ICMIEE-PI-140123

Numerical Investigation of fluid flow Over a Lid-driven Square Cavity

Mohammad A.Hossain [1], Mominul Huq [2]

[1] Department of Mechanical Engineering, The University of Texas at El Paso, El Paso, TX, USA
[2] Department of Mechanical and Production Engineering, Ahsanullah University of Science and Technology, Dhaka, BANGLADESH

ABSTRACT
This work is focused on numeric investigation of fluid flow over a lid-driven cavity. A square cavity is chosen to do the simulation. The simulation is done for both laminar, transient and turbulent flow for both 2D and 3D case. Reynolds number is considered between 1 < Re < 7500. First the mesh independence test is done by different mesh size. Five different quadratic mesh configuration is used for mesh independence test. Finally a 200*200 cell elements are used for the final simulation. Commercial software ANSYS Fluent is used for the simulation. A pressured based solver is used with QUICK solution schema to find the horizontal velocity components at different Reynolds number. The results are compared with available published data. There are significant agreement with the experimental data. Streamlines at different Reynolds number, position of the primary and secondary vortex are compared with the experimental data and presented.

Keywords: lid driven cavity, vortex, CFD

1. Introduction

Flow through a lid driven cavity is one of the classic fluid mechanics problem used for validation and verification of fluid flow models. Due to the simplicity of the geometry and complex flow field, this problem has been used for verification of 2D Navier Stokes equation as it is very difficult to capture the flow phenomena near the singular points near the corner of the cavity [1-2]. The cavity analysis is used for material process, metal casting, designing journal bearing and many more.

Many researcher have done significant amount of work on cavity flow. Ghia el al [3] have done the detail analysis of cavity flow. N.A.C Sidik et al [4] considered the cubic interpolated pseudo particle method validate their results with shear driven flow in shallow cavity. There have been some works devoted to the issue of heat transfer in the shear driven cavity. Manca et al [5] 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. Numerical simulation of unsteady mixed convection in a driven cavity using an externally excited sliding lid is conducted by Khanafer et al [6]. 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 [7] 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.

2. Objective

The purpose of the study is to investigate the flow behavior through the lid-driven cavity and to calculate flow separation and recirculation zone at different Re. In order to validate the result, the simulated data would have to compare with the experimental data [1].

3. Theory

The simulation is done by solving 2D Navier-Stoke equation for constant density flow. The continuity and momentum equation for a steady state 2D constant density flow are given as follows-
Continuity Equation,

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

For Constant density,

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

* Mominul Huq. Tel.: +88-01673729672
E-mail addresses: mhpurab@gmail.com

Momentum Equation,

X-Momentum:

$$[illegible]$$

Y-Momentum:

$$[illegible]$$

Vorticity:

$$[illegible]$$

Here [illegible] are the horizontal and vertical velocity component of the flow. The boundary conditions are assumed as-

[illegible]

4. Numerical modeling

4.1 Geometry

The geometry is developed based on the cavity size. A square cavity of height L is assumed during the development of geometry. Other parameters are assumed as a function of L. Figure 1 shows the schematic of the geometry of a square cavity. The depth of the cavity L is assumed as 100mm.

