Computational Fluid Dynamics: Project 4

Hari Balaji (22110092)

Department of Mechanical Engineering

Prof. Dilip Srinivas Sundaram

1. Problem Statement

Lid-Driven Cavity Flow

The lid-driven cavity problem is a validation case used for testing new codes or CFD solution methods. It involves a simple geometry and a simple set of boundary conditions. Usually, the domain is defined as a two-dimensional square with three stationary sides and one side with a velocity parallel to its length [3].

Governing Equations

Continuity equation:

$$\frac{d}{dt}\int_{CV}\rho dV + \int_{CS}\rho(\vec{v}\cdot\hat{n})dA = 0 \tag{1}$$

2-D steady-state incompressible Navier-Stokes equations:
X-momentum equation:

$$\frac{d}{dt}\int_{CV}\rho u dA + \int_{CS}\rho u(\vec{v}\cdot\hat{n})dA = -\int_{CS}P(\hat{n}\cdot\hat{i})dA + \int_{CS}(\mu\vec{\nabla}u)\cdot\hat{n}dA \tag{2}$$

Y-momentum equation:

$$\frac{d}{dt}\int_{CV}\rho v dA + \int_{CS}\rho v(\vec{v}\cdot\hat{n})dA = -\int_{CS}P(\hat{n}\cdot\hat{j})dA + \int_{CS}(\mu\vec{\nabla}v)\cdot\hat{n}dA \tag{3}$$

There is no external force in the x or y-directions, and hence the body-force term is absent in both momentum equations. Also, the equations are solved at steady state, and hence the first term involving the time derivative becomes zero for all three equations.

Simulation Requirements

- Utilise the Finite Volume Method (FVM) for discretisation
- Use a staggered mesh and the SIMPLE Algorithm to solve the equations
- Choose various grid sizes (at least three) and demonstrate grid-independence of the results.
- Plot the contours of the x-velocity, y-velocity and pressure field for the finest grid.
- Plot the variation of the x-velocity, y-velocity and pressure along the $x = 0.5$ and $y = 0.5$ centrelines for the finest grid.
- Show the velocity streamlines for the various grid sizes considered and discuss the effects of grid refinement on the streamlines.

2. Mesh Details and Discretisation Approach

2.1. Mesh Details

The computational is shown as follows:

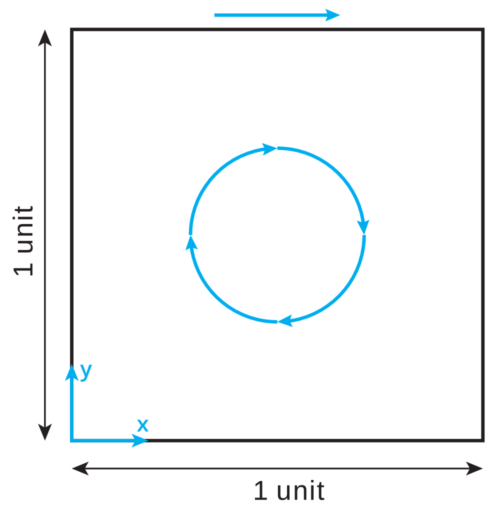

Figure 1. *Computational Domain*

The domain needs to be meshed in a staggered manner. This means that three separate meshes need to be defined, one each for the continuity, x-momentum and y-momentum equations. A staggered mesh is used to prevent odd-even decoupling between pressure and velocity. Odd-even coupling is a discretisation error which causes checkerboard patterns to arise in the solutions, invalidating the results [2]. So, there will be a total of three grids. The main continuity grid, the x-momentum grid staggered in the forward direction and the y-momentum grid staggered in the backward direction.
Assume that the domain is divided into a grid of $n \times n$ points.

1. u-velocity: There are $(n+1)\times(n+2)$ grid points. There are 2 extra grid points in the y-direction to include the exterior points, but only 1 extra point in the x-direction since the x-momentum grid is staggered in the forward direction.
2. v-velocity: There are $(n+2)\times(n+1)$ grid points. There are 2 extra grid points in the x-direction to include the exterior points, but only 1 extra point in the y-direction since the y-momentum grid is staggered in the upward direction.
3. Pressure: There are $(n+2)\times(n+2)$ grid points. There are 2 extra grid points in each direction to include the exterior points.

