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 $B=0.020\text{m}=2.0\text{cm}$. 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 steady-state 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 steady state flow? That is, in what region of your flow domain is the flow unchanging? Discuss your answer with reference to appropriate graphs.
3. Compare the total force/length on the wall (either side) with a hand calculation of the same quantity.

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). 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 steady velocity profile. We find the true steady state solution only in the portion of the flow domain in which the velocity profile is unchanged.

Once the Comsol flow calculations are complete, we can calculate the velocity gradient $-\frac{\Delta p}{L}$ 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.

Note also that when calculating the force/length on the top wall, we should only include the portion of the flow that is well developed (do not include the upstream portion when the velocity profile is not yet at steady state).

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.
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

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 4.3b**.

   The program opens up in the Wizard

2. Select 2D from the **Select Space Dimension** list.
3. Click the blue arrow for next page.
4. In the **Add Physics** window select (double click) **Fluid Flow, Single-Phase Flow, Laminar Flow (spf)**.
5. Click the blue arrow for next page.
6. Select **Stationary** at the **Present Studies Module** (Stationary=steady state; Time Dependent=unsteady).
7. Click the flag to finish.

Part 2: Set up the Flow Geometry and Physics Settings

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

Geometry Setting

8. To bring up the Rectangle/square dialog select the **Draw Rectangle** icon from the menu bar and drag any sized rectangle in the **Graphics Window**.

9. 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)**. Enter the following: Width= 0.020m; Height= 0.500m; Base Corner (x,y) = (−0.01,0.0).
10. To enter these new dimensions click on the **Build Selected** icon . 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, $v_y(x)$.

![Graph of velocity profile]

**Physics Settings**

Our fluid is water. We choose water from the Material Browser.

11. From the **Model Builder** menu, Right-click on **Materials** and select **Open Material Browser**.
12. Click the arrow at the **Liquids and Gases**, then **Liquids**, then select **Water**.
13. Right click on water and select **Add Material to Model**.

**Part 3: Boundary Conditions and Mesh Generation**

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**

14. In **Model Builder**, Right click **Laminar Flow (spf)** and select **Inlet**.
15. In the **Graphics Window**, select the bottom boundary by clicking on it, and under **Boundary Selection**, choose $\mathbb{1}$. Boundary 2 adds to the dialog box.
16. In the Inlet menu under **Velocity**, select **Velocity Field**. Enter the number 0 for $v_x$ and 0.0175 m/s for $v_y$.

17. Right click **Laminar Flow (spf)** from the Model Builder and select **Outlet**.

18. In the Graphics window, select the top boundary and click on +. Boundary 3 adds to the dialog box.

20. Under Laminar Flow (spf), click on Wall 1. In the wall window the 1 and 4 boundary segments will already be selected (both side walls). In boundary condition select No Slip (Default).

Mesh Generation

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

21. Right click on Mesh 1 from the Model Builder menu and select Free Triangular.
22. In Model Builder select Size under Mesh 1, and in the Size window under Element Size, choose Calibrate for Fluid Dynamics and Predefined Fine.
23. In the Model Builder box, right click on Free Triangular 1, and choose Build Selected to build the mesh 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.
Part 3: Solve

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

24. In Model Builder, right click on Study 1 and select Compute. Messages will appear under the graphics display indicating the progress of the calculation. After a few 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.

Part 4: Generate Plots and Visualize the Solution

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 (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 measure—$\Delta p / L$). We also create a series of plots of the cross-sectional velocity distribution at various positions down the axis of the flow. These plots will allow us to assess which parts of the flow domain actually represent steady state flow.

Arrow Surface

25. Right click on Results and select 2D Plot Group. Right click on 2D Plot Group 3 and select Rename and rename it Arrow Plot.
26. Right click on Arrow Plot and select Arrow Surface.

27. In the “Arrow Surface”, under Arrow Positioning select 10 points in x grid points and 25 points in y grid points.
28. Click on Plot to draw the arrow plot. Again use Zero Extents to zoom out or Zero 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**

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.

29. In the Model Builder, Right click on Arrow Surface 1 and select Disable to clear the plot view.
30. Right click on Data Sets and select Cut Line 2D. Right click on Cut Line 2D 1 and rename it Velocity Slices.

![Model Builder window](image)

31. In the 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 the Plot icon and Zoom Extents to see the line highlighted). This is the first cross section. Now we create additional cross sections.

![Cut Line 2D window](image)
32. In Cut Line 2D, check Additional parallel lines and under Distances click on the icon to the right to bring up the range menu. Enter Start= 0, Step= 0.025 $m$ and Stop= 0.500 $m$ and click on Replace.

33. Click on Plot icon to see the lines.
34. To select which data to plot at the selected locations, in Model Builder right click on Results and select 1D Plot Group. Right click on 1D Plot Group 4 and rename it Part I Plots.
35. Right click on Part I Plots and select Line Graph. Rename Line Graph 1 to Velocity Slices Plot.
36. Click on Velocity Slices Plot and go to the dialog window under Data and under Data set select Velocity Slices from the drop down menu.

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

39. Then select **Plot**.

**Pressure Profile Generation**

Now we want to create a plot of the variation for pressure down the centerline of the flow (from point \((x, y) = (0, 0)\) to point \((x, y) = (0, 0.500)\).

40. In **Results, Part I Plots, Velocity Slices Plot** right click and select **Disable**.

41. In **Results**, right click on **Data Sets** and select **Cut Line 2D**. Rename the cut line **Centerline Axis**.

42. In the **Cut Line 2D window** under **Line Data** enter \((x_1, y_1) = (0, 0)\) and \((x_2, y_2) = (0, 0.500)\).

43. Click on **Plot** to see the line, which goes down the centerline of the flow.
44. Select **Part I Plots** and then right click on **Part I Plots** and select **Line Graph**. Rename **Line Graph 2** to **Pressure Profile**.
45. To select where to evaluate the pressure, we select the appropriate cut line. In the **Line graph** window under **Data, Data set**, select **Centerline Axis** from the drop down menu.
46. Next, we select what variable to plot. In **y-Axis Data**. Click on **Replace Expression** icon and select **Laminar Flow, Pressure (p)**.

![Line Graph](image)

47. In **Data, x-Axis Data** select **Arc length (Default)**.
48. Then select **Plot**. Note that near the inlet, the pressure profile is not linear but some distance down stream the flow reaches steady state, and the pressure gradient (-slope) becomes constant.
49. To integrate a quantity over a surface, in **Model Builder** go to **Results, Derived Values** and right click on **Derived Values**. Choose **Integration, Line Integration**, and make the appropriate choices in the dialog box.

**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.
50. Note that under **Part I Plots**, *Pressure Profile* is in black text and *Velocity Slices Plot* is greyed out.

Choose **Part I Plots** and the **Export Data** icon will appear in the toolbar at the top.

51. Click on the **Export Data** icon and browse to where you wish to save the data; click **Save**. At the top of the *Plot* window, click on the **Export Icon** to export the chosen data. Because the *Pressure Profile* plot was active, these are the data that will be exported.

52. Under **Part I Plots**, disable *Pressure Profile* enable the *Velocity Slices Plot*. Export as before.

**Part 6: Import Data in Excel**

53. 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 as type .xlsx.

**Part 7: Write the Report**

54. Be sure to address all objectives in your report.