



Computer Simulation of Lid Driven Deep Cavity Flow

Z. A. K. BALOCH, M. E. BALOCH*, R. A. MEMON**, H. SHAIKH****

Department of Petroleum Engineering, University of Stavanger, Norway.

Received 13th July 2014 and Revised 28nd August 2014

Abstract: A numerical investigation of lid driven deep cavity flows of incompressible fluid is presented, adopting a finite element method (FEM). The formation of counter-rotating primary vortex, secondary vortex, left corner and right corner vortices are analysed, at various widths to height aspect ratios of cavities and increasing inertia is considered. The impact of corresponding aspect ratios, taken from 1:1, 1:2, 1:3 and 1:4. The effects of the moderate inertial values from $Re = 100$ to 3000 are examined. The influence of both aspects, ratios and inertia on the formation of vortices, intensity and magnitude of the recirculating region are observed. Various number of vortices and their location of centres, and many other flow features in the deep cavities are also investigated. The results are compared with other numerical studies.

Keywords: Cavity Flow, Finite Element Method, Newtonian Fluids, Deep Cavities, Aspect Ratios.

1. INTRODUCTION

The flow through lid-driven cavities of incompressible Newtonian fluid is one of the most interesting research problems for scientists and engineers because many important flow phenomena, such as corner vortices, longitudinal vortices, transition and turbulence all occur in the same closed geometry (Aidun *et al.*, 1991), Alleborn *et al.*, 1999), Baloch, *et al.*, 2013), (Benjamin, and Denny, 1979, Shankar, and Deshpande, 2000). The accurately prediction of formation of eddies, movement of vortices centre, shape of flow structure and pressure drop remain challenge problem and encountered in several industrial fields such as extrusion injection moulding and in polymer rheology (Schreiber, and Keller, 1983), Mehta, and Lavan, 1969), and Shankar, 1993).

While numerical modelling and simulations of fluid flows are generally carried out with Finite Difference Method, Finite Elements Method, Finite Volume Method and etc. Initially, the Finite Difference Method (FDM) is comparatively simple and easy and not required high computations for natural problems, nevertheless the results are not guaranteed. Finally, Finite element method is adopted here.

Flow of lid-driven rectangular cavity for Non-Newtonian fluids was presented by Cheng, and Hung, (2006). A model for both square and shallow cavities in the ratio 4:1 was developed. For non-Newtonian fluids, when Power Law index decreases, at initial critical

inertia increases form shear-thickening to shear thinning fluids. The Carreau model was also selected to develop the model for Non-Newtonian fluids and observed the rheological constraints in different ranges. The numerical predictions were achieved by using finite difference scheme of second order and compared with other numerical solutions.

The study of fluid flow through deep and as well as shallow cavities are also suggested by Haque, (2012) and employing Lattice Boltzmann method. The influence of inertia ($Re = 100$ to 1000) is predicted with changing various aspect ratios (0.25, 0.5, 1.5, 4) on the flow and different features were described broadly. The mechanism of the initial counter-swirling vortex, secondary vortex below the initial vortex, left and right salient corner vortices are developed. The results of LBM were compared with computational fluid dynamics package Fluent.

After detail study of the literature, the flow through the deep cavity is the one of the most important problem that has stress wall boundaries nearby the domain such as entry flow geometry is particularly important due to presence of singularities at re-entrant corner and development of vortices. The isothermal Newtonian flow is considered through the deep cavity which is driven by the constant velocity of the moving upper top lid. The analysis on vortex development and pressure is presented adopting finite element method (FEM).

++Corresponding Author: H. SHAIKH , email: hisam.shaikh@salu.edu.pk

*Department of Electrical Engineering and Information Technology. University of Duisburg-Essen, Germany.

** Department of Basic Sciences and Related studies, Mehran UET, Shaheed Z. A. Bhutto Campus Khairpur

*** Department of Mathematics, Shah Abdul Latif University, Khairpur (Mir's), Sindh, Pakistan.

2. MATERIAL AND METHODS

Problem Definition

The incompressible Newtonian fluid flow through deep lid-driven cavity of aspect ratio 1:1, 1:2, 1:3 and 1:4 are considered. Four different deep cavities and recirculation flow rate are presented. In two dimensional domains, total number of elements, total number of nodes and degree of freedom are described and triangular elements are chosen for all domains. Fully developed boundary conditions are expressed to gratify the Dirchlet boundary conditions and no-slip boundary conditions that are predetermined on the boundary walls (Fig.1AND 2).