The staggered grid is shown below:

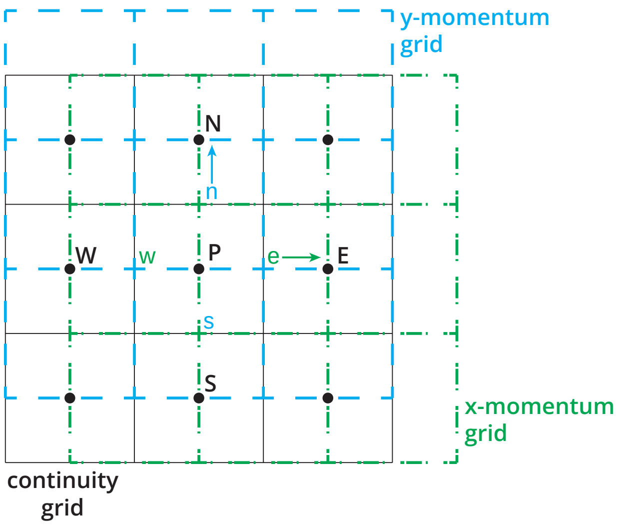

Figure 2. *Staggered Grid*

2.2. Boundary Conditions

The boundary conditions needed to be applied on the domain are shown as follows:

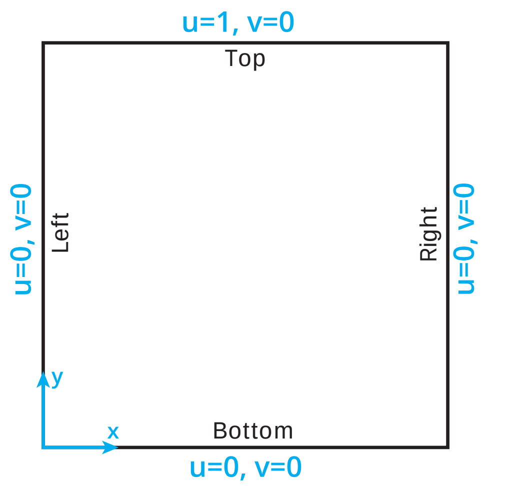

Figure 3. *Boundary Conditions*

The left, right and bottom walls are stationary and have zero velocities. The top wall has a velocity in the horizontal direction which essentially drives the flow.

Boundary	Velocity Condition
Left Wall	$u(0, y) = 0$ and $v(0, y) = 0$
Right Wall	$u(1, y) = 0$ and $v(1, y) = 0$
Top Wall	$\mathbf{u(x, 1) = 1}$ **and** $\mathbf{v(x, 0) = 0}$
Bottom Wall	$u(x, 0) = 0$ and $v(x, 0) = 0$

Table 1. *Boundary Conditions*

2.3. Variables and Constants Utilised

Variable	Value
Kinematic Viscosity (ν)	0.01
Grid points (n)	21 41 81

The viscosity μ can be written as equal to μ/ρ where rho is the density. In the code, ν has been used for ease since it reduces one variable. Since a grid independence study needs to be performed, three example grid sizes were considered.

2.4. Approach for discretising the equations

3. Discretised Equations

Continuity equation

$$\int_{CS} \rho(\vec{v} \cdot \hat{n}) dA = 0 \tag{4}$$

$$\int_E (\vec{v} \cdot \hat{n}) dA + \int_W (\vec{v} \cdot \hat{n}) dA + \int_N (\vec{v} \cdot \hat{n}) dA + \int_S (\vec{v} \cdot \hat{n}) dA = 0 \tag{5}$$

