Homework 7: Poiseuille Flow with Symmetry Boundary

ME 590: Applied CFD and Numerical Heat Transfer

Graham Wilson, AER

Problem Statement

Consider laminar flow between parallel flat plates whose length and width are much greater than the distance between the plates (Poiseuille Flow). Assuming a uniform velocity profile at the inlet and a constant pressure at the outlet, develop a CFD model of the system to visualize the development of the velocity boundary layers, as illustrated in Figure 1. Use the following flow properties for your analysis:

$$\rho = 1\,\mathrm{kg/m^3} \qquad \nu = 0.01\,\mathrm{m^2/s}$$
$$L_y = 0.1\,\mathrm{m} \qquad u_\infty = 1\,\mathrm{m/s} \qquad N_y = 20$$

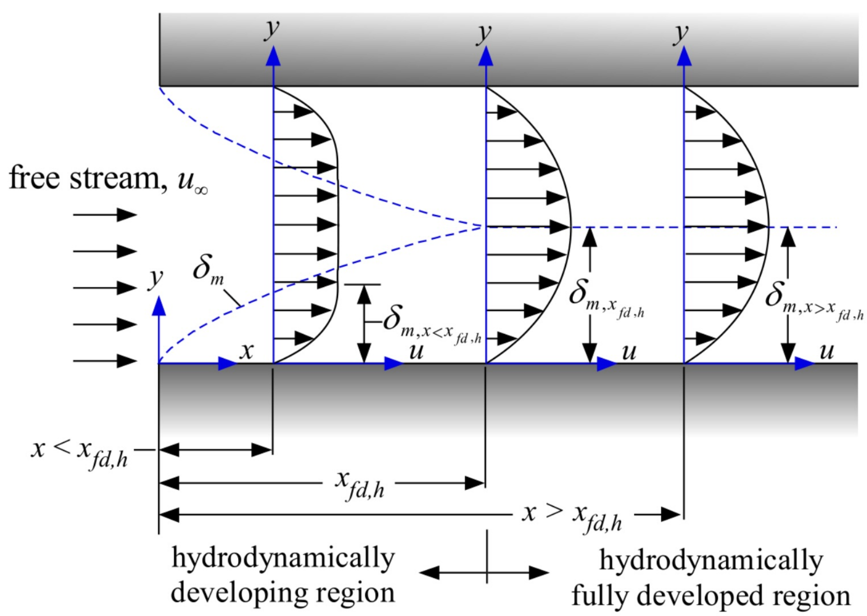

Figure 1: Internal flow hydrodynamic development

Solution

(a) CFD Solution Development Using OpenFOAM

The objective is to modify the grid geometry, boundary conditions, and timestep in the OpenFOAM lid-driven cavity tutorial to develop a CFD solution for fully developed flow using the PISO algorithm. We need to determine:

1. An appropriate timestep.
2. The minimum length of the domain in the x-direction for the flow to become fully developed.
3. The minimum number of timesteps for the system to reach steady state.

Flow Properties

Given:

- Density (ρ): $1\,\mathrm{kg/m^3}$
- Kinematic viscosity (ν): $0.01\,\mathrm{m^2/s}$
- Plate spacing (L_y): $0.1\,\mathrm{m}$
- Inlet velocity (u_∞): $1\,\mathrm{m/s}$
- Number of grid points in y-direction (N_y): 20

Grid Geometry

We consider a two-dimensional domain representing half of the channel due to symmetry. The dimensions are:

- Length in x-direction (L_x): To be determined.
- Height in y-direction ($L_y/2$): $0.05\,\mathrm{m}$

Boundary Conditions

Inlet ($x = 0$):

$$u = u_\infty \quad v = 0 \quad \frac{\partial p}{\partial x} = 0$$

Outlet ($x = L_x$):

$$\frac{\partial u}{\partial x} = 0 \quad v = 0 \quad p = 0$$

Symmetry Plane ($y = 0$):

$$\frac{\partial u}{\partial y} = 0 \quad v = 0 \quad \frac{\partial p}{\partial y} = 0$$

Wall ($y = L_y/2$):

$$u = 0 \quad v = 0 \quad \frac{\partial p}{\partial y} = 0$$

Implementation in OpenFOAM

We use the `icoFoam` solver in OpenFOAM, which employs the PISO algorithm for transient incompressible laminar flow.

Mesh Generation
The mesh is generated using the `blockMesh` utility with the specified domain dimensions and grid resolutions.

Boundary Conditions Setup
In the `0` directory:

- `U` file: Specifies velocity boundary conditions.
- `p` file: Specifies pressure boundary conditions.