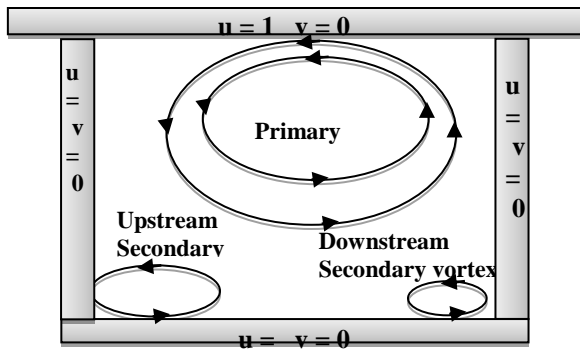


Fig.1:Schematic diagram of lid-driven cavity.

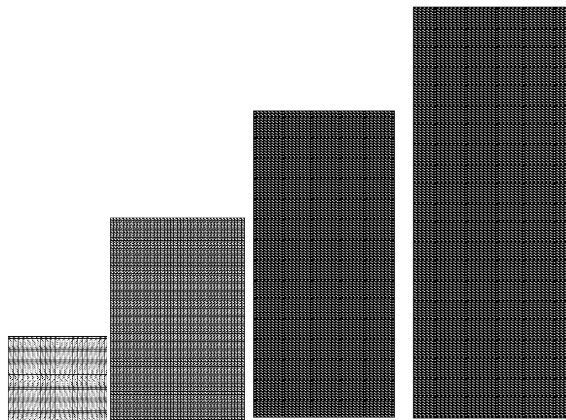


Fig.2: Finite Element meshes in the aspect ratio 1:1, 1:2, 1:3 and 1:4.

Governing system of equations and numerical scheme

For isothermal flows of incompressible fluids in the absence of body force can be modelled through the conservation laws which are expressed by the following vector notation:

1. Conservation of mass: $\nabla \cdot \mathbf{v} = 0$, where ∇ denotes the divergence operator and \mathbf{v} is the velocity vector field.
2. Momentum conservation

law: $\rho \left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = \nabla \cdot (2\mu \mathbf{D} - p\delta)$, where p is the isotropic pressure, ρ is the fluid density per unit mass, δ is the

Kroneker delta (identity tensor), t is the time, μ is the Newtonian fluid viscosity, $\mathbf{D} = \frac{1}{2}(\nabla \mathbf{v} + \nabla \mathbf{v}^T)$ is the rate-of-strain tensor, T is the transpose.

A fractional-staged finite element scheme is employed, for time derivative that discretises initially in Taylor series. To achieve second order accuracy in time employs a two-step Lax-Wendroff approach is adopted. Taylor-Galerkin approximation used for primitive variables, initially proposed by Donea, (1984) for transient solution of viscous incompressible flows in combination of pressure-correction method (Van 1986)) which plays a vital role in accuracy and stability. Semi-implicit form of this scheme was developed by various researchers (Baloch, Townsend, and Webster, (1994), and Townsend, and Webster, 1987) to capture complex flow phenomena.

This scheme was applied by (Townsend, and Webster, 1987 Baloch, 1994), (Baloch, 2013) for different complex flows and predicted the steady-state solution of Newtonian and non-Newtonian fluids. The same scheme is employed here to achieve the steady-state solution of flow through deep cavities in the various ratios of width and height 1:1, 1:2, 1:3 and 1:4. The numerical results are compared against to ref. (Mussa *et al.*, 2008 and Haque *et al.*, 2012).

