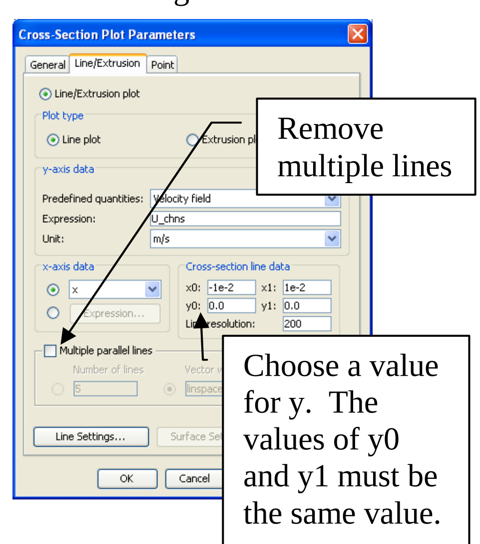
Exporting data to excel

You can export this data to excel in a number of ways. From the postprocessing user interface you can save the data to a text file or you can use the commands **File>Export>Postprocessing Data**.

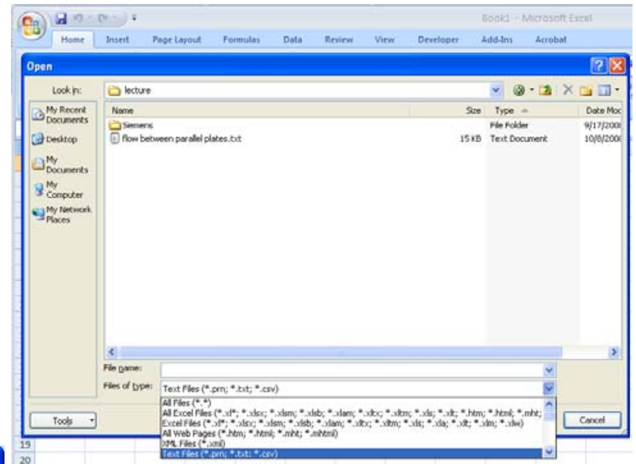




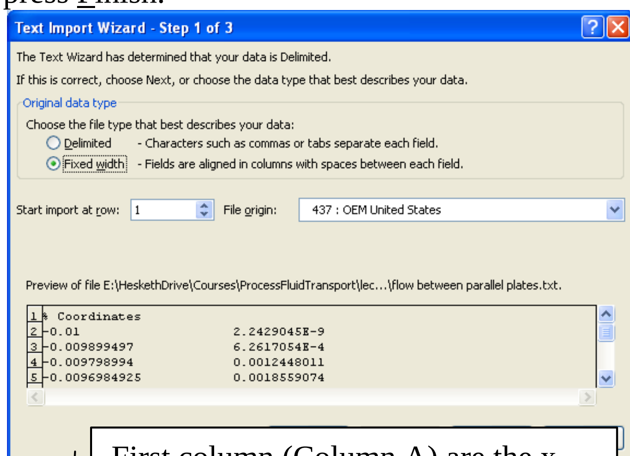
1. Make an additional plot in excel comparing the analytical solution to the fully developed flow profile in this solution. You will only need one of the COMSOL solutions. Which solution plotted is a fully developed solution? (e.g. at what y value is the flow fully developed?)
2. Once you have decided what should be the fully developed flow solution, then make a second plot by first selecting a New figure and clicking the box "Keep current plot." For this new plot you should only plot the fully developed flow profile. Go the the Line/Extrusion tab and remove the multiple parallel lines check mark and specify the y value. The y value plotted with the numbers given in the image on this page would be the inlet flow velocity (y0=0 and y1=0). This is not the fully developed flow profile!
3. ***Exporting the Current COMSOL Plot to a File***
   There are several methods to export the data to a text file that can be imported into a spreadsheet. I recommend using the short cut button. The short cut button is located on the Figure 1 plot of Comsol shown above. Save this data as a text file and then import it into a spreadsheet.



4. Open the text file. The data will be in the following form: x,y coordinates for each time that is given in the legend of the Comsol Figure 1 plot. You will need to open this data with an excel spreadsheet. Make sure you are searching for a *.txt file in the Files of type: line.

