Vortex Formation in Lid-driven Cavity with Disturbance Block

Maysam Saidi, Hassan Basirat Tabrizi, Reza Maddahian

***Abstract*—**In this paper, numerical simulations are performed to investigate the effect of disturbance block on flow field of the classical square lid-driven cavity. Attentions are focused on vortex formation and studying the effect of block position on its structure. Corner vortices are different upon block position and new vortices are produced because of the block. Finite volume method is used to solve Navier-Stokes equations and PISO algorithm is employed for the linkage of velocity and pressure. Verification and grid independency of results are reported. Stream lines are sketched to visualize vortex structure in different block positions.

***Keywords*—**Disturbance Block, Finite Volume Method, Lid-Driven Cavity

I. Introduction

Lid-driven cavity is one of the most widely used benchmarks to validate numerical codes. Although it seems a simple problem in the first glance, a square with three fixed walls and a moving wall which drives the internal fluid, but many aspects of fluid dynamics can be observed in its flow behavior. Internal circulation can be divided into two categories: primary and secondary. Primary vortex is in the center and secondary vortices are near corners. Their size and shape are changed via Reynolds number.

Burggraf [1] performed numerical simulation to see flow behavior inside a lid-driven cavity for the first time. Ghia et al. [2] were among the first researchers who present results for lid-driven cavity in different Reynolds number (100 to 10,000). Their results are among the most cited results for numerical validations and experimental calibration. Lid-driven cavity problem has been solved with different methods. Some researchers have solved the lid driven cavity in different geometries to see the geometry effect on its flow field. Polar cavity with a driven lid was investigated by Fuchs and Tillmark [3] experimentally and numerically. Glowinsky et al. [4] revealed the vortex structure in a semi-circular lid-driven cavity using finite element method. Patil et al. [5] studied the effect of depth to with ratio on vortex structure of rectangular cavities. Erturk and Dursun [6] studied the skewed lid-driven cavity vortices. Mercan and Atalik [7] studied vortex formation in arc-shape lid-driven cavities. Zhang et al. [8] simulated trapezoidal cavities using lattice Boltzmann method.

Although the simplicity of the problem but as shown in the literature reviews, researchers are working on lid-driven cavities by changing the geometry, boundaries, operational parameters and solution methods.

M. Saidi is Ph.D. Student at Mechanical Engineering Department of Amirkabir University of Technology (Tehran Polytechnic), 424 Hafez Ave, P.O. 15875-4413, Tehran, Iran (Phone: +98-21-64543410; Fax: +98-21-66419736; Email: maysam.saidi@aut.ac.ir).
H. Basirat Tabrizi is Professor at Mechanical Engineering Department of Amirkabir University of Technology (Tehran Polytechnic), Tehran, Iran (Email: hbasirat@aut.ac.ir).
R. Maddahian is Ph.D. Student at Mechanical Engineering Department of Sharif University of Technology, Tehran, Iran (Email: r_maddahian@mech.sharif.edu).

One of interesting studies would be to investigate flow behavior in presence of a disturbance block. This study is performed to compare vortex formation of nine different block positions in a standard lid-driven cavity with the standard one without block. The size and shape of primary and secondary vortices are shown and discussed.

II. Problem Description

Different geometries are used to simulate lid-driven cavity by inserting a block to different positions of classical square cavity. The geometry is a 1×1 m^2 square which has three fixed walls and a moving top wall with the velocity of 1 m/s. The disturbance block is a 0.2×0.2 m^2 square. To study the effect of block on cavity flow field nine different positions are selected to insert block which are shown in Fig. 1. The fluid is Newtonian incompressible and the Reynolds number is 10,000 in simulations as its value is in the highest bound of laminar flow and the vortex structure is more distinguishable.

