Sindh Univ. Res. Jour. (Sci. Ser.) Vol.45 (2) 277-280 (2013)

SINDH UNIVERSITY RESEARCH JOURNAL (SCIENCE SERIES)

Computer Simulation of Lid Driven shallow Cavity Flows: Effects of Fluid Inertia and Aspect Ratios

Z. A. K. BALOCH[++], M. E. BALOCH*, H. SHAIKH**, M. A. SHAR***

Institute of Environment Engineering, Mehran University of Engineering and Technology Jamshoro.

Received March 2013 and Revised July 2013

Abstract: The paper presents the computational modelling of liquid flow through lid–driven shallow cavity explained by a semi–implicit pressure/correction finite element scheme. The impact of Reynolds number ranges from 200 to 2500 and consequent ratios 2:1, 2:2, 2:3 and 2:4 of the shallow cavities and vortex intensity in terms of the fluid inertia are examined, and develops the descending rounded primary vortex to the centre of the cavity and two small left corner and right corner secondary vortices in all geometries of deep cavities .Superb agreement of numerical results are obtained and contrasted with the other numerical results presented in the research material.

Keywords: Mmoving lid–driven shallow Cavity Flow, Finite Element, fluid dynamics.

1. INTRODUCTION

Present study considered a flow of incompressible liquid through shallow lid–driven cavities. In computational fluid dynamics, the shallow lid–driven cavity fluid flow is the highly important problem because of its ease of the cavity geometry by applying a several numerical techniques such as finite difference, finite volume, Successive over Relaxation (SOR) method, least square method and Lattice Boltzmann method (Fletcher, 1991, Sahin 2003, Mussa, *et al.* 2008 and Erturk 2009).

Initially, Burggraf (1996) proposed the analytical method and computational numerical scheme for the steady solution of separated flow in complex geometries. After that, the various scholars spawned many of the papers of computational scientific algorithms for the lid–driven cavity geometry, only worried about the requisite accuracy and stability (Gatski *et al..*,1986), Ghia *et al.* (1986) and Gustafson et al.(1986)). Subsequently, A finite–volume technique is employed for the steady–state and unsteady solution of Navier–Stokes equations with effect of fluid inertia up to 10000 by Sahin (2003). Three (coarse, medium and fine) meshes were described in order to examine the grid dependency of the steady –state and unsteady solution. Analytical and numerical methods were applied and visualized the primary vortex transfer towards the right of the cavity wall, two secondary upstream vortex and downstream vortex in the shallow cavities.

Mussa, *et al.* (2008) adopted the computational fluid dynamics software Fluent and Lattice Boltzmann method to predict the solution for the incompressible fluids through deep and as well as shallow lid–driven cavities. The influence of the fluid inertia (= 100 to 1000) with various aspect ratios (0.25, 0.5, 1.5 and 4) was examined on the flow structures. The numerical results were compared with the CFD software Fluent and same results as Sahin (2003) were envisioned the primary vortex and two small upstream vortex and downstream eddy because of increase a Reynolds number.

A robust time–dependent finite element numerical method is adopted to obtain the steady solutions of shallow cavities. At both low and high–level Reynolds numbers (200 to 2500), various ratios of shallow cavities (1: 1, 2:1, 3:1 and 4:1) are simulated by adopting adequate grid mesh. The numerical results are contrasted with the Sahin 2003, Mussa, *et al.* 2008 and other numerical results of CFD package Fluent available in literature. Excellent agreements of results are accomplished here with.

2. Problem description and solution strategy

The liquid flow of Newtonian fluid through shallow lid–driven cavity the characteristic ratios 1:1, 2:1, 3:1 and 4:1 with different Reynolds numbers 200 to 2500 are chosen. The benchmark problem is extensively used to evaluate the accuracy, stability and required the level of convergence measures.