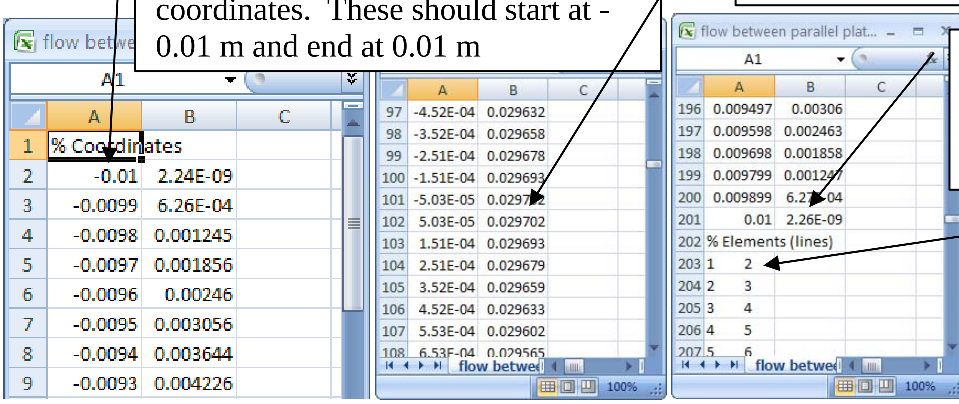


5. Next a Text Import Wizard will open. Select fixed width and then Next. Check to see that the data is being imported in two columns by scrolling down with the scroll bar and then press Finish.



First column (Column A) are the x coordinates. These should start at -0.01 m and end at 0.01 m

The second column contains the velocity values. The maximum velocity is at 0.0297 m/s in this

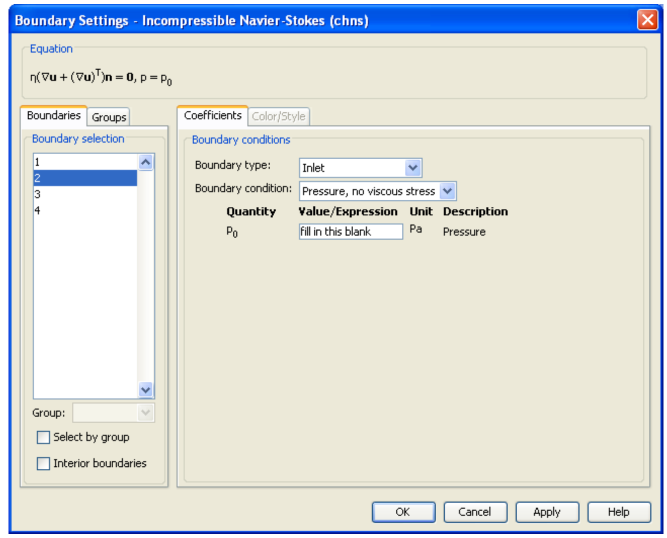


This is the last velocity value at 0.01 m. Notice that it is not zero because of numerical errors

After the data are given information on the number of elements used. We do not use this for the plot

6. You should now have 2 columns of data in the spreadsheet. The data in the column A is the distance coordinate in meters and the data in column B is the velocity in m/s. If you are exporting more than one data set, (which you should not do for this exercise – but may do in later problems) then each data set corresponds to a new value of y in your Comsol plot.

Problem 2: Fully developed flow between 2 parallel plates



In this problem we will assume that the flow is fully developed at the inlet of the volume. You will then specify the inlet pressure and obtain a velocity profile. To do this you will need to calculate the pressure drop by hand.

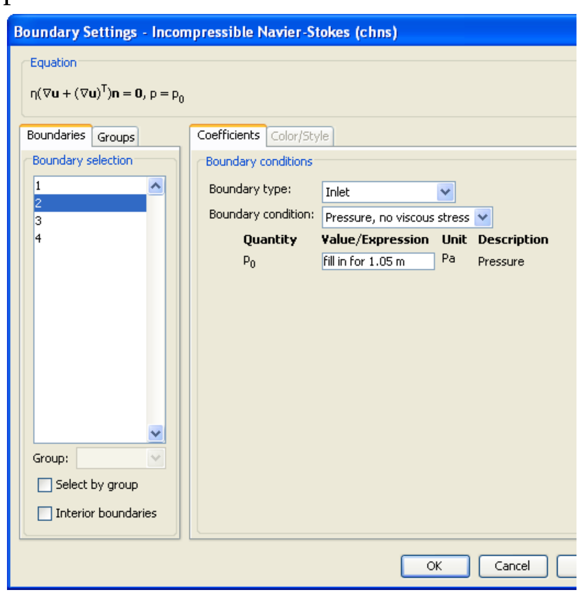
The solution given in many text books for this problem is $v_y = -\dfrac{\delta^2}{2\mu}\dfrac{dp}{dy}\left(1-\left(\dfrac{x}{\delta}\right)^2\right)$

