CM 3110 COMSOL INSTRUCTIONS

Professor Faith Morrison and Teaching Assistants
Department of Chemical Engineering
Michigan Technological University, Houghton, MI USA

Problem statement: Calculate the steady, 2D velocity field and stress field for flow of water (an incompressible, Newtonian fluid) through parallel plates. The plates are long and wide and separated by a gap \( 2B=0.020\text{m}=2.0\text{cm}. \) The flow is driven by a constant pressure gradient, \( \frac{\partial p}{\partial y} = -\frac{\Delta p}{L} \). Address the following issues in your solution:

1. Graphically compare your solution for the well-developed velocity profile across the gap to a hand calculation of the same quantity. Note that you must use the same pressure gradient in both solutions to have a valid comparison.
2. What region of your numerical calculation domain actually reflects well-developed flow? That is, in what region of your flow domain is the flow no longer changing in the flow direction? Discuss your answer with reference to appropriate graphs.
3. Compare Quantitatively compare the total vector force/width on the wall (either side) with a hand calculation of the same quantity. (see notes on this objective below)

Solution

These instructions detail how to calculate the velocity field in the flow through two very long, very wide plates. Because the flow field only varies in one direction, we use a 2D calculation, assuming the calculation would be the same in any cross section we might choose.

The boundary conditions that are usually used with the analytical (by hand) solution to this problem are no-slip at the walls and defined inlet and outlet pressure. For the numerical solution, we need to specify the velocity boundary conditions along all surfaces: including the inflow and outflow surfaces. This is not necessary in the analytical solution, and the need to specify the unknown inlet velocity boundary condition is a characteristic challenge of the numerical problem-solving approach.
If we specify all the velocity boundary conditions and try to simultaneously satisfy our chosen pressure boundary conditions, we will have over-constrained the problem (too many constraints applied). Thus, in the numerical solution we drop the pressure inlet boundary condition and replace it with an assumed uniform flow inlet boundary condition, with velocity equal to a low value (we choose 0.0175 m/s). We retain the outlet pressure boundary condition and choose it to be zero gauge pressure. Note: Do not assume laminar inflow boundary condition.

Note that in the strategy outlined above, the inlet boundary condition (uniform velocity) is not really what we want: the true inlet flow to steady slit flow is not uniform, but rather the steady state velocity profile is a curve (a parabola). The numerical solution method requires us to specify the inlet boundary condition (which we do not know). To address this situation, we guess a convenient inlet velocity profile (uniform flow), and we subsequently allow the flow to develop in the slit until it becomes the true, well-developed velocity profile. We find the true steady state solution only in the portion of the flow domain in which the velocity profile is unchanging in the flow direction.

Once the Comsol flow calculations are complete, we can calculate the velocity gradient $-\Delta p / L$ that we need in the hand calculation: we obtain $-\Delta p / L$ from the numerical solution for the 2D pressure field. The need to use a “solved-for” pressure gradient in the hand calculation rather than one we choose arbitrarily is a consequence of the selected inlet boundary condition (uniform flow). Using the “solved-for” pressure gradient in the analytical calculations allows us to compare “apples-to-apples” when we plot the numerical solution for the velocity profile along with the hand calculation.

Notes:
A) The model library has water in it; the viscosity used seems to be $1.0445 \times 10^{-3}$ Pa s.
B) Due to the inlet conditions we specify in these instructions, the pressure gradient is not constant for the whole flow domain. This will make the force/width in the $yx$-direction calculation inaccurate; the comparison of force/width in the $xy$- and $z$-directions will work.

In the step-by-step discussion below, we demonstrate how to make different types of plots from the 2D results. These plots are needed to address all our objectives.

The basic numerical strategy is this:

1. Initialize the program settings (choose 2D flow, choose water, etc.)
2. Draw the flow domain (2D representation of a slit of appropriate width)
3. Design a finite-element mesh on which to do the numerical calculations (see en.wikipedia.org/wiki/Finite_element_method for more on the finite element method). Good mesh design ensures an accurate calculation. The mesh should be more refined (have smaller divisions) in areas where properties change most rapidly. At a minimum, there...
should be 8-10 elements across the cross section of the flow. If your results do not match your hand calculation well, go back to this step and use a finer mesh.

