Sindh Univ. Res. Jour. (Sci. Ser.) Vol.45 (2) 253-262 (2013)

SINDH UNIVERSITY RESEARCH JOURNAL (SCIENCE SERIES)

Finite Element Simulation of Newtonian Lid-Driven Cavity Flow

M. S. CHANDIO[++], M. U. JHATIAL S. F. SHAH*

Institute of Mathematics and Computer Science, University of Sindh, Jamshoro.

Received 28th April 2013 and Revised 2nd June 2013

Abstract: In this paper we have proposed a numerical scheme for the solution of two dimensional, steady state isothermal and incompressible Navier-Stokes equations**.** The proposed method is the modified version of semi-implicit Taylor-Galerkin-Pressure-Correction algorithm (Chandio and Webster 2002). The projection method, originally developed for unsteady flow problems by Chorin and Temam (1968), is used here for solving incompressible Navier-Stokes equations for the solution of primitive variables. The velocity and pressure fields are computed in a decoupled manner. To validate the accuracy and consistency of the proposed scheme, one-sided lid driven square cavity is considered here. The behaviour of primary, secondary and tertiary vortices are investigated at high Reynolds numbers. The governing equations are discretized using Galerkin finite element method. Simulation is performed for a various Reynolds numbers ranging from 500 to 20000. The obtained results are compared with the experimental and numerical results available in the literature and are in good agreement with them.

Keywords: Finite Element Method, Lid-Driven Cavity, Newtonian incompressible flow

1. INTRODUCTION

The rotating flows of viscous, incompressible and Newtonian fluids have gained great significance from industrial point of view. In particular, lid-driven cavity flow has been a subject of main interest among the researchers in which the fluid is induced to move by the motion of one of the walls of the cavity. A good deal of work on lid-driven cavity flow has been carried out by the researchers since sixties when Kawaguti (Aidun *et al*, 2002) conducted the pioneering study on the subject. The literature on the investigation of vortex structure inside the 2D square and rectangular cavities is in abundance and covers a considerable amount of computer-based simulations for considering different aspect ratios of length to width of the cavity.

Experimental phase of the study of lid-driven cavity has been the source of inspiration for the investigators. Practical-oriented work has attached a great importance to this class of flows.

Taneda (1979) has experimentally visualized the behaviour of shear driven flow inside the rectangular cavities at very small Reynolds number. He examined four cavities of various lengths to width aspect ratios. He found two corner vortices in the cavity with D=L/W=3 and only one primary vortex in the remaining cavities was presented.

Migeon *et al* (2000) carried out experiments to examine flow phenomenon inside three standard industrially modeled square, rectangular and semi-circular cavities with aspect ratios of 2:1 for Re=1000. The cavity was driven by one of its vertical walls. Besides a primary vortex, one secondary vortex developed in square and two secondary vortices in rectangular; while semi-circular cavity generated only one primary vortex.

Number of the research articles pertaining to the study of flow in a driven cavity may roughly fall into two categories, *i.e.* Physical aspect of the flow and Validation of the newly developed numerical schemes.

Plenty of the work on the physics of the flow is concerned with the paradigm shift from laminar to periodic flow and the localization of the first Hopf bifurcation (Kosinski *et al.*, 2009)

Yih-FerngPenget *et al.,* (2003) studied transition process from laminar to chaotic flow with two-branch solutions in square lid driven cavity. They presented that the flow in the first branch turned out to be periodic from steady flow at Re=7402 $\pm$ 4% and then entered into the second branch as Reynolds number is increased to 11000; which finally routed to chaotic flow for Reynolds number greater than 11000.

Charles-Henri *et al.,* (2006) performed numerical simulations to investigate the limitations of steady-state solution for high Reynolds numbers in a square cavity. Their results showed that steady solution is unstable at Re=10000. They identified a large eddy with two secondary eddies in the two bottom corners for Re=1000 and a secondary and tertiary eddies in the bottom left corner.

Auteri *et al* (2009) used singular subtraction technique to investigate the stability of the flow in the

[++] Corresponding author: mschandio@hotmail.com +929213181 Ext. 2030.
*****Department of Basic Sciences and related studies, MUET, Jamshoro

driven cavity with singular corners and localized the Reynolds numbers in the interval [8017.6, 8018.8] for the first Hopf bifurcation.

Aidun *et al* (1991) studied the stability of a lid-driven cavity flow and related such flows directly with coaters. Their results suggested the local stability of the flow at low Reynolds numbers however unstable globally.

Pan and Acrivos (1967) studied rectangular cavities with aspect ratios ranging from 0.25 to 5 both numerically and experimentally to investigate the steady flow for Reynolds numbers up to 4000. They observed that the enlargement in the structure of vortex in the cavity was dependent on the depth of the cavity.

Shankar (1993) concluded that the primary corner vortices are the prime source of the evolution of the large vortex structure in the deep cavity. He used an efficient computational procedure, based on an eignfunction expansion, to investigate the dependence of every secondary main vortex on the merger of each pair of developing primary corner vortices in 2D rectangular cavity.

Shankar and Deshpande (2000) elegantly reviewed the driven cavity flows and emphasized its importance theoretically and practically. They were of the view that the simple geometry of the cavity stands as yardstick to test new computational schemes and it also helps compare among the results obtained from theory, numerical simulation and experiment. Secondly, the flow domain in this class of flows remains unchanged at very high Reynolds numbers. Thirdly, many important features of cavity flows such as primary vortices, corner vortices, longitudinal vortices, transitions and turbulence all prevail in the same closed geometry.

Cheng and Hung (2006) performed a remarkable work on the vortex structure of steady flow in a lid-driven cavity of different aspect ratios for various Reynolds numbers up to 5000. They used Lattice Boltzmann method to investigate the impact of aspect ratios and Reynolds number on the formation of vortices in the cavity. They showed that when Reynolds number gets closer to zero, the flow displays symmetric vortex structure and asymmetric behaviour as Reynolds number increases.

