Lid Driven Cavity Flow Simulation using CFD & MATLAB

Jagram Kushwah[1] K. C. Arora[2] Manoj Sharma[3]
[1]M.Tech. Student [2]Dean [3]Assistant Professor
[1,2,3]Department of Mechanical Engineering
[1,2,3]NITM Gwalior

Abstract— Steady Incompressible Navier-Stokes equation on a uniform grid has been studied at various Reynolds number using CFD (Computational Fluid Dynamics). Present paper aim is to obtain the stream-function and velocity field in steady state using the finite difference formulation on momentum equations and continuity equation. Reynold number dominates the flow problem. Taylor's series expansion has been used to convert the governing equations in the algebraic form using finite difference schemes. MATLAB has been used to draw to flow simulations inside the driven-cavity.

Key words: CFD, Lid Driven Cavity Flow Simulation

I. INTRODUCTION

The lid-driven cavity flow problem has been studied my many authors. The problem has been for low Reynolds to Large Reynolds number. The lid-driven cavity flow is the motion of a fluid inside a rectangular cavity created by a constant translational velocity of one side while the other sides remain at rest. Fluid flow behaviours inside lid driven cavities have been the subject of extensive computational and experimental studies over the past years. Applications of lid driven cavities are in material processing, dynamics of lakes, metal casting and galvanizing.

This paper aims to provide a CFD simulation study of incompressible viscous laminar flow in cavity flow using MATLAB package.

A. Problem Statement

Fig. 1 shows the Schematic of cavity. Upper lid of the cavity is moving with velocity u. While other boundaries have no-slip velocity boundary conditions. Fluid flow is laminar inside the square cavity. Stream-function and vorticity approach has been used to simulate the governing equations.

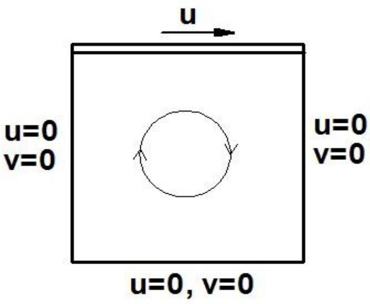


Fig. 1: Schematic of square cavity

II. NUMERICAL METHODOLOGY

As for incompressible flow pressure is difficult variable to handle because there is no direct equation available for pressure calculation, stream function-vorticity approach has been adopted to solve governing equation. Finite difference technique has been used to solve vorticity transport equation. A nonuniform collocated grid has been used in which all the flow variables are stored at the same location.

A. Governing Equations

Governing equation are those of 2D incompressible Navier - Stokes equations includes the continuity equation, x-momentum and y-momentum equation.

B. Continuity Equation

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

C. Momentum Equations

1) *X-Momentum Equation*

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+v\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)$$

2) *Y-Momentum Equation*

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+v\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)$$

3) *Stream-Vorticity Implementation*

As governing equation involves the pressure term and there is no direct equation for calculating pressure it is difficult to calculate pressure in the incompressible flow. Stream-vorticity implementation will eliminate the pressure term from governing equation by cross-differentiation of the x-momentum and y-momentum equation and makes the problem easy to construct numerical schemes.

4) *Velocity and Stream-Function Relationship*

$$u=\frac{\partial \psi}{\partial y}, v=-\frac{\partial \psi}{\partial x}$$

5) *Vorticity and Stream-Function Relationship*

$$\omega=\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y}=-\left(\frac{\partial^2 \psi}{\partial x^2}+\frac{\partial^2 \psi}{\partial y^2}\right)$$

6) *Vorticity Transport Equation*

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

The above equations can be non-dimensionalized by non-dimensional parameters listed below

$$X=\frac{x}{L},\qquad Y=\frac{y}{L},\qquad U=\frac{u}{u_\infty},$$

$$V=\frac{v}{u_\infty},\qquad \psi^*=\frac{\psi}{u_\infty L},\qquad \omega^*=\frac{\omega}{u_\infty/L}$$

7) *Vorticity and Stream-Function Relationship*

$$\omega^*=-\left(\frac{\partial^2 \psi^*}{\partial x^{*2}}+\frac{\partial^2 \psi^*}{\partial y^{*2}}\right)$$

8) *Vorticity Transport Equation*

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

9) *Grid used*

Cavity has been discretized into small elements. A uniform grid has been used. Figure 2 shows the uniform grid where element φ (i, j) represents the velocity component, stream-function, vorticity and temperature at the i^{th} and j^{th} node.

