

COMSOL Project

Instructions for Solving

Flow between Two Parallel Plates

Professor Faith A. Morrison
Department of Chemical Engineering
Michigan Technological University
11 November 2009

Problem Statement:

Using COMSOL 3.5a, calculate the steady state velocity field for the flow of an incompressible Newtonian fluid between parallel plates. The plates are 0.100m long and infinitely wide. The two plates are separated by a gap of 0.020m. The density of the fluid is 1000 kg/m³ and its viscosity is 0.0100 Pa s. The inlet flow is a uniform flow of velocity 0.0175m/s; the outlet pressure is zero gauge pressure. In your report, please answer the following questions and attach the appropriate plots:

1. How does your answer for the steady velocity profile compare to the analytical solution? Be quantitative (e.g. “The numerical and the analytical results differed by at most 4% . . .”). Note that you must match the pressure drops between your hand calculation and your Comsol calculation (see hint at the end of this handout). Plot the exit velocity profile on the same graph as the analytical result (e.g. in Excel, v_y/v_{max} versus $x/width$).
2. The inlet flow is a uniform velocity profile (same velocity at every x -position), and at the exit of the flow domain the velocity profile has reached its steady, parabolic profile. How far down the tube does the flow need to go before the flow reaches its steady profile? Include a plot of various cross-sections of the flow.
3. What does the two-dimensional velocity field look like? (provide a 2D arrow plot)
4. How long did it take you to complete the calculations for this part of the assignment? How long to work on the report?

Strategy:

First we will create a flow domain that is the shape specified in the problem and enter the constants (density, viscosity, inlet average velocity) and the boundary conditions.

Second, we will set up the geometry that is needed to carry out the numerical solution. To do the numerical solution to the continuity equation (mass balance) and the Navier-Stokes equations (momentum balance), we first create a calculation mesh. The mesh is a digitalization of the flow domain. The numerical algorithms used by COMSOL calculate the velocities, pressure, and stresses on each discrete cell of the mesh. For higher accuracy we will choose the mesh to be made of smaller elements near the walls since this is where the stresses are highest.

Third, we instruct COMSOL to solve the problem.

Finally we perform various plots to visualize the results. All the data from plots may be exported to ASCII (plain text) files that can then be imported into other programs (e.g. Excel) for the preparation of the final plots. Present your results in the form of a report with a cover page (title, name, date).

Contact the TA, Jifei Liu (jifeil@mtu.edu) if you have any questions about this procedure.

Instructions

Part 1: Create the flow Domain

Start-up

1. Click the **Start** button, point to **All Programs** → **Other Apps** → **COMSOL 3.5a** → **COMSOL Multiphysics 3.5a**

The program opens up in the Model Navigator

2. Select **2D** from the **Space dimension** drop-down list
 3. Click the **New** page
 4. Click the plus sign at the **Chemical Engineering Module, Momentum Transport, Laminar Flow, Incompressible Navier-Stokes** and select the **Steady-state analysis**. Note the variables are u (x -velocity), v (y -velocity), and p (pressure).
 5. Click **OK**
-

Part 2: Set up the Flow Geometry

Geometry Setting

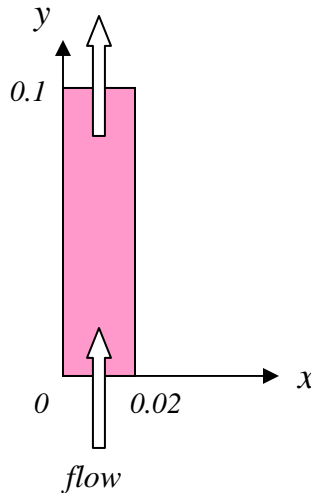
6. To bring up the Rectangle/Square dialog box, simultaneously press <Shift> and click the **Rectangle/Square** button on the left drawing toolbar *HINT: You can find out the name or function of any button by moving the mouse over the button and “hovering”*.
7. Enter the following dimensions in the **Rectangle** dialog box (all are in SI units, i.e. meters, kg, seconds)

Size	
Width:	0.02
Height:	0.10

8. Click **OK**; alternately you may draw the same solid by holding the shift key and choosing the Line segment. A dialog box opens and you can enter all the x -coordinates of your solid (0.0, 0.02, 0.02, 0.0), all the y -coordinates (0,0,0.1,0.1)

and chose style closed polyline (solid). This method is more flexible and gives the same result.

- Click the **Zoom Extents** button (above, looks like a magnifying glass with red arrows pointing north, south, east, and west) on the Main toolbar; this is your flow domain.