Simplifying the above equation, we get the discretised continuity equation as:

$$\boxed{u_e A_E - u_w A_W + u_n A_N - u_s A_S = 0} \tag{6}$$

X-momentum equation

$$\int_{CS} \rho u(\vec{v} \cdot \hat{n}) dA = -\int_{CS} P(\hat{n} \cdot \hat{i}) dA + \int_{CS} (\mu \vec{\nabla} u) \cdot \hat{n} dA \tag{7}$$

Considering the first term of the above equation,

$$\int_{CS} \rho u(\vec{v} \cdot \hat{n}) = \dot{m}_E u_E + \dot{m}_P u_P + \dot{m}_{ne} u_{ne} + \dot{m}_{se} u_{se} \tag{8}$$

Replacing the velocity terms by the average of the faces, we get:

$$\int_{CS} \rho u(\vec{v} \cdot \hat{n}) = \frac{\dot{m}_E}{2}(u_e + u_{ee}) + \frac{\dot{m}_P}{2}(u_e + u_w) + \frac{\dot{m}_{Ne}}{2}(u_e + u_{Ne}) + \frac{\dot{m}_{Se}}{2}(u_e + u_{Se}) \tag{9}$$

$$\int_{CS} \rho u(\vec{v} \cdot \hat{n}) = u_e \left\{ \frac{\dot{m}_E}{2} + \frac{\dot{m}_P}{2} + \frac{\dot{m}_{Ne}}{2} + \frac{\dot{m}_{Se}}{2} \right\} + u_{ee}\frac{\dot{m}_E}{2} + u_w \frac{\dot{m}_P}{2} + u_{Ne}\frac{\dot{m}_{Se}}{2} + u_{Se}\frac{\dot{m}_{Se}}{2} \tag{10}$$

Rewriting the above equation in a convenient form,

$$\int_{CS} \rho u(\vec{v} \cdot \hat{n}) = a_e^c u_e + \sum_{nb} a_{nb}^c u_{nb} \tag{11}$$

Considering the second term of equation 7,

$$-\int_{CS} P(\hat{n} \cdot \hat{i}) dA = P_P A_P - P_E A_E \tag{12}$$

Considering the third term of equation 7,

$$\int_{CS} (\mu \vec{\nabla} u) \cdot \hat{n} dA = \mu \int_E \vec{\nabla} u \cdot \hat{n} dA + \mu \int_P \vec{\nabla} u \cdot \hat{n} dA + \mu \int_{Ne} \vec{\nabla} u \cdot \hat{n} dA + \mu \int_{Se} \vec{\nabla} u \cdot \hat{n} dA \tag{13}$$

$$\int_{CS} (\mu \vec{\nabla} u) \cdot \hat{n} dA =$$
$$\mu \left\{ \frac{\partial u}{\partial x}\bigg|_E A_E - \frac{\partial u}{\partial x}\bigg|_P A_P + \frac{\partial u}{\partial y}\bigg|_{Ne} A_{Ne} - \frac{\partial u}{\partial y}\bigg|_{Se} A_{Se} \right\} \quad (14)$$

Now, we apply a central difference scheme to calculate the partial derivatives:

$$\frac{\partial u}{\partial x}\bigg|_E = \frac{u_{ee} - u_e}{\Delta x}$$

$$\frac{\partial u}{\partial x}\bigg|_P = \frac{u_e - u_w}{\Delta x}$$

$$\frac{\partial v}{\partial y}\bigg|_{Ne} = \frac{u_{Ne} - u_e}{\Delta y}$$

$$\frac{\partial v}{\partial y}\bigg|_{Se} = \frac{u_e - u_{Se}}{\Delta y}$$

Substituting these in equation 14, we get:

$$\int_{CS} (\mu \vec{\nabla} u) \cdot \hat{n} dA = u_e \left(-\frac{\mu A_E}{\Delta x} - \frac{\mu A_P}{\Delta x} - \frac{\mu A_{ne}}{\Delta y} - \frac{\mu A_{ee}}{\Delta y} \right)$$
$$+ u_{ee} \left(\frac{\mu A_E}{\Delta x} \right) + u_w \left(\frac{\mu A_P}{\Delta x} \right) + u_{Ne} \left(\frac{\mu A_{Ne}}{\Delta x} \right) + u_{Se} \left(\frac{\mu A_{Se}}{\Delta x} \right) \quad (15)$$