Zdanski *et al.,* (2003) numerically simulated the comparison between laminar and turbulent flows over shallow cavities and investigated the effect of aspect ratio, Reynolds number and inlet turbulence level on turbulent flow; in which case the external flow touched the bottom of the cavity for a certain value of each of the above parameters. Their results suggested that the increasing Reynolds decreased the center of laminar structure of vortex. On the contrary, the center of both the large and small structure for turbulence case was observed unchanged.

Trianta fillaopoulos *et al.,* (1998) visualized numerically flow observations in a diagonal lid-driven cavity, supposed to be a main source for coating process. They proposed that the machine speed affects the flow in cavity which results in the formation of vortex the nip of the blade. Thus the changing speed of the machine influenced the vortex size considerably but the location of the vortex remained fixed.

Ercan Erturk (2009) studied lid-driven cavity flow in detail considering physical, mathematical and numerical aspects, and suggested that finer grids are important for the resolution of the flow at very high Reynolds numbers. Quaternary vortex at the bottom left corner was showed at Re=7500, which was improvement over previous results.

Zoran *et al.,* (2006) proposed a numerical scheme with the combination of Boundary Element and Finite Element methods to make better the accuracy of boundary vorticities computation. A 3D lid-driven cavity was used to test the algorithm.

Fuat G¨urcan (2003) carried a work on rectangular cavity with two moving walls. The symmetric flow behavior was displayed for Aspect ratio A=0.9 at the origin (0, 0). As A exceeded 0.9, the centre (0, 0) changed to saddle point; which turned to centre again when A approached zero.

Flow with temperature effects in cavity is another industrial-oriented and an interesting aspect of lid-driven cavity. Tanmay Basak *et al* (2009) used a lid-driven square cavity to analyze the natural and forced convection flows and showed that the increasing Reynolds number decreases the effect of natural convection and increases the forced convection. Rejane De *et al.,* (2009) analyzed numerically entropy generation inside rectangular cavities. Youssef Stiriba (2008) conducted a numerical study of the flow and heat transfer past an open cubical cavity. Steady three-dimentional structure of the stable flow was showed at moderate Grashof number and unstable was observed at large Richardson number.

Kosinski *et al.,* (2009) simulated the interaction between fluids and solid particles in a driven cavity and reported the effect of the particle clouds on the flow.

Wang (2009) studied fibrous medium mixed with a viscous fluid contained in a lid-driven rectangular

cavity by using Eigen function method. He presented that the effect of porous medium diminished the strength and the number of developing vortices.

Numerical Scheme

For an incompressible and isothermal laminar flow, the mathematical model consist the system of governing equations which may be described by the conservation of momentum and the continuity equations. Here, the problem under study is incompressible, steady state and 2D in Cartesian coordinate systems. In the absence of body force, the governing equations can be described in vector calculus form as:

$$\left.\begin{aligned} \rho\frac{\partial u}{\partial t} &= \mu\nabla^2 u - \rho u.\nabla u - \nabla p \\ \nabla.u &= 0 \end{aligned}\right\} \qquad (1)$$

where, $\boldsymbol{u}(x,y)$ is the velocity, p is the pressure, ρ is the density and μ is the dynamic viscosity. Adopting appropriate non-dimensional character scale for u, v, p, ρ, and μ, an equivalent non-dimensional system of equations to (1) is obtained as:

$$\left.\begin{aligned} Re\,\frac{\partial u}{\partial t} &= \nabla^2 u - Re\; u.\nabla u - \nabla p \\ \nabla.u &= 0 \end{aligned}\right\} \qquad (2)$$

where $Re = \dfrac{\rho UL}{\mu}$ is the non-dimensional *Reynolds* number.

For the solution of governing equations (2), we have used projection method as proposed by Chorin and Temam (1968)[21]. Adopting the notation of Cuvelier *et al.,* (1992) on the velocity and pressure fields on the specified domain, the Galerkin finite element method is chosen for the discretization of the system of governing equations, details may be seen on fractional stages in Chandio and Webster (2002).

2. RESULTS AND DISCUSSION

We have considered a lid-driven square cavity flow. The cavity of unit width and height is filled with constant density and viscosity Newtonian Fluid. Top wall of the cavity is set to move with constant unit velocity in the x-direction, where as the bottom and side walls are kept non-moving, see figure 1. Here, the goal is to validate accuracy, stability and consistency of the proposed numerical scheme for high Reynolds numbers. The density of the fluid is taken as ρ = 1000 g/m^3 and viscosity, μ, of the fluid is computed from $Re = \dfrac{\rho UL}{\mu}$. Here L is the characteristic length of the cavity, (1m) and U is the characteristic velocity of the lid (1m/s). The normal pressure gradient on the fixed walls is taken as zero and the initial conditions for u, v, and p are set to zero.