The constant velocity (**u**) is presented on the lid and no–slip boundary conditions are used on the cavity walls. The schematic diagram of the cavity domain in **(Fig. 1)** and different fine grids in (**Fig. 2)** are displayed. Total number of elements (800, 1600, 2400 and 3200), total number of nodes (1681, 3321, 4961and 6601) and degrees of freedom (3803, 7503, 11203 and 14903) respectively with triangular elements are chosen in all geometries.

[++]Corresponding Author: Hisam Uddin Shaikh, , email : hisam.shaikh@salu.edu.pk

*Department of Computer System Engineering, Technical University Duisburg, Germany.

**Department of Mathematics, Shah Abdul Latif University, Khairpur Mirs, Sindh.

***Centre of Excellence research in Engineering materials Advanced Manufacturing Institute, King Saud University Riyadh Kingdom of Saudi Arabia.

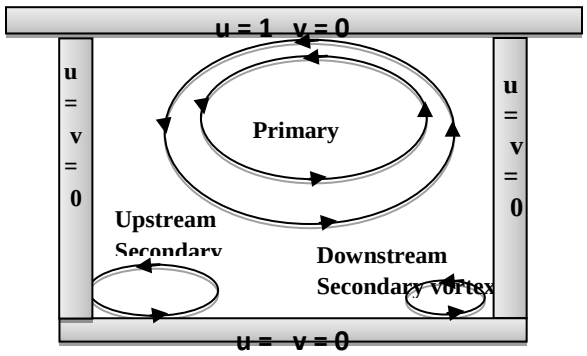


Fig. 1 Schematic diagram of lid–driven cavity.

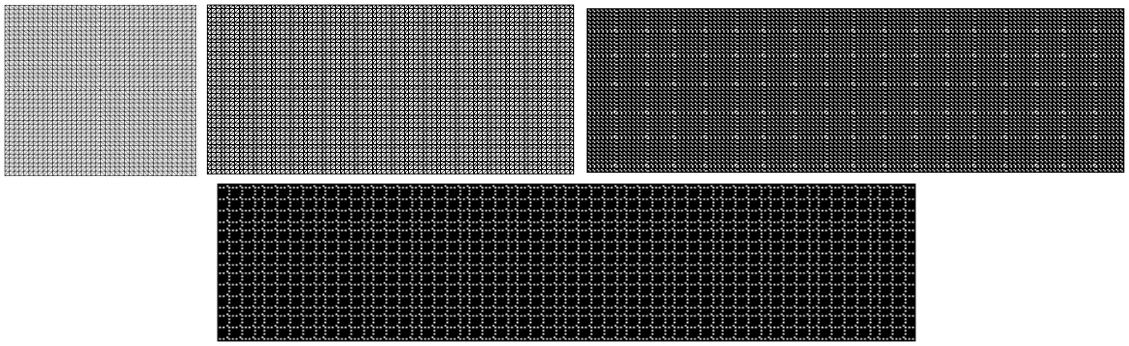


Fig.2: Finite Element meshes in the aspect ratio 1:1, 2:1, 3:1 and 4:1.

3. Governing system of equations and numerical scheme

The flow of Newtonian fluid is considered though lid–driven shallow cavities which are governed by the continuity and momentum equations in the absence of body force as follows:

1. Continuity equation: with zero divergence due to incompressibility and explained as the amount of fluid inflowing the cell is equal to the amount of fluid exiting from the cell.

$$2.\ \frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0.$$

Where u and v are represents the velocity components in axial (x) and transversal (y) directions respectively.

2. Momentum equation

$$\frac{\partial u}{\partial t}+(u\cdot\nabla)u=\frac{1}{Re}\nu\nabla^2 u-\frac{\partial p}{\partial x}$$

$$\frac{\partial v}{\partial t}+(v\cdot\nabla)v=\frac{1}{Re}\nu\nabla^2 v-\frac{\partial p}{\partial y}$$

Here $\frac{\partial \mathbf{u}}{\partial t}$ denotes the time derivative and for steady–state solution the time derivative vanishes and $(u\cdot\nabla)u$ denotes the advection term and is used for the direction in which the surrounding fluid thrusts a minor region of fluid. Re defined as the Reynolds number as $Re=\frac{\rho UL}{\mu_0}$ in which ρ, μ and L denotes the density, constant viscosity of the fluid and length of the channel and $\nu\nabla^2 u$ denotes the momentum diffusion term termed as how rapidly the fluid damps out deviation in the viscosity nearby a given fluid and $\frac{\partial p}{\partial x}$ describe d as the pressure gradient that defines how a fluid is shoved in a direction from high–level to low–level pressure.

