

Analysis of Lid driven Cavity

-Rahul Kumar Meena (23117106)

-Mridul Agrawal (23117082)

Objective and Introduction.

This report presents the development and implementation of a numerical solver for the **two-dimensional lid-driven cavity flow**, a classical and widely studied benchmark problem in **computational fluid dynamics (CFD)**. The primary goal is to simulate and analyze the unsteady behavior of an **incompressible, viscous fluid** confined within a square cavity, where the **top boundary (lid) moves horizontally** with a uniform velocity while all other boundaries remain stationary and impermeable.

The lid-driven cavity problem is especially significant in CFD due to its well-defined geometry, boundary conditions, and sensitivity to Reynolds number

In this study, we employ a **finite difference method (FDM)** to discretize the Navier-Stokes equations in their **primitive variable formulation** (velocity-pressure form). A **projection method** is used to enforce the incompressibility condition by solving a **Poisson equation for pressure correction**. The simulation is carried out on a uniform Cartesian grid, with the flow evolved in time using an explicit time integration scheme.

Although this work uses the primitive-variable approach instead of the vorticity-stream function formulation mentioned in other benchmark studies, the solver remains consistent with core CFD methodologies. Our implementation is validated by examining the **flow characteristics at $Re = 50$** , including velocity fields, streamlines, and velocity profiles along the cavity centerlines.

This project not only demonstrates the numerical solution of the Navier-Stokes equations for internal flows but also deepens our understanding of:

- Pressure-velocity coupling in incompressible flows,
- Boundary-layer formation and vortex development,
- Grid-based discretization schemes and their accuracy.

By exploring flow behavior at moderate Reynolds numbers and visualizing **velocity magnitude, streamlines, and centerline profiles**, this study provides valuable insight into the dynamics of confined viscous flows and strengthens practical knowledge of **numerical methods in CFD**.

Governing Equations.

The non-dimensional form of the governing equations for two-dimensional incompressible flow is given by:

1. Continuity Equation (Incompressibility condition)

$$\partial u / \partial x + \partial v / \partial y = 0$$

2. x-Momentum Equation

$$\partial t / \partial u + u * \partial x / \partial u + v * \partial y / \partial u = - \partial p / \rho \partial x + n u * (\partial^2 u / \partial x^2 + \partial^2 u / \partial y^2)$$

3. y-Momentum Equation

$$\partial v / \partial t + u * \partial v / \partial x + v * \partial v / \partial y = - 1 \partial p / \rho \partial y + n u * (\partial^2 v / \partial x^2 + \partial^2 v / \partial y^2)$$

Where:

- u , v are the velocity components in the x and y directions respectively
- p is the pressure
- $n u$ is the kinematic viscosity ($n u = 1 / Re$)
- ρ is the density
- t is time
- x , y are the spatial coordinates

These equations govern the time evolution of velocity and pressure fields within the cavity.

Numerical Methods.

The solver uses a **finite difference method (FDM)** on a **uniform Cartesian grid**, employing a second-order central difference scheme for spatial derivatives and a forward Euler method for time integration. The computational domain is discretized into 51×51 grid points, spanning a unit square cavity of size 1.0×1.0 .

The numerical solution advances through the following steps:

1. Boundary Conditions

- **No-slip conditions** are applied on all walls.

$$u=v=0 \text{ on left, right, and bottom walls}$$

- The **top lid** moves with a uniform velocity:

$$u=1.0, v=0 \text{ at the top wall}$$

2. Right-Hand Side (RHS) Construction for Pressure Poisson Equation

A source term \mathbf{b} is constructed to enforce mass conservation by incorporating divergence of the intermediate velocity field. This step ensures the resulting pressure field will correct velocities to be divergence-free.

3. Pressure Poisson Solver

To enforce incompressibility, a **Poisson equation for pressure** is solved iteratively using **Jacobi iteration**:

$$\nabla^2 p = \text{RHS}(\mathbf{b})$$

Dirichlet and Neumann boundary conditions are applied to ensure physical realism (e.g., zero normal pressure gradient at walls and $p=0$ at the top-right corner to anchor the solution).

4. Velocity Field Update

With the new pressure field, the velocity components are updated using the momentum equations. Advection, pressure gradient, and diffusion terms are all discretized using finite differences.