Rewriting the above equation in a convenient form, we get:

$$\int_{CS} (\mu \vec{\nabla} u) \cdot \hat{n} dA = a_e^v u_e + \sum_{nb} a_{nb}^v u_{nb} \quad (16)$$

Now re-substituting the simplified terms into equation 7, we get:

$$a_e^c u_e + \sum_{nb} a_{nb}^c u_{nb} = P_P A_P - P_E A_E + a_e^v u_e + \sum_{nb} a_{nb}^v u_{nb}$$

Simplifying the above equation, we get the discretised x-momentum equation as:

$$\boxed{a_e u_e = \sum_{nb} a_{nb} u_{nb} + P_P A_P - P_E A_E \quad (17)}$$

Y-momentum equation

We can perform a similar procedure for the y-momentum equation, and doing so, we get its discretised form as:

$$\boxed{a_n v_n = \sum_{nb} a_{nb} v_{nb} + P_P A_P - P_N A_N \quad (18)}$$

Pressure-Correction Equation for the SIMPLE Algorithm

4. Solution Methodology

The algorithm used for solving the discretised equations is the SIMPLE algorithm. *SIMPLE* stands for *Semi-Implicit Method for Pressure Linked Equations*. The methodology for the SIMPLE algorithm is as follows:

1. Guess the pressure field P^*.
2. Solve for the predicted velocities (u^* and v^*) using equations 17 and 18.
3. Define the corrections:
$$u = u^* + u' \quad (19)$$
$$v = v^* + v' \quad (20)$$
$$P = P^* + P' \quad (21)$$
4. Use the pressure corrector equation to obtain P':
$$a_P P'_P = \sum_{nb} a_{nb} P'_{nb} + b$$
$$b = (u_e^* A_E - u_w^* A_W) + (v_n^* A_N - v_s^* A_S)$$
Solve this equation to convergence.
5. Correct u, v and P using the corrections from step 2:
$$u = u^* + u' \qquad = u^* + \alpha_u (P'_P - P'_E) d_e \quad (22)$$
$$v = v^* + v' \qquad = v^* + \alpha_v (P'_P - P'_N) d_n \quad (23)$$
$$P = P^* + \alpha_P P' \quad (24)$$
Where $d_e = A_E / a_e$ and $d_n = A_N / a_n$, and α_P, α_u and α_v are under-relaxation factors.
6. Check for overall convergence.