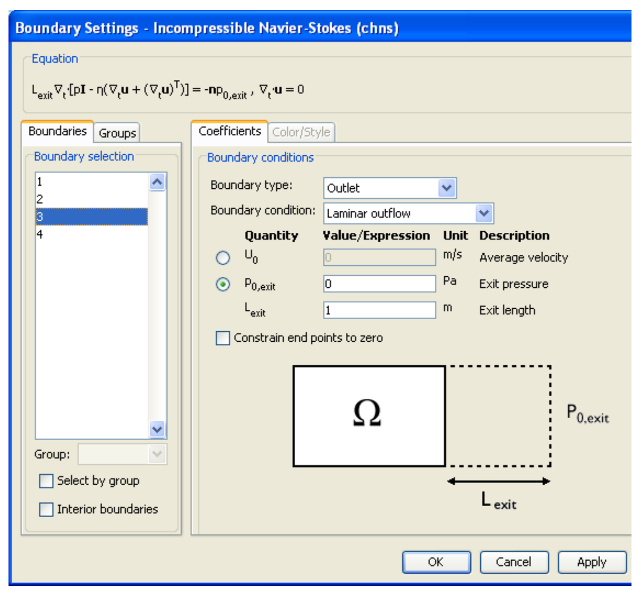
The maximum velocity is given as $v_{y\,\max} = -\dfrac{\delta^2}{2\mu}\dfrac{dp}{dy}$ which occurs at the center of the channel $x = 0$. Using the maximum velocity of the fully developed flow from your previous simulations and physical properties and dimensions calculate the pressure drop.

Use this value of pressure to specify the inlet boundary condition. Remember to convert pressure drop into pressure.

For the outlet boundary condition there is a special setting:

Adjust your pressure drop for a distance of 1.05 m and then use a gauge pressure with the outlet pressure at zero