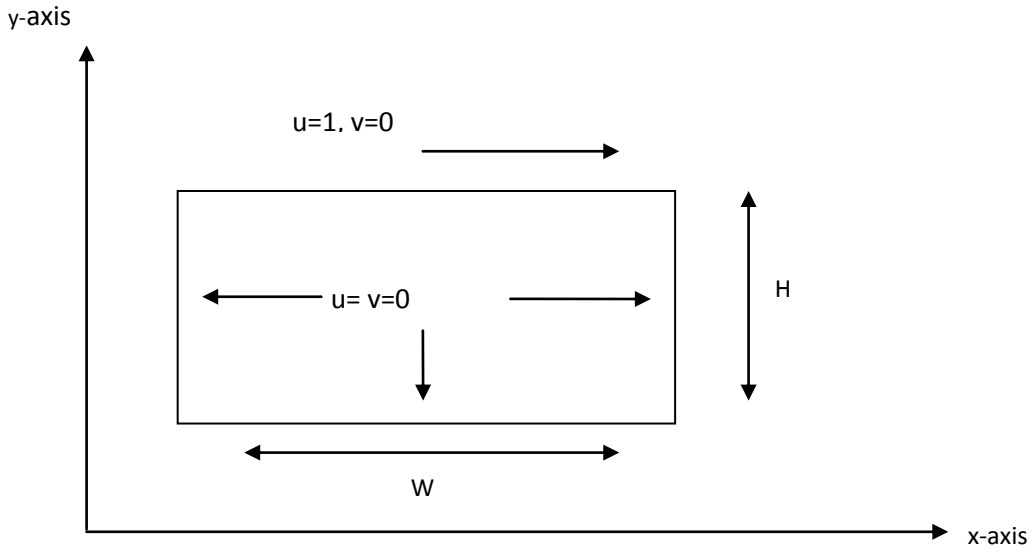
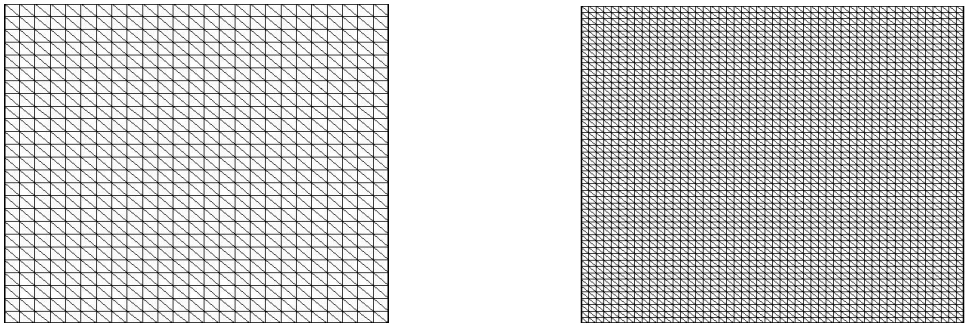
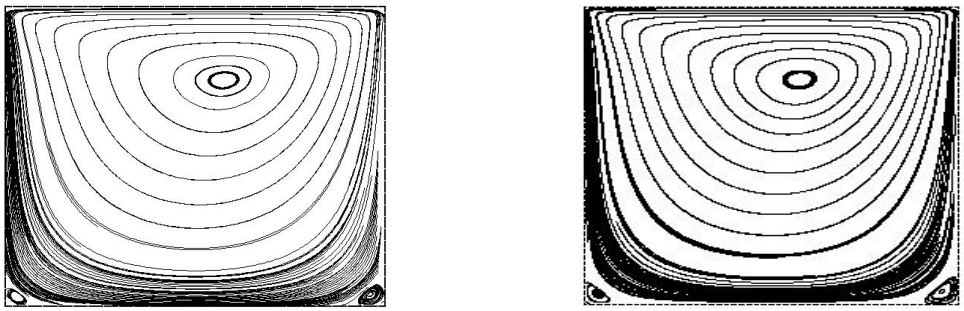


Fig.1. Schematic diagram of the square width lid-Drive cavity flow



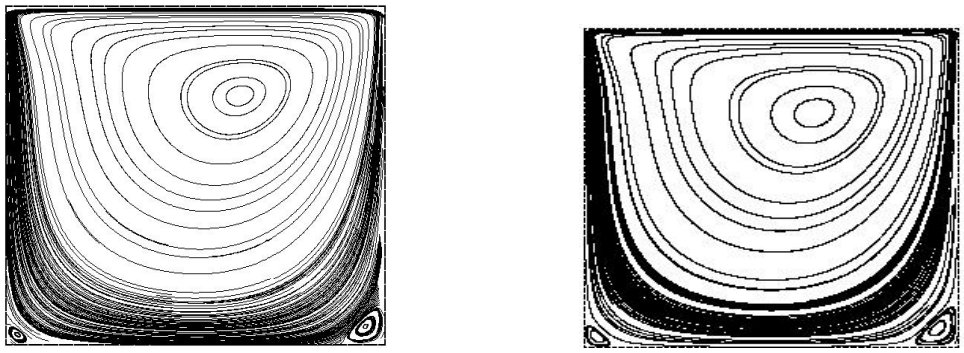
(a) **(b)**

Fig.2. (a) Course Mesh 25x 25 (b) Dense Mesh 50x50



(a) **(b)**

Fig.3. Streamline pattern at Re=500 for course and Dense Meshes.

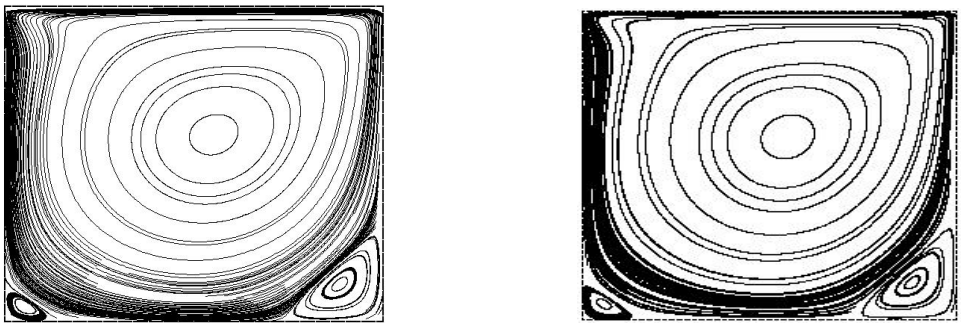


(a) (b)

Fig.4. Streamline patterns at *Re*=1000 for course and Dense Meshes.

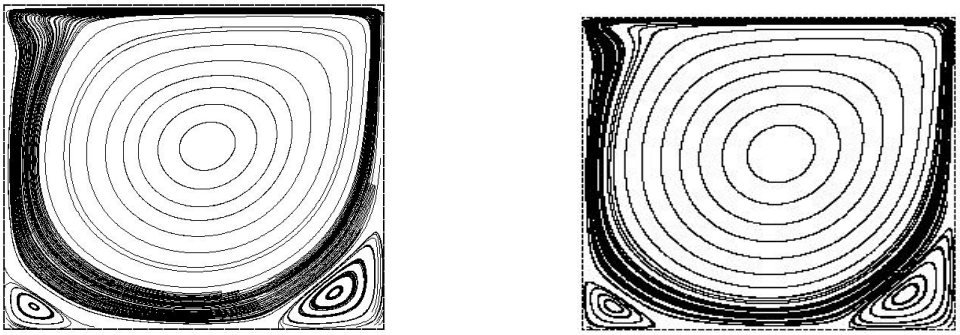
In this study we have used course mesh of size 25x25 and dense mesh of size 50x50 grids (see figure 2.a. and 2.b.) In figure 3.a, the stream line patterns in the cavity of grid 25x25 at *Re*=500 are shown. It is observed that a large vortex fills the whole space of the cavity with two bottom corner vortices. The centre of the vortex is slightly away from the vertical line of the cavity and seems to be moving towards the right wall of the cavity. The size of the right bottom corner is larger than that of the left one. Almost the same flow behaviour is shown for the large vortex when the mesh is further refined to 50x50, **(see Fig. 3.b.),** where a small enhancement in the size of corner vortices is observed.