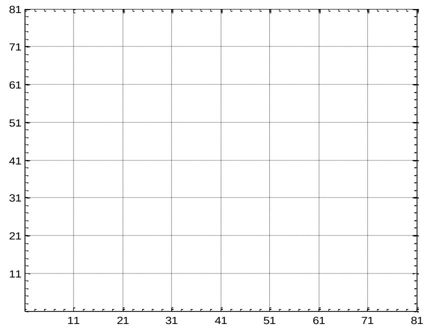


Fig. 2: Schematic diagrams of computational domain and grid layout

III. RESULTS

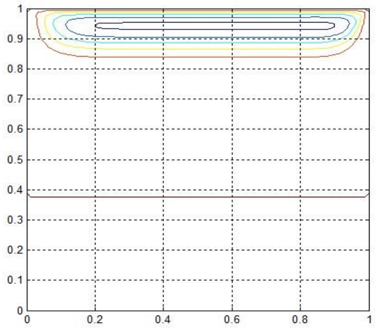


Fig. 3(a): 50

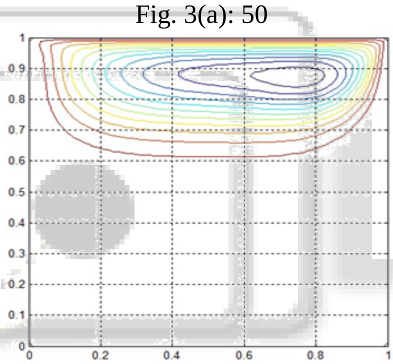


Fig. 3(b): 200

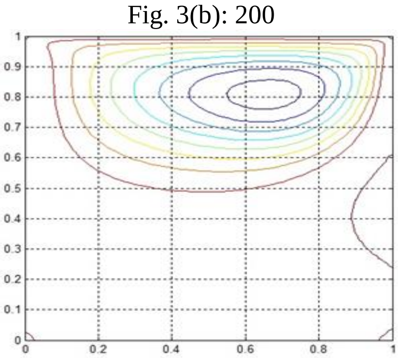


Fig. 3(c): 400

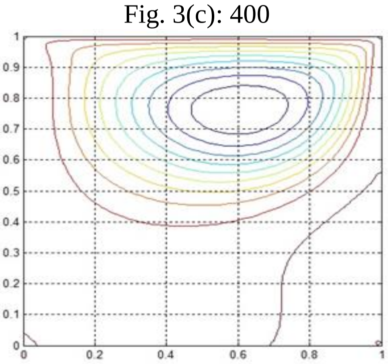


Fig. 3(d): 800

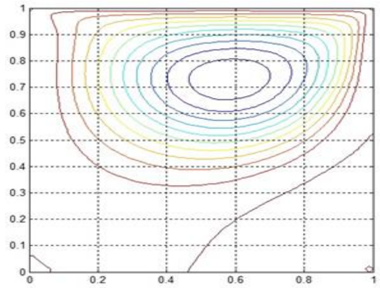


Fig. 3(e): 1000

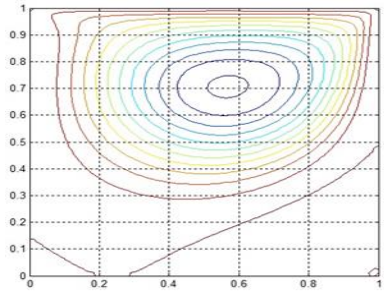


Fig. 3(f): 1200

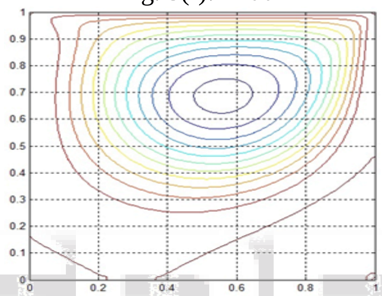


Fig. 3(g): 1400

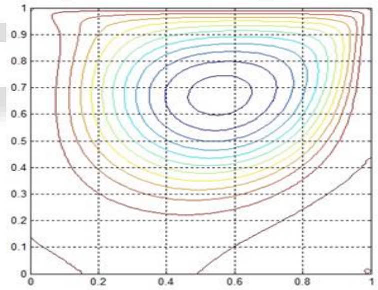


Fig. 3(h): 2000

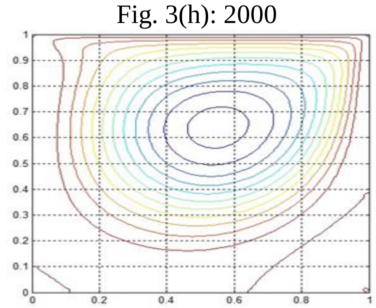


Fig. 3(i): 2500

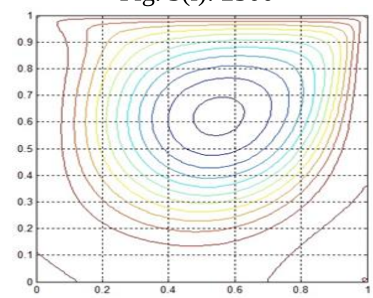


Fig. 3(j): 4000

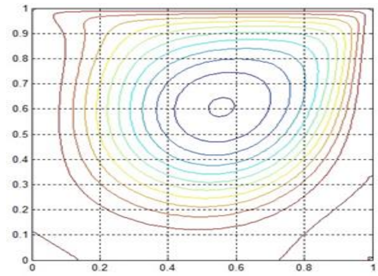


Fig. 3(k): 5000

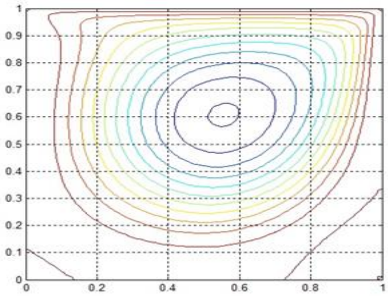
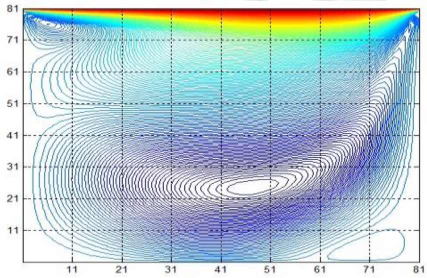


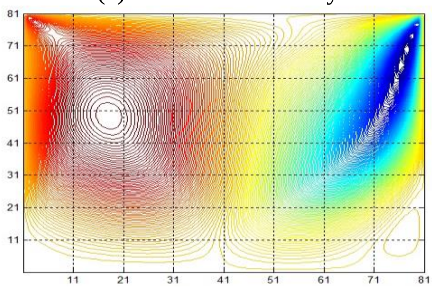
Fig. 3(l): 10000

Fig. 3 (a-l) Stream function flow contours for number of iteration

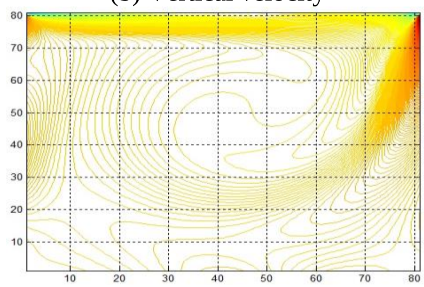
Figure 3 represents the contours of stream-function for different number of iteration. One can observe the growth of the stream-function from figure 'a' to figure 'l'. One can also notice that increasing the number of iteration after 4000 intervals does not change the growth of the stream-function. Increasing the iteration after 4000 to 5000 or 10000 only increases the time consumption of simulation.



(a) Horizontal velocity



(b) Vertical velocity



(c) Vorticity

Fig. 4: Contours of horizontal, vertical velocities and vorticity