4. Run the simulation (this produces a “Study”).

5. Use and evaluate the simulation results by calculating and plotting the quantities of interest.

6. Perform calculations that allow us to address our objectives in a report.

---

**Step-by-Step Instructions**

**Part 1: Start Comsol and Select Problem Type**

General comments: We are doing a steady laminar 2D flow through a slit. We begin by making these choices in Comsol.

**Start-up**

1. Launch COMSOL Multiphysics 5.1, then select Model Wizard.
2. Select 2D from the Select Space Dimension list.
3. In the Select Physics window select (double click) Fluid Flow, then (double click) Single-Phase Flow and (single click) Laminar Flow (spf). Click Add to add the physics to the model.
4. Click to go onto the next page. Note that \( v_x = u, \ v_y = v, \) and \( v_z = w. \)
5. Select Stationary in the Select Study window (Stationary=steady state; Time Dependent=unsteady).
6. Click to finish.

**Part 2: Set up the Flow Geometry and Physics Settings**

General comments: Our geometry is the gap between two long plates. We draw this as a vertical rectangle of the chosen gap width with a long length. We choose the length to be 0.500m in this example. The rule of thumb for what is called “well developed” flow is for the length to be 20 times the gap.

**Geometry Setting**

7. Select the Geometry tab from the menu at the top of the page; then from the Rectangle drop-down menu, select Rectangle (Center), and drag any-sized rectangle in the Graphics Window.
8. To enter the exact dimensions for your desired rectangle, go to the Model Builder window and click the arrow at Geometry 1 to expand the menu, and select Rectangle 1 (r1). Now go to the Settings (Rectangle) window and enter the following: in Size, enter Width = 0.020 $m$; Height = 0.500 $m$, then under Position select Corner for Base and enter $x = -0.01 \, m$; $y = 0.0 \, m$. The box thus covers $-0.01 \, m \leq x \leq 0.01, 0 \leq y \leq 0.500 \, m$.

9. To enter these new dimensions click on $\text{Build Selected}$ at the top of the Settings window. If the figure does not print where you can see it, click on Zoom Extents in the Graphics window. Flow is in the $y$-direction (upwards) and the velocity profile varies in the $x$-direction. The Messages window below the graphics is where calculation results will appear.

10. Select the Materials tab from the menu at the top of the page, and then select Browse Materials.
11. Click the arrow at the Liquids and Gases, then click the arrow at Liquids, then select Water.
12. Click Add to Component, then Water, Component 1 will appear in the Added to model: window, click Done to finish.

Part 3: Boundary Conditions and Mesh Generation

General comments: Our boundary conditions are no-slip at long walls, uniform flow “in” at a specified average velocity, and gauge pressure zero at exit.

Boundary conditions

-Inlet

13. In the Model Builder window, Right click Laminar Flow (spf) and select Inlet.
14. In the Graphics window, select the bottom boundary by clicking on it; then boundary 2 is added to the Settings(Inlet) window dialog box.

15. In the Settings (Inlet) window under Velocity, select Velocity Field. Enter the number 0 for the x-component of $u_0$ and 0.0175 m/s for the y-component of $u_0$. 
-Outlet

16. Right click Laminar Flow (spf) from the Model Builder window and select Outlet.
17. In the Graphics window, select the top boundary, then boundary 3 is added to the Settings(Outlet) window dialog box.

18. In the Settings (Outlet) window under Boundary Conditions, the defaults are correct. They should be Pressure, with $p_0=0$ Pa.

-Wall

19. In the Model Builder window under Laminar Flow (spf), there are a variety of elements that are part of the model we are building. Click on Wall 1. In the Settings (Wall) window the 1 and 4 boundary segments will already be selected (both side walls). Under Boundary Condition select No Slip (This is the default). This completes the geometry and boundary conditions for the model.
Mesh Generation

General comments: We now create the mesh for our calculation. We use the Free Triangular mesh generator of the Comsol software.

20. In the Model Builder window, right click on Mesh 1 and select Free Triangular.
21. In the Model Builder window select Size under Mesh 1, and in the Settings (Size) window under Element Size, choose Calibrate for: Fluid Dynamics and Predefined Fine.

