Flow Between Parallel Plates – Modified from the COMSOL ChE Library module rev 10/13/08

Modified by Robert P. Hesketh, Chemical Engineering, Rowan University Fall 2008

# **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-demensional 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 <u>Equation 4-1</u> 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.

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

(Accumulation of momentum per unit volume) – Shear Stress + rate of momentum + pressure forces = gravity forces

where  $\eta$  denotes the dynamic viscosity (M L<sup>-1</sup> T<sup>-1</sup>) (e.g. Newtonian viscosity with units kg/(m s), u the velocity vector (L T<sup>-1</sup>),  $\rho$  the density of the fluid (M L<sup>-3</sup>), *p* the pressure (M L<sup>-1</sup> T<sup>-2</sup>) and F is a body force term (M L<sup>-2</sup> T<sup>-2</sup>) 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,  $\nabla$ , is a mathematical operator. The gradient or "grad" of a scalar field such as pressure is defined as:

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

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

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

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

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

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

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

Where the substantial time derivative is defined as

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

# **Boundary Conditions**

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

Boundary	<b>Boundary</b> 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	$\mathbf{u} = -\mathbf{n}u_0 \text{ or}$ $\mathbf{u} = \mathbf{u}_0$	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 $\eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}) \mathbf{n}$	For this condition the inflow of fluid must be perpendicular to the boundary
Fluid Outlet			
	Velocity	$\mathbf{u} = -\mathbf{n}u_0 \text{ or}$ $\mathbf{u} = \mathbf{u}_0$	Becareful not to overspecify
	Pressure, No viscous Stress	$p = p_0$ and $\eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}) \mathbf{n}$	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	$\mathbf{u} \cdot \mathbf{t} = 0$ and $p = p_0$	This is used for fully developed flow in which the flow is only perpendicular to the area.

 Table 1: Summary of Common Boundary Conditions for Fluid Flow

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  $2 \times 10^{-2}$  m/s, viscosity of 0.01 kg/(m s) and density of 1000 kg/m<sup>3</sup>. 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
- 4. Select Chemical Engineering Module, Momentum Transport, Incompressible Navier-Stokes, Steady-state analysis
- 5. Take note of the variables: u, v and  $\rho$ . The variable u is in the x direction and the variable v is in the y direction for this 2diemsional problem.
- 6. Click OK.

Variables used u, v and  $\rho$ .

Number of

dimensions

Space dimension:





equations

en Settings

2D

# **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  $2 \times 10^{-2}$  m/s, viscosity of 0.01 kg/(m s) and density of 1000 kg/m<sup>3</sup>. Table 2: Constants

) 🚅 层	Acces/Grk	d Settings		£.		= 🗎	49 5	) Q (	😓 🔍 🖣	🖌 🖌	20.06	i 🛞 🐓	8	
todel Tree	DF Update S	ymbols						_						
te ba b	Constant	s					,							-
Geom1	Expressio	ms		•										
Incor	Integrati	on Coupling V	ariables	• •										- 1
	Extrusion	n Coupling Var	iables	•										
	Projection	n Coupling Va	riables	×										
	Identity (	Conditions		•										
	Boundary	y Distance Var	iables											
	Functions	s												- 1
	Materials	/Coefficients	Library											
	Viouslinat	ion/Selection	Settions											
	View Geo	metries	Jorna apr											
	Zoom			1.										
	Suppress													
	Labels													
-				-										- 1
	Preferen	ces												
	~ D	-0.4												- 1
	🗰 🕃													
	a 🚟	-0.6												
	0													
	22													
	5	-0.8												1
nbbled]														
		-1 -2	-1	-0.8	-0.6	-0.4	-0.2	0	0.2	0.4	0.6	0.8		
	<u>_</u>	-116	.,	-0.0	-0.0	-0.1	-0.1		0.6	0.1	0.0	0.0		
	1961													

