CFD (AE 706)

Assignment no.5

A report on

Computation of Lid-Driven Cavity Flow Using Vorticity-Stream Function Formulation

Instructor:

Prof. J C Mandal

Submitted By:

Ram Naresh Mahato

Roll No: 24M0065

Department of Aerospace Engineering

Indian Institute of Technology Bombay

Date: April 18, 2025

Table of Contents

1 Introduction

The lid-driven cavity flow is a classic benchmark problem in computational fluid dynamics (CFD) used to validate numerical methods for incompressible flows. This assignment focuses on solving the steady-state flow in a two-dimensional square cavity with a moving lid using the vorticity-stream function formulation. The problem involves computing the flow field for a Reynolds number of 100, where the flow is governed by the Navier-Stokes equations expressed in terms of vorticity and stream function.

1.1 Governing Equations

The flow is described by the following equations:

- **Vorticity Transport Equation**:

$$\frac{\partial \omega}{\partial t} + u\frac{\partial \omega}{\partial x} + v\frac{\partial \omega}{\partial y} = \nu \left(\frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2} \right) \tag{1}$$

  where ω is the vorticity, defined as:

$$\omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}. \tag{2}$$

- **Stream Function Equation (Poisson Equation)**:

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = -\omega, \tag{3}$$

  where the velocity components u and v are derived from the stream function ψ:

$$u = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x}. \tag{4}$$

1.2 Boundary Conditions

The boundary conditions for the lid-driven cavity are as follows:

- **Lid (Top Wall)**:
  - Velocity: $u = 1\,\mathrm{m/s}$, $v = 0\,\mathrm{m/s}$.
  - Vorticity: $\omega_{i,J} = \frac{2(\psi_{i,J-1} - \psi_{i,J})}{\Delta y^2} - \frac{2u_{i,J}}{\Delta y}$.
- **Bottom and Side Walls**:
  - No-slip condition: $u = 0\,\mathrm{m/s}$, $v = 0\,\mathrm{m/s}$.

– Vorticity:

$$\omega_{1,j} = -\frac{2(\psi_{2,j} - \psi_{1,j})}{\Delta x^2} \quad \text{(left wall)},$$
$$\omega_{I,j} = -\frac{2(\psi_{I-1,j} - \psi_{I,j})}{\Delta x^2} \quad \text{(right wall)},$$
$$\omega_{i,1} = -\frac{2(\psi_{i,2} - \psi_{i,1})}{\Delta y^2} \quad \text{(bottom wall)}.$$

1.3 Convergence Criteria

The convergence of the numerical solution is assessed using the following criteria:

- **Stream Function Residual**: The Root Mean Square (RMS) Residual of the Poisson equation is calculated as:

$$R_2 = \sqrt{\frac{1}{N} \sum_{i,j}^{I-2,J-2} R_{i,j}^2}, \tag{5}$$

  where $R_{i,j}$ is the residual at grid point (i, j). The iteration terminates when $R_2 \leq 10^{-2}$.

- **Velocity Residual**: The RMS residuals for the velocity components u and v are defined as:

$$\text{RMS}_f = \sqrt{\frac{1}{N} \sum_{i,j}^{I-2,J-2} \left(f_{i,j}^{(n+1)} - f_{i,j}^{(n)} \right)^2}, \tag{6}$$

  where f represents either u or v. The time iteration stops when RMS_u and RMS_v fall below 10^{-8}.

1.4 Time Step Calculation

The time step Δt is determined based on stability conditions:

- **Convective Time Step**:

$$\Delta t_c = \sigma_c \frac{\Delta x \Delta y}{|u_{\max}|\Delta y + |v_{\max}|\Delta x}, \tag{7}$$

  where $\sigma_c = 0.4$ is the Courant number.

- **Diffusive Time Step**:

$$\Delta t_d = \sigma_d \frac{1}{2\nu} \left(\frac{\Delta x^2 \Delta y^2}{\Delta x^2 + \Delta y^2} \right), \tag{8}$$

  where $\sigma_d = 0.6$ is the diffusion number.