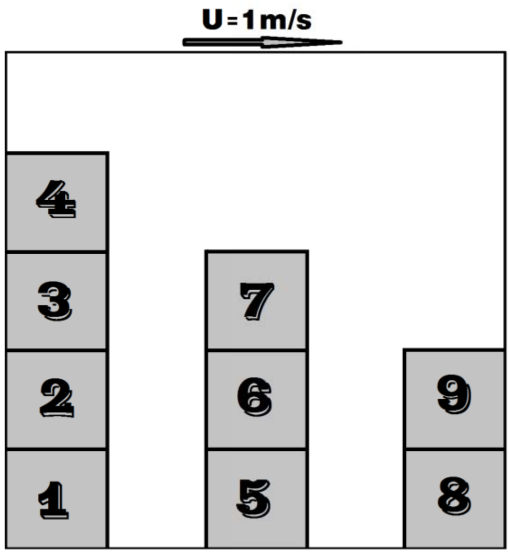


Fig. 1 Positions of block in lid-driven cavity

III. Numerical Solution and Results Validation

The Navier-Stokes (continuity and momentum) equations are solved in two dimensional incompressible flows in the absence of body forces which have the form of Eq. 1-3.

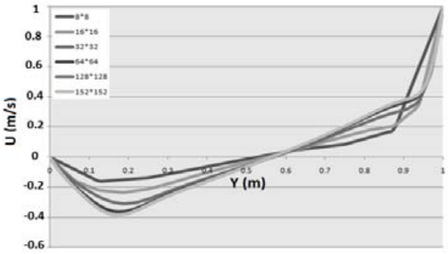
$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

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

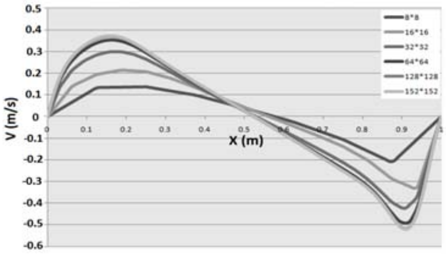
$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{\partial p}{\partial y}+\frac{1}{Re}\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

To solve these equations the domain is divided to discrete volumes to use finite volume method. Transient solver for incompressible laminar named icoFoam is employed to solve above equations. icoFoam is a C++ based solver which has been implemented in OpenFOAM code. The Pressure-Implicit Split Operator (PISO) algorithm handles the linkage between the velocities and pressure (Issa [9]). Preconditioned conjugate gradient solver for pressure and preconditioned bi-conjugate gradient solver for velocity are used. Upwind differencing method for convection, linear central differencing method for diffusion and Euler scheme for time is used.

The generated mesh for simulating flow inside lid-driven cavity is selected as 200×200 grids in the fine mesh and with assurance about independency of results on grids. The selection of this grid is upon grid independency study for Re=1000. As shown in Fig. 2 the 128×128 grids could be a wise choice for the standard classical lid-driven cavity.



(a)



(b)

Fig. 2 Velocity profile along a) vertical and b) horizontal center line in different grid sizes

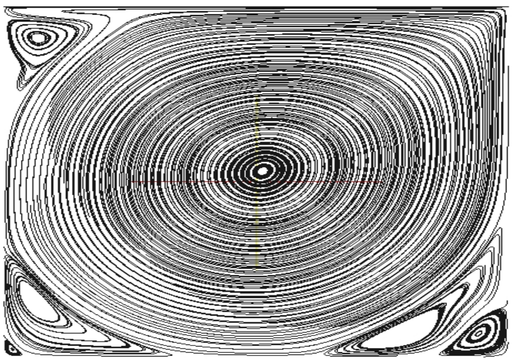
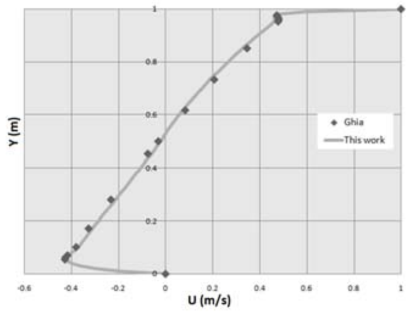


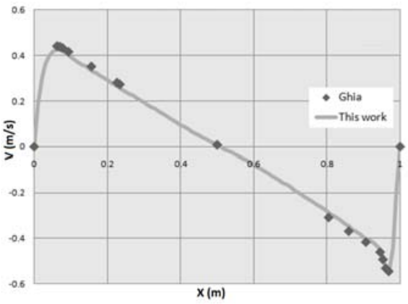
Fig. 3 Vortex structure in standard cavity

Stream lines in the standard classical geometry are shown in Fig. 3. Primary vortex in the center and secondary ones in the corners can be seen and will be used to compare with new studied geometries in the next section. For better later references upper left, lower left and lower right secondary vortices are called UL, LL and LR vortex, respectively.

