

The Navier-Stokes Equations

The Navier-Stokes equations describe flow in viscous fluids through momentum balances for each of the components of the momentum vector in all spatial dimensions. They also assume that the density and viscosity of the modeled fluid are constant, which gives rise to a continuity condition.

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. Following this is also a description of how the Navier-Stokes equations can be scaled.

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.

$$\begin{aligned}\rho \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} + \nabla p &= \mathbf{F} \\ \nabla \cdot \mathbf{u} &= 0\end{aligned}\tag{4-1}$$

where η denotes the dynamic viscosity ($\text{M L}^{-1} \text{T}^{-1}$), \mathbf{u} the velocity vector (L T^{-1}), ρ the density of the fluid (M L^{-3}), p the pressure ($\text{M L}^{-1} \text{T}^{-2}$) and \mathbf{F} is a body force term ($\text{M L}^{-2} \text{T}^{-2}$) 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:

$$\nabla p = \mathbf{i} \frac{\partial p}{\partial x} + \mathbf{j} \frac{\partial p}{\partial y} + \mathbf{k} \frac{\partial p}{\partial z} \tag{Equation 3.6-11}$$

In addition the dot product of the symbol, ∇ , with another vector is called the divergence or “div” of a vector \mathbf{v} .

$$\nabla \cdot \mathbf{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} \tag{Equation 3.6-12}$$

For constant density fluids at steady-state the continuity equation gives

$$\nabla \cdot \mathbf{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0 \quad \text{Equation 3.6-24}$$

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

$$\rho \frac{D\mathbf{v}}{Dt} = \rho \frac{\partial \mathbf{v}}{\partial t} + \rho(\mathbf{v} \cdot \nabla \mathbf{v}) = -\nabla p + \rho \mathbf{g} + \mu \nabla^2 \mathbf{v} \quad \text{Equation 3.7-39}$$

Boundary Conditions

The boundary conditions can be any of a number described below. For an imposed velocity, the velocity vector normal to the boundary can be specified by:

$$\mathbf{u} \cdot \mathbf{n} = u_0 \quad (4-2)$$

which is denoted as the **Inflow/Outflow boundary condition**. In the above equation \mathbf{n} is a unit vector that has a direction perpendicular to a boundary or normal to a boundary. For example in horizontal pipe flow the inlet velocity is perpendicular to the cross sectional area of the pipe. We can instead impose a certain pressure in the **Outflow/Pressure boundary condition**:

$$p = p_0 \quad (4-3)$$

There is a special boundary condition that combines both these descriptions. This is the **Normal flow/Pressure or “straight-out” boundary condition**, which sets the velocity components in the tangential direction to zero, and sets the pressure to a specific value. It is provided to simulate channels that are long in length where flow is assumed to have stabilized so that all velocity occurring in the tangential direction is negligible. Imposing this condition removes the necessity of simulating this length.

$$\mathbf{u} \cdot \mathbf{t} = 0 \text{ and } p = p_0 \quad (4-4)$$

Where the unit vector \mathbf{t} , is tangent to a boundary.

The **Slip/Symmetry condition** states that there are no velocity components perpendicular to a boundary:

$$\mathbf{u} \cdot \mathbf{n} = 0 \quad (4-5)$$

The **Neutral boundary condition** states that transport by shear stresses is zero across a boundary. This boundary condition is denoted neutral since it does not put any constraints on the velocity and states that there are no interactions across the modeled boundary.

$$\eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \mathbf{n} = 0 \quad (4-6)$$

The **No-slip boundary condition** eliminates all components of the velocity vector:

$$\mathbf{u} = 0 \quad (4-7)$$

SCALING

Many modeling domains may be large in one dimension as opposed to the others. This can create unreasonably large computational problems, which may not be able to be solved. Scaling the equations and dimensions of the geometry can create a more reasonable model size.

The scaling process ideally involves specifying the Navier-Stokes equations in a dimensionless form. In this case, we can introduce a characteristic length, d , and a characteristic velocity, u_0 . For flow in a circular tube, d is often taken to be the tube diameter and u_0 the average velocity. In this case, new dimensionless variables are defined according to:

$$\begin{aligned}\tilde{\mathbf{u}} &= \frac{\mathbf{u}}{u_0}; & \tilde{p} &= \frac{(p - p_0)}{\rho u_0^2}; & \tilde{t} &= \frac{t u_0}{d}; \\ \tilde{x} &= \frac{x}{d}; & \tilde{y} &= \frac{y}{d}; & \tilde{z} &= \frac{z}{d};\end{aligned}\tag{4-8}$$

where p_0 denotes a reference pressure. The momentum equations, in dimensionless form, become

$$\frac{\partial \tilde{\mathbf{u}}}{\partial \tilde{t}} - \tilde{\nabla} \cdot \left[\frac{\eta}{d u_0 \rho} \right] (\tilde{\nabla} \tilde{\mathbf{u}} + (\tilde{\nabla} \tilde{\mathbf{u}})^T) + (\tilde{\mathbf{u}} \cdot \tilde{\nabla}) \tilde{\mathbf{u}} + \tilde{\nabla} \tilde{p} = \left[\frac{g d}{u_0^2} \right] \frac{\mathbf{F}}{\rho g}\tag{4-9}$$

where the differential operators are also expressed in dimensionless form. This implies that the boundary conditions and domain geometry must also be defined in dimensionless form. The inverse of the dimensionless numbers within the square brackets are the Reynolds and Froude numbers, respectively.