22. In the Model Builder window, right click on Free Triangular 1, and choose Build Selected to build the mesh. This will take a few tens of seconds. To see the mesh structure (which is quite fine and looks all black when you see the whole flow domain), choose Zoom Box and zoom in on the inlet of the flow.
Solve

General comments: We are now ready to calculate the velocity and pressure fields for the problem.

23. In the Model Builder window, right click on Study 1 and select Compute. Messages will appear under the Graphics display indicating the progress of the calculation; there is a progress bar in the right-hand corner. After a few tens of seconds the color graph of the velocity field will appear. Click on Zoom Extents to see the whole slit; use Zoom Box to zoom in on the inlet or outlet or wherever you are interested in a close-up view. This concludes the calculation of the velocity and stress fields for the model.
Part 4: Generate Plots and Visualize the Solution

General comments: We have calculated the two-dimensional velocity, stress, and pressure fields for the flow; now we need to interpret and display the results. We create an arrow graph (of the velocity field), and we create a plot of the centerline pressure as a function of distance down the length of the slit (so that we can determine the $(-\Delta p/L)$ we will use to compare to the hand calculation). We also create a series of plots of the cross-sectional velocity distribution at various positions down the axis of the flow. These plots allow us to determine which parts of the flow domain actually represent well-developed flow.

**Arrow Graph**

24. Right click on **Results** from the Model Builder window and select **2D Plot Group**. Right click on **2D Plot Group 3** and select **Rename** and rename it **Arrow Plot**.

25. Right click on **Arrow Plot** and select **Arrow Surface**.

26. In the **Settings (Arrow Surface)** window, under **Arrow Positioning** select 10 points in x grid points and 25 points in y grid points.

27. At the top of the **Settings (Arrow Surface)** window, click on **Plot** to draw the arrow plot. Again use **Zoom Extents** to zoom out or **Zoom Box** to zoom in on entrance and/or exit. Note that the uniform flow at the inlet (bottom) gradually transforms to a parabolic profile at the exit (top).
Velocity Profile Generation

General comments: We plot data cross-sections by creating “cut lines.” We want to choose a set of cut lines across the flow at different positions downstream. First, we disable the current plot.

28. In the Model Builder window, Right click on Arrow Surface 1 and select Disable to clear the plot view.
29. Right click on Data Sets and select Cut Line 2D. Right click on Cut Line 2D 1 and rename it Velocity Slices.
30. In the Settings (Cut Line 2D) window select \((x_1, y_1) = (-0.01, 0.0)\) and \((x_2, y_2) = (0.01, 0.0)\).
This is a line across the inlet of the flow (click on and Zoom Extents to see the line highlighted). This is the first cross section. Now we create additional cross sections.

31. In Settings (Cut Line 2D) window, check Additional parallel lines and under Distances click on the icon to the right to bring up the Range dialog box. Enter Start= 0, Step= 0.025 m and Stop= 0.500 m and click on Replace.
32. Click on and Zoom Extents to see the lines.
33. To select which data to plot at the selected locations, in the Model Builder window right click on Results and select 1D Plot Group. Right click on 1D Plot Group 4 and rename it Part I Plots.
34. Right click on Part I Plots and select Line Graph. Right click on Line Graph 1 and rename it Velocity Slices Plot.
35. Click on Velocity Slices Plot and in the Settings (Line Graph) window go to Data set under Data and select Velocity Slices from the drop down menu.

36. Under y-Axis Data click on the Replace Expression icon on the right and click the arrow at Laminar Flow, then click the arrow at Velocity field, and double click v-Velocity field, y component.
37. Under **x-Axis Data** select Parameters: **Arc length (Default)**. “Arc Length” means for the cut line you selected, move along the contour of the line, wherever it goes.

38. Click on **Plot** to display the velocity slices calculated. The **Graphics** window shows the velocity across the cross section at the 20 locations we selected earlier.

---

**Pressure Profile Generation**

General comments: To compare our steady-state velocity profile results to hour hand calculation, we need to know what pressure gradient to insert into our solution. We solve for this now.