- **Final Time Step**:

$$\Delta t = \min(\Delta t_c, \Delta t_d). \tag{9}$$

2. Results and plots

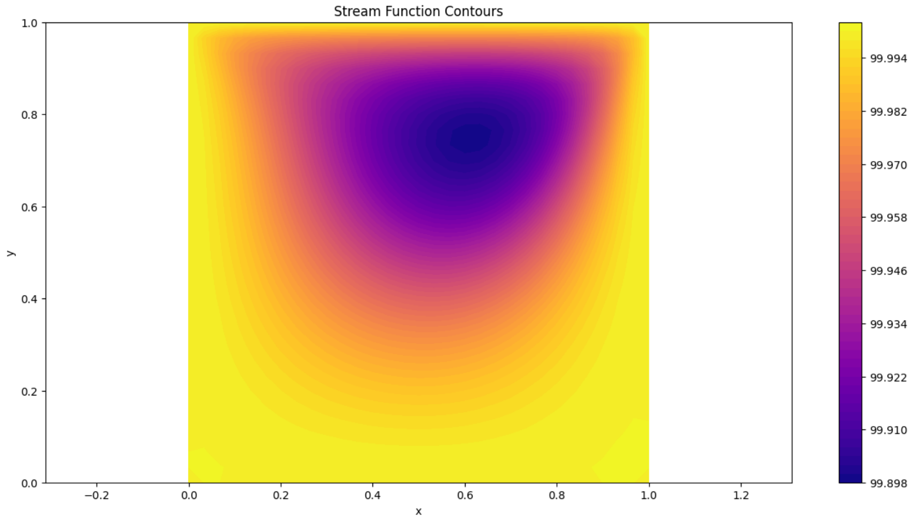

Figure 1: Stream Function Contours

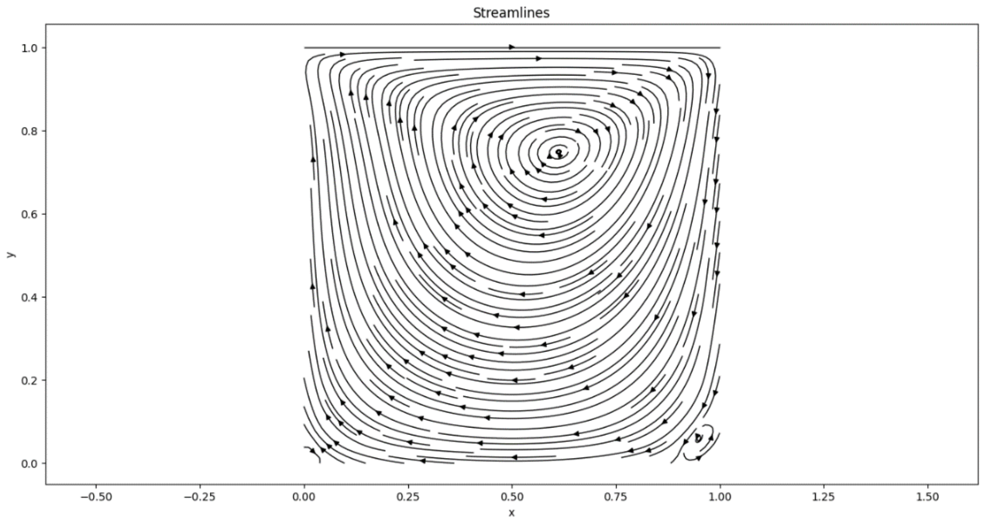

Figure 2: Streamline plot