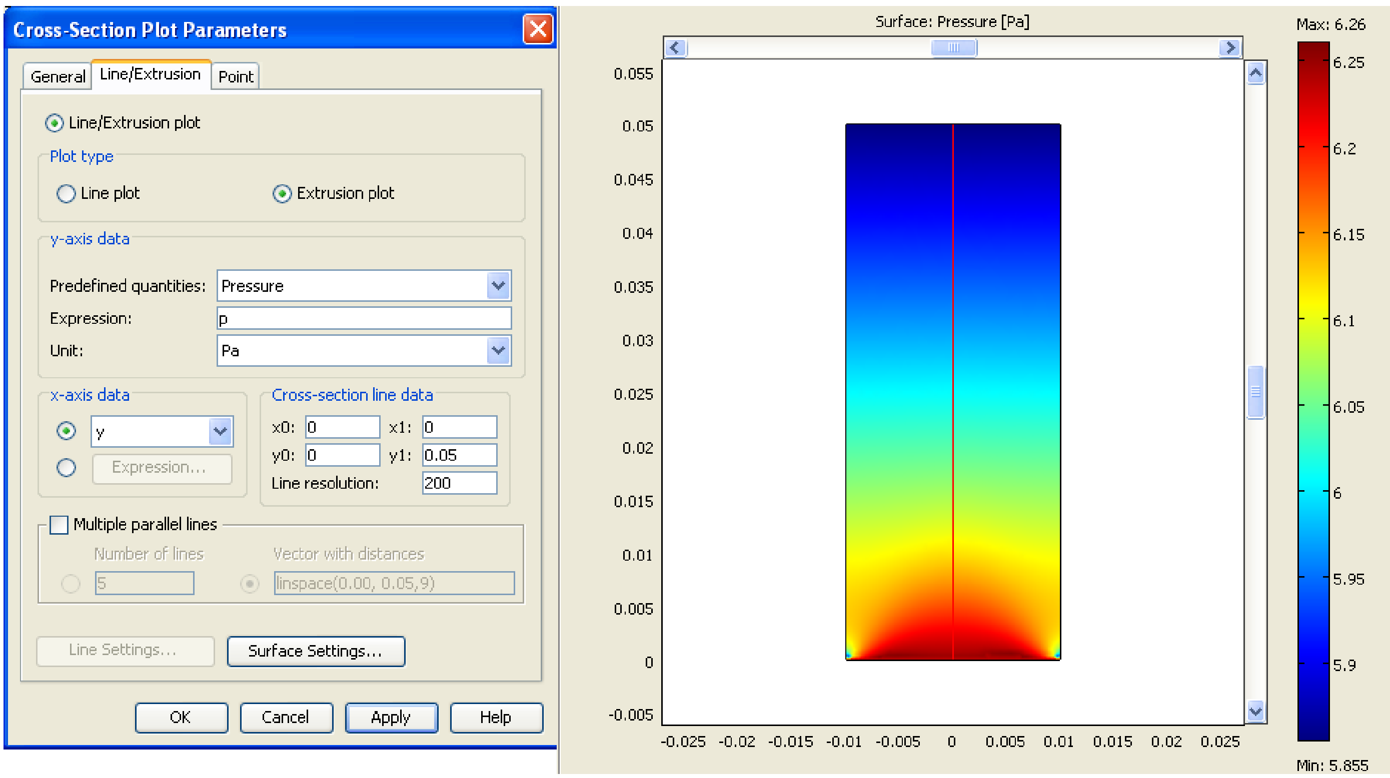


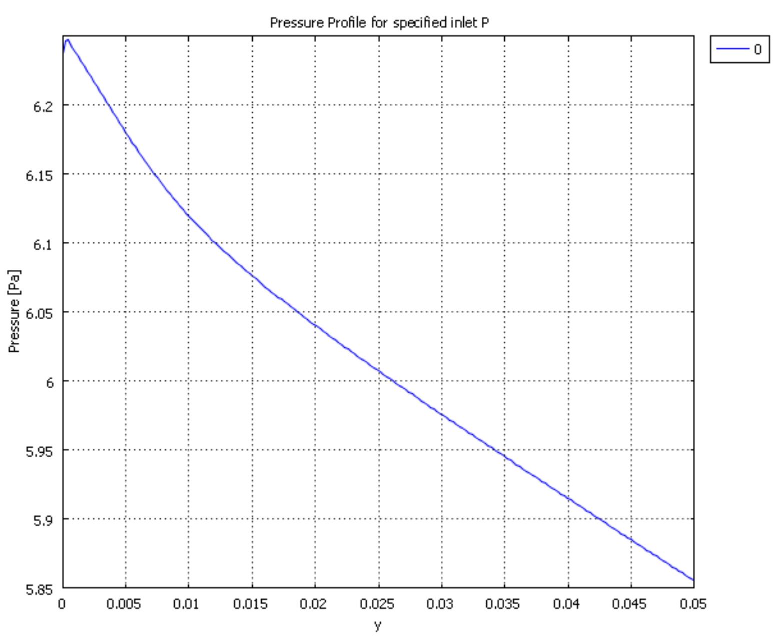


Since this is not a fully developed flow for the entire channel you should examine where the flow is fully developed.

Plot the pressure profile using the surface plot pressure and a Cross section plot.

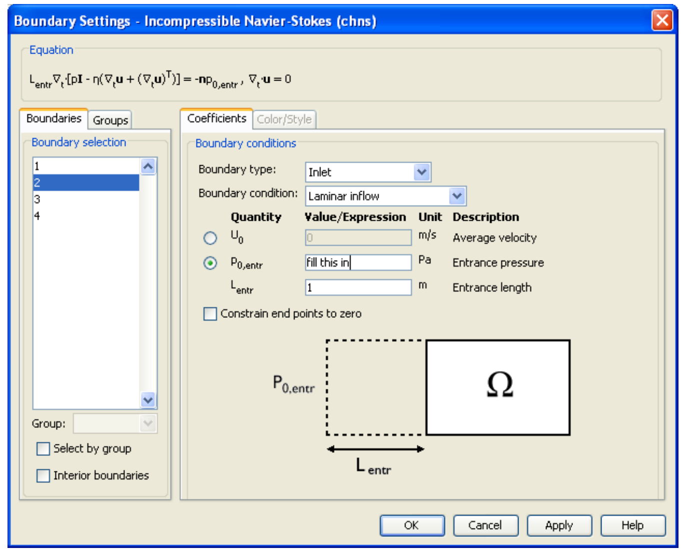
Explain why there is a difference between the finite element method solution (COMSOL) and the analytical solution.