To validate results, velocity profiles along horizontal and vertical centerlines are compared with Ghia et al. [2] in Fig. 4 and good agreement is noticed.



(a)



(b)

Fig. 4 Velocity profile along a) vertical and b) horizontal center line in standard lid-driven cavity (Re=10000)

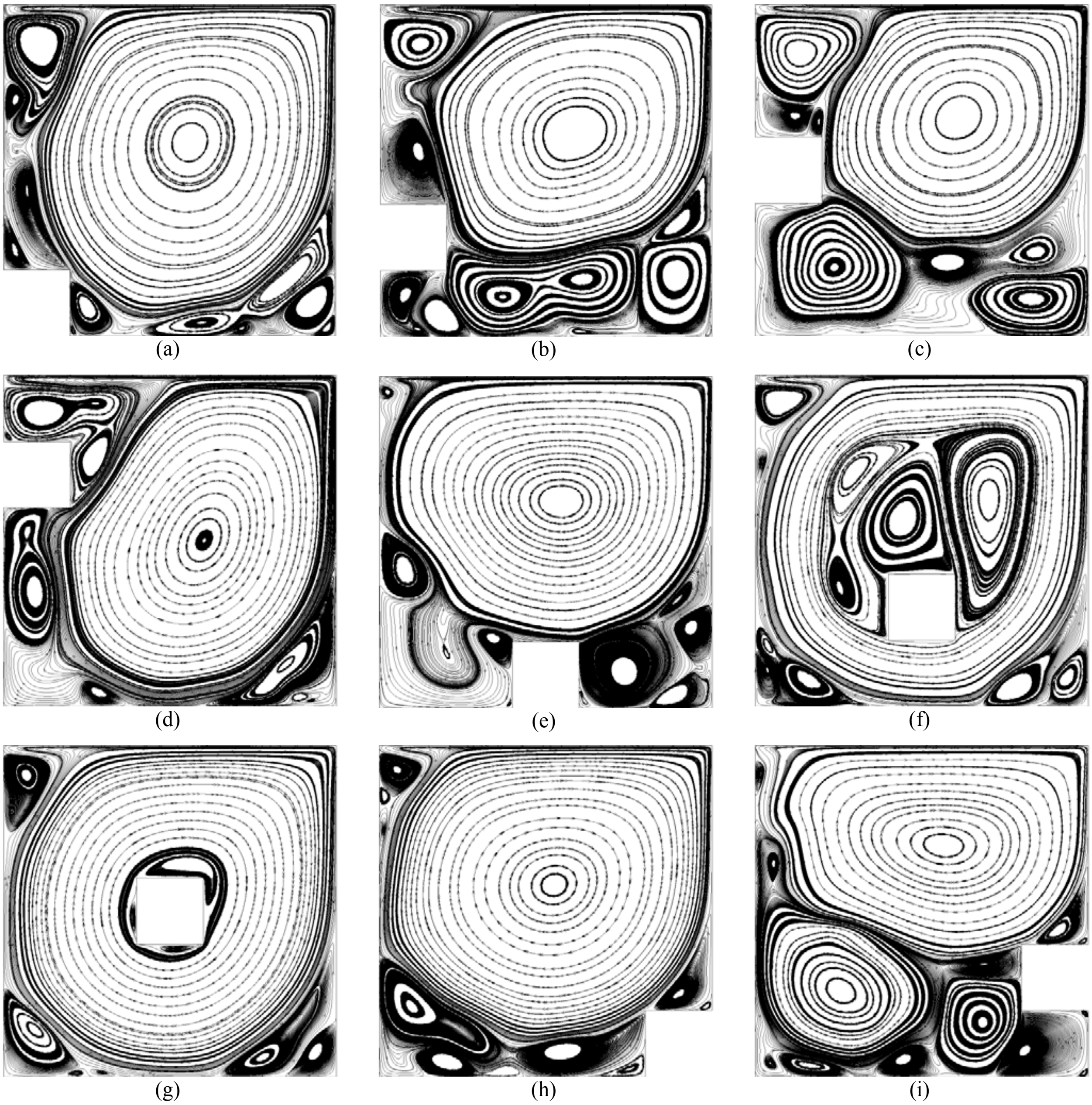


Fig. 5 Stream lines in different geometries (a-i: Positions 1-9 of Fig. 1)