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

🌀 Constar	nts				$\mathbf{X}$		
Name	Expression	Value	Description				
rho	1e3	1000	density - (kg/	m^3)	~		
eta	1e-2	0.01	viscosity - kg,	'(m s)			
v0	2e-2	0.02	velocity - m/s				
					С <mark>(</mark> С	lass Kit Licens	e) COMSOL
					File	Edit Options Di	raw Physics
							- ۲ 🖻 🛱
<b>Geometry</b> 1. Pres brin	<b>Modeling (Create P</b> as the <b>Shift</b> key and c g up the menu for th	<b>Parallel Plates</b> ) Elick the <b>Rectan</b> e Rectangle/Squ	<b>gle/Square</b> b are.	utton to		Tree L: L: om1	angle/Square
2. Typ	e the values shown is	n the adjacent fig	gure for 🛛 🕞	ctangle			
the n 3. 2e-2 Mai 4. Nov	rectangle dimensions 2 Click the <b>Zoom</b> n toolbar. v you have created a	s. E <b>xtents</b> button in domain that has	n the	Size Width: 2e-2 Height: 5e-2		Rotation angle α: 0 (	(degrees)
bou: bou	ndaries. Comsol lab ndaries and the interi	els as subdomair or of the rectang	ns the 4 gle.	Position Base: Corner k: -1e-2 y: 0	<b>v</b>	Style: Solid Name: R1	<b>~</b>
					ОК	Cancel	Apply
	emical Engineering Multiphysics Help	g Module - Incomp	oressible Navie	r-Stokes (cl	nns) : [Ui	ntitled] 💶 🗖	
	== \% \\$\\$ 	Zoom Ex	tents				0

#### Subdomain Settings



## **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 <sub>0</sub>			0





8



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.

Refine only the

by the mouse



## **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



Streamline Parl General Surfa
Plot type v Surface Contour Boundary Arrow Streamline Particle tracing Max/min marker Deformed shape v Geometry edges

Plot Parameters 🛛 🛛 🗙
Streamline Particle Tracing Max/Min Deform Animate General Surface Contour Boundary Arrow
Arrow plot     Plot arrows on: Subdomains
Subdomain Data Boundary Data Height Data
Predefined quantities:     Velocity field       x component:     u       y component:     v
Unit: m/s
Arrow positioning       Number of points     Vector with coordinates       × points:     15       y points:     15
Arrow parameters
Arrow type: Arrow Scale factor: V Auto
Arrow length: Proportional V Color
OK Cancel Apply Help



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

Cross-Se	ction Plot Par	amete	rs				
General	Line/Extrusion	Point					
💿 Line	e/Extrusion plot						
Plot ty	/pe						
💿 Lii	ne plot		🔘 Extrus	ion plot			
y-axis	data					$\equiv$	
Predel	fined quantities:	Velocit	y field			~	
Expre:	ssion:	U_chns	5				
Unit:		m/s				~	
x-axis	data		Cross-sec	tion line da	ta	51	
•	x	~	x0: -1e-2	2 x1:	1e-2	]	
0	Expression		yu: U Line resolu	ution:	U 200	-	
- 🔽 Mu	ltiple parallel lines	(					
	Number of lines		Vector wit	th distance:	s		
0	5	۲	linspace((	0.00, 0.05,	9)		
		Li	ne Settings	S			×
Line	e Settings	L	ne color:	Cycle	<b>v</b>	olor	
	ОК		ne style:	Solid line	*		
			Legend	None			
					ОК	Can	:el

🐲 FEMLAB - Geom1/Chemical Engineering Module - I... 🔳 🗖 🛿

Max: 0.0297

0.025

0.02

Surface: Velocity field

۲

**.** 

Ⅲ 米□ ≦

/

0.05

0.04

0.03

- 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**.

ross-Section Plot Parameters	0.02	0.01
Plot type         Inne/Extrusion plot       Point plot         Solutions to use       Solution at angle (phase):         Select via:       Image: the select of	0.01	0.005
Plot in: Figure 1 Color	Title/Axis Settings       X         Title:       Auto       Velocity Profile: Uniform Inlet Velocity         Axis settings for line and point plots       Log scale         First axis label:       Auto       Log scale         Second axis label:       Auto       Log scale         OK       Cancel	0.01 Min: 0
OK Cancel Apply Help		12