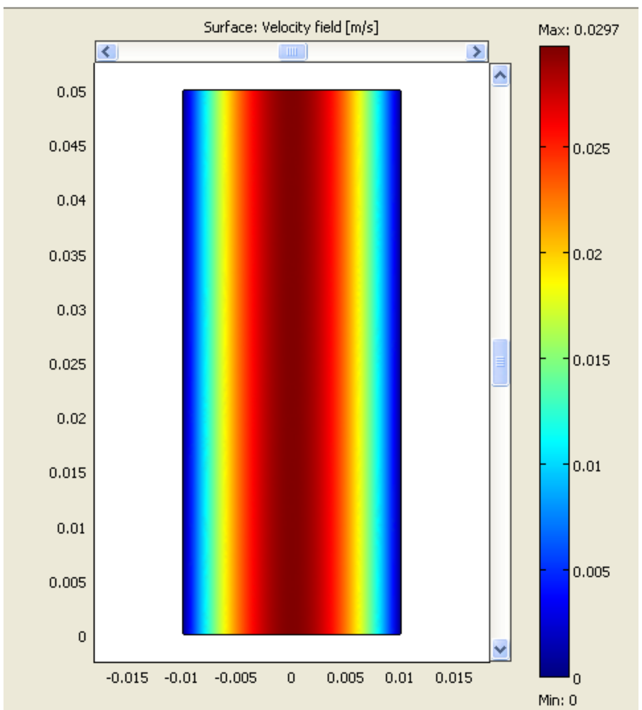


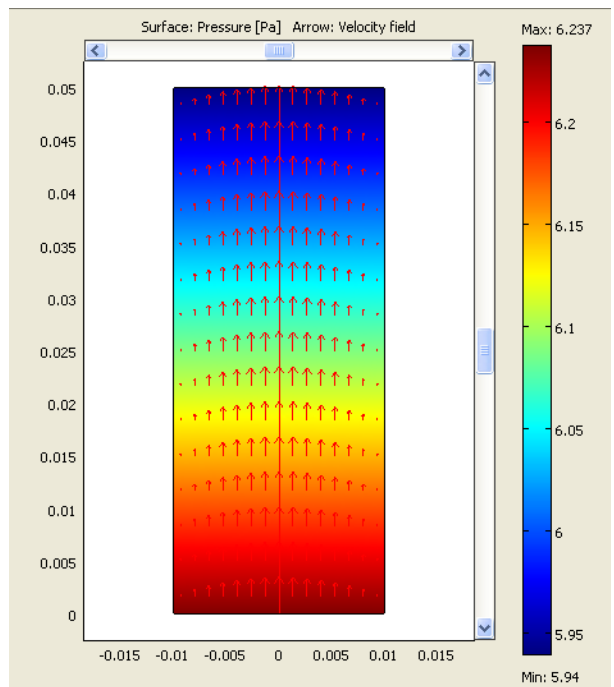


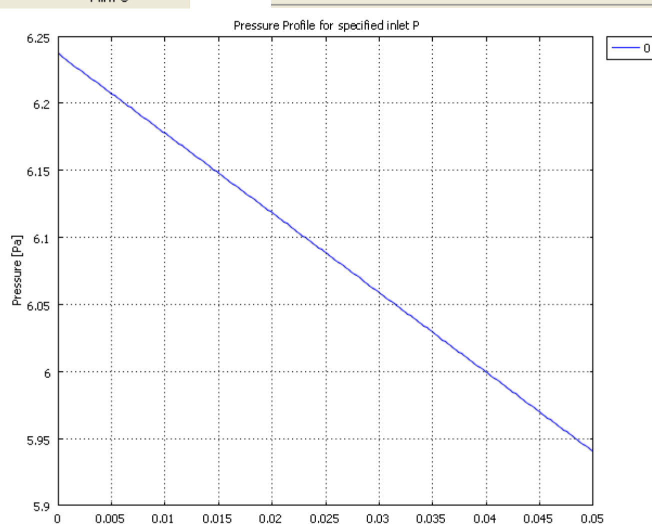
Comment on why this plot is not a straight line.