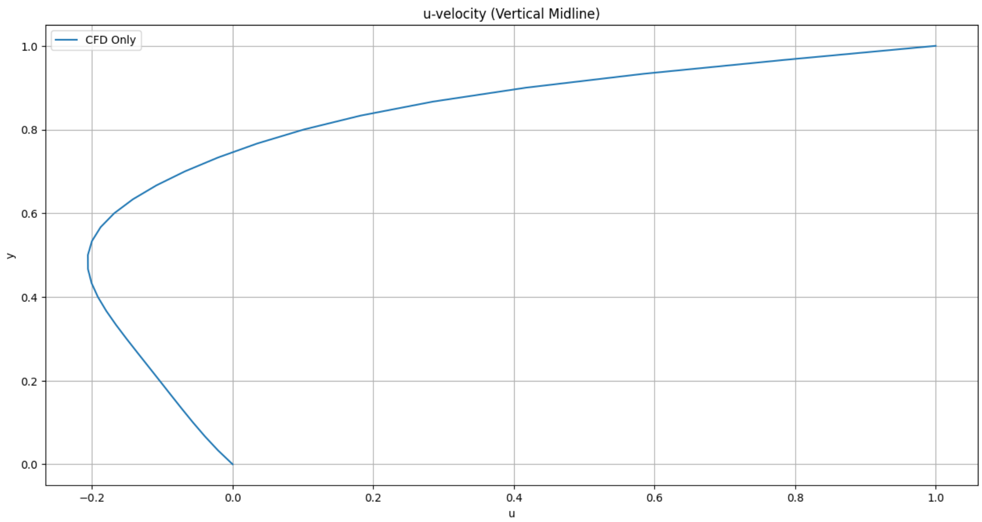

Figure 3: Distribution of the x-component of velocity vector (u) along the mid vertical line.

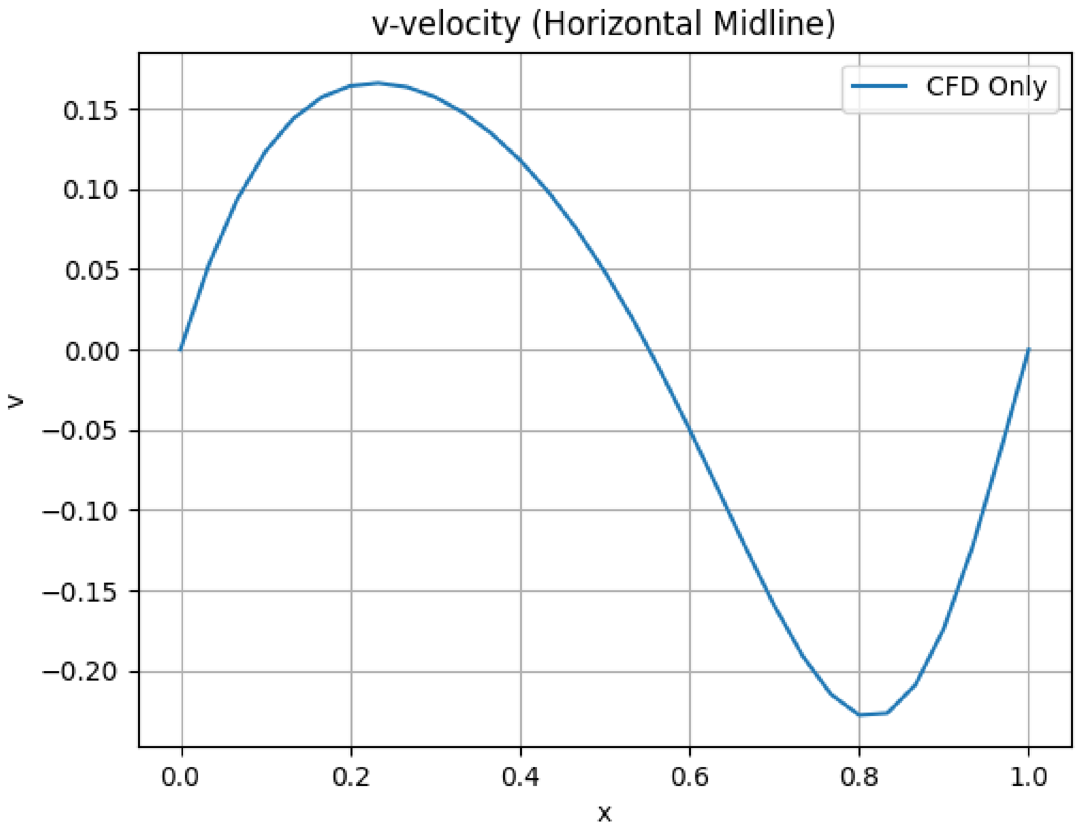

Figure 4: Distribution of the y-component of velocity vector (v) along the mid horizontal line