At *Re*=1000, the centre of the vortex further moves towards the right wall of the cavity and corner vortices grow in size slightly **(see Fig. 4.a.).** The shape of the large vortex is little bit circular as compared with the vortex formed when Re=500. The corner vortices seem to be growing in size for the refined mesh of size 50x50 **(see Fig. 4.b.)**



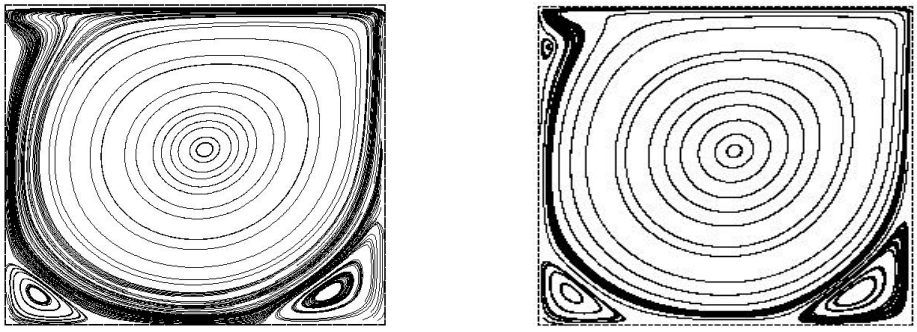
(a) Course mesh (b) Dense Mesh

Fig.5. Streamline pattern at Re=5000 for course and Dense Meshes.



(a) **(b)**

Fig.6. Streamline pattern at Re=10000 for course and Dense Meshes.



(a) **(b)**

Fig.7. Streamline pattern at Re=20000 for course and Dense Meshes.

A considerable change in the flow pattern is observed at Re=5000 **(Fig. 5.a.).** Centre of the vortex lies at the centre of the cavity and the large vortex is developing circular in shape. The curved pattern of streamlines at the top left corner indicates the possible emergence of the fourth vortex at high Reynolds number. A significant growth in the bottom corner vortices is visible. Right bottom corner vortex grows at faster growth rate than the left corner vortex when Reynolds number increases. We observe that the refinement in the mesh affects the size of bottom corner vortices and gives a little rise in their sizes **(Fig. 5.b.).**

When Reynolds number reaches 10000, the flow behaviour looks more circular **(Fig. 6.a.).** No significant effect on the size of right bottom corner vortex is shown at Re=10000; while left bottom corner appears to be enhanced in the shape as compared with the left bottom corner vortex formed when Re=5000.

The fourth vortex emerging at the top left corner at Re=20000 for the refined mesh of size 50x50 **(Fig.7.b);** whereas no such vortex is visible in the cavity for the mesh of size 25x25 **(Fig. 7.a.).** Both the bottom corner vortices have further grown in size but the right bottom corner vortex is still larger than the left one.