For the steady–state solution of continuity and momentum equation adopting a semi–implicit pressure/correction finite element scheme and this scheme consisting three stages referred from Baloch, A *et al.* (1994). This scheme involve a pair of schemes as finite difference and finite element methods that is proposed and corrected by Donea, (1984) and Kan, (1986) and achieved a required accuracy and stability. Consequently, the various researchers employed the semi–implicit schemes for Newtonian, Non–Newtonian and viscoelastic fluid flows in simple as well as in complex geometries in the meadow of computational fluid dynamics.(Townsend and Webster, (1987) and Hawken, *et al*. (1990).

4. Numerical results and Discussion

4.1 ***The influence of fluid inertia:*** The streamlines contours in 2–Dimensional through square and shallow cavities are examined by adopting a semi–implicit pressure/correction finite element scheme. The Reynolds number varies from 200 to 2500 with aspect ratios 2:1, 2:2, 2:3 and 2:4 are analysed in the domain. In (**Fig. 3)** at low Reynolds number primary vortex occurred and moves downward in whole of geometry and small downstream vortex activity starts. With increasing Reynolds number the primary vortex is visualised, downward stream vortex on left of cavity grows in size and intensity and small activity in size and intensity of upstream secondary vortex developed in right of cavity. Consequently with increasing aspect ratios of geometry, the flow phenomena is altered as primary–spinning vortex descend in downwards and proceed towards right of the cavity wall with increasing Reynolds number. Bigger upstream secondary vortex are visualized and with increasing Reynolds number, the upstream vortex expands in size and intensity but downstream vortex remains small at low and high Reynolds number. The vortex intensity is also calculated of all geometries and displayed in (**Fig. 4-7).**

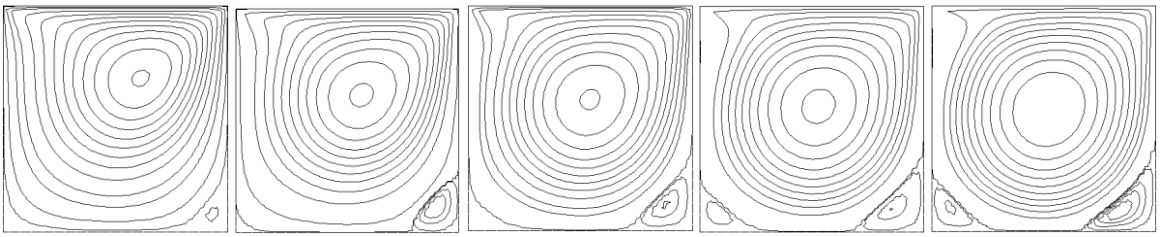


Fig. 3: **Streamline contours of 1:1 cavity for Re = 200, 400, 7500, 1500 and 2500.**