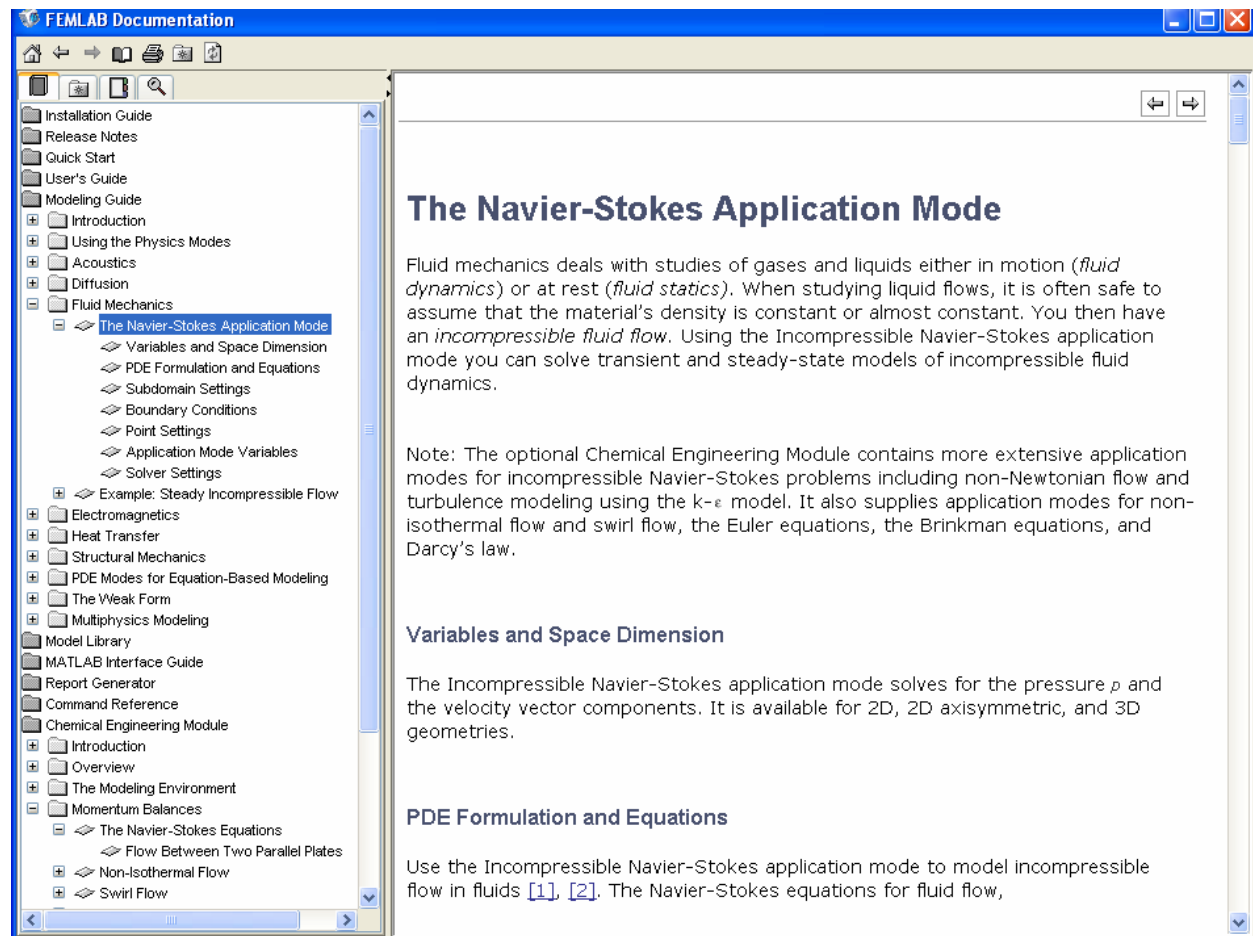
The Incompressible Navier-Stokes application mode also comes with special element types with different shape functions. They include the discontinuous element (`shdisc`) and the bubble element (`shbub`). They are summarized in the *Application Mode Implementation* chapter under the section for *Incompressible Navier-Stokes*.

The Navier-Stokes equations not only describe all types of Newtonian incompressible flow, but, theoretically, can also describe turbulent flow. However, modeling of turbulent flow with the Navier-Stokes equations is impractical in most engineering applications, since it requires that even the smallest eddies are to be resolved, which creates an unreasonably large number of nodes in the discretization process. Furthermore, flow cannot be said to be stationary since the eddies move mostly randomly through the flow.

These types of time-dependent simulations are very demanding in the number of computational operations and memory required, and they are too large to be handled by most computers. It is therefore necessary to use simplified models for the modeling of turbulent flow. One such approach is included in the k - ϵ application mode available in the module.

Also note that the Chemical Engineering Module contains an application mode for non-Newtonian fluids with predefined viscosity models.

More information about the Navier-Stokes Application Mode can be found in the Modeling Guide, Fluid Mechanics Chapter.



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 (Example 3.8-1 in Geankoplis)

In this problem you will model the steady-state flow of a fluid 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 .

1. The initial 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 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.