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 Look in C) lecture Nome My Recent Documents Date M Figure 1 plot. You will need to open this data Siemens File Folder 15 KB Text Docu en parallel plates.txt Desitop My Documents with an excel spreadsheet. Make sure you are S My Computer searching for a \*.txt file in the Files of type: My Ne 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 Text Files (\* grey, \* loty, \* cerv) ER Files (\* s) ER files (\* s) ER for out Files (\* s) ER for out Files (\* s)\* \* close, \* den, \* den, \* den, \* den, \* den, \* close, \* cl scrolling down with the scroll bar and then Files of type Cancel press Finish. Tools + Text Import Wizard - Step 1 of 3 ?× The Text Wizard has determined that your data is Delimited. If this is correct, choose Next, or choose the data type that best describes your data Original data type Choose the file type that best describes your data: - Characters such as commas or tabs separate each field. O Delimited • Fixed width - Fields are aligned in columns with spaces between each field. File origin: 437 : OEM United States Start import at row: 1 Preview of file E:\HeskethDrive\Courses\ProcessFluidTransport\lec...\flow between parallel plates.txt. 1 & Coordinates -0.01 2.2429045E-9 2 -0.01 3 -0.009899497 4 -0.009798994 5 -0.0096984925 6.2617054E-4 0.0012448011 0.0018559074 ¥ The second column contains the velocity values. The maximum First column (Column A) are the x velocity is at 0.0297 m/s in this coordinates. These should start at flow between parallel plat... 🔽 flow bet 0.01 m and end at 0.01 m This is the last velocity Δ1 - 6 A A В В C value at 0.01 m. Notice 97 -4.52E-04 0.029632 196 0.009497 0.00306 В С 197 0.009598 0.002463 that it is not zero because 98 -3.52E-04 0.029658 % Coordinates 1 198 0.009698 0.001858 99 -2.51E-04 0.029678 of numerical errors 199 0.009799 0.00124 2 -0.01 2.24E-09 100 -1.51E-04 0.029693 200 0.009899 6.27 04 101 -5.03E-05 0.0297 3 -0.0099 6.26E-04 102 5.03E-05 0.029702 201 0.01 2.26E-09 4 -0.0098 0.001245 After the data are given 103 1.51E-04 0.029693 202 % Elements (lines) 5 104 2.51E-04 0.029679 203 1 -0.0097 0.001856 2 information on the 105 3.52E-04 0.029659 204 2 6 -0.0096 0.00246 106 4.52E-04 0.029633 205 3 4 number of elements 7 -0.0095 0.003056 107 5.53E-04 0.029602 206 4 5 207 5 6 If I + F flow betwee 1 used. We do not use 8 -0.0094 0.003644 108 6.53F-04 0.029565 I → → flow betwee 4 ..... > > -0.0093 0.004226 9 this for the plot 田口口 100
- 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 
$$v_y = -\frac{\delta^2}{2\mu} \frac{dp}{dy} \left( 1 - \left(\frac{x}{\delta}\right)^2 \right)$$

The maximum velocity is given as

$$v_{y \max} = -\frac{\delta^2}{2\mu} \frac{dp}{dy}$$
 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

Boundary Settings - Incompressible Navier-Stokes (chns)	Boundary Settings - Incompressible Navier-Stokes (chns)
Equation $n(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)\mathbf{n} = 0, \ p = p_0$	$\begin{split} & \text{Equation} \\ & L_{exit} \nabla_t \left[ p \mathbf{I} \cdot \eta (\nabla_t \mathbf{u} + (\nabla_t \mathbf{u})^T) \right] = -n \rho_{0,exit} \;, \; \nabla_t \cdot \mathbf{u} = 0 \end{split}$
Boundaries Groups Coefficients Color/Style	Boundaries Groups Coefficients Color/Style
Boundary selection Boundary conditions	Boundary selection Boundary conditions
Boundary type: Inlet	Boundary type: Outlet
3 Boundary condition: Pressure, no viscous stress	3 Boundary condition: Laminar outflow
4 Quantity Value/Expression Unit Description	4 Quantity Value/Expression Unit Description
P <sub>0</sub> fill in for 1.05 m Pa Pressure	U0 m/s Average velocity
	P <sub>0,exit</sub> D     Pa Exit pressure
	L <sub>exit</sub> 1 Exit length
	Constrain end points to zero
Group:	Group: ☐ Select by group ☐ Interior boundaries
OK Cancel	OK Cancel Apply

