Studies on Vortex Structures formed inside a Lid Driven Cavity – Influence of Extending the Length of the Cavity

D J Ranjith1, K Sreenath2 and S Ajith Kumar3
123Department of Mechanical Engineering, Amrita Vishwa Vidyapeetham, Amritapuri, India

E-mail : ajithkumars@am.amrita.edu

Abstract. Finite volume methods based on pressure-velocity coupling in incompressible flow has been evolved as the basic guideline and is successfully implemented in numerous industrial and aerospace fields. In our studies, we use this class of techniques for analysing flow inside the lid driven cavity. Our simulation analysis is based on the finite-volume method using non-uniform staggered mesh. An issue of storing and coupling pressure-velocity in finite volume method is solved by using SIMPLE algorithm are reviewed. Current works aim to study the formation of vortex for varied Reynolds numbers based on numerical simulations and its influence over mid-plane velocities as extending the length of the cavity. The simulation results are represented in terms of contours of streamlines and velocity plot.

1. Introduction

Lid driven cavity flow is achieved by the translation motion of a moving wall which induces flow field inside a box. The lid driven cavity is a bench-mark problem for numerical solution and computational fluid dynamics because of its geometry and simplicity. Still flow inside a lid driven cavity holds its richness due to the formation of multiple primary, secondary vortices along with corner vortices on the influence of aspect ratio and Reynolds number. Flow inside a lid driven cavity is governed by the aspect ratios and Reynolds number for fluids under a Newtonian category.

Early work of analytical and numerical studies carried out by Burggraf [1] uses FMG-FAS algorithm to solve the 2D N-S equation on a staggered grid which exhibit well representation of corner vortices in compare with earlier studies. Lid driven cavity analysis has been studied by many researchers in order to validate the Navier-Stoke equations. Shankar and Deshpande[2] work considered to be a very promising one as both experimental and computational analysis are carried for different Reynolds number. The industrial application such as high grade papers and photographic films are manufactured using short dwell coaters and flexible coaters, microcrystalline materials by the melt-spinning process which involves well understanding flow through these cavities. Instabilities exist in the flow will affect the quality of the product. Study of flow structure inside a lid driven found to be most useful as it exhibits formation of the vortex, 3-D flow pattern, instabilities, and transition from laminar to turbulence as varying Reynolds number. Further literature survey suggests that comparison of experimental, theoretical and computational results of this class of flows can be made in detail and also platform for new computational schemes in comparisons with the other class of flows. In late 1950 the seminal work of Harlow et al.[3,4], introducing pressure-velocity coupling in
steady flows by Patankar and Spalding [5], the computational work by Jameson et al. [6], are emerged as a benchmark work for the development of CFD.

Studies of Ghia et al. [7] is considered to be the platform for CFD simulation by considering uniform mesh for Re=1000 which leads to the additional formation of corner vortices in the flow structure. Based on these many kinds of researches were carried for various Reynolds number. Flow analysis for higher Reynolds number till Re=65000 is performed by Yapici et al. [8]. They have used QUICK technique to discretize the spatial derivatives of N-S equation by using 768x768 grid size. Wiggle formation and convergence problem at higher Reynolds number been resolved by adapting above computational methodology. Sahin and Rober [9] used implicit cell-vertex based finite volume method which eliminated pressure term from Navier–Stokes equations at high Reynolds numbers. Thus they represented N-S equation in terms of velocity components thus eliminates difficulties exist in velocity and pressure coupling technique. They have used this technique both for 2-D steady and unsteady flow up to Reynolds number 10000.

Sundaresan et al.[10] solutions were based on clustered grids and multi-grid code. This result has shown that in order to get the clear visualisation of corner vortices it is necessary to have proper distribution of meshes at boundary walls. Sun et al.[11] used Inner Doubly Iterative Efficient Algorithm for Linked Equations (IDEAL) algorithm which result in great converging rate and stability in the iteration process. IDEAL algorithm found to be robust in computation. A new algorithm has been compared with existing codes based on SIMPLE and SIMPLER. Peric et al.[12] research presents comparisons of staggered and collocated numerical grids in terms of convergence rate, under relaxation parameter dependent, computational time and accuracy. Ramesh [13] article illustrates SIMPLE algorithm to solve inclined side wall cavity on the staggered mesh. This article represents case studies carried for different angles for Reynolds number 100 and 1000. Uniform staggered grid of size 81x81 and 101x101 based on SIMPLE algorithm has been used. Mehdi and Ashrafizadeh[14]article consider co-located grids by finite volume method based on proper closure equations for solving incompressible flow. The results indicate that at higher Reynolds number does not require severe under relaxation parameter. Ming and Tang [15] and Erturk et al.[16] articles expressed N-S equations in stream function- vorticity form for higher Reynolds number( <7500 and <21000 respectively) where quaternary vortex and a new tertiary vortex are observed and also convergence rate found to be faster. Chen and Przekwas [17] paper propose a pressure and velocity using pressure-correction coupled solution approach for both structured and unstructured meshes. The same method is used for incompressible and subsonic compressible flow. They have carried 1D,2D and 3D simulation case studies. Results were found to be excellent and improved while testing on various CFD benchmark problems.