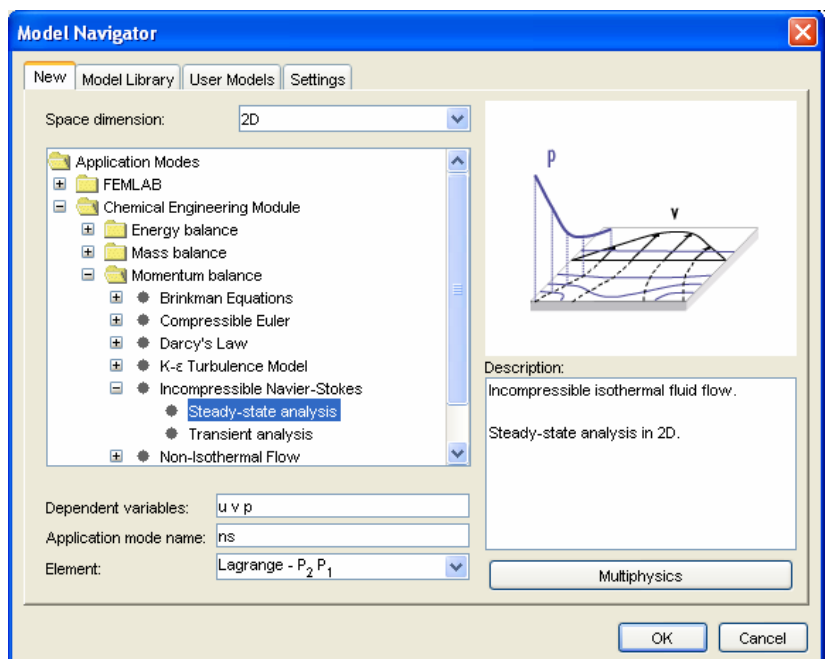
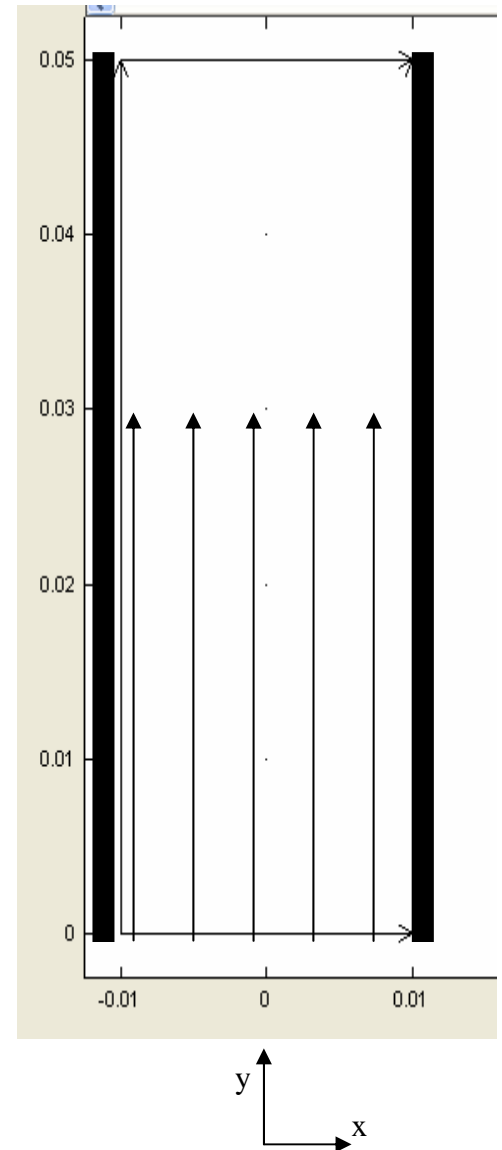
Modeling using the Graphical User Interface

1. Start COMSOL Multiphysics
2. In the **Model Navigator**, click the **New** page
3. Select **Chemical Engineering Module, Momentum Balance, Incompressible Navier-Stokes, Steady-state analysis**
4. Click **OK**.

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/m³.

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

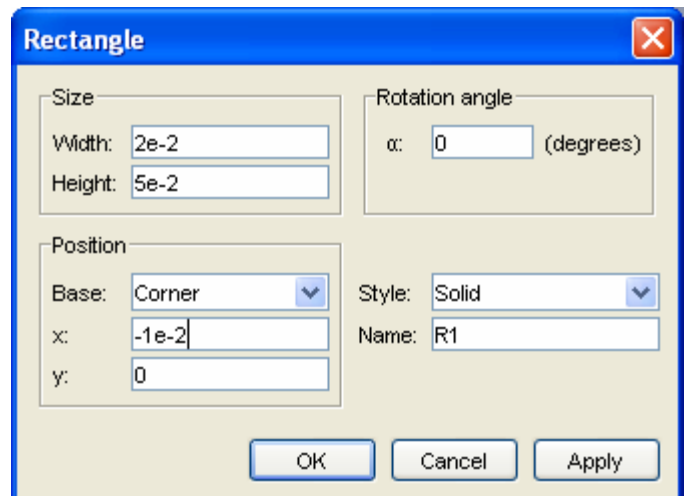


Geometry Modeling (Create Parallel Plates)

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. Click the **Zoom Extents** button in the Main toolbar.

Subdomain Settings

Select **Subdomain Settings** from the **Physics** menu. Define the physical properties of the fluid, in this case viscosity and density, according to the table below.