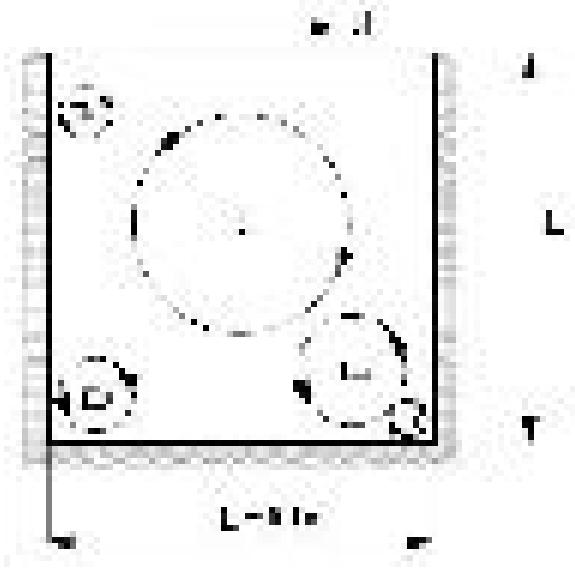


Figure 1. Schematic of the square cavity

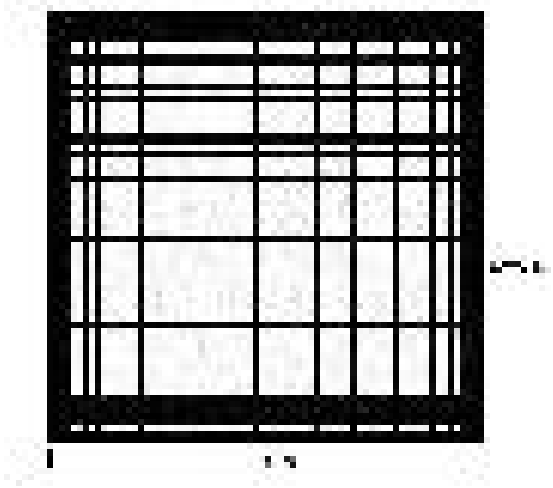


Figure 2. Mesh generation of the model

4.2 Mesh Generation

Around 40,000 structured quadrilateral mesh is used within the entire domain having 200 * 200 cells. The mesh elements were clustered near the wall boundary (figure 2) in order to resolve the boundary layer. Maximum face size of the mesh is assumed as 0.5mm. The orthogonal quality of the mesh remains 0.98 and maximum aspect ratio is 1.8. A mesh independence test was done based on u-velocity at x = 0.5L at Re = 400. Different mesh configurations were used to check mesh independence.

Figure 3 shows the mesh dependency on u-velocity. 3 different cases are checked for this, keeping Re = 400. From the table it's clearly shown that the problem become mesh independent at mesh size of 100 * 100. To find more accurate result the final mesh is done with 200 * 200 cells.

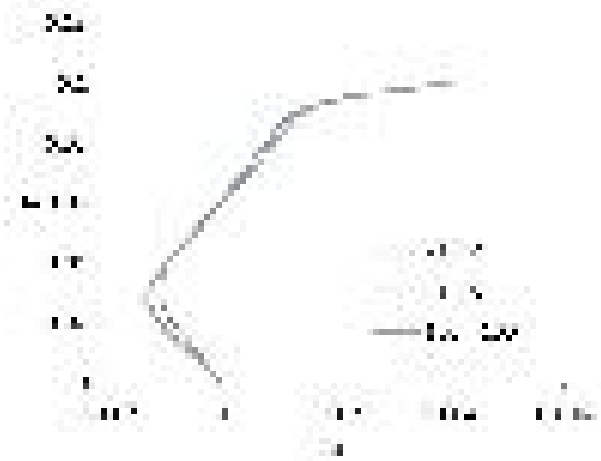


Figure 3. Mesh independence test for different cell number

5. Case setup and Solution

A 2D planner pressure based solver is used for the simulation. Water is considered as a working fluid. Inlet

boundary condition is determined based on Re. For Re calculation the characteristic length is assumed as L, where L is the depth.

Standard least square discretization technique is used. A second order upwind solution method with QUICK solver is used. Re is set from 1 to 5000 as inlet condition. For convergence, the residue for continuity and velocity is set as 1e-10. Each solution is allowed to run 10,000 iteration to meet the minimum convergence criteria. To validate the CFD model, u-velocity is at x=0.5L and Re = 400 is compared with the published data. Figure 4 shows the comparison of CFD results which have a good agreement with experimental data.

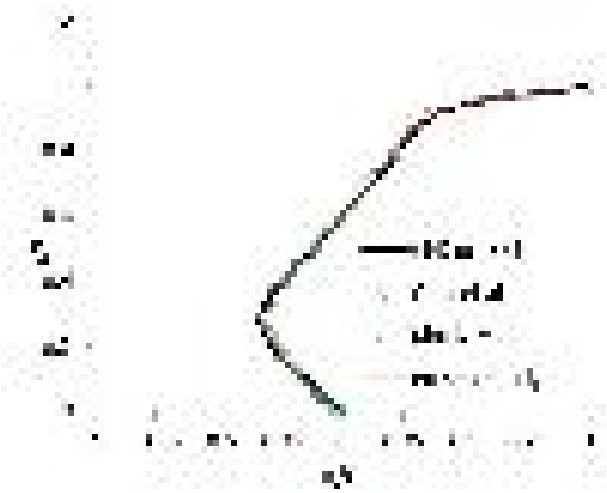


Figure 4. Validation of result at x =0.5L and Re=400

6. Results and Discussion

Horizontal velocity have been calculated for different Re ranging from 100 to 5000 in order to observe the flow behavior in Laminar, transition and turbulent regimes. The primary vortex form initially and as Re increases the vortex change its position. Figure 5 shows a typical streamline to demonstrate the formation of primary, secondary and tertiary vortex. Table 1 shows the center position of the primary vortex at different Re. It also shows the published data. The results are significantly close enough. Figure 6 shows the x-y coordinates of the primary vortex. It also shows the deviation from the published result. It is observed that, the center of the primary vortex become close to the center of the cavity as Re increases.