In our current work, numerical simulations are analysed for 5 different cases by varying Reynolds numbers and aspect ratios (AR) using non uniform mesh. Aspect ratio is defined as ratios of length to height of the lid-cavity. Case 1(with Re=1000 and AR=1) is considered as a standard case for validation. In cases 2, 3, 4 and 5 Reynolds number is varied (Re=100,200,300,500 and 1000) keeping the aspect ratio different (AR=1.2 and 3). As the length of the cavity (i.e. aspect ratio) increases the primary corner vortex grows in size and fluid motion near the moving wall is unaffected. Primary vortex shifts towards the centre of the cavity with an increase in Reynolds number at fixed aspect ratio.

2. Methodology

2.1 Governing equation

Flow inside a lid driven cavity is considered in 2-D cartesian coordinate system by assuming steady, incompressible and isothermal flow. Velocity components are represented as $u$ and $v$ in stream-wise $(x)$ and span-wise $(y)$ respectively. Non-dimensionless forms of continuity and momentum equations are as follows:
\begin{equation}
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0
\end{equation}

\begin{equation}
\frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{1}{\text{Re}} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)
\end{equation}

\begin{equation}
\frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \frac{1}{\text{Re}} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)
\end{equation}

2.2 Boundary conditions
Applying boundary conditions for stationary and moving wall.

Figure 1. Boundary condition applied at moving and stationary walls of the lid driven cavity.

2.3 Numerical method
2.3.1 Discretization. The finite volume approach is used to numerically discretize the governing equations on a staggered grid. The domain of solution is divided into a number of small control volumes of a non-uniform structured grid in a cartesian coordinate as shown in figure 2 for aspect ratios 1, 2 and 3. An equation of continuity and components of momentum equations are discretized into approximated linear algebraic equations and each small control volumes consist of values of unknown \( u \) and \( v \) at the faces of control volume which represents the surface flux integrals and \( p \) at the centre of the control volume.

Figure 2. Grid Structures for Aspect Ratios: (a) 1 (b) 2 (c) 3

The general transport equation is used in the finite-volume method where each component of velocity and momentum equations in transport equations is replaced by dependent variable \( u \) and \( v \). Here we use staggered mesh to store values of pressure at the centre of scalar control volume and
velocity components at cell faces. N-S equation is basically convection-diffusion equation which can be solved using the following
\[ \text{div}(\rho \mathbf{u}) = \text{div}(\mu \nabla \mathbf{u}) - \nabla p \] (11)

Discretized momentum equations are as follows:
\[ a_{i,j} u_{i,j}^* = \sum a_{nb} u_{nb} + (p_{i-1,j} - p_{i,j}) A_{i,j} + b_{i,j}, \] (12)
\[ a_{i,j} v_{i,j}^* = \sum a_{nb} v_{nb} + (p_{i,j-1} - p_{i,j}) A_{i,j} + b_{i,j}. \] (13)

Values of coefficient of \( a_{i,j} \) and \( a_{nb} \) solved using upwind scheme.

2.3.2 Algorithm. The incompressible continuity and momentum equations are discretized using finite volume method which results in set of linear algebraic equations. Since the flow is incompressible and isothermal, the pressure variable cannot be determined through continuity and state equation. Momentum equations cannot be used as a solution variable as such both velocity and pressure are dependent variable in this equation. This problem is resolved by SIMPLE algorithm (Semi-Implicit Method for Pressure Linked Equations) technique which is based on pressure-velocity coupling and this method extensively used for solving incompressible flows. The sequential calculations are required as momentum equations are coupled with other scalars and method is iterative.

In our study case, iteration will continue until residuals fall below 1e-6. The accuracy of results obtained is compared with well benchmark literature and detailed comparison of the stream-wise and span-wise velocity profiles along the symmetry of cavity for different cases has been noted.