If you wanted a fully developed flow entering the inlet and leaving the outlet then you would need to change you inlet boundary condition and recalculate your needed inlet pressure.







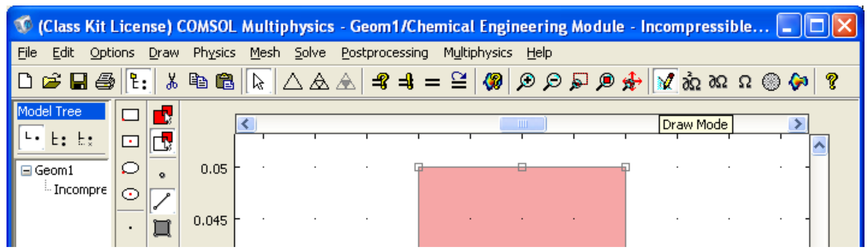


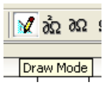
Laminar Flow – Falling film

Now solve a problem using a gravity term. In this case you will simulate a vertical plate with liquid flowing down the plate. You have no applied pressure terms, but you will use a force term. Compare your simulation to an analytical solution that you obtain from first principles using an excel plot. Show this derivation.

Geometry

Change your geometry so that the film thickness is 0.002m and the length of the plate is 0.01 m. Set the free film at x = 0 and the wall at x = 0.002 m. Use the Draw mode button which is a triangle and

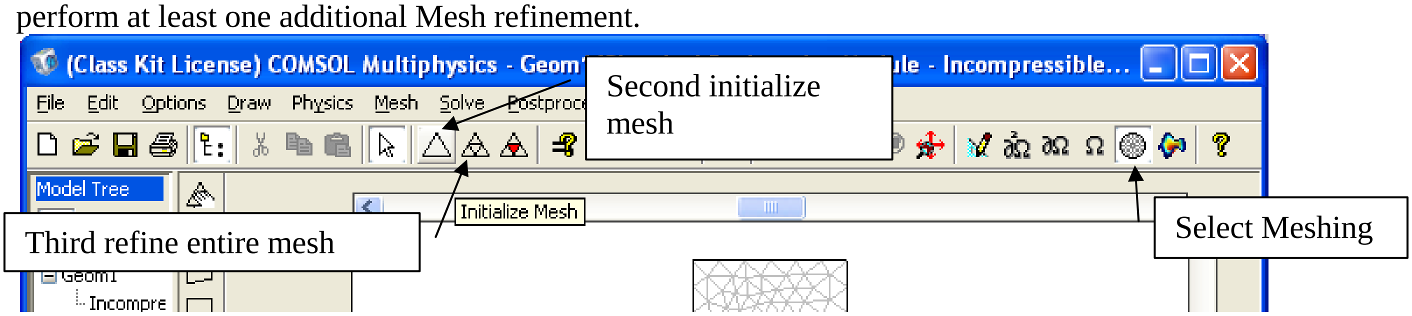


pencil. 

Then double click on the figure and make the rectangle smaller. Also place the orgin (corner of the rectangle) at point 0,0.