Rectangle

Size

Width: 2e-2

Height: 5e-2

Rotation angle

α: 0 (degrees)

Position

Base: Corner

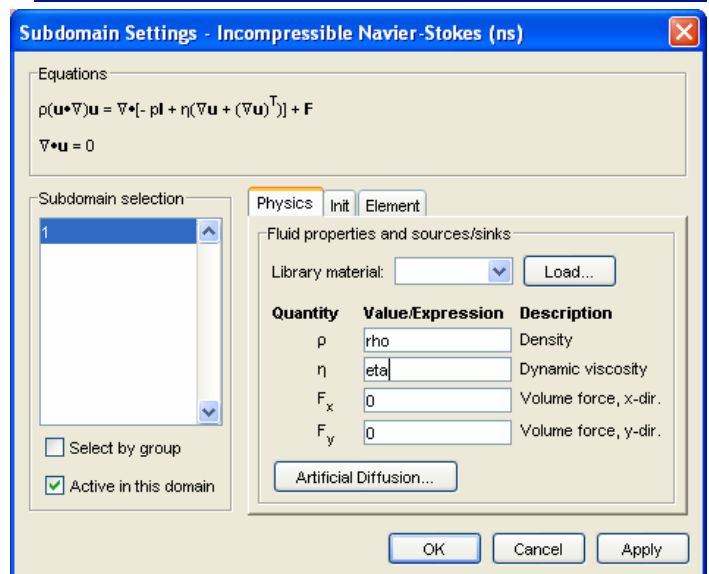
x: -1e-2

y: 0

Style: Solid

Name: R1

OK Cancel Apply



Subdomain Settings - Incompressible Navier-Stokes (ns)

Equations

$$\rho(\mathbf{u} \cdot \nabla) \mathbf{u} = \nabla \cdot [-p\mathbf{I} + \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \mathbf{F}$$

$$\nabla \cdot \mathbf{u} = 0$$

Subdomain selection

1

☐ Select by group

☒ Active in this domain

Physics Init Element

Fluid properties and sources/sinks

Library material: Load...

Quantity	Value/Expression	Description
ρ	rho	Density
η	eta	Dynamic viscosity
F_x	0	Volume force, x-dir.
F_y	0	Volume force, y-dir.

Artificial Diffusion...

OK Cancel Apply

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	No-slip	Inflow/Outflow velocity	Outflow/Pressure
u0		0	
v0		v0	
p0			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 and is not solving for an 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

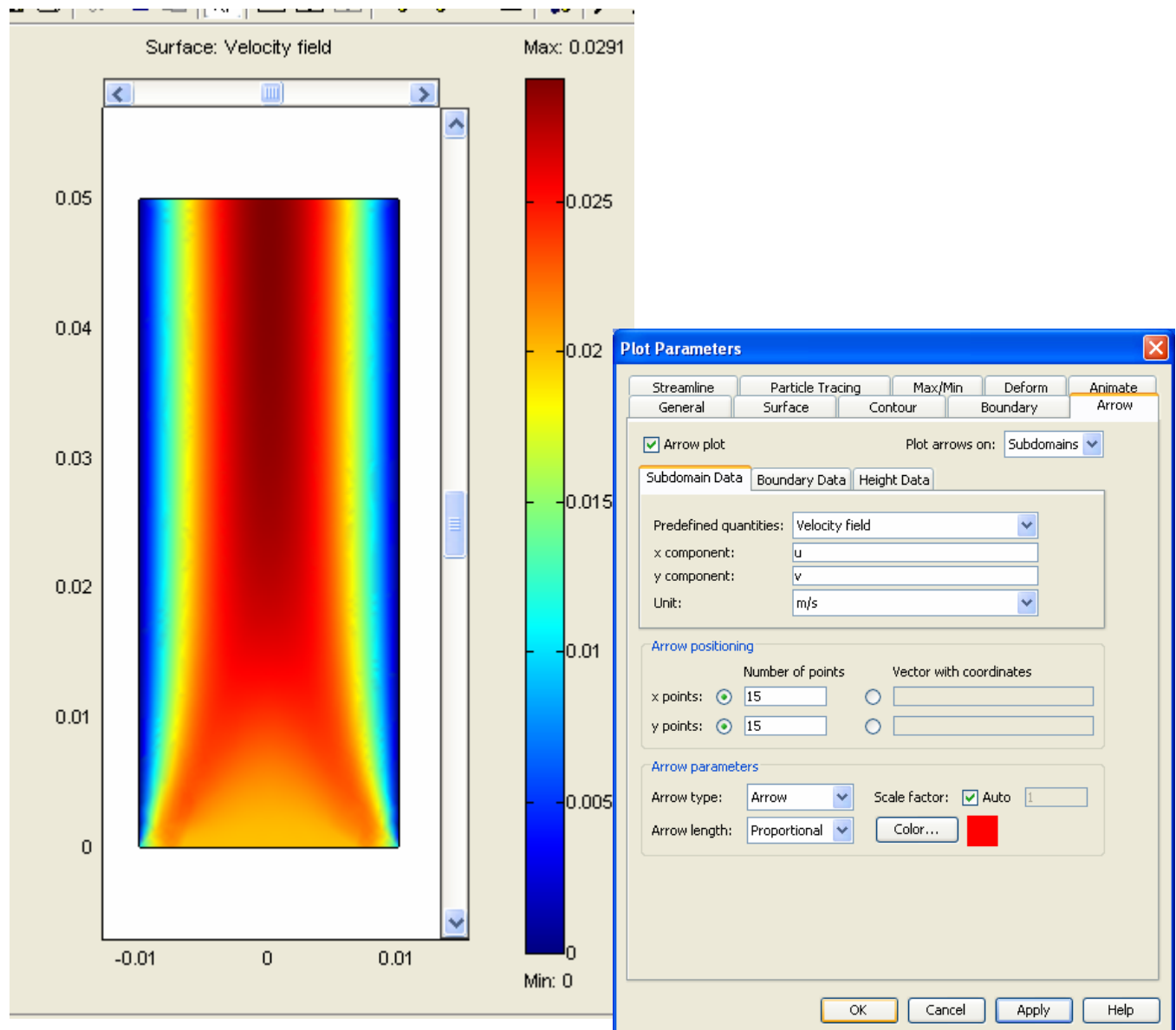
Computing the Solution

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. Remember to select the Arrow Plot Box.