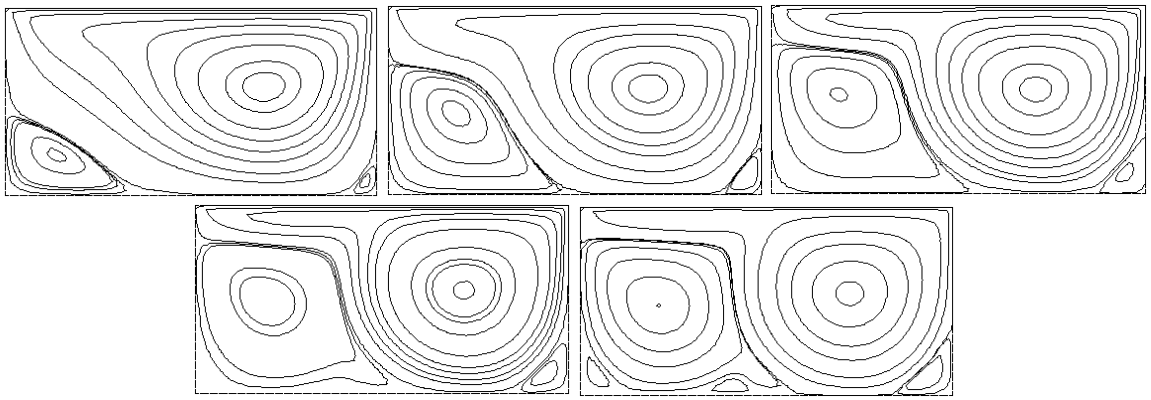


Fig. 4: Streamline contours of 2:1 cavity for Re = 200, 400, 7500, 1500 and 2500.

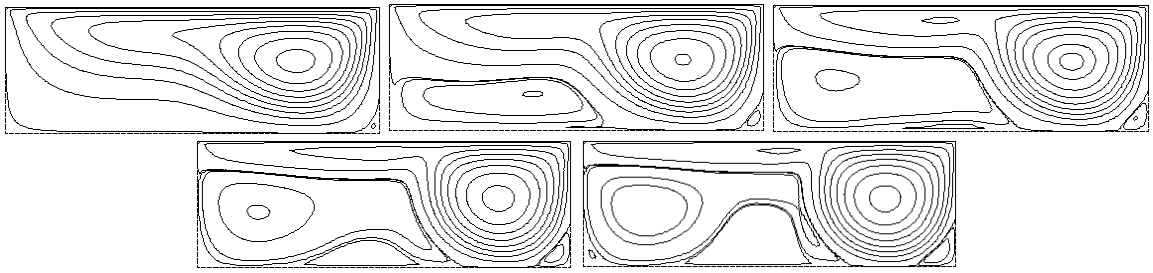


Fig. 5: Streamline contours of 3:1 cavity for Re = 200, 400, 7500, 1500 and 2500.

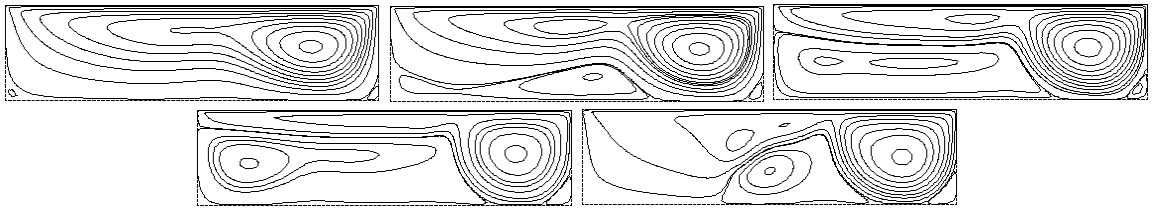


Fig. 6: Streamline contours of 4:1 cavity for Re = 200, 400, 7500, 1500 and 2500.

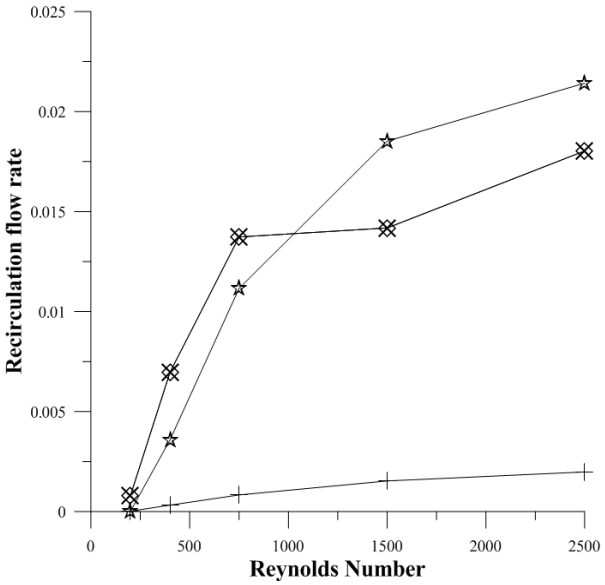


Fig.7 Recirculation flow rate of Fluid inertia (Re)