2.4 Validation

In this report, numerical analysis of steady flow driven cavity for Reynolds number (1000) is analysed by using non-uniform structured mesh by considering 129x129. Results are plotted in terms of velocity profiles (\( u \) and \( v \)) along a vertical and horizontal centre of the cavity respectively as shown in figures (a) and (b). Both figures show that the results obtained are in good agreement with the well-established literature results of Ghia et al. [7] and Erturk et al. [16]. In our analysis, we used the staggered non-uniform mesh of 129x129 by implicit Green Gauss Cell Based (GGCB) using SIMPLE algorithm of second order accuracy with under-relaxation until iteration continues till it converges below 1e-6.

**Figure 3.** Variation of velocities along the centre of the cavity: (a) \( u \) (b) \( v \)
2.5 Grid independence

Grid independence of our problem statement is achieved by the transition from coarse mesh to finest mesh for steady flow driven cavity for Re=300 using different mesh size as tabulated in table 1 and figure 4. As fine mesh used computation and convergence time taken will be more. Figure 3 illustrates, for location (y=4), increase in mesh size (after 121x121) moreover velocity remains parallel to the horizontal axis. Mesh size error is below (1e-5) hence we constrained numerical analysis to mesh 121x121 of staggered non uniform for different aspect ratio under criteria of convergence 1e-6.

![Figure 4. u velocity for different values of mesh size at y=4 along centre of mesh cavity at Re=300](image)

| Mesh size | No of iterations | Mesh size | No of iterations | Mesh size | No of iterations |
|-----------|-----------------|-----------|-----------------|-----------|-----------------|
| 11x11     | 130             | 61x61     | 1221            | 121x121   | 3776            |
| 21x21     | 251             | 71x71     | 1562            | 129x129   | 4166            |
| 31x31     | 434             | 81x81     | 1931            | 141x141   | 4858            |
| 41x41     | 657             | 91x91     | 2348            |           |                 |
| 51x51     | 919             | 101x101   | 2762            |           |                 |

Table 1. Number of iteration for different mesh size.

3. Results and Discussion

A study is carried out for different aspect ratios and Reynolds number. It is found that streamlines are highly dependent on Reynolds number and aspect ratios (AR). Flow characteristics of our solutions show main features such as the main vortex, secondary vortices and tertiary vortices in the lower corners for higher aspect ratio. Figure 5 to figure 10 depicts streamlines for Re=100, 200, 300, 500, 1000 and 1500 for different Aspect Ratios (AR) 1, 2 and 3.

From figure 5 to figure 9, for case studies (a) i.e. aspect ratio=1 (flow is towards the right side) as Reynolds number increased, bottom right corner vortex grows in size and influences the flow field beside primary vortex move towards the centre of cavity. It can be seen from figure 5 to figure 9, for case studies (b) i.e. aspect ratio=2, the transformation from bottom left corner vortex into the main vortex. These corner vortices are strongly dependent on Reynolds number. Figure 10, for a case study (a) i.e. aspect ratio=2 at Reynolds number =1500, shows the formation of two primary vortex along with three secondary vortices which may be formed due to flow detachment. These vortices formed...
are opposite in rotating direction. Above detailed analysis shows that as Reynolds number increased the centre of corner vortex shift towards centre of cavity with enlargement in size. Figure 5 to figure 9, for case studies (b) i.e. aspect ratio=3, differs that from a case study of aspect ratio=2, where the centre of corner vortex move towards a boundary of the cavity towards left with the influence of Reynolds number. From this, it is clear that the transformation of bottom left corner vortex to the main vortex is highly dependent on a length of cavity and Reynolds number. In comparison with aspect ratios 2 and 3 from figure 5, bottom left corner vortex doesn’t exhibit enlargement in case of aspect ratio=3 for an increment of Reynolds number from 100 to 200.
Figure 8. Streamlines for Re=500 at Aspect Ratios(AR): (a) 1 (b) 2 (c) 3

Figure 9. Streamlines for Re=1000 at Aspect Ratios(AR): (a) 1 (b) 2 (c) 3

Figure 10. Streamlines for Re=1500 at Aspect Ratios(AR): (a) 2 (b) 3

Figure 11 to figure 12 illustrate the variation of $u$-velocity and $v$-velocity profile at the centre of mesh cavity for different Reynolds number at different AR. From figure 11 and figure 12, it is clear that different aspect ratios exhibit varied velocity profiles. Figure 12 shows that as the length of the cavity increased maximum velocity moves away from the boundary wall. From figure 6 to figure 9, for case studies (b) and (c), it is clear that transition of corner vortex to primary vortex influences the maximum $v$ velocity to move away from the boundary wall. $u$ velocity also affected by the
enlargement of corner vortices which is known from figure 11. Considering case study for aspect ratio = 3, from figure 11 (c), for Reynolds number 100 to 300 \( u \) velocity profile doesn’t differ in shape because of corner vortex influence is negligible. Figure 5 to figure 7 case studies (c) shows formation of vortex contours. When Reynolds number further increased from 500 to 1000 till cavity height = 3.5 \( u \) velocity is almost negligible due to effect of corner vortex and gradually increases due to primary vortex. It is evident from figure 8 and figure 9 in case studies (c).