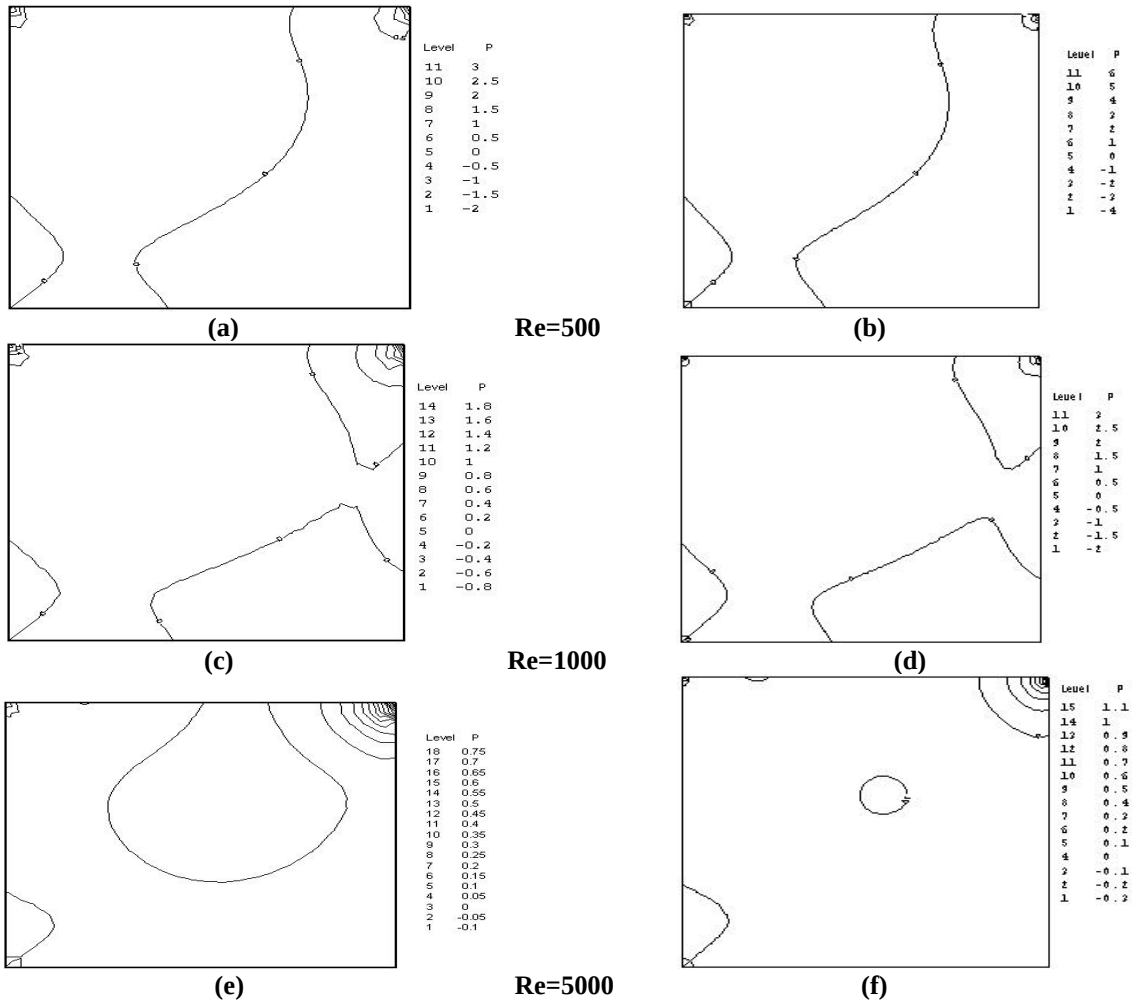


Fig.8. Pressure Contours at various Reynolds numbers

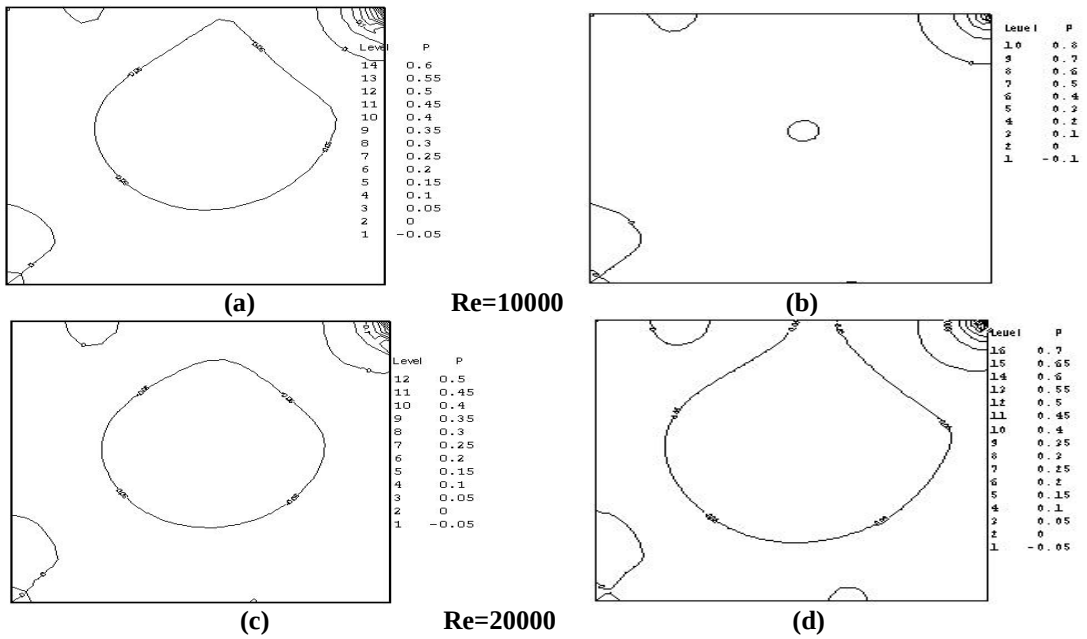


Fig.9. Pressure Contours at various Reynolds numbers

Pressure line contours are shown in **(Fig. 8 and Fig. 9)** at various Reynolds numbers ranging from 500 to 2000. Here, we can see that the pressure of the fluid inside the lid-driven cavity is decreasing with the increasing Reynolds number **(see Fig. 8-9).** The maximum pressure of the flow in the cavity occurs at the top right corner of the cavity and the minimum pressure appears at the top left corner of the cavity. The fluid, with the motion of the moving lid, starts flowing and it spreads the whole cavity after striking with the right wall of the cavity. In result, the pressure gradually decreases in the lower part of the cavity **(see Fig. 8-9).** For the refined mesh of size 50x50, the magnitude of pressure is large as compared with the pressure of flow in the cavity with course mesh of size **25x25 (see Fig. 8 (a-d)).** At low Reynolds numbers (500 and 1000), it is observed that the magnitude of pressure for the refined mesh increases rapidly in comparison with course mesh at high Reynolds numbers (Re>1000), **(see Fig. 8-9).** No significant change in the magnitude of pressure is observed for refined mesh, **(see Fig. 9 (a-d).**