3. RESULTS AND DISCUSSION

The impact of fluid inertia:

The semi-implicit finite element scheme is adopted to analyse fluid flow through deep cavities with characteristic ratios 1:1, 1:2, 1:3 and 1:4. The thirteen streamline contours are described in all geometries. Numerical results are presented with increasing Inertia (Re) from 100 to 3000. The main counter-whirling flow at centre, secondary vortex flow, left silent corner and right silent corner vortex are visualized in (Fig.3 to 6). For square cavity in figure-3, at low Inertia (Re =100), a counter-whirling vortex is developed right at the bottom. With increasing Inertia (Re) to 500, the centre of vortex is starting to proceed beneath the driven lid. As we increase the depth of cavity to ratio 1:2 in Fig.4, the formation of vortex in the flow structure is comparatively different to square cavity. In this case the secondary eddy is developed right beneath the primary vortex at low Inertia. As Inertia is increased the secondary eddy starts to strengthen. In figure-5 for ratio 1:3, the formation of tertiary eddy is observed right beneath the secondary eddy at high Inertia. Similarly in figure-6 for ratio 1:4 tertiary eddy is observed at moderate (Re) i.e., (Re = 1000) and continues to strengthen as (Re) is increased and also another small vortex is observed in the vicinity of downstream close to the top moving surface. The numerical results are

compared with the numerical solutions of (Mussa *et. al.*, 2008) and other numerical results available in literature, and realised in close agreement.

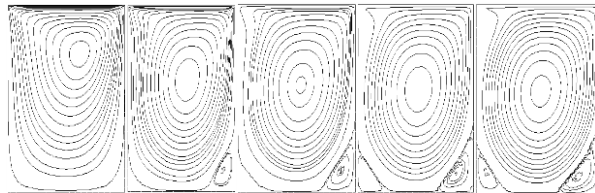


Fig. 3: Streamline contours of 1:1 cavity in favour of Re varies from 100 to 3000.

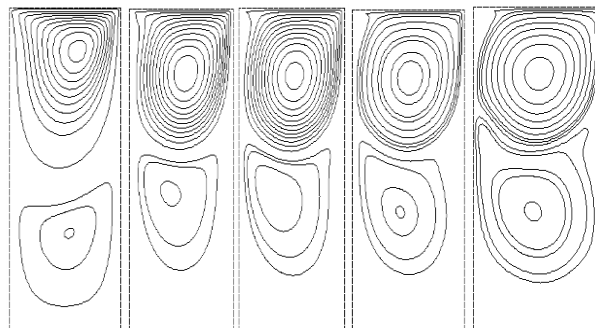


Fig. 4: Streamline contours of 1:2 cavity in favour of Re varies from 100 to 3000.

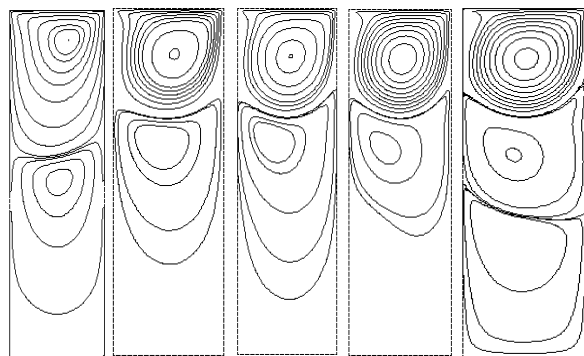


Fig. 5: Streamline contours of 1:3 cavity in favour of Re varies from 100 to 3000.

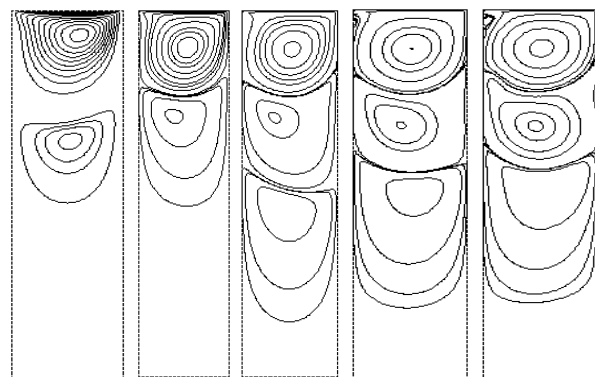


Fig. 6: Streamline contours of 1:4 cavity in favour of Re varies from 100 to 3000.