5. Convergence Check

Every 500 time steps, the solver checks for convergence by evaluating the difference between successive velocity fields:

$$\text{Difference} = \sum |u^{n+1} - u^n| + \sum |v^{n+1} - v^n|$$

If the difference falls below a pre-defined tolerance (1×10^{-5}), the simulation is considered converged.

Time stepping and stability.

The time step $dt=0.001$ is chosen to satisfy stability requirements (CFL condition), especially important given the explicit nature of the scheme. Although computationally expensive due to the number of iterations (up to 5000), this ensures accuracy and stability of the unsteady solution.

Summary of key numerical features.

Feature	Method
Grid	Uniform Cartesian grid
Discretization	Second-order central difference (space), Forward Euler (time)
Pressure Solver	Iterative solution of Poisson equation
Time Step	Fixed $dt=0.001$
Velocity-Pressure Coupling	Projection method using pressure correction
Boundary Conditions	No-slip, lid-driven at top boundary
Convergence Criteria	Velocity residual $< 10^{-5}$

Simulations Parameters.

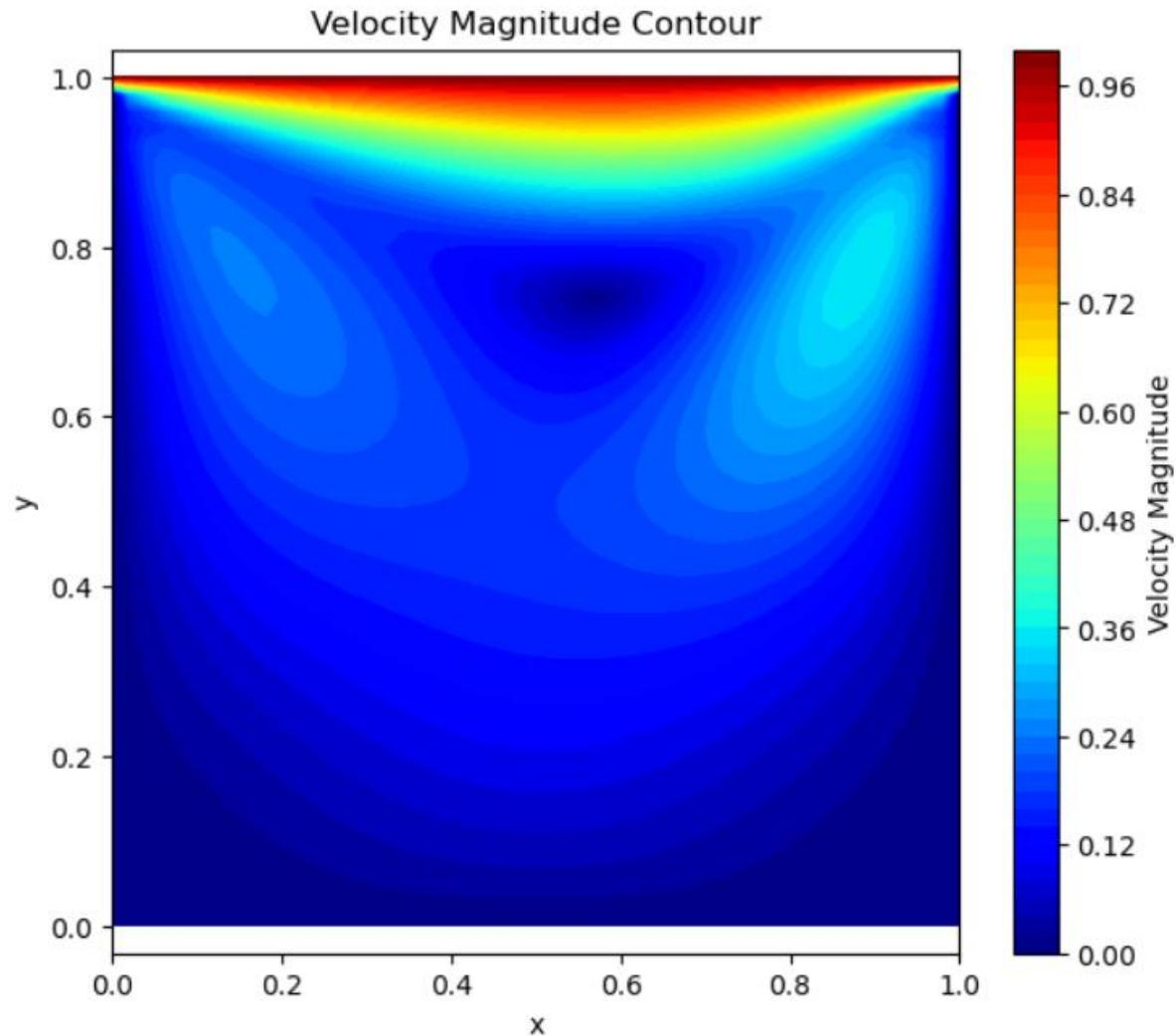
Parameter	Value
Domain size $L_x=L_y$	1.0
Grid size $n_x = n_y$	51
Reynolds number (Re)	50
Kinematic viscosity ν	$1/Re=0.021$
Time step dt	0.001
Total steps nt	5000
Density ρ	1.0
Convergence tolerance	10^{-5}