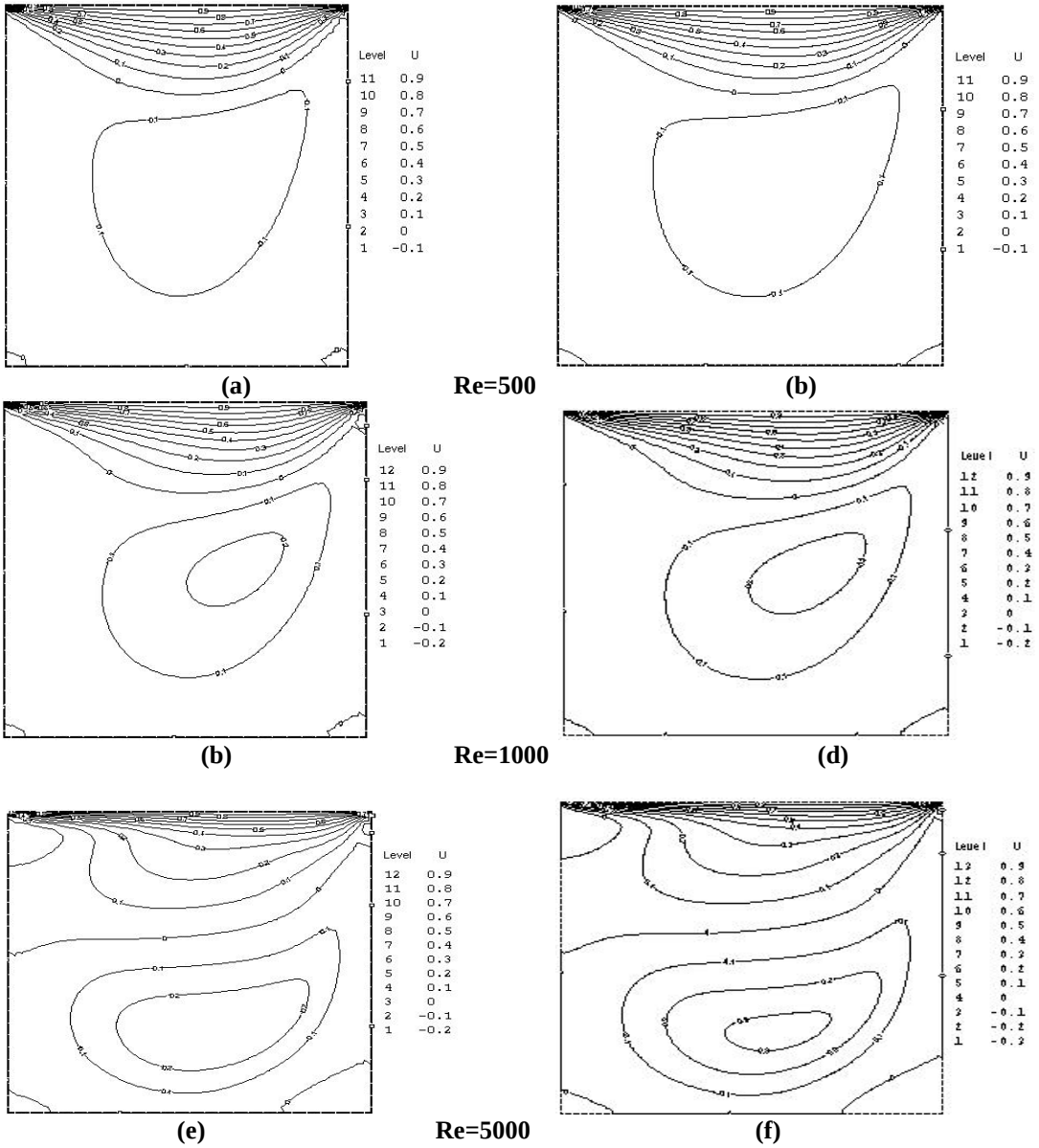


Fig.10. U-Component of velocity at various Reynolds numbers

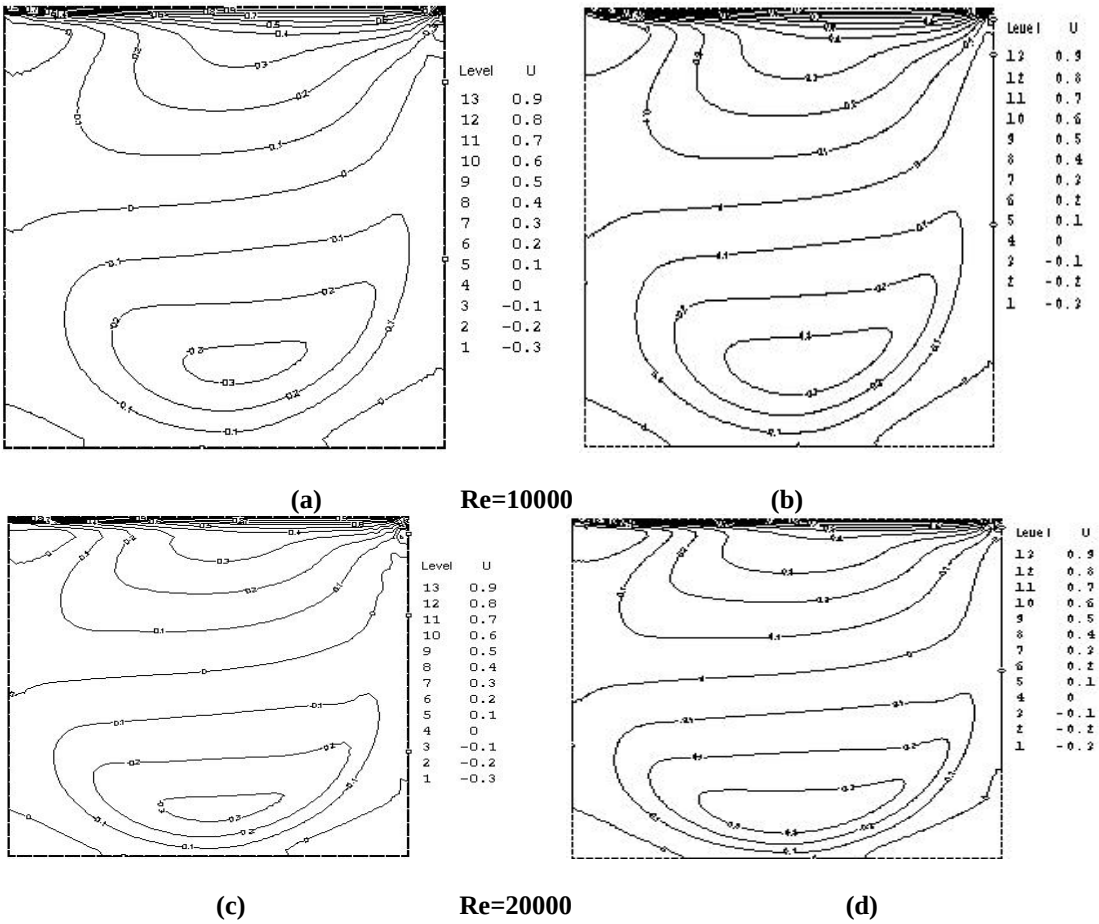
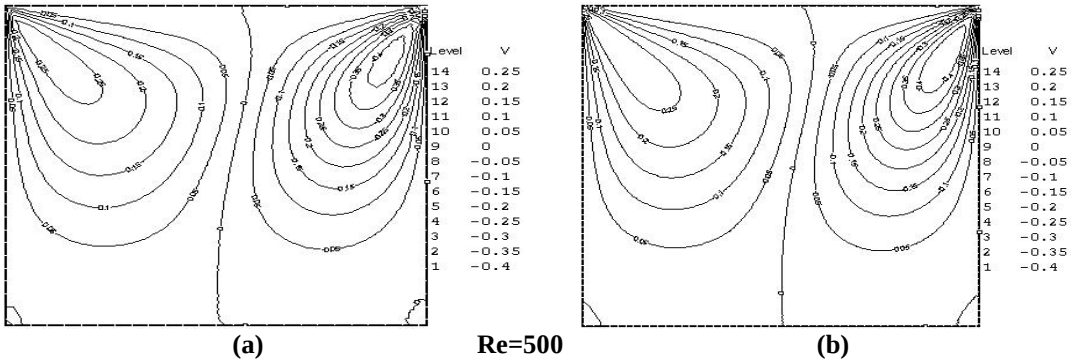