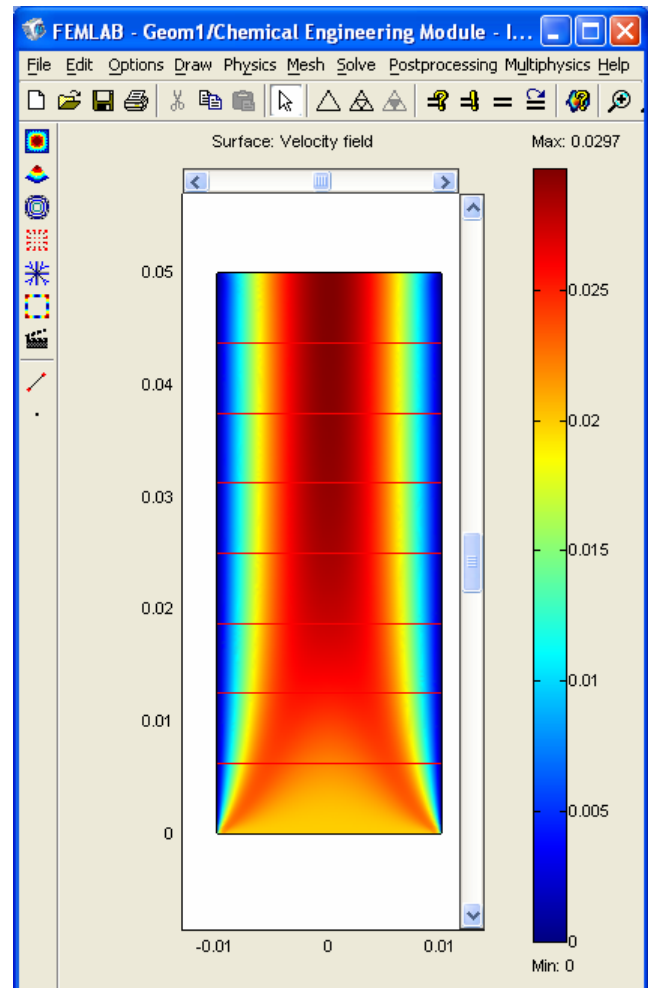
It is also possible to create cross-sectional plots of the velocity along horizontal lines at different positions along the channel.

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

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

7. Select the **Multiple parallel lines** check box.
8. 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.
9. Select **Line Settings** and click on the **Legend** box
10. Click **OK**.