5. Results and Discussions

Grid-Independence Study

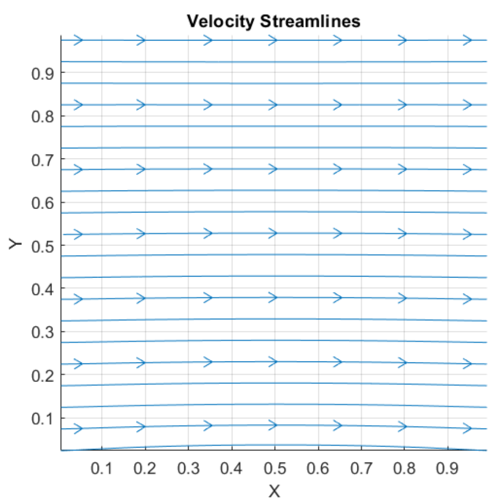

Figure 4. *For* $n = 21$

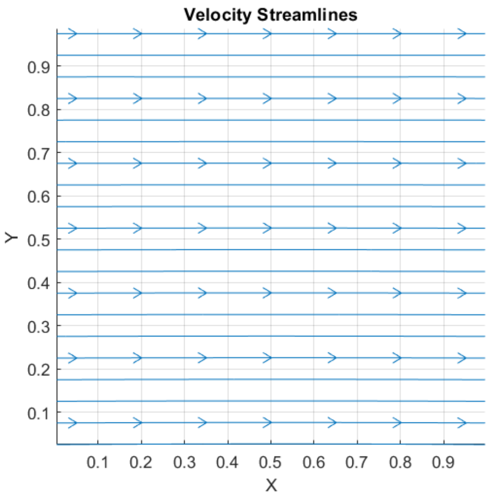

Figure 5. *For* $n = 31$

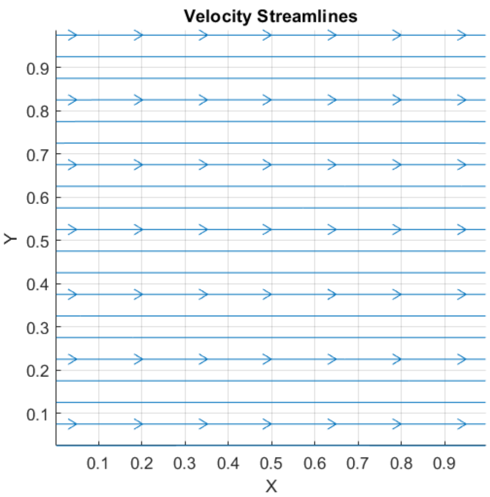

Figure 6. *For* $n = 41$

From the above figures, we can see that there is not much change in the velocity streamlines which were obtained for grid sizes of 31, 41 and 51 respectively. Even though there are slight changes and deviations in them, the general trend is relatively the same.
We can therefore say that the results are grid-independent.