Fig.11. U-Component of velocity at various Reynolds numbers

U-velocity distribution indicates that the flow is dominant along the lid of the cavity due to non-moving right and left walls **(see Fig. 10-11).** No significant influence of the increasing Reynolds number on U-velocity distribution is observed.



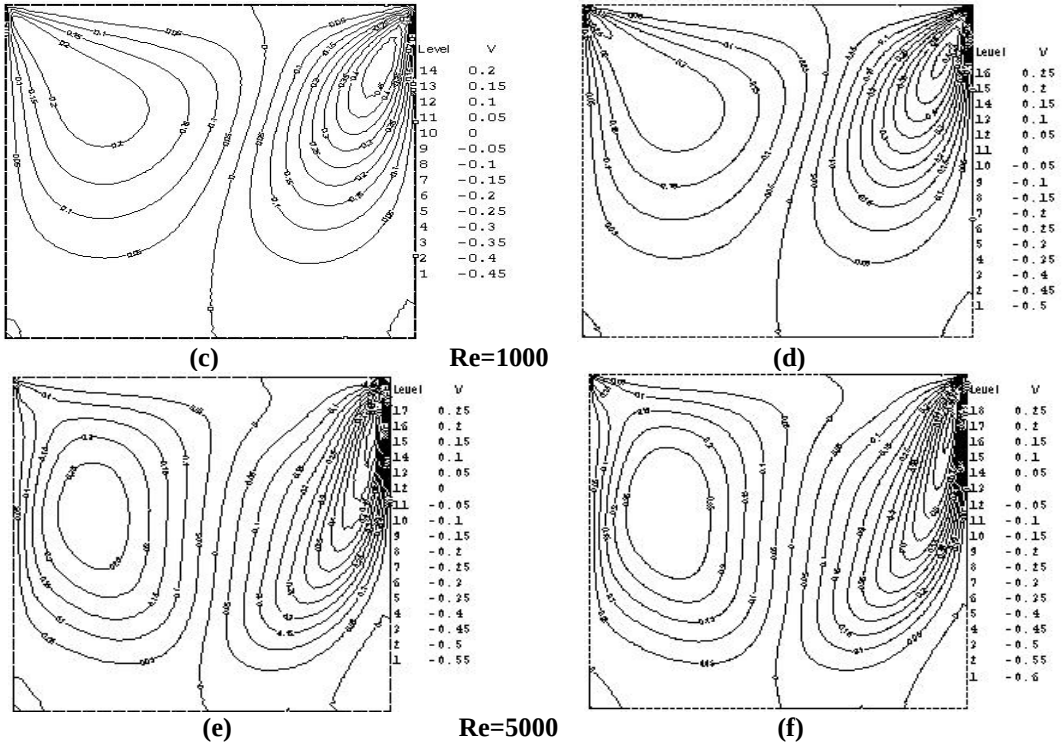


Fig.12. V-Component of velocity at various Reynolds numbers

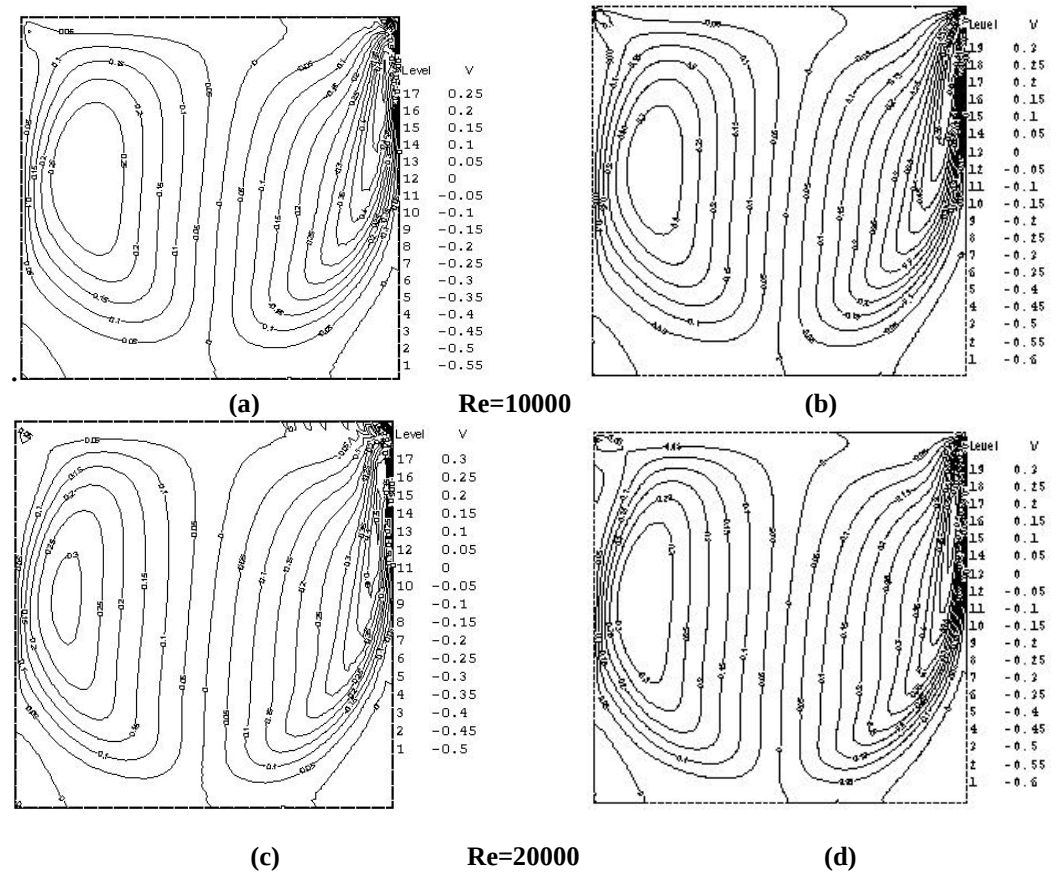


Fig.13. V-Component of velocity at various Reynolds numbers

In V-velocity distribution, behaviour of the flow shows its dominancy along the left wall of the cavity and the reverse flow on the right wall **(Fig. 12-13).** No significant influence of the increasing Reynolds number on V-velocity distribution is observed.

3. CONCLUSION

In the current paper, the numerical simulation of the behaviour the structure of the vortices of viscous flow in 2D lid-driven square cavity is studied. Here, the obtained results are in good agreement with the numerical results available in the literature. In the present study we observed that the proposed modified Semi-implicit Taylor-Galerkin method remained stable for large Reynolds number with refined mesh size of 50x50. However, in the literature (Auteri *et al*, 2002), the stability and convergence of the numerical scheme is achieved with refined mesh size of 1260x1260. Thus we can conclude that the proposed scheme has shown consistency, stability and accuracy for large Reynolds numbers for mesh size of 50x50. We will use the proposed method for rectangular meshes of various aspect ratios with both top and bottom lid-driven cavity viscous flows.

REFERENCES:

Aidun C.K., N. G. Triantafillaopoulos and J. D. Benson (1991) *Global stability of a lid-driven cavity with through flow*. Flow visualization studies. Institute of Paper Science and Technology, School of Mechanical Engineering, Georgia Institute of Technology, 575 Fourteenth Street, N W, Atlanta, Georgia 30318.

Auteri F., N. Parolini and L. Quartapelle (2002) *Numerical Investigation on the Stability of Singular Driven Cavity Flow*. Journal of Computational Physics, vol. **(183):** 1–25

Chandio M. S. and Webster M. F. (2002) *Numerical study of transient instabilities in reverse-roller coating Flows*. International Journal of Numerical Methods for Heat and Fluid Flow, vol. **(12):** 4, 375- 403.

Charles-Henri B. and M. Saad (2006), *The 2D lid-driven cavity problem Revisited*. Computers and Fluids, vol. **(35):** 326–348.

Cheng M. and K.. C. Hung (2006) *Vortex structure of steady flow in a rectangular cavity*. Computers and Fluids, vol. **(35):** 1046–1062.

Chorin, A. J. (1968) *Numerical Solution of the Navier-Stokes Equations*. Math. Comp.vol. **(22):** 745–762.