![Figure 11](image)

**Figure 11.** \( u \) velocity at the centre of mesh cavity for different Reynolds number at Aspect Ratios: (a) 1 (b) 2 (c) 3

![Figure 12](image)

**Figure 12.** \( v \) velocity at the centre of mesh cavity for different Reynolds number at Aspect Ratios: (a) 1 (b) 2 (c) 3

4. Conclusion

In this work, numerical simulations are carried out for a flow inside lid driven cavity using 2D steady Navier-Stokes equations by varying aspect ratios and Reynolds number. Finite volume method is applied on the staggered grid which uses non-uniform meshes for coupling pressure-velocity in 2D steady N-S equation and solves dependent variables \( u, v \) and \( p \) present in the approximated discretised equation. A non-uniform staggered mesh of UPWIND scheme with SIMPLE technique is used. Discretization of flow structure by non-uniform grids tends to increase in accuracy and improved visualization of corner vortices. Studies and flow visualization carried out at low Reynolds number. From velocity profiles and streamlines, it was found that the primary vortex structure is strongly influenced by the geometry of the cavity as well as Reynolds number. The results obtained are in good accordance with the established results. Streamlines and mid plane stream-wise and span-wise graphs are plotted. A stream-wise and span-wise velocities graph shows the effect of vortices on mid-cavity
velocities. With the increase in Reynolds number, it is found that the primary vortex is moving towards the centre of the cavity. Interesting factor found that formation of tertiary vortex for AR=3 at Re=1500.

References

[1] Burggraf R Odus 1966 J. Fluid Mech. 24 113-151
[2] Shankar P N and Deshpande M D 2000 Annu. Rev. Fluid Mech. 32 93–136
[3] Harlow F H and Fromm J E 1965 Sci. Am. 212 104–110
[4] Harlow F H and Welch J E 1965 Phys. Fluids 8 2182–2189
[5] Patankar S V and Spalding D B 1972 Int. J. Heat Mass Transfer 15, 1767–1806
[6] Jameson, A, Baker T J and Weatherill N P 1986 Calculation of Inviscid Flow over a Complete Aircraft (Reno: 24th Aerospace Sciences Meeting)
[7] Ghia U, Ghia K N and Shin C T 1982 J. Comput. Phys. 48 387-411
[8] Yapici K and Uludag Y 2013 Braz. J. Chem. Eng. 30 923 – 937
[9] Mehmet Sahin and Robert G Owens 2003 Int. J. Numer. Meth. Fluids 42 57–77
[10] Sundaresan S, Nagarajan S, Deshpande S M and Narasimha R 1998 16th Int. Conf. on Numerical Methods in Fluid Dynamics (Arcachon:Soringer) pp 231-236
[11] Sun D. L, Qu Z G, He Y L and Tao W Q 2008 Int. J. Comput. Method. Part B 53 1–17
[12] Peric M, Kessler R and Scheuerer G 1988 Computers and Fluids 16 389-403
[13] Ramesh Chandra Mohapatra 2016 Study on Laminar Two-Dimensional Lid-Driven Cavity Flow with Inclined Side Wall Open Access Library Journal 3 e2430
[14] Mehdi Nikfar and Ashrafizadeh Ali 2016 Int. J. Comput. Method. Part B 69 447-472
[15] Ming Li and Tao Tang 1995 Int. J. Numer. Meth. Fluids 20 1137-1151
[16] Erturk E, Corke T C and Gokcol C 2005 Int. J. Numer. Meth. Fluids, 48 747–774
[17] Chen Z J and Przekwas A J 2010 J. Comput. Phys. 229 9150–9165
[18] Ferziger J H and Peric M 2002 Computational Methods for Fluid Dynamics (Newyork: Springer)
[19] Patankar S V 1980 Numerical Heat Transfer and Fluid Flow (Washington: Taylor and Francis)
[20] Versteeg H K and Malalasekera W 2007 An Introduction to Computational Fluid Dynamics(Harlow: Pearson)