Physics Settings

- Select **Constants** from the **Option** menu
- Enter the following constant in the **Constants** dialog box (all are in SI units)

Name	Expression
rho	1000
eta	0.0100
vo	0.0175

- Click **OK**
- Select **Subdomain Settings** from the **Physics** menu
- Select **1** from the **Subdomain selection** box. Note that the Navier-Stokes equation in Gibbs notation appears at the top of the page; also the continuity equation for incompressible fluids (in Gibbs notation) is shown. (Question: What is **F** in the Navier-Stokes shown there? Answer: see reference list)
- Enter the following expression in the **Physics** page

Quantity	Value/Expression
ρ (density)	rho
η (viscosity)	eta

- Click **OK**
- Select **Boundary Settings** from the **Physics** menu
- Select **1** from the **Boundary selection**, **Wall** for **Boundary type**, and **No slip** from the **Boundary condition** drop-down list.

19. Select **2** from the **Boundary selection, Inlet** for **Boundary type**, and **Velocity** from the **Boundary condition** drop-down list. Choose to set u_0 and v_0 (x- and y-velocity). Set x-velocity: 0, y-velocity: v_0
20. Select **3** from the **Boundary selection, Outlet** for **Boundary type**, and **Pressure** from the **Boundary condition** drop-down list. Set Pressure: 0
21. Select **4** from the **Boundary selection, Wall** for **Boundary type**, and **No slip** from the **Boundary condition** drop-down list.
22. Click **OK**

Mesh Generation

23. Select **Free Mesh Parameters** from the **Mesh** menu
24. Click the **Boundary** tab
25. Simultaneously press <Ctrl>, select **1** and **4** from the **Boundary selection**, and set Maximum element size: 0.001 (this makes the mesh finer near the walls)
26. Click the **Remesh** button; Comsol creates a mesh for calculating the results.
27. Click **OK**

Part 3: Solve

Solution Computation

28. Click the **Solve** button on the Main toolbar (looks like “=”)

Part 4: Generate Plots and Visualize the Solution

Velocity Profile Generation

29. Select **Cross-Section Plot Parameters** from the **Postprocessing** menu
30. In the **General** page, select **Line/Extrusion plot** from the **Plot type**
31. In the **Line/Extrusion** page, select **Velocity field** from the **Predefined quantities** drop-down list; set the **Cross-section line data** $x_0: 0, y_0: 0, x_1: 0.02, y_1: 0$, click the **Multiple parallel lines** check box, select the **Vector with distances** and click **Edit**. For first velocity choose 0; for last velocity choose 0.10, and for number of values choose 21. Click **Replace**.
32. Click **OK**; a plot is generated with 20 curves at planes $x=0, 0.005, 0.010, \dots 0.1$.
33. To save the data from the plot, in the plot window click the **ASC II** button on the Main toolbar of the **Figure** window, browse to where you wish to save the data, and click **Save**.

Pressure Profile Generation

34. Select **Cross-Section Plot Parameters** from the **Postprocessing** menu
35. In the **Line/Extrusion** page, select **Line/Extrusion plot** and **Line plot** from the **Plot type**
36. Select **Pressure** from the **Predefined quantities** drop-down list; set the **Cross-section line data** $x_0: 0.01, y_0: 0, x_1: 0.01, y_1: 0.1$, unclick the **Multiple parallel lines** check box. In the **x-axis data box** choose to plot versus y. Click **OK** or **Apply**; a plot is generated of pressure down the centerline of the flow. Save data as instructed above.

Part 5: Import Data into Excel

37. Open MS Excel and choose **Open** and filename *.txt.
38. Choose your data file.
39. Choose Delimited, Next, space, Next, Finish.
40. The data file will have all the data points in the first two columns. There are 200 data points per plot of v_y versus x ; there are 21 lines.

Hint

For this of the Comsol assignment, I ask you to compare your results with the hand calculations that you do. One missing number that is not specified above is the value of the inlet pressure.

The problem that you solved numerically did not specify the inlet pressure. Instead, the inlet velocity was specified to be a given number, and the velocity profile was assumed to be flat (plug flow or uniform flow). The difficulty, then, is in sorting out what the $dp/dy = \Delta p/L$ is for the problem in order to be able to make the final comparison.

I intend for you to use a steady-state pressure gradient that you get from your Comsol solution as the pressure gradient in the comparison. You can obtain the pressure gradient found by Comsol by plotting the pressure at the center of the slit as a function of length down the tube. Near the flow entrance, the slope dp/dy will not be constant, but after the velocity profile re-arranges into the steady-state velocity profile, the pressure gradient dp/dy will become constant. You can plot P versus y and measure dp/dy in the linear portion and use that to compare your analytical result with your numerical results.

Answer to question above: \mathbf{F} is density multiplied by the gravity vector.