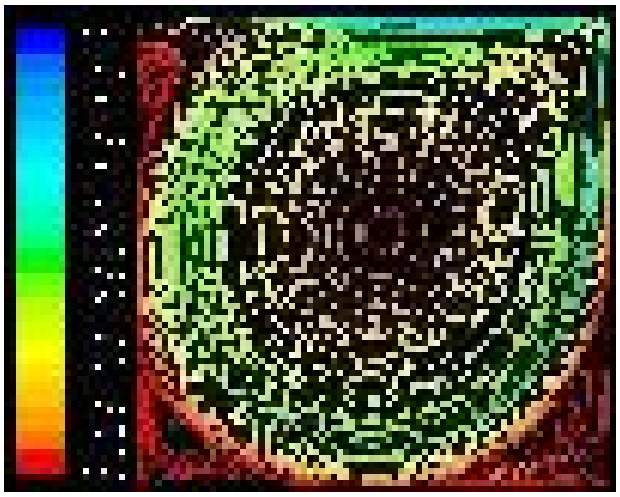


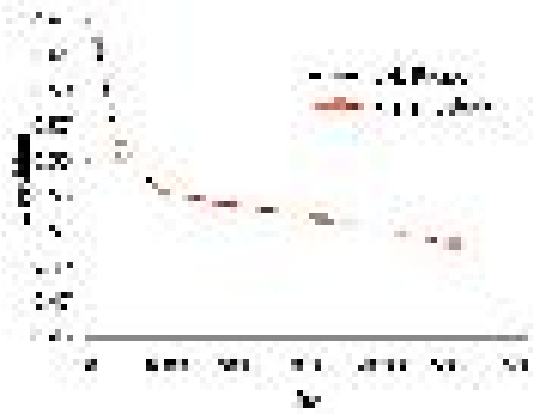
Figure 5. Different vortex formation at Re = 5000

Table 1. Position of the center of the primary vortex

Re	100	400	1000	3200	5000
Present	(0.619,0.742) 21 * 21	(0.560, 0.607) 31 * 31	(0.532, 0.564) 41 * 41	(0.518, 0.540) 61 * 61	(0.502, 0.514) 61 * 61
Ghia 82	(0.617, 0.734) 129 * 129	(0.555, 0.606) 257 * 257	(0.537, x) 257 * 257	(x , 0.547) 257 * 257	(0.512, 0.535) 257 * 257
CFD Project	(0.614, 0.734) 250 * 250	(0.553, 0.606) 250 * 250	(0.532, 0.562) 250 * 250	(0.540, 0.518) 250 * 250	(0.535, 0.502) 250 * 250

Figure 6. Coordinate of primary vortex at different Re

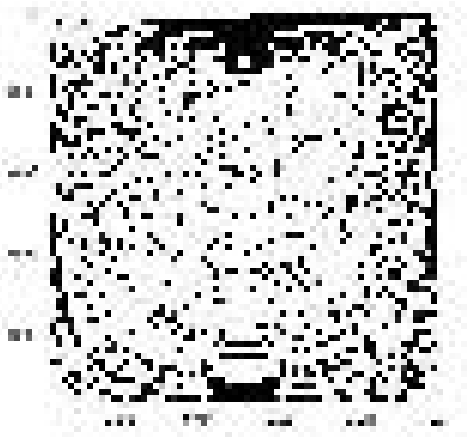
Figure 7(a) and 7(b) also show the change of position of the primary vortex in X and Y coordinate. The results are also compared with the published data and the deviation is significantly low. Figure 8(a) - 8(g) shows the streamlines at different Re. The primary and the secondary vortices are clearly visible there. The simulation successfully resolved the tertiary vortex at the corner of the cavity at Re = 5000 showed in figure 8(g).