Results.

```
Iteration 0, Difference = 2.450000
Iteration 500, Difference = 0.282107
Iteration 1000, Difference = 0.126835
Iteration 1500, Difference = 0.068827
Iteration 2000, Difference = 0.039824
Iteration 2500, Difference = 0.023605
Iteration 3000, Difference = 0.014099
Iteration 3500, Difference = 0.008432
Iteration 4000, Difference = 0.005037
Iteration 4500, Difference = 0.003005
```

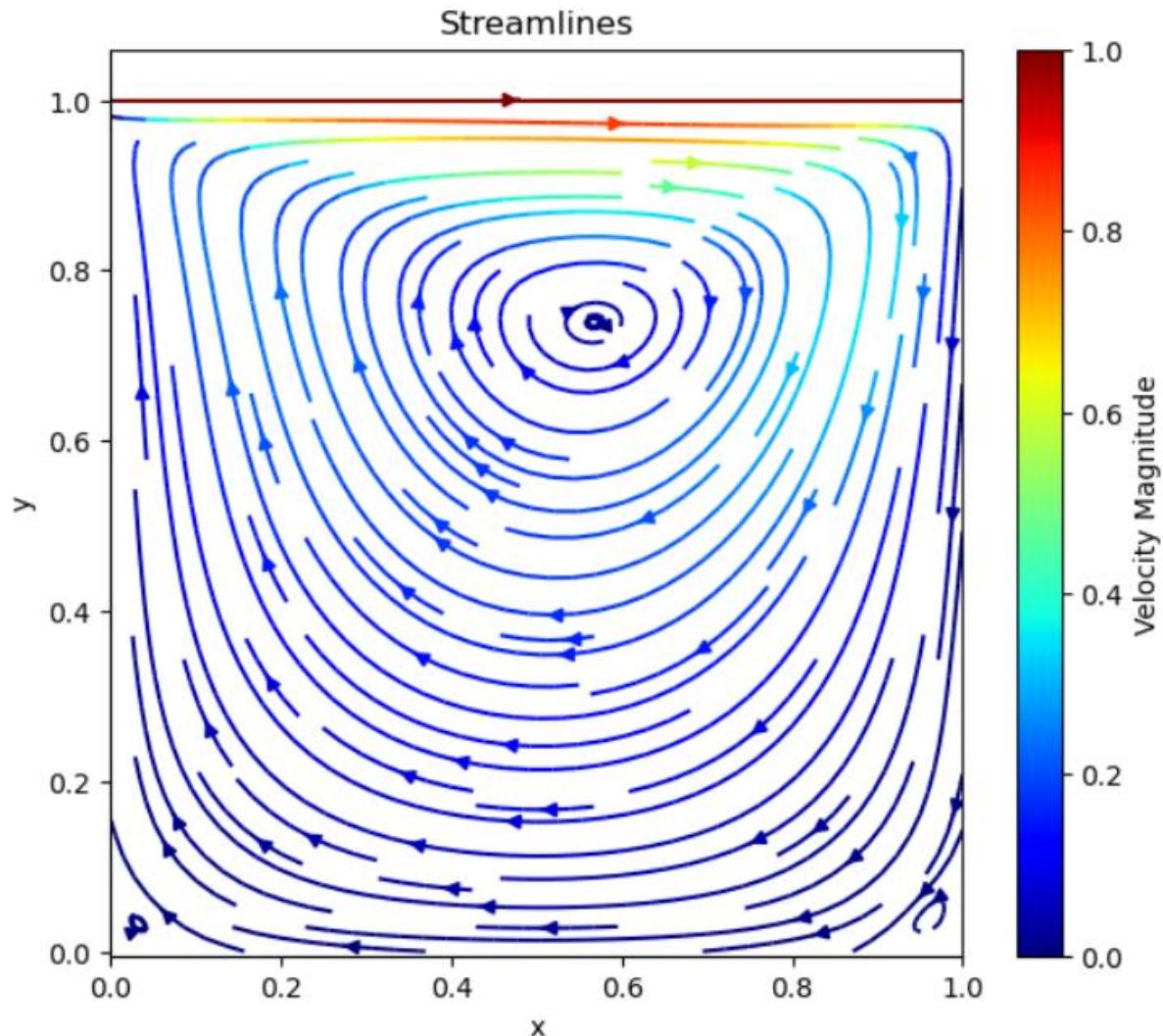
◆ Velocity Magnitude Contour

A contour plot of velocity magnitude shows the highest speed near the moving lid and a well-developed primary vortex at the cavity centre. Secondary vortices near corners may also appear depending on the Reynolds number.