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

Boundary Settings - Inco	mpressible Navier-S	tokes (chns)				D
Equation $\eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)\mathbf{n} = 0, \ \mathbf{p} = \mathbf{p}_0$	)					
Boundaries Groups	Coefficients Color/Sty	/le				
Boundary selection	Boundary conditions					
1	Boundary type:	Inlet	~			
3	Boundary condition:	Pressure, no viscous	stress	~		
4	Quantity	Value/Expression	Unit	Description		
	P <sub>0</sub>	fill in this blank	Pa	Pressure		
Group: V Select by group Interior boundaries						
		0	к	Cancel	Apply	Help

#### 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



26 26 🕅

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,  $\mathbf{F} = (F_x, F_y, F_z)$ , 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 m/s<sup>2</sup> to your constants. Your flow will now be downward which is opposed to the y-direction

Subdomain Settings - Inco	mpressible I	Navier-Stokes (ch	ns)	×			
Equations							
$\rho(\mathbf{u} \cdot \nabla)\mathbf{u} = \nabla \cdot [-p\mathbf{I} + \eta(\nabla \mathbf{u} + 0)]$	(∇u) <sup>T</sup> )] + F						
$\nabla \cdot \mathbf{u} = 0$							
Subdomains Groups	Physics Toit	Flement Color					
	et al						
Subdomain selection Fluid properties and sources/sinks							
	Library material: Voad						
	Quantity	Value/Expression	Unit Description				
	ρ	rho	kg/m <sup>3</sup> Density				
	η	eta	Pais Dynamic viscosity				
	F <sub>x</sub>	0	N/m <sup>3</sup> Volume force, x-dir				
	Fy	-rho*g	N/m <sup>3</sup> Volume force, y-dir				
Group:	Autoficial Diffusion						
Select by group							
Active in this domain							
	_			_			
	L						

# **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 conditi	Symmetry ion 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 <sub>0</sub> =0	f <sub>0</sub> =0	
Boundary Settings - Incompres	sible Navier-Stokes (chns)	×		
Equation $f_{UU} = 0  p_{1}(-p_{1} + p(\nabla U + (\nabla U)^{T}))p_{1}(\nabla U + (\nabla U)^{T})$	= -f n			
	- 1 <sub>0</sub> 1			
Boundaries Groups Coel	inders Color/Style	]		
1 Bo	oundary type: Stress			
Bo	Oundary condition: Normal stress, normal flow	♥ Description		
T .	f <sub>0</sub> 0 N/m <sup>2</sup>	Normal stress		
	Templing p. or f.			
	Implies $p \approx r_0$			
Group:				
Interior boundaries				
Boundary Settings - Incompres	sible Navier-Stokes (chrs)			
Equation	sible navier-stokes (clilis)			
$\mathbf{t} \cdot \mathbf{u} = 0,  \mathbf{n} \cdot [ \cdot \mathbf{p} \mathbf{I} + \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) ] \mathbf{n}$	= -f <sub>0</sub> n			
Boundaries Groups Coef	fficients Color/Style			
Boundary selection Bo	undary conditions			
1 Bo	oundary type: Stress			
3 4	Quantity Value/Expression Unit	Description		
	f <sub>0</sub> 0 N/m <sup>2</sup>	Normal stress		
	Implies p ≈ f.			
Group:				
Interior boundaries				
	ОК	Cancel Apply Help		

Cross-Section Plot Parameters	×					
General Line/Extrusion Point						
Line/Extrusion plot     Plot type						
Cine plot     O Extrusion plot						
Predefined quantities:     Velocity field       Expression:     U_chns       Unit:     m/s						
x-axis data       Cross-section line data         x       x         Expression       x0:         Line resolution:       200						
Multiple parallel lines         Number of lines       Vector with distances         5       Inspace(0.00,0.01,9)						
Line Settings Surface Settings						
OK Cancel Apply Help						

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