Solver Settings
In the `system` directory:

- `controlDict`: Sets the timestep ($\Delta t = 0.001\,\text{s}$), end time ($t_{\text{total}} = 0.5\,\text{s}$), and write interval.
- `fvSchemes` and `fvSolution`: Define numerical schemes and solver settings.

Running the Simulation

The simulation is run using the following commands:

```
blockMesh
icoFoam
```

Results

(i) Pressure Contours

The pressure contours obtained from the simulation show a linear pressure gradient along the flow direction, consistent with Poiseuille flow theory.

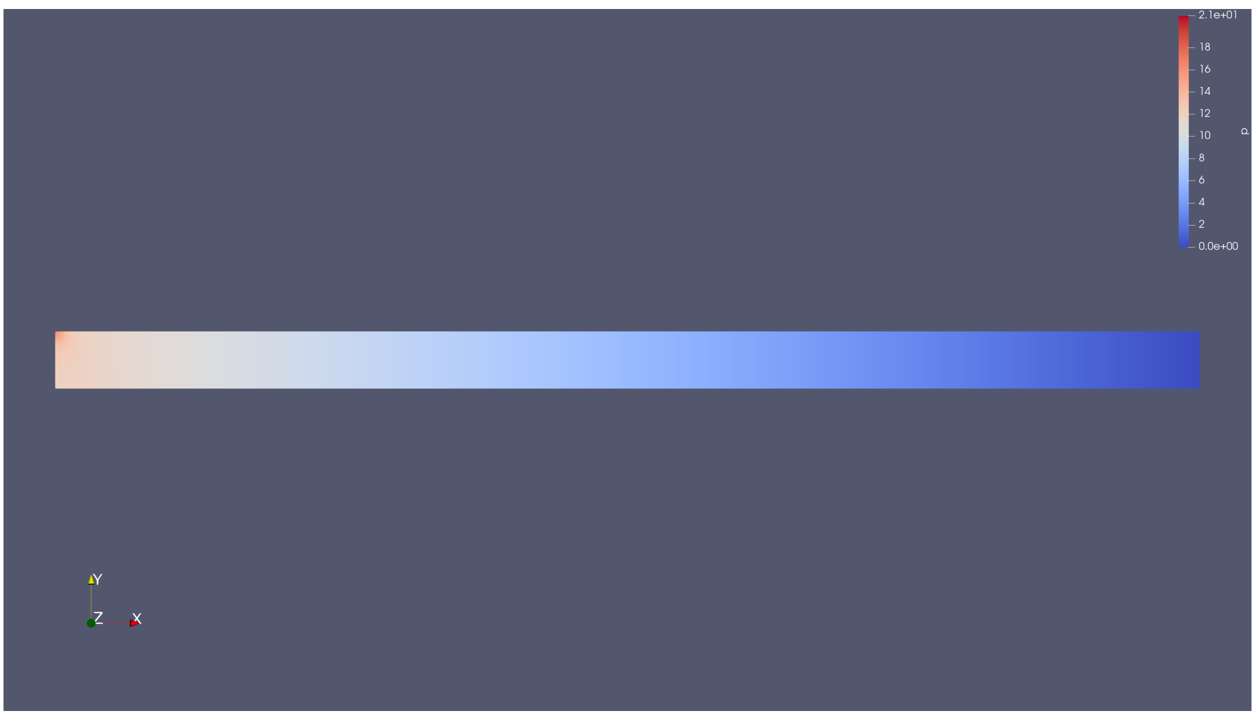

Figure 2: Pressure contours of the flow field

(ii) Velocity Contours

The velocity contours illustrate the development of the velocity profile from the uniform inlet condition to the fully developed parabolic profile.

Figure 3: Velocity contours showing flow development

(iii) Streamlines

The streamlines depict the flow patterns and confirm the laminar nature of the flow and the symmetry about the centerline.