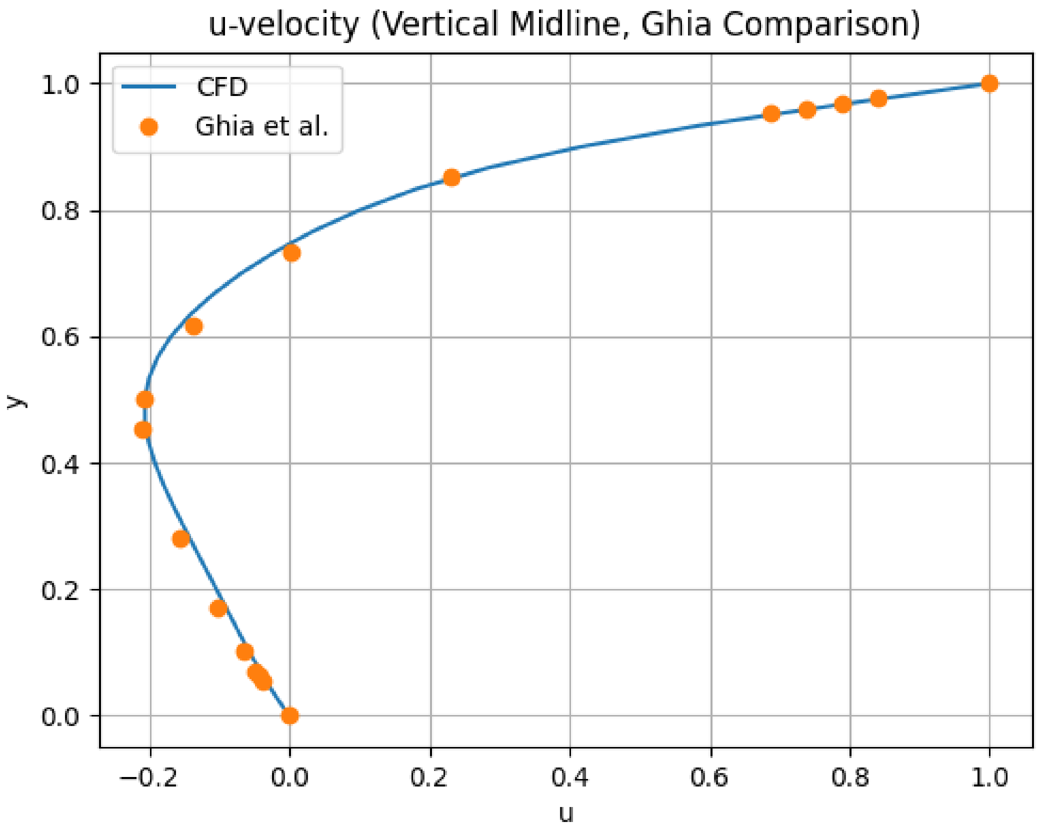

Figure 5: u-velocity along mid vertical line compared between CFD result and Ghia [1]

