CFD Simulation of Lid Driven Cavity Flow

Jagram Kushwah[1] K. C. Arora[2] Manoj Sharma[3]
[1]M.Tech Student [2]Dean Academic and Head ME [3]Assistant Professor
[1,2,3]NITM Gwalior

Abstract— This project aims at simulating lid driven cavity flow problem using package MATLAB. Steady Incompressible Navier-Stokes equation with continuity equation will be studied at various Reynolds number. The main aim is to obtain the velocity field in steady state using the finite difference formulation on momentum equations and continuity equation. Reynold number is the pertinent parameter of the present study. Taylor's series expansion has been used to convert the governing equations in the algebraic form using finite difference schemes.
Key words: CFD, Navier-Stroke, Stream-Vorticity Implementation

I. INTRODUCTION

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.

The macroscopic variables of interest in conventional numerical methods, such as velocity and pressure are usually obtained by solving the Navier-Stokes equation. Such numerical methods for two dimensional steady incompressible Navier-Stokes equations are often tested for code validation. The one sided lid-driven square cavity flow has been used as a benchmark problem for many numerical methods due to its simple geometry and complicated flow behaviors. It is usually very difficult to capture the flow phenomena near the singular points at the corners of the cavity (Chen, 2009 and 2011. Migeon et al. (2003)

(N. A. C. Sidik and S. M. R. Attarzadeh, 2011) considered the cubic interpolated pseudo particle method and validate their results with the shear driven flow in shallow cavities. (Li et al., 2011) applied the new version of multiple relaxation time lattices Boltzmann method to investigate the fluid flow in deep cavity.

There have been some works devoted to the issue of heat transfer in the shear driven cavity. (Manca et al., 2003) presented a numerical analysis of laminar mixed convection in an open cavity with a heated wall bounded by a horizontally insulated plate. Results were reported for Reynolds numbers from 100 to 1000 and aspect ratio in the ranges from 0.1 to 1.5. They presented that the maximum decrease in temperature was occurred at higher Reynolds. The effect of the ratio of channel height to the cavity height was found to be played a significant role on streamlines and isotherm patterns for different heating configurations. The investigation also indicates that opposing forced flow configuration has the highest thermal performance, in terms of both maximum temperature and average Nusselt number.

(Gau and Sharif, 2004) reported mixed convection in rectangular cavities at various aspect ratios with moving isothermal side walls and constant flux heat source on the bottom wall. Numerical simulation of unsteady mixed convection in a driven cavity using an externally excited sliding lid is conducted by (Khanafer et al. 2007). They observed that, Re and Gr would either enhance or retard the energy transport process and drag force behavior depending on the conduct of the velocity cycle.

(Ch.-H. Bruneau and M. Saad, 2006, Batchelor, (1956), have pointed out that, driven cavity flows exhibit almost all the phenomena that can possibly occur in incompressible flows: eddies, secondary flows, complex three-dimensional patterns, chaotic particle motions, instability, and turbulence. Thus, these broad spectra of features make the cavity flows overwhelmingly attractive for examining the computational schemes.

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 square cavity. From fig. 1 one can notice that the upper lid of the cavity is moving with velocity u. While other boundaries have no-slip tangential and zero normal velocity boundary condition. Fluid flow is laminar inside the two-dimensional square cavity. As there is no direct equation available for pressure calculation for incompressible flow stream-vorticity approach has been used to solve the governing equations
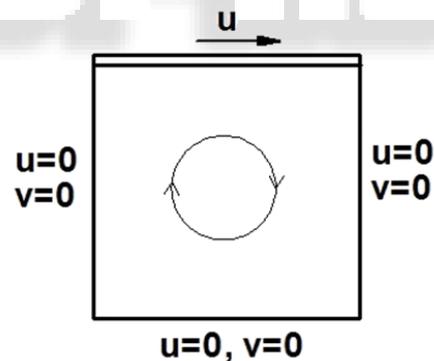

Fig. 1: Schematic of square cavity

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

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

2) Y-Momentum Equation

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

D. 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.

1) Velocity And Stream-Function Relationship

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

2) Vorticity And Stream-Function Relationship

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

E. Vorticity Transport Equation

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

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

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

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

F. Vorticity and Stream-Function Relationship

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

G. Vorticity Transport equation

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

H. Taylor's Series Expansion

The difference approximation can be done using Taylor-series expansion or Taylor's formula. Taylor-series expansion for T(x+Δx, y) about (x, y) can be represented as

$$T(x + \Delta x, y) = T(x, y) + \frac{\partial T(x,y)}{\partial x}\frac{\Delta x}{1!} + \frac{\partial^2 T(x,y)}{\partial x^2}\frac{(\Delta x)^2}{2!} + O(\Delta x)^3$$