IV. Results and Discussions

The indicated geometries have different flow behavior depend on the block position. Because of the selected high Reynolds number, results are not fully steady state but the final form of vortex structure remains nearly constant. Vortices structures are shown in Fig. 5. In the Fig. 5-a the block is located in the LL vortex position and leads to enlargement of LR and UL vortex size. The LL vortex ruptures and secondary vortices are produced in two sides of the block. By heightening the block in Fig. 5-b the primary vortex size decreases. LL and LR vortices reach to each other and secondary vortices are produced in top and bottom faces of the block.

The clockwise part of LR vortex is increased in size obviously in comparison with Fig. 5-a. Locating the block in the center of left wall increases the size of secondary vortices and decreases the size of primary vortex which are shown in Fig. 5-c. In the Fig. 5-d LR has minor changes rather than LL and UL. Locating the block in the center of bottom wall in Fig. 5-e pushes the primary vortex upward and the size of UL decreases. The interesting event occurs in Fig. 5-f by heightening the block to internal domain. UL, LL and LR shape returns to standard form but new secondary vortices are created among the primary vortex. This behavior repeated in Fig. 5-g with better stability and more similarity to standard lid-driven cavity by inserting the block to the center of cavity.

Comparing Fig. 5-h with Fig. 5-a smaller UL vortex size and different shape of LL and LR vortices can be seen. This difference is seen in comparison between Fig. 5-i and Fig. 5-b. increase of LL vortex size is another results of locating the block in Fig. 5-i.

V. Conclusion

The effect of disturbance block on flow field of lid-driven cavity is investigated. Presence of wall is the reason of secondary vortices production. Smaller changes in the secondary vortex occur by inserting the block in the center of primary vortex. Rupture of primary vortex and production of secondary vortices among it occurs by positioning the block in the primary vortex but with distance from center (Fig. 5-f). Minimum primary vortex size is in the cavity with block on the center of vertical wall. Positioning the block on left and right wall with same height doesn't lead to same vortex structure because the flow field is not symmetric.

References

[1] O. R. Burggraf, "Analytical and numerical studies of the structure of steady separated flows", *J. Fluid Mech.*,vol. 24, 1966, pp. 113–151.
[2] U. Ghia, K. N. Ghia, C. T. Shin, "High-Re solutions for incompressible flow using the Navier–Stokes equations and a multigrid method", *J. Comput. Phys.*, vol. 48, 1982, pp. 387–411.
[3] L. Fuchs, L. N. Tillmark, "Numerical and experimental study of driven flow in a polar cavity", *Int. J. Numer. Methods Fluids*, vol. 5, 1985, pp. 311–329.
[4] R. Glowinski, G. Guidoboni, T. W. Pan, "Wall driven incompressible viscous flow in a two-dimensional semi-circular cavity", *J. Comput. Phys.* Vol. 216, 2006, pp. 76–91.
[5] D. V. Patil, K. N. Lakshmisha, B. Rogg, "Lattice Boltzmann simulation of lid-driven flow in deep cavities", *Computers & Fluids*, vol. 35, 2006, pp. 1116-1125.
[6] E. Erturk, B. Dursun, "Numerical solutions of 2-D steady incompressible flow in a driven skewed cavity", *J. Appl. Math. Mech.,* vol. 87, 2007, 377-392.
[7] H. Mercan, K. Atalik, "Vortex formation in lid-driven arc-shape cavity flows at high Reynolds numbers", *Europ. J. Mech. B/Fluids,* vol. 28, 2009, pp. 61–71.
[8] T. Zhang, B. Shi, Z. Chai, "Lattice Boltzmann simulation of lid-driven flow in trapezoidal cavities", *Computers & Fluids,* vol. 39, 2010, pp. 1977–1989.
[9] R.I. Issa, "Solution of the implicitly discretised fluid flow equations by operator-splitting", *J. Comput. Phys.,* vol. 62, 1986, pp. 40-65.