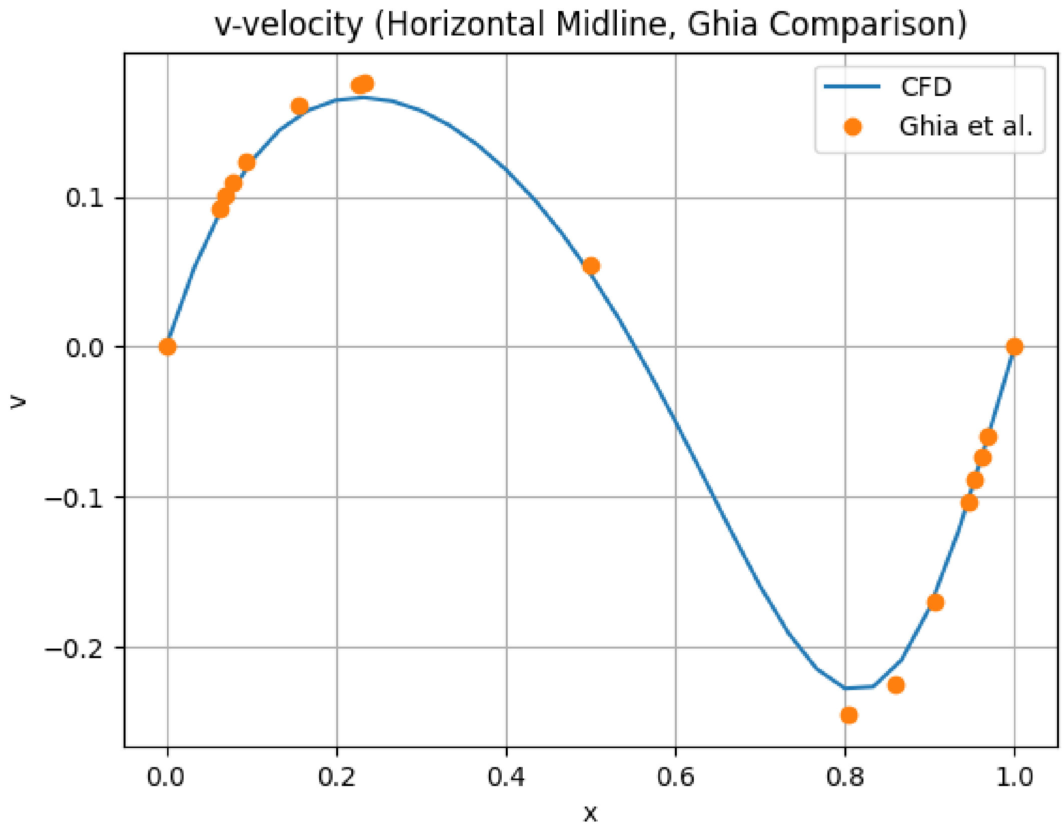

Figure 6: v-velocity along mid horizontal line compared between CFD result and Ghia [1]