Contour Plots

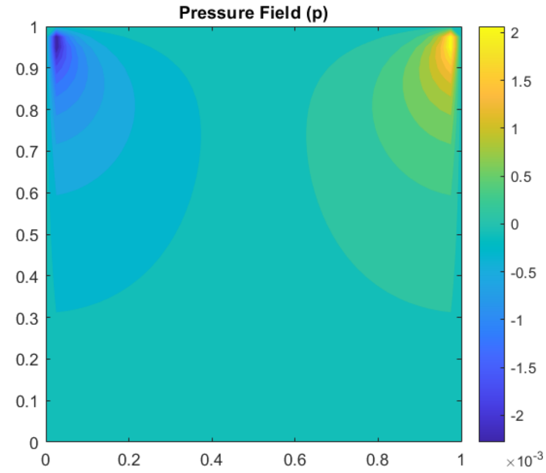

Figure 7. *Pressure Contours*

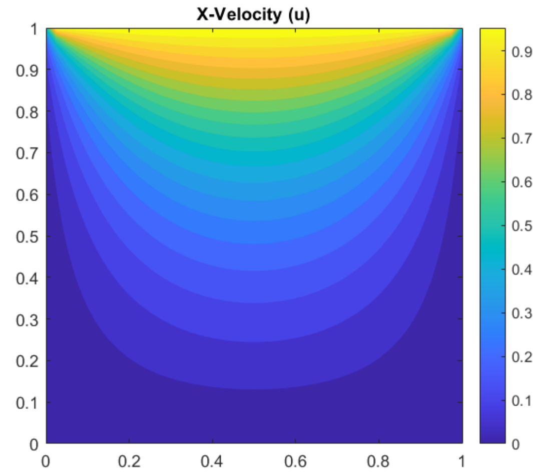

Figure 8. *u-velocity contours*

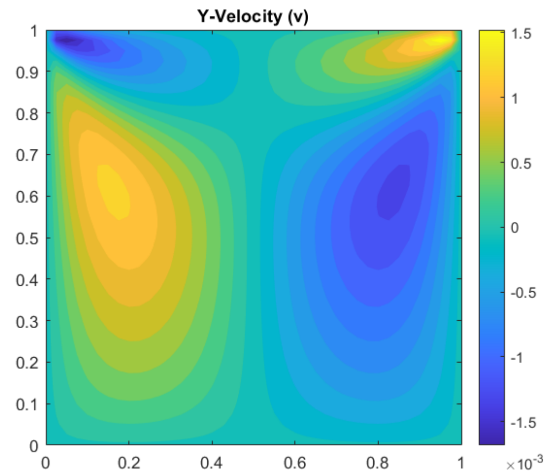

Figure 9. *v-velocity contours*

Plots along the x = 0.5 centreline

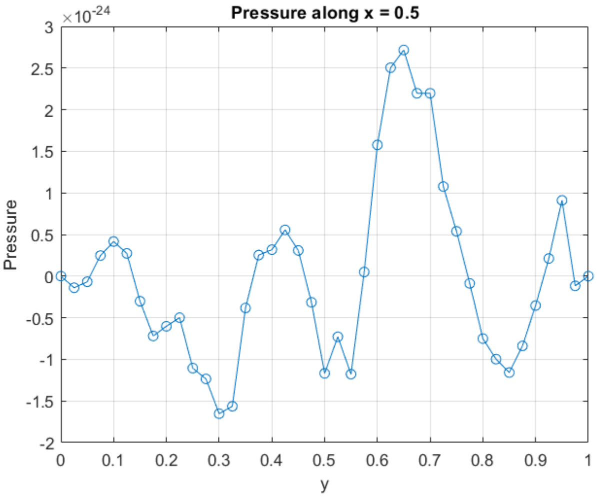

Figure 10. *Pressure along* $x = 0.5$

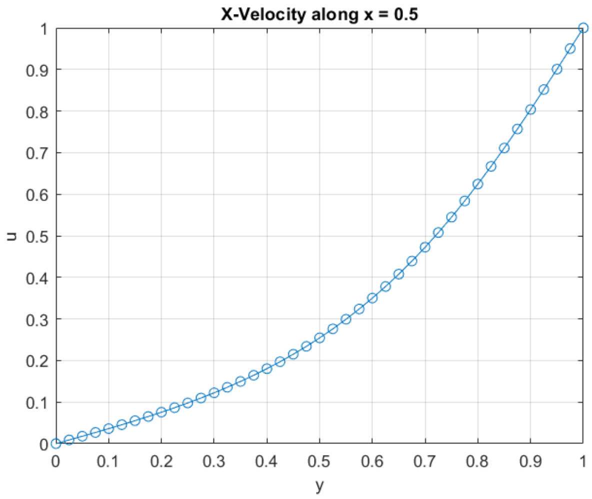

Figure 11. *u-velocity along* $x = 0.5$

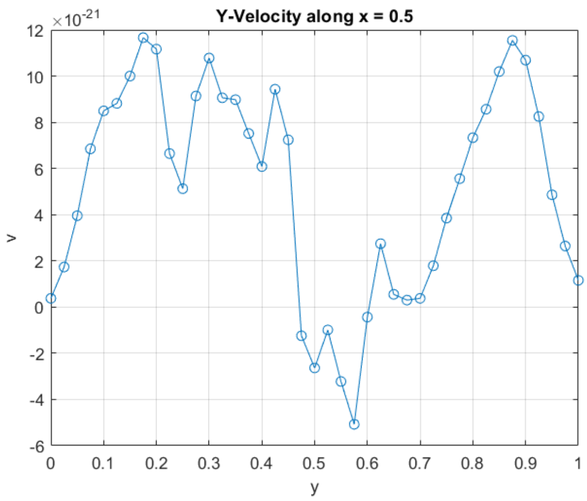