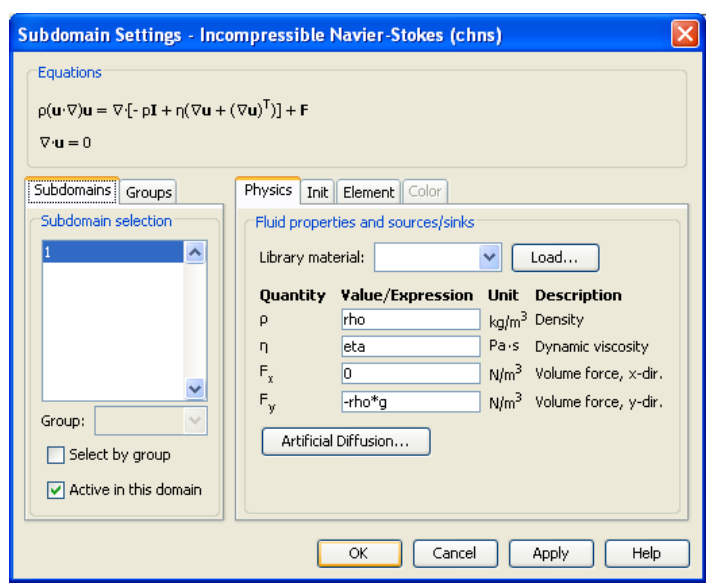
Mesh

Re-grid your new geometry by first selecting the Mesh button, then initialize the mesh and then perform at least one additional Mesh refinement.



Subdomain Settings

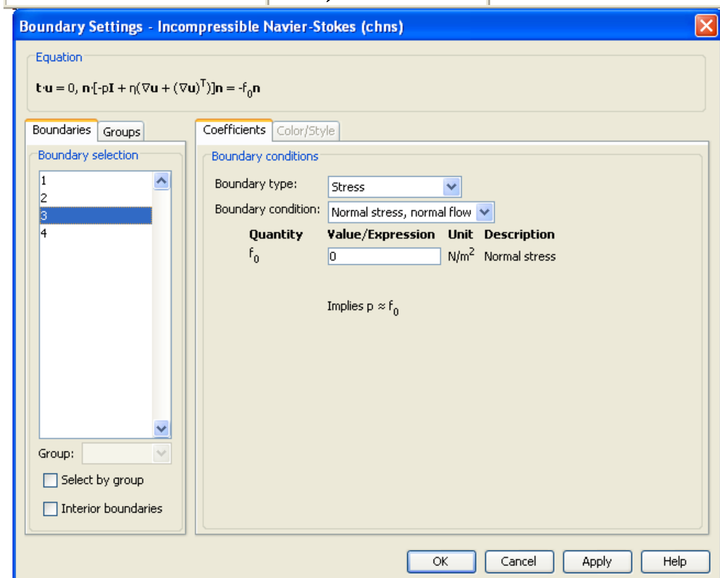
The volume force vector, **F** = (F_x, F_y, F_z), describes a distributed force field such as gravity. The unit of the volume force is force/volume. In this model you will need a force per area term which operates in the negative y-direction. Add a g=9.81 m/s^2 to your constants. Your flow will now be downward which is opposed to the y-direction