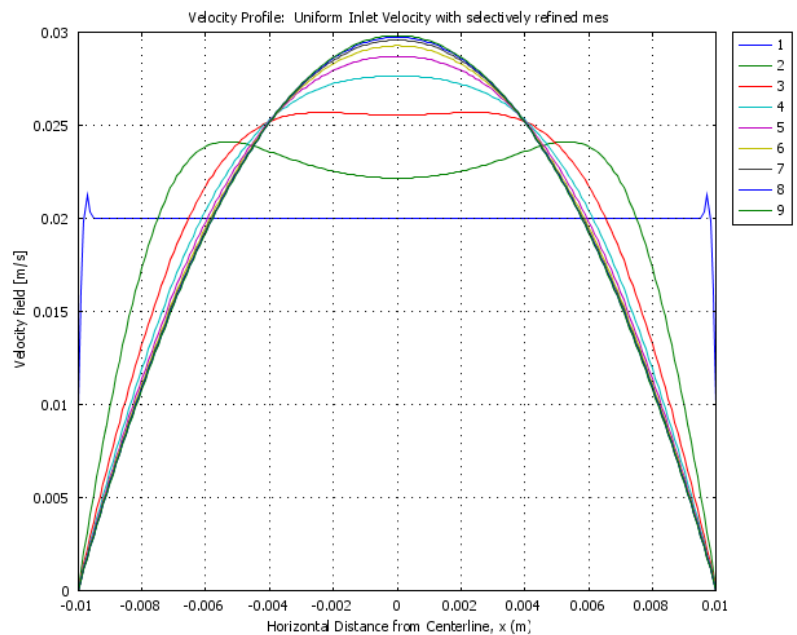
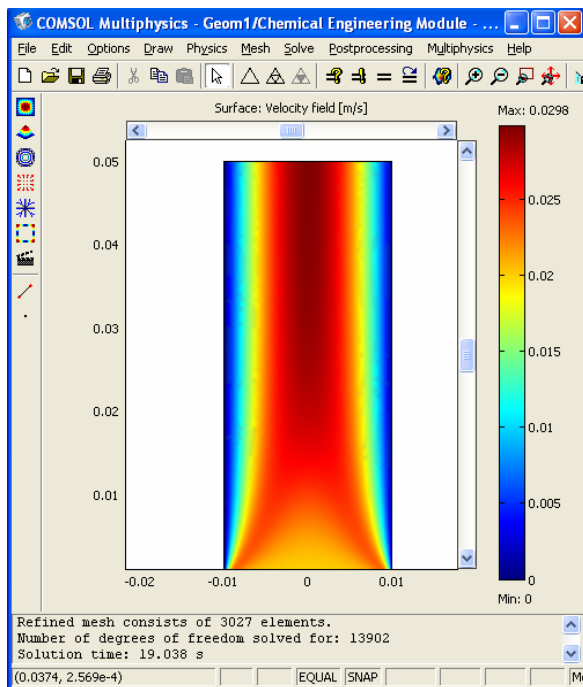
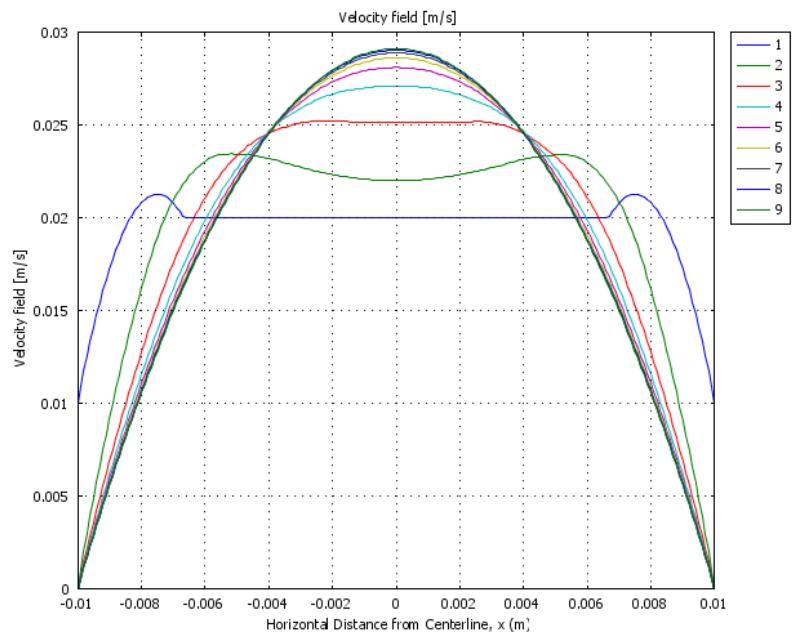
This gives a plot in which velocity is on the y axis and the width of the plate is on the x-axis. Each curve starting with the blue line corresponds to a given distance in the flow direction (y direction). Notice the appearance of the horizontal red lines on the original post process contour solution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



Problem 2: Fully developed flow between 2 parallel plates

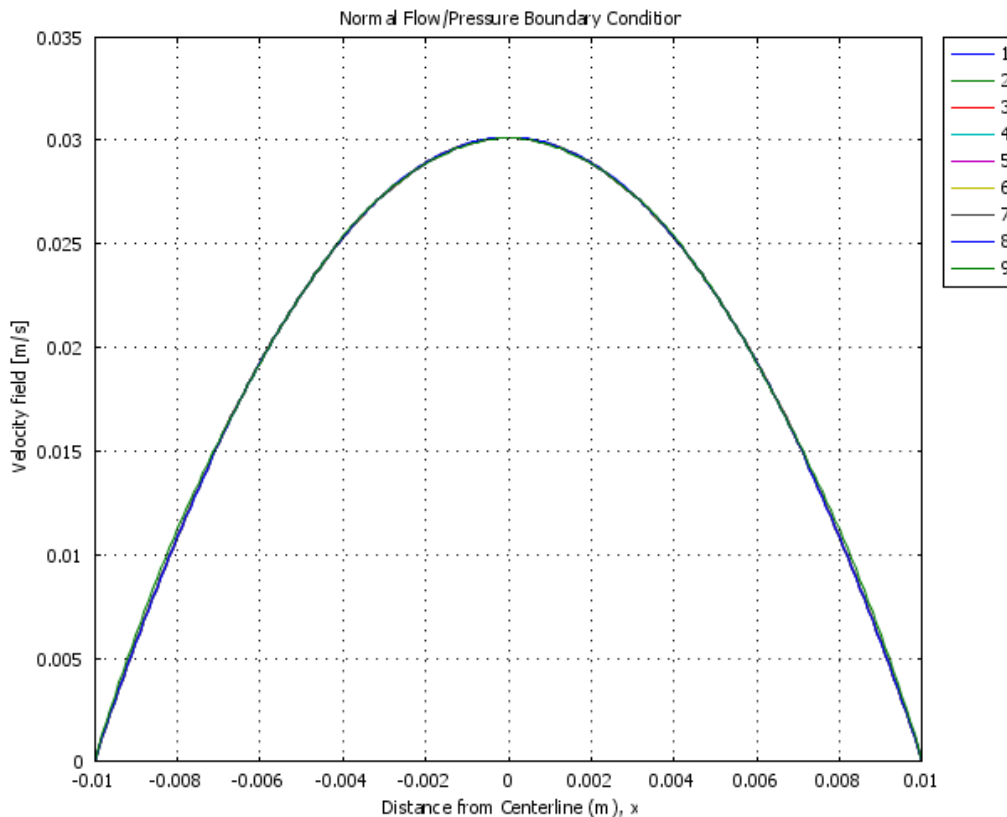
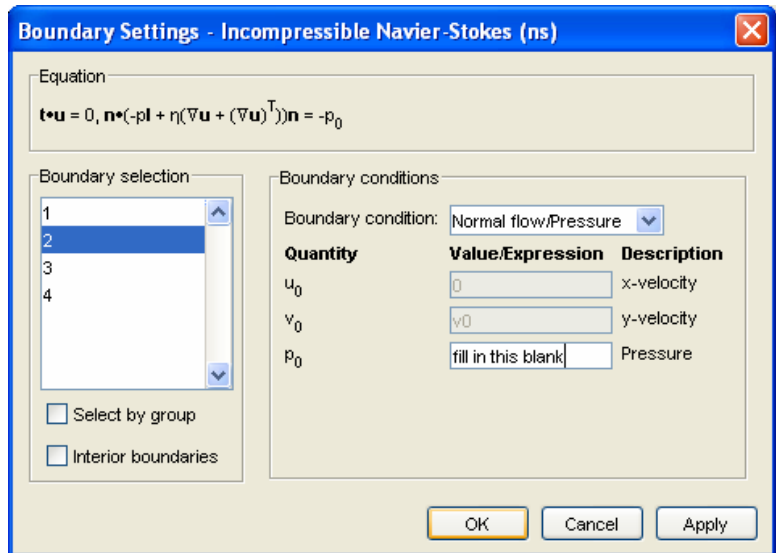
Now let's look at fully developed flow between 2 parallel plates. COMSOL has a boundary condition that allows us to specify the inlet pressure. For this problem what is the pressure drop per unit length?