Figure 12. *v-velocity along* $x = 0.5$

Plots along the y = 0.5 centreline

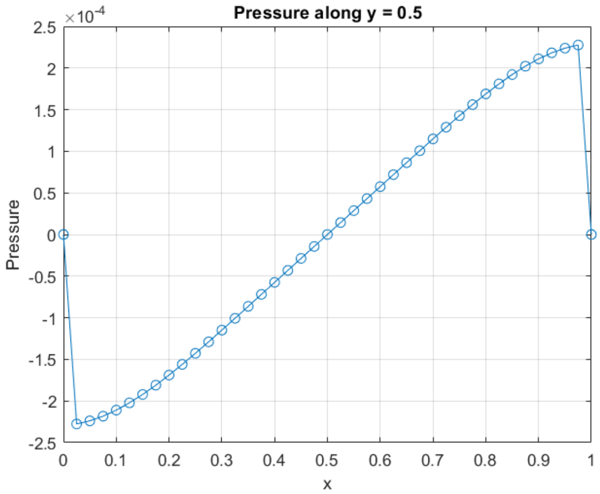

Figure 13. *Pressure along* $y = 0.5$

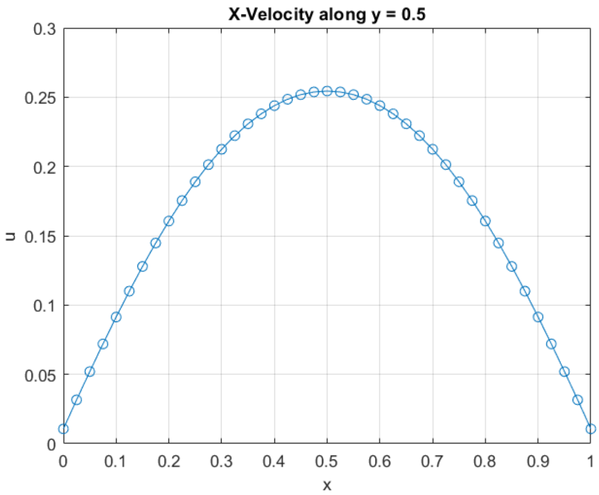

Figure 14. *u-velocity along* $y = 0.5$

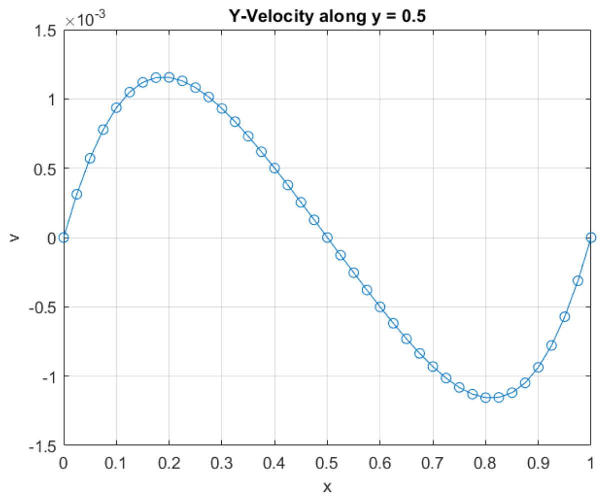

Figure 15. *v-velocity along* $y = 0.5$

Expected Results:

The below figures for the expected results have been obtained by the simulations of *Ghia et al., 1982* [1].