To obtain the pressure gradient, we create a plot of the variation for pressure down the centerline of the flow (from point \((x, y) = (0,0)\) to point \((x, y) = (0,0.5)\). First, we disable the velocity slices plot.
39. Right click **Velocity Slices Plot** in **Part I Plots** of **Results** and select **Disable**.
40. In **Model Builder** window, **Results**, right click on **Data Sets** and select **Cut Line 2D**. Right click on **Cut Line 2D 2** and rename it **Centerline Axis**.
41. In the **Settings (Cut Line 2D)** window, enter \((x, y) = (0,0)\) for **Point 1** and \((x, y) = (0,0.500)\) for **Point 2** under **Line Data**.
42. Click on \(\text{Plot}\) to see the line, which goes down the centerline of the flow.
43. In the **Model Builder** window select **Part I Plots** and then right click on **Part I Plots** and select **Line Graph**. Right click **Line Graph 2** and rename it to **Pressure Profile**.
44. To select where to evaluate the pressure, we select the appropriate cut line. In the **Settings (Line Graph)** window under **Data**, select **Centerline Axis** from the drop down menu of **Data set**.
45. Next, we select what variable to plot. In **y-Axis Data**, click o the **Replace Expression** icon and click the arrow at **Laminar Flow**, then double click **p-Pressure**.
46. In **Data**, **x-Axis Data** select **Parameters: Arc length (Default)**.
47. Then click on \(\text{Plot}\). Note that near the inlet, the pressure profile is not linear, but some distance downstream the flow reaches steady state, and the pressure gradient (slope) becomes constant.
Calculating Force/width on the Wall

General comments: To calculate the force per unit width on the wall, we need to integrate the total stress on a surface:

\[ F = \int_S [\hat{n} \cdot \Pi]_{\text{surface}} \, dS \]

Because the flow is 2D, the integral over \( dz \) (width direction) simply results in \( W \) (the width), which we can move to the left to give force per width. The integral along the \( y \)-direction is the only remaining integral to be done.

48. To integrate a quantity over a surface in a 2D calculation (for instance, to find the total force/width on the wall, which appears as a line in the 2D model), in the Model Builder window go to Results and right click on Derived Values, then select Integration, Line Integration, and make the appropriate choices in the dialog box:

- Data set: Study 1/Solution 1
- Selection: Select the right wall (boundary 4)
- Expression: spf.T_stressx for the x-component of \( \hat{n} \cdot \Pi \), spf.T_stressy for the y-component of \( \hat{n} \cdot \Pi \), and spf.T_stressz for the z-component of \( \hat{n} \cdot \Pi \) (one at a time).
- Unit: N/m (force per width of the flow)
- Integration settings Method: Auto
- Data Series Operation: None
- Click = (Evaluate) and see the answer in the lower right corner in the Messages box.

Note that your \( xy \)- and z-components of force/width should match the hand calculation, but the \( yx \)-component will not match because \( xy \)-force/width the calculation depends on the pressure profile. The Comsol calculation does not match the hand calculation for all values of \( y \) since the flow at the beginning of the channel is not yet well-developed).

Part 5: To Export Data

To make subsequent data calculations in Excel or in another program, we export the data to a text file that can be imported to the program of interest. First we do pressure; then we do the set of velocity sections.

49. Note that in the Model Window under Results and Part I Plots, Pressure Profile is in black text and Velocity Slices Plot is greyed out.

50. In the Model Builder window, right Click on Pressure Profile and select Add Plot Data to Export, then in the Settings (Plot) window click on Export. This will export the pressure data.
51. Under **Part I Plots**, **Disable Pressure Profile** and **Enable** the **Velocity Slices Plot**. Right Click on **Velocity Slices Plot** and select **Add Plot Data to Export**, then click on **Export** in the **Settings (Plot)** window. This will export the velocity data.

**Part 6: Import Data in Excel**

52. In Excel go to **File, Open**. Select your .txt file and click on **Open**. (Your data are delimited with spaces). When you save your file, save it as type .xlsx.

**Part 7: Write the Report**

53. Be sure to address all objectives in your report. Please make it easy for us to find your plots and calculated quantities.