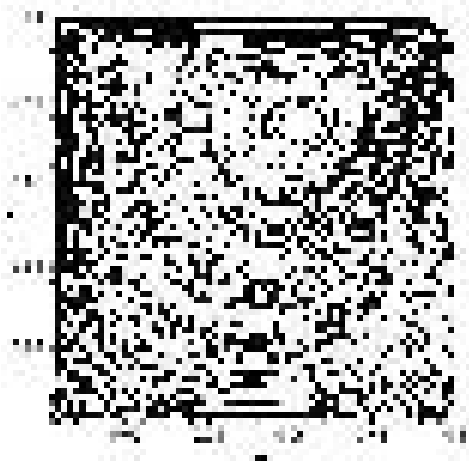
7(a)

7(b)

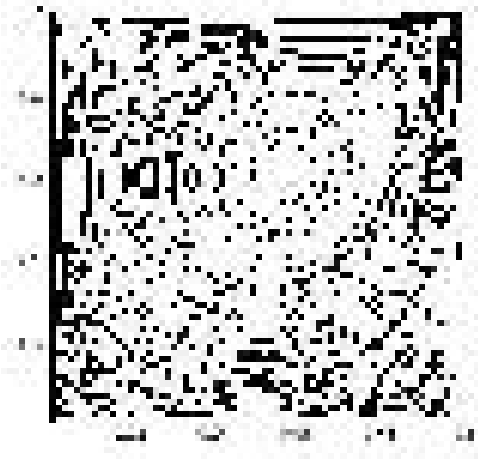
Figure 7. Position of the primary vortex at different Re. (a) X-coordinate, (b) Y - coordinate



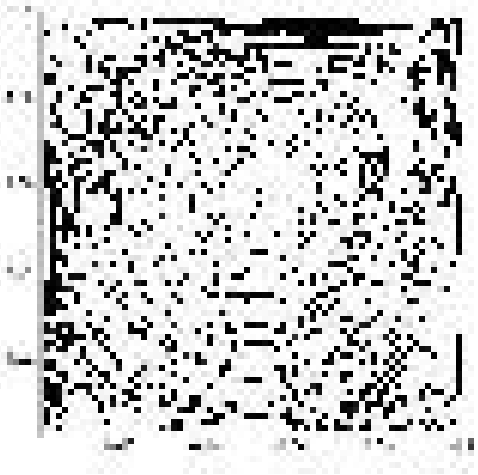
8(a) Re = 1



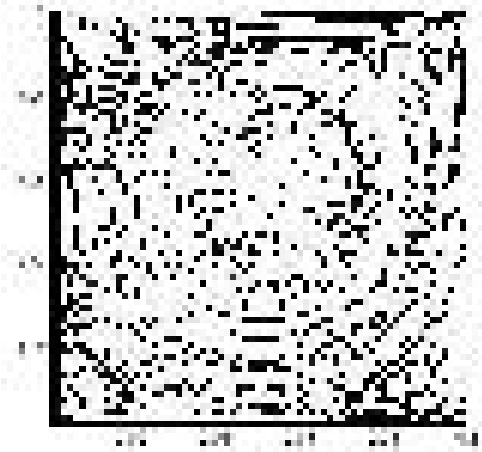
8(b) Re = 100



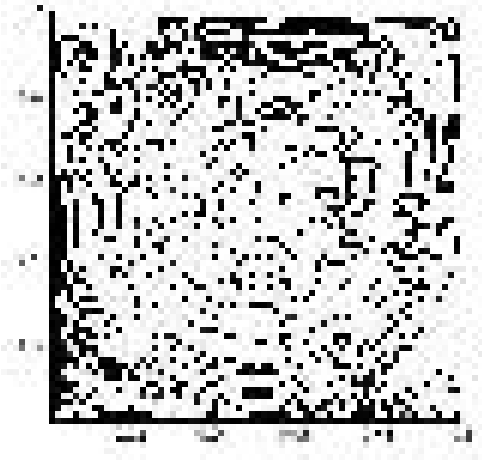
8(c) R = 400



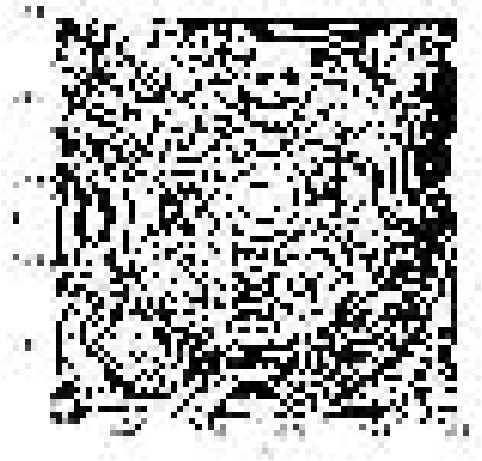
8(d) Re = 1000



8(e) Re = 2000



8(f) Re = 3200



8(f) Re = 5000

A 3D flow simulation is also done for the cavity. Figure 9 shows the velocity contour at the center of the cavity. Figure 10 shows the volumetric streamline for the cavity. The streamline is showed with particle form.