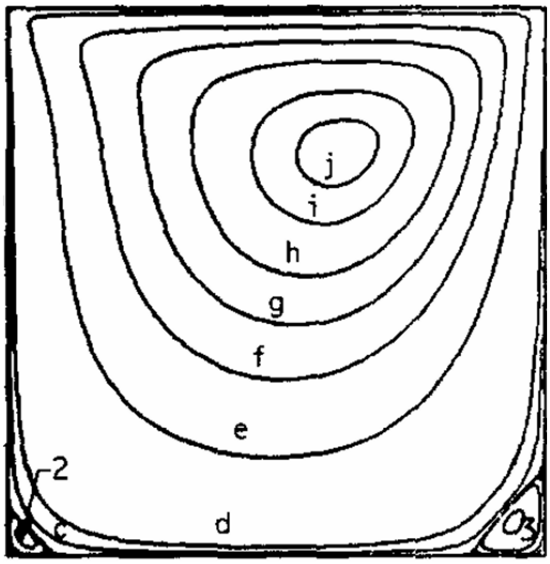

Figure 16. *Expected u-contours*

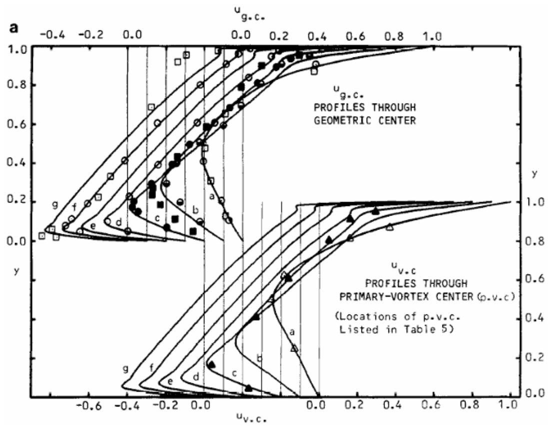

Figure 17. *Expected u-velocity about the centreline*

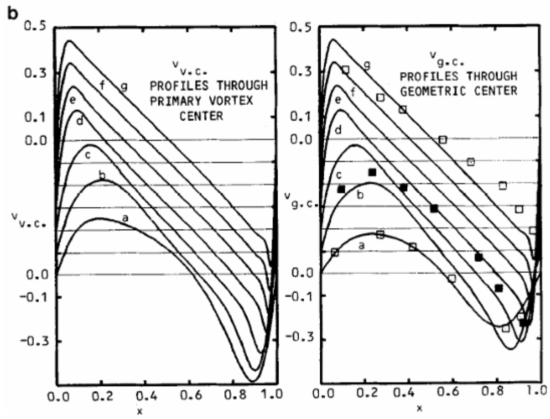

Figure 18. *Expected v-velocity about centreline*

Discussions

The plots of u-velocity and v-velocity The obtained results are not close to the expected results and this may be due to several reasons, such as instabilities in the code, along with incorrectly defined convergence or boundary conditions.

It can be seen that there are instabilities in the pressure and the v-velocity at the $x = 0.5$ centreline. Also, the u-velocity contour is not exactly as expected. Due to the lack of time, I was unable to satisfactorily identify the errors in the code and fix them so that the correct solution could be obtained.

Future debugging of the code can be done in the following manner:

- Check the convergence criteria.
- Check the implementation of the boundary conditions.
- Try multiple values of the under-relaxation factors.
- Check the implementation of the link coefficients and the staggered grid.
- Optimise the code and eliminate any redundancies.

6. Conclusion

In this project, I attempted to simulate the standard test case for CFD simulations - The lid-driven cavity flow. I was partially successful in doing so, and I am confident that with a little more time I would have been able to completely simulate the case.

The discretisation was performed using the finite volume method (FVM) and the solution algorithm used was the *Semi-Implicit Method for Pressure-Linked Equations* (SIMPLE) algorithm which utilises a staggered grid and a pressure correction equation. For this project, the contours of the u and v velocities and the pressure were plotted for the given boundary conditions: three fixed walls and one moving wall. Also, the variations of the velocities and the pressure along the centrelines $x = 0.5$ and $y = 0.5$ were also plotted. The obtained results were compared with existing literature and discrepancies were noted, along with ways to improve the simulation.

A grid-independence study was also performed to show that the simulation was independent of the chosen grid-size.

The project overall offered great insights into modern CFD simulations,

7. Acknowledgements

I would like to thank to Prof. Dilip S. Sundaram for teaching me the principles to enable me to successfully complete this project. His expertise and support greatly helped us.

I would also like to thank the TAs Malay Vyas and Pardha Sai for helping me to complete this project.

I would also like to thank IIT Gandhinagar for providing me the platform and support necessary to complete this project.

Lastly, I would like to thank my peers, for their discussions with me which were instrumental in completing this project successfully.

References

[1] U. Ghia, K. N. Ghia, and C. Shin, "High-re solutions for incompressible flow using the navier-stokes equations and a multigrid method", *Journal of computational physics*, vol. 48, no. 3, pp. 387–411, 1982.

[2] CFD Online, *Staggered grid*, Accessed: 2024-11-21, 2024. [Online]. Available: https://www.cfd-online.com/Wiki/Staggered_grid#:~:text=Using%20a%20staggered%20grid%20is,checkerboard%20patterns%20in%20the%20solutions..

[3] *Lid-driven cavity problem*, https://www.cfd-online.com/Wiki/Lid-driven_cavity_problem, Accessed: 2024-11-17.