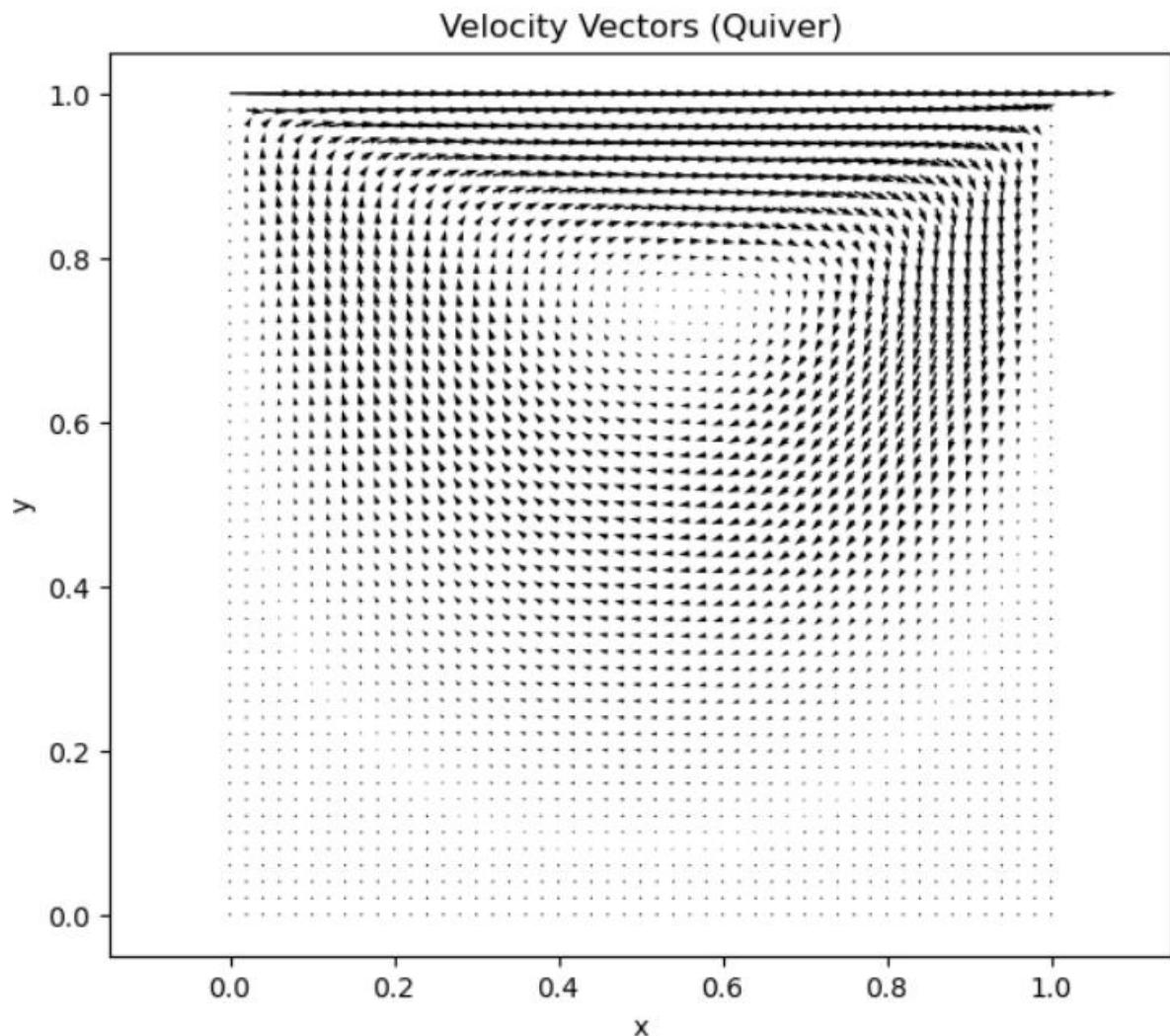
◆ Streamlines

Streamlines depict the circular motion inside the cavity, indicating the formation of the main recirculation zone and the development of boundary layers along the walls.