From equation 4.1 we can find the algebraic form of first order partial differential term for forward scheme by rearranging the term as,

$$\frac{\partial T}{\partial x} = \frac{T(x + \Delta x, y) - T(x, y)}{\Delta x} - \frac{\partial^2 T(x,y)}{\partial x^2}\frac{(\Delta x)^2}{2!}$$

Changing to i, j notation, we can write the above equation as,

$$\frac{\partial T}{\partial x} = \frac{T_{i+1,j} - T_{i,j}}{\Delta x} + T.E.$$

Where truncation error (T.E.) is the difference between the partial derivative and its finite-difference representation. We can also write the above eq. as,

$$\frac{\partial T}{\partial x} = \frac{T_{i+1,j} - T_{i,j}}{\Delta x} + O(\Delta x)$$

Where $O(\Delta x)$ represents the order of term Δx and higher, which has been neglected during finding the approximation.

Similarly,

$$\frac{\partial T}{\partial x} = \frac{T_{i,j} - T_{i-1,j}}{\Delta x} + O(\Delta x)$$

$$\frac{\partial T}{\partial x} = \frac{T_{i+1,j} - T_{i-1,j}}{\Delta x} + O(\Delta x)^2$$

$$\frac{\partial^2 T}{\partial x^2} = \frac{T_{i,j} - 2T_{i-1,j} + T_{i+2,j}}{(\Delta x)^2} + O(\Delta x)^2$$

III. GRID GENERATION

Physical domain has been discretized into small rectangular four nodded elements. A non-uniform collocated grid has been used for better accuracy at the walls of the enclosure. A collocated grid is what in which all the field variables (vectors as well as scalars) are defined at the same point of a cell. Figure 2 shows the non-uniform collocated grid where element φ (i, j) represents the velocity component, stream-function, vorticity and temperature at the ith and jth node.
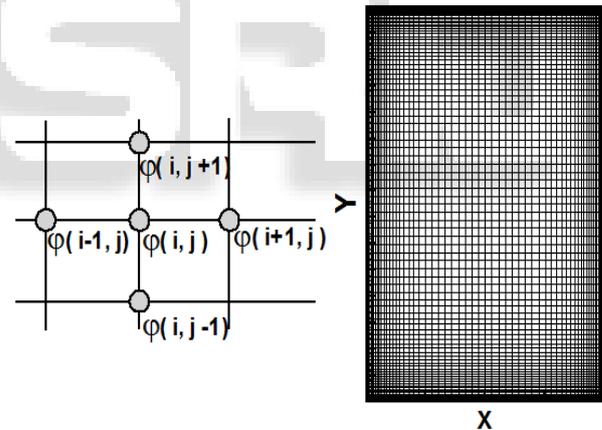

Fig. 2: Schematic diagrams of computational domain and grid layout

IV. RESULTS

Figure 3 and 4 represents the contours of stream lines, vertical velocity and horizontal velocity for different grid size, stream lines actually represents the flow of the fluid inside the cavity. From the contours one can notice that difference between the stream lines near the upper lid is very small which means that flow of the fluid is very high which actually represents the lid-driven upper wall.

Grid=81*81
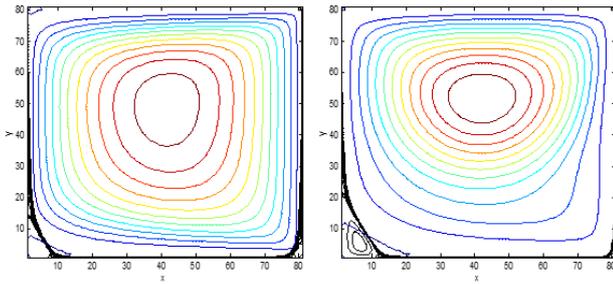
Re=100

Grid=81*81

Re=400

Fig. 3: Contour plot of stream function for characteristics

Vertical velocity
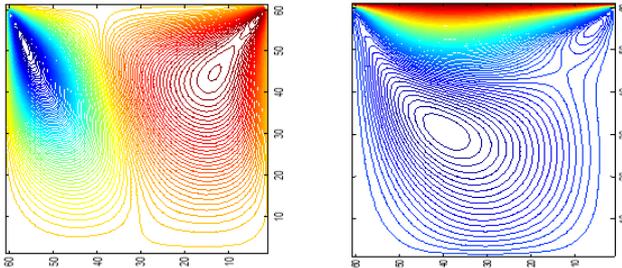
Horizontal velocity

Fig. 4: Contour plot of vertical and horizontal velocities for characteristics

V. CONCLUSION

- Taylor series helps in converting the partial equations into algebric form.
- Software like MATLAB can be used to simulate the Navier-stokes equation.
- Contours of strem function and velocities have been presented.
- Increment in grid size affects the solution of the problem.
- From the contours one can notice that difference between the stream lines near the upper lid is very small.
- 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.