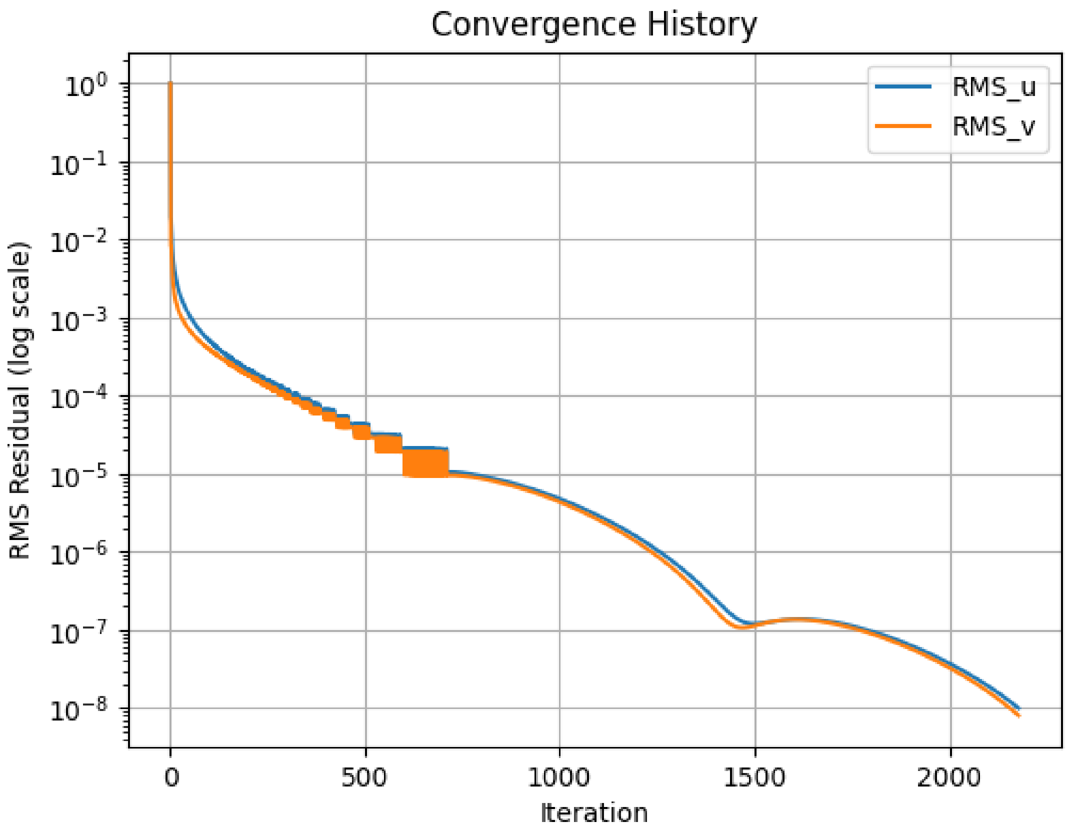

Figure 7: Convergence history for (RMS)u and (RMS)v with iterations

3. Flowchart

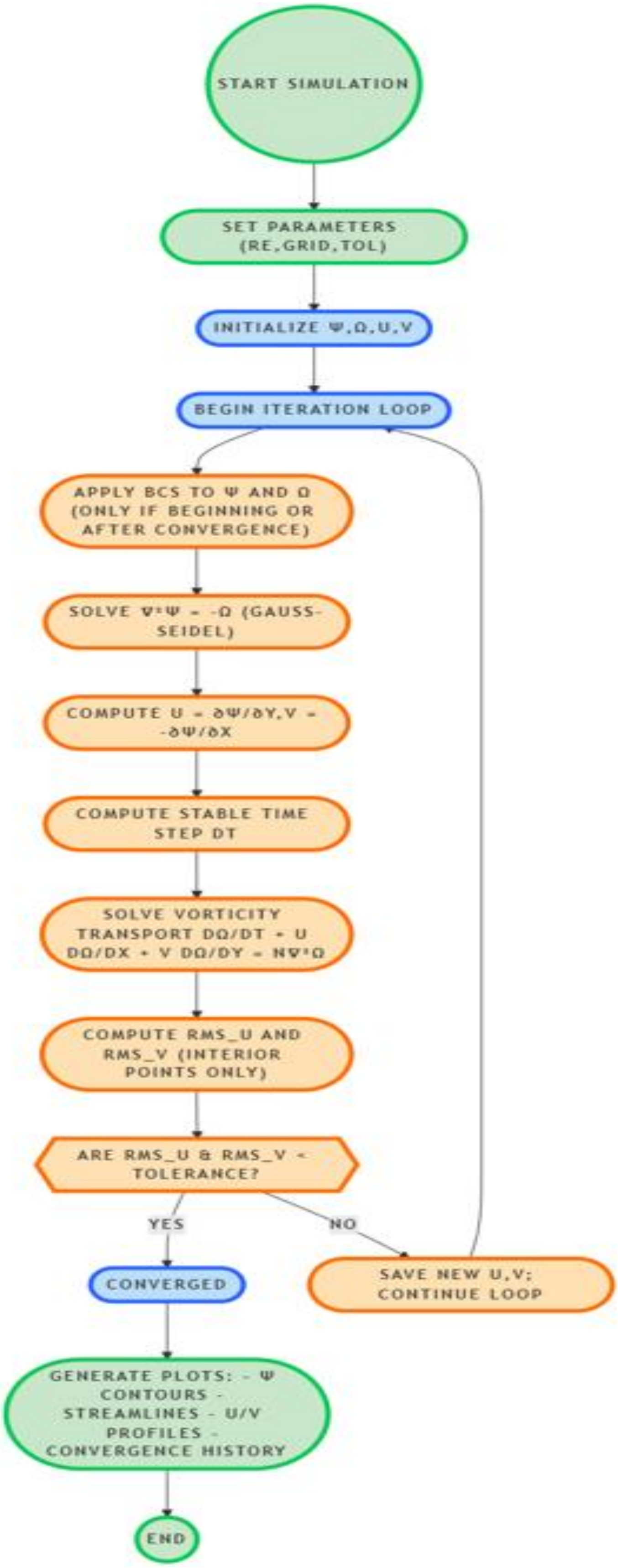

Figure 8: Flow chart

4. Conclusion

(i) All the results are listed as expected.

(ii) The velocity plots, u-velocity along mid vertical line and v-velocity along mid horizontal line matches with results of Ghia [1] as shown in figure 5 and figure 6 respectively.

5. References

[1] Ghia, U., Ghia, K. N., & Shin, C. T., High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method, Journal of Computational Physics, 48(3), 387–411.

[2] Hoffmann, K. A. and Chiang, S. T., Computational Fluid Dynamics for Engineers, Vol. I, 4th ed., Engineering Education Systems (2000).

[3] Pletcher, R. H., Tannehill, J. C., and Anderson, D. A., Computational Fluid Dynamics and Heat Transfer,3rd ed., Taylor & Francis (2011).

[4] Roache, P. J. Fundamentals of Computational Fluid Dynamics, 2nd ed., Hermosa Pub. (1998).