Figure 4 represents the contours of horizontal velocity, vertical velocity and vorticity. The contours have been plotted for 400 Reynolds number. A grid size of 81×81 has been taken for developing the above contours. The code has been run for 5000 number of iteration. From the contours one can observe that the distance between the contour lines for vorticity, horizontal and vertical velocity is less near the moving side of the cavity. Red lines represent the highest velocity peaks. From the contours of horizontal and vertical velocities it can be noticed that near the bottom right corner of the cavity extra contours are forming which physically represents the counter velocity contours.

IV. CONCLUSION

– Increment in number of itration helps in getting the accurate flow inside the lid-driven cavity.
– Near the moving side distance netween the contour lines are less for strem-function, vorticity and horizontal velocity.
– Software like MATLAB can be used to simulate the Navier-stokes equation.
– Flow of the fluid is very high which actually represents the lid-driven upper wall.

REFERENCES

[1] Chen S, A large-eddy-based lattice Boltzmann model for turbulent flow simulation. App. Mathe. & Comp., 215, pp. 591–595.
[2] N. A. C. Sidik and S. M. R. Attarzadeh, An accurate numerical prediction of solid particle fluid flow in a lid-driven cavity. Intl. J. Mech., 5(3), pp. 123-128.
[3] S.L. Li, Y.C. Chen, and C.A. Lin. Multi relaxation time lattice Boltzmann simulations of deep lid driven cavity flows at different aspect ratios. Comput. & Fluids, 45(1), pp. 233-240.
[4] A.J. Chamkha, Hydromagnetic combined convection flow in a vertical lid-driven cavity with internal heat generation or absorption, Numer. Heat Transfer, Part A, vol. 41, pp. 529-546.
[5] K.M. Khanafer, A.M. Al-Amiri and I. Pop, Numerical simulation of unsteady mixed convection in a driven cavity using an externally excited sliding lid, European J. Mechanics B/Fluids, vol. 26, pp.669-687.
[6] G.K. Batchelor. On Steady Laminar Flow with Closed Streamlines at Large Reynolds Numbers. J. Fluid Mech., 1, pp. 177–190.