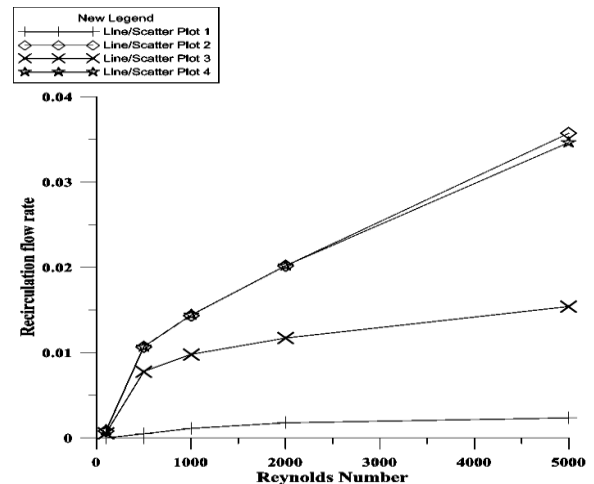


Fig. 7: Recirculation flow rate in the function of Inertia

The recirculation flow rates are also displayed in (Fig.7) and it is clearly observed that the recirculation flow rates are monotonically increased with increasing Inertia and with increasing corresponding ratio of the geometry the recirculation flow rates are grown, which are displayed in Fig.7. The numerical predictions are compared with (Mussa *et. al.*, 2008) and other numerical results available in literature and good numerical results are achieved.

4. CONCLUSIONS

The flow in lid driven deep cavities is simulated for an incompressible Newtonian fluid. The numerical predictions are computed adopting semi-implicit Taylor-Galerkin/pressure-correction scheme.

At low level of inertia, in square cavity the primary vortex is appear in the centre of the cavity nearby the lid. With increasing inertia the centre of vortex shift below towards the centre and two embryo vortices are developed in the bottom of the cavity on both left and right silent corner. The vortex on the right corner strengthen further at high inertial value. Whilst, increasing the depth of the cavity, the numbers of vortices appears with increasing the ratio of cavity and inertia. That clearly demonstrate the numerical results are influenced by both the depth of cavity and fluid inertia on the formation of number, intensity and size of vortices. Numerical predictions are in good agreement with other numerical results. In all cavities, the recirculation flow rate is significantly increasing linearly with enhancement of inertial level.

5. ACKNOWLEDGEMENT

Authors greatly acknowledge with thanks to Department of Mathematics, Shah Abdul Latif University, Khairpur (Mir's), Sindh.

REFERENCES:

- Aidun, C. K., N. G. Triantafillopoulos, and J. D. Benson, (1991), 'Global Stability of a Lid-Driven Cavity with Through Flow: Flow Visualization Studies', *Phys. and Fluids*, Vol. 3, 2081–91.
- Alleborn, N., H. Raszillier, and F. Durst, (1999), 'Lid-Driven Cavity with Heat and Mass Transport', *Int. J. Heat Mass Transfer*, Vol. 42, 833–53.
- Baloch, Z. A. K., M. E. Baloch, H. Shaikh, and M. A. Shar, (2013), 'Computer Simulation of Lid Driven shallow Cavity Flows: Effects of Fluid Inertia and Aspect Ratios', *SURJ*, Vol. 45, no. 2, 277–280.
- Benjamin, A. S. and V. E. Denny, (1979), 'On the Convergence of Numerical Solutions for Two-dimensional Flows a Cavity at Large Re', *J. Comput. Phys.*, Vol. 33, 340–58.
- Cheng, M. and K. C. Hung, (2006), 'Vortex Structure of Steady Flow in a Rectangular Cavity', *Computers and Fluids*, Vol. 35, 1046–1062.
- Haq, S., I. Lashgari, F. Giannetti, and L. Brandt, (2012), "Stability of Fluids with Shear-Dependent Viscosity in the Lid-Driven Cavity", *Journal of Non-Newtonian Fluid Mechanics Elsevier*, Vol. 173–174, 49–61.
- Mussa, M. A. S., Abdullah, C. S. Norazwadi, N. Muhammad, and K. Sopian, (2008), 'Numerical Simulation of Lid-Driven Cavity Flow Using the Lattice Boltzmann Method', *Proceedings of the 13th WSEAS International Conference on Applied Mathematics*. Malaysia.
- Peng, Y. F., Y. H. Shiau, and R.R. Hwang, (2003), 'Transition in a 2-D Lid-Driven Cavity Flow', *Comput. and Fluids*, Vol. 32, 337–52.
- Shankar, P. N. and M. D. Deshpande, (2000), 'Fluid Mechanics in the Driven Cavity', *Ann Rev Fluid Mechanics*, Vol