◆ Velocity Vectors (Quiver Plot)

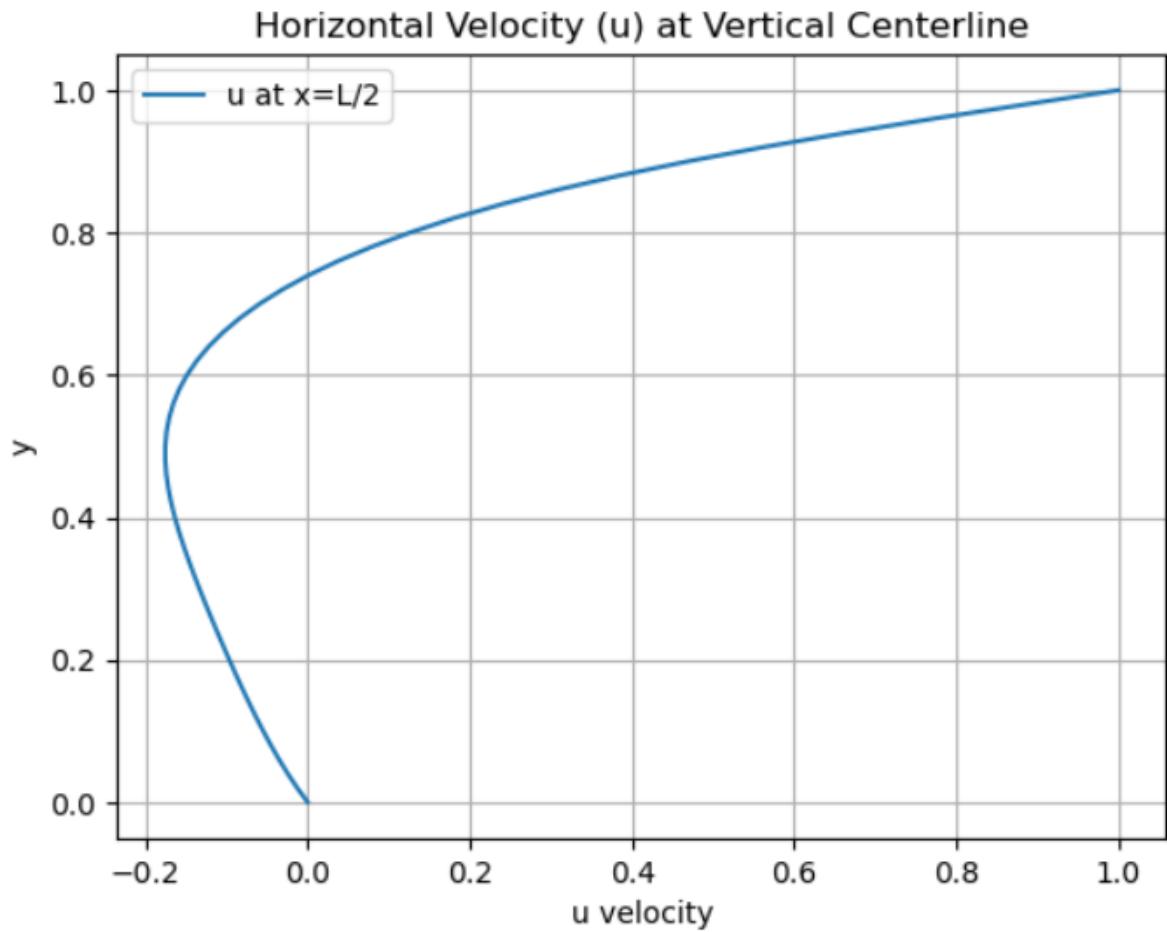
The quiver plot provides directional flow information. The top lid drives the flow, which circulates through the cavity forming a single dominant vortex.