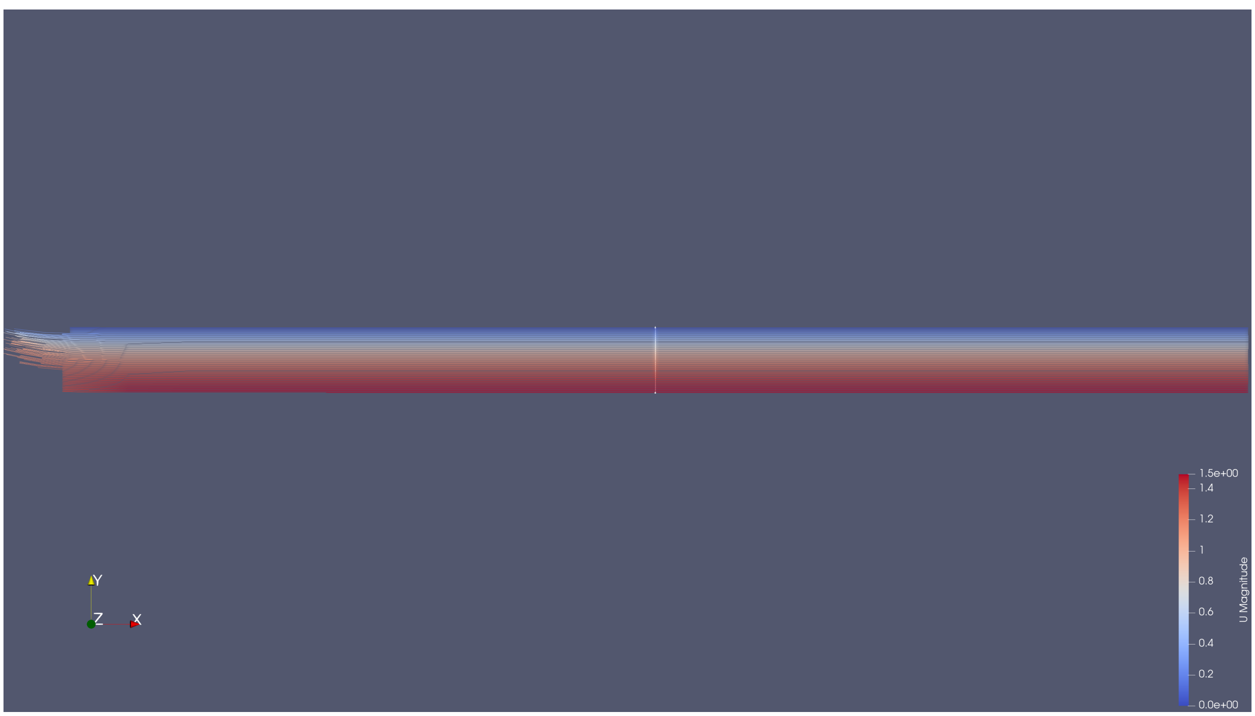

Figure 4: Streamlines of the flow field

(iv) Comparison with Full Domain Solution from Homework 6

In Homework 6, the same flow was simulated using MATLAB over the full channel height. The figure displays the following aspects:

Velocity and Pressure Contours with Streamlines

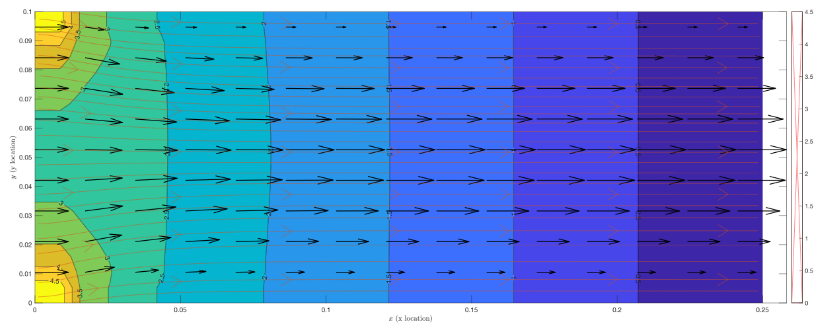

Figure 5: Velocity and Pressure contours with Streamlines from MATLAB (full domain)

Velocity Distribution
Figure 3 compares the velocity profiles at a location downstream where the flow is fully developed.

The OpenFOAM simulation (symmetry solution) matches the MATLAB full domain solution when mirrored about the centerline.

Pressure Distribution
Figure 2 compares the pressure profiles at a location downstream where the flow is fully developed.

Both simulations show a linear pressure drop along the x-direction, indicating consistent viscous effects in the flow.

Entrance Length
The entrance length estimated from the simulations is approximately $x_{fd,h} = 0.05\,\text{m}$, agreeing with the theoretical value calculated earlier.

Centerline Velocity
The fully developed centerline velocity from the OpenFOAM simulations is:

$$u_{\text{max}} \approx 1.5\,\text{m/s}$$

Which aligns with the theoretical prediction:

$$u_{\text{max}} = \frac{3}{2}u_{\infty} = 1.5\,\text{m/s}$$

However, the MATLAB full domain solution shows a slightly lower value of:

$$u_{\text{max}} \approx 1.41\,\text{m/s}$$

This difference may be attributed to the numerical discretization and solver settings used in the two simulations.