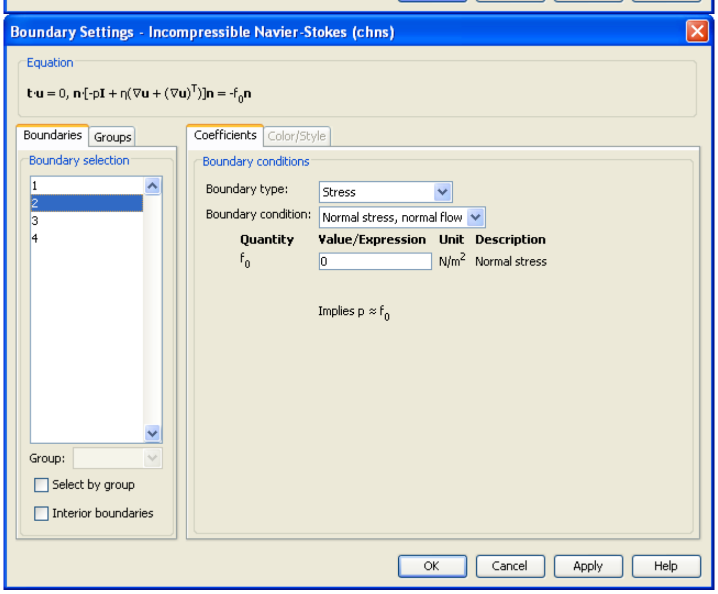


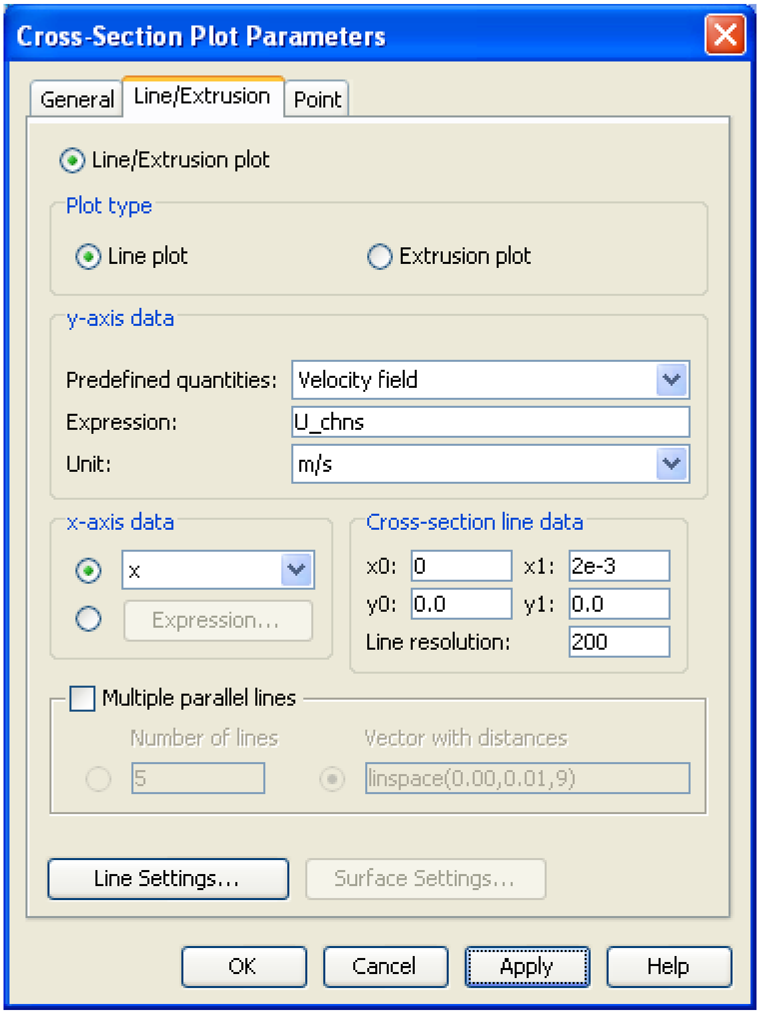
Boundary Conditions

The following boundary conditions will result in a fully developed flow from the inlet to the outlet. Select **Boundary Settings** from the **Physics** menu and enter the boundary conditions according to the following table:

BOUNDARY	1	2	3	4
Boundary condition	Symmetry Boundary or Wall with slip	Stress Normal stress, normal flow with	Stress Normal stress, normal flow with	No-slip
	(This sets the stress at the liquid/air interface to zero)	f_0=0	f_0=0	







To make your cross section plot and excel plot use the above as a guide.

Submit:

1. **Derivations**
   1.1. of velocity profile for flow between parallel plates
   1.2. of velocity profile for falling film
2. **Cross Section Plots:**
   2.1. Developing flow between parallel plates
   2.2. Fully Developed flow between parallel plates
   2.3. Falling Film
3. **Contour Plots (Basic Solution)** Add arrows on top of plot to signify velocity
   3.1. Developing flow between parallel plates
   3.2. Fully Developed flow between parallel plates
   3.3. Falling Film
4. **Excel Plots:**
   4.1. comparison of Fully Developed flow between parallel plates with analytical Solution
   4.2. comparison of falling film velocity profile with analytical solution.
5. **Answers**
   5.1. Determine at what y distance the velocity profile can be considered fully developed for the flow between 2 parallel plates with uniform inlet velocity. Explain your answer.
   5.2. pressure drop calculation for fully developed flow
   5.3. Comment on comparison between analytical and simulated results in the two fully developed flow models.

Fun for experts and those who finish early!
Laminar Flow Between Horizontal Plates with One Plate Moving
Now have the motion of the fluid created by the movement of one plate. Set the 2 boundary conditions for the entrance and exit to give the fully developed flow solution (Normal Flow/Pressure, with pressure equal to zero). You don't have to change the geometry. Just exclude gravity from any of the model terms. (you already have done this). Use the fully developed boundary condition. Please note that you are solving a linear equation.
Plot: Cross section plot