◆ Velocity Profiles

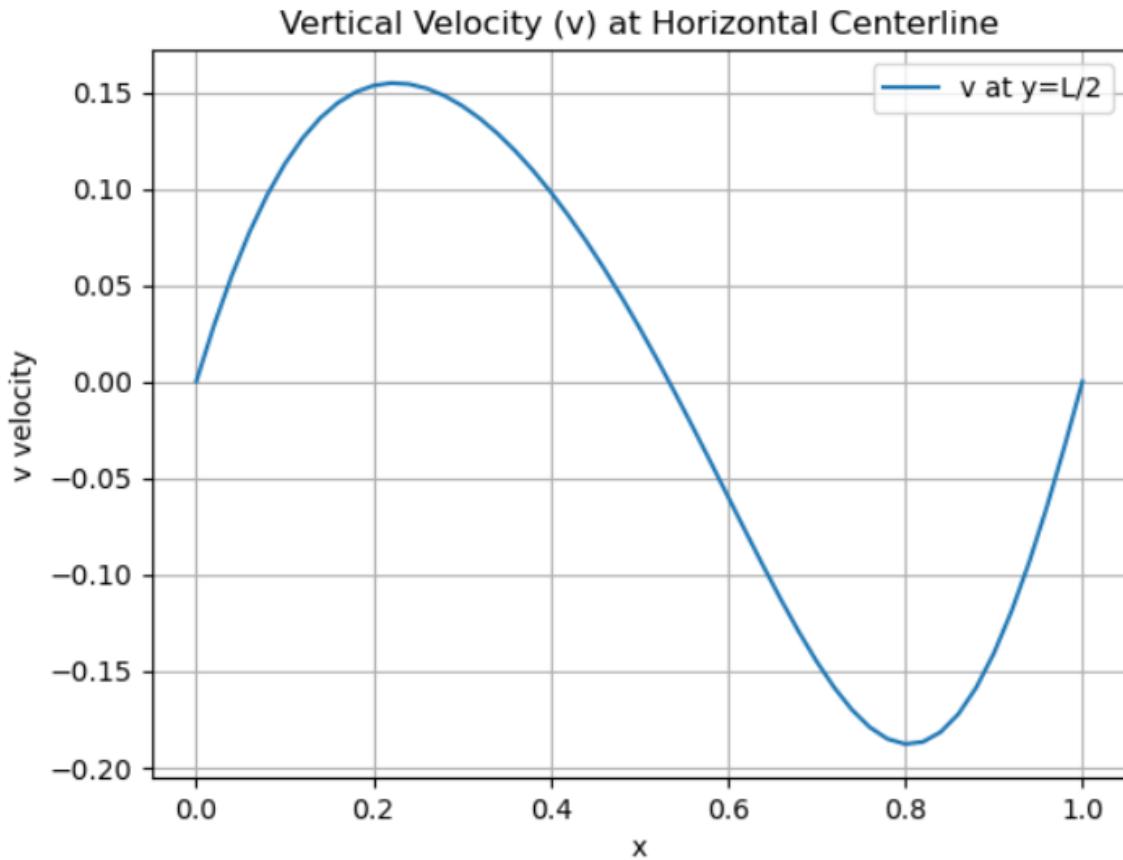
▪ Horizontal Velocity u at the Vertical Centreline:

This profile shows how the horizontal velocity varies along the y -direction at the midpoint of the cavity. A peak is observed near the moving lid.



- **Vertical Velocity v at the Horizontal Centreline:**

This shows vertical velocity variations along the x-direction. The profile is symmetric about the cavity centre and confirms the flow recirculation.



Discussion

- The simulation captures the essential features of lid-driven cavity flow.
- The primary vortex is centered with symmetric structure, validating correctness at $Re = 50$.
- As the Reynolds number increases, the secondary corner vortices become more pronounced (this can be further studied).

Conclusions

This 2D lid-driven cavity flow simulation demonstrates the use of the finite difference method to solve incompressible Navier-Stokes equations. The code accurately simulates the velocity field and pressure distribution inside the cavity. Such simulations are foundational for understanding internal viscous flows.

Future Works

- Extend to higher Reynolds numbers (e.g., $Re = 1000$) to observe secondary and tertiary vortices.
- Use higher-order or implicit schemes for stability and accuracy.
- Compare with benchmark data from Ghia et al. (1982).