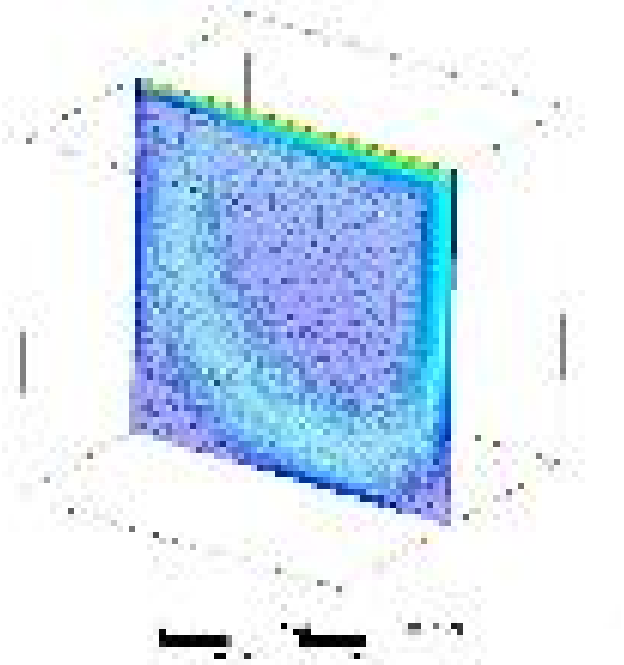


Figure 9. Velocity contour at the center plane of the cavity at Re = 400

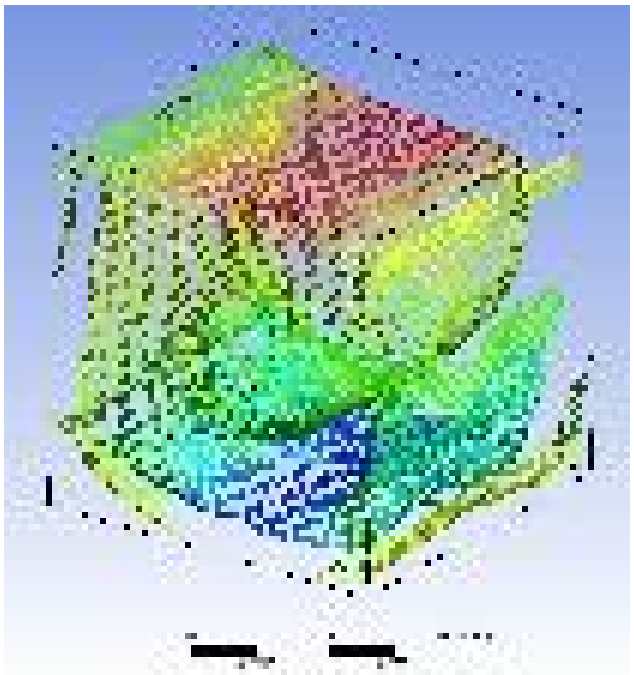


Figure 10. Streamline in particle form at Re = 400

7. Conclusion

The flow through a square cavity is simulated for both 2D and 3D case for different Re and the position of the vortices are presented. Mesh dependency is done for different cell number. The results are compared with the published data and it shows good agreement with published work.

NOMENCLATURE

[illegible]

REFERENCE

[1] Chen S, A large-eddy-based lattice Boltzmann Model for turbulent flow simulation. Applied Mathematics and Computation, 215, 591–595.

[2] 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, 2011, 45(1): 233-240

[3] U Ghai et al, High-Re solution for incompressible flow using the Navier stokes equation and using a multi grid method, J of computational physics, 48, 387-411(1982)

[4] 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. 2011, 5(3): 123-128.

[5] O. Manca, S. Nardini, K. Khanafer and K. Vafai, Effect of Heated Wall Position on Mixed Convection in a Channel with an Open Cavity, *Numer. Heat Transfer, Part A*, vol. 43, pp.259–282, 2003

[6] 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, 2007.

[7] Ch.-H. Bruneau, M. Saad, The 2D lid-driven cavity problem revisited, Computers & Fluids35 (3), 2006.