5. CONCLUSIONS

The dynamic numerical results of steady–state solution of shallow cavity are achieved and visualized the flow features of the schematic diagram of the domain at low and high Reynolds number. The counter–spinning primary vortex declined downwards in the domain 1:1 and move on right of the cavity wall in other ratio geometries. In the ratio 1:1, the downstream vortex is bigger than upstream stream vortex is visualized. Consequently with increasing ratio of the domain, the flow features are altered as the upstream vortex is bigger than downstream vortex are envisioned at low and high Reynolds number. The vortex intensity in the function of Reynolds number is also developed of various ratio geometries.

ACKNOWLEDGEMENT

Authors greatly acknowledge with thanks to the Mehran University of Engineering and Technology, Jamshoro and Department of Mathematics, Shah Abdul Latif University, Khairpur (Mir's), Sindh.

REFERENCES:

Baloch, A., P. Townsend, and M. F. Webster, (1994) 'On two– and three–dimensional expansion Flows', Computers & Fluids Vol. **(24):** No, 8, 863–882.

Burggraf O. R. (1966) "Analytical and numerical studies of the structure of steady separated flows. Journal of Fluid Mechanics; 24 **(1):**113–151.

Donea, J., (1984) 'A Taylor–Galerkin method for convective transport problems', International Journal for Numerical Methods in Engineering. Vol. **(20):** 101-119, Willey online Library.

Erturk, E., (2009) "Discussions on Driven Cavity Flow", International Journal for Numerical Methods in Fluids, Vol. (**60):** 275-294.

Fletcher, C.A.J. (1991) "Computational Techniques for Fluid Dynamics" (2nd edn.) Springer-Verlag. New York.

Gatski T. B., C. E. Grosch M.E. Rose (1982) "A numerical study of the two-dimensional Navier–Stokes equations in vorticity–velocity variables", Journal of Computational Physics; 48 **(1):** 1–22.

Ghia U., K. N. Ghia and C. T. Shin (1982) "High-Re solutions for incompressible flow using the Navier–Stokes equations and a multigrid method", Journal of Computational Phsics; 48 **(3):** 387–411.

Gustafson K., and K. Halasi (1986), "Vortex dynamics of cavity flows", Journal of Computational Physics; 64 **(2):** 279–319.

Hawken, D. M., H. R. Tamaddon–Jahromi, P. Townsend, and M. F. Webster, (1990) 'A Taylor–Galerkin–based algorithm for viscous incompressible flow', Int J Num Meth Fluids, Vol. **(10):** 327–35.

Mussa, M. A., S. Abdullah, C. S. Norazwadi, N. Muhamad, and K. Sopian, (2008) "Numerical Simulation of Lid-Driven Cavity Flow Using the Lattice. Boltzmann Method" Math'08 Proceedings of the 13th World Scientific and Engineering Academy and Society (WSEAS) 236-240 Stevens Point, Wisconsin, USA.

Sahin, M. and R. G. Owens, (2003) "A Novel Fully-Implicit Finite Volume Method Applied to the Lid-Driven Cavity Flow Problem. Part II. Linear Stability Analysis", International Journal for Numerical Methods in Fluids 42, 79-88.

Townsend, P. and M. F. Webster, (1987) 'An algorithm for the three–dimensional transient simulation of non–Newtonian fluid flows', In Proc of Numeta Conf, numerical methods in Engg: theory and applications, NUMETA 87, Vol. **(2):** T12/1–I 1.

Van Kan, J., (1986) 'A second–order accurate pressure–correction scheme for viscous incompressible flow', SIAM J Scientific Statistical Computing, Vol. **(7):** 3, 870Pp.