Ercan E. (2009) *Discussions On Driven Cavity Flow*. International Journal For Numerical Methods In Fluids, vol. **(60):** 275-294

Frank P. and A. Andreas (1967) *Steady flows in rectangular cavities*. J. Fluid Mech., vol. **(28):** part 4, 43-655.

Fuat G¨urcan (2003) *Streamline Topologies in Stokes Flow within Lid-Driven Cavities*, Theoretical and Computational Fluid Dynamics, vol. **(17):** 19-30.

Kosinski P., A. Kosinska and A. C. Hoffmann. (2009) *Simulation of solid particles behaviour in a driven cavity flow*. Powder Technology, vol. **(191):** 327-339.

Migeon C., A. Texier and G. Pineau (2000) *Effects of Lid-Driven cavity shape on the flow establishment phase*. J. of Fluids and Structures, vol. **(14):** 469-488.

Rejane De C., M. Oliveski, H. Macagnan and B. C. Jacqueline (2009) *Entropy generation and natural convection in rectangular cavities*. Applied Thermal Engineering, vol. **(29):** 1417–1425.

Shankar P. N. (1993) *The eddy structure in Stokes flow in a cavity*. J Fluid Mech., Vol. **(250):** 371–83.

Shankar P.N. and Deshponde M.D. (2000), *Fluid Mechanics in the Driven Cavity*. Ann Rev Fluid Mech., vol. **(32):** 93–136.

Taneda S. (1979), Visualization of separating Stokes flows. J Physical Society of Japan, vol. **(46):** No.6.

Tanmay B., S. Royb, P. K. Sharma and I. Pop (2009) *Analysis of mixed Convection flows within a square cavity with uniform and non-uniform heating of bottom wall*. International J. of Thermal Sci. vol. **(48):** 891–912.

Temam, R. (1968) *Une méthode d'approximation des solutions des équations Navier-Stokes*. Bull. Soc. Math. France vol. **(98):** 115-152.

Triantafillaopoulos N., G. Rudemiller, T. Jr.Farrington and J. Lindsay, (1998) *Numerical simulation of Short-dwell coater pond flows*. TAPPI Engineering Conference in Chicago.

Wang C. Y. (2009) The *recirculating flow due to a moving lid on a cavity containing a Darcy–Brinkman medium*. Applied Mathematical Modelling, vol. **(33):** 2054–2061.

Yih-Ferng P., S. Yuo-Hsien and R. R. Hwang (2003) *Transition in a 2-D lid-driven cavity flow*. Computers and Fluids, vol. **(32):** 337–352.

Youssef S. (2008) *Analysis of the flow and heat transfer characteristics for assisting incompressible laminar flow past an open cavity*. International Communications in Heat and Mass Transfer, vol. **(35):** 901–907.

Zdanski P.S.B., M. A.. Ortega G. C. R. Nide Jr. Fico (2003) *Numerical study of the flow over shallow cavities*. Computers & Fluids, vol. **(32):** 953–974.

Zoran Z, M. Hribersek, L. Skerget and J. Ravnik (2006) *3D Lid Driven Cavity flow by Mixed Boundary and Finite Element Method*. European Conference on Computational Fluid Dynamics ECCOMAS CFD.