The solution given in Geankoplis for this problem is $v_y = \frac{1}{2\mu} \frac{dp}{dy} (x^2 - x_0^2)$

Where x is half of the distance between the plates which in our geometry is equal to 0.01m. The maximum velocity is given as

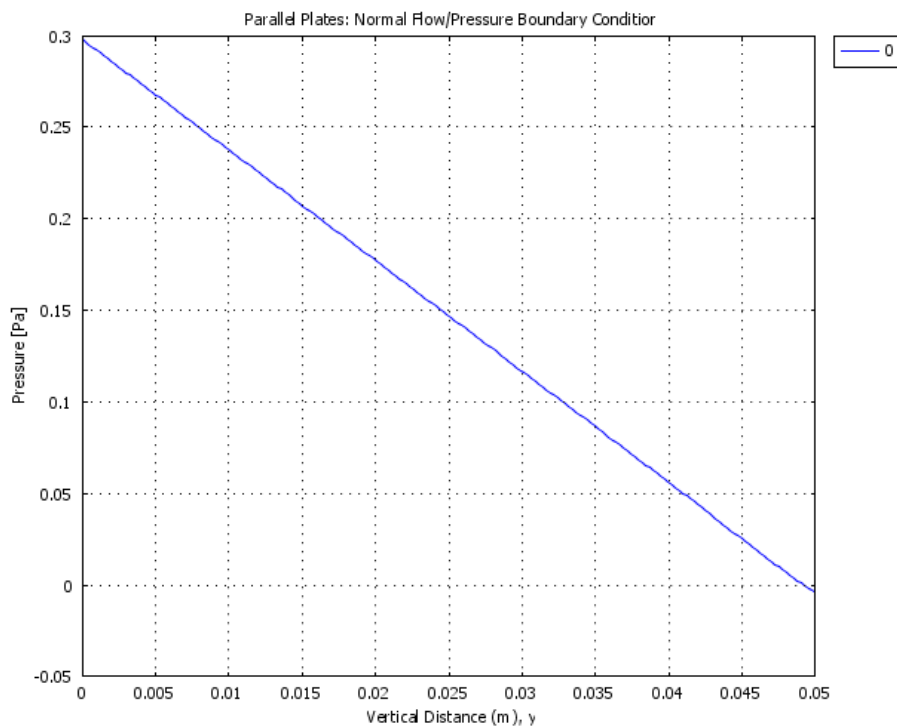
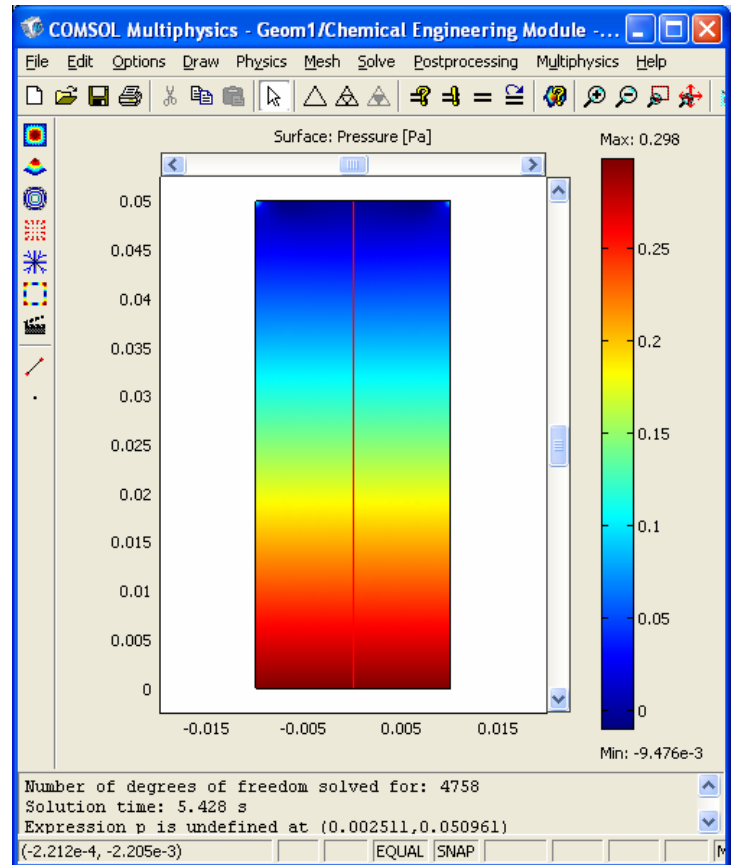
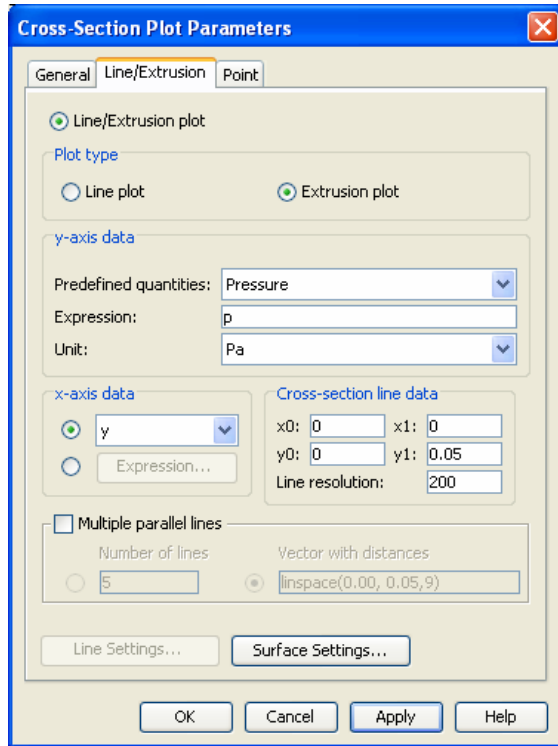
$$v_{y \max} = \frac{1}{2\mu} \frac{dp}{dy} (-x_0^2).$$

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.



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.



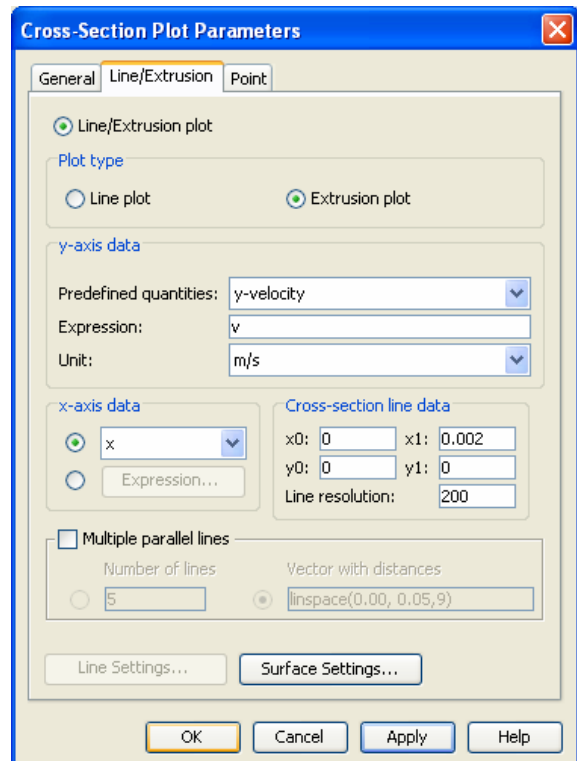
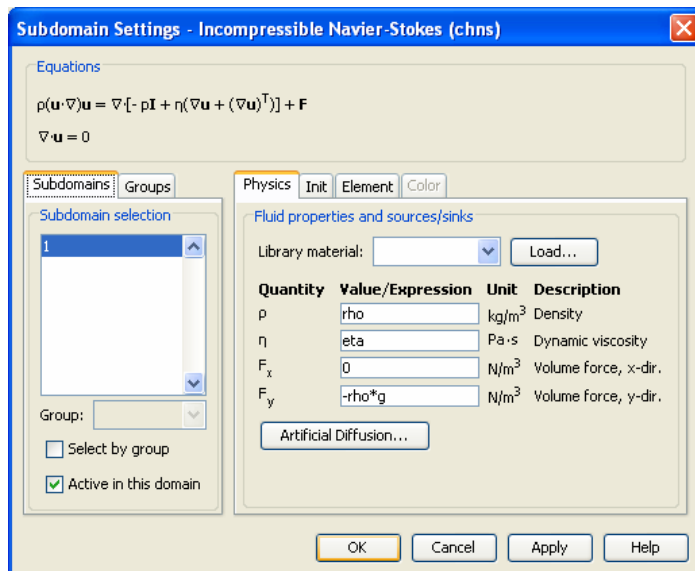
Laminar Flow – Falling film

Now solve a problem using a gravity term. In this case you will simulate a vertical plate with liquid flowing down the plate. 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. You have no applied pressure terms, but you will use a force term. The volume force vector, $\mathbf{F} = (F_x, F_y, F_z)$, 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 $g=9.81 \text{ m/s}^2$ to your constants. Compare your simulation to an analytical solution that you obtain from first principles. Show this derivation.

Boundary Conditions

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

BOUNDARY	1	2	3	4
Boundary condition	Slip/Symmetry	Outflow/Pressure	Normal flow/Pressure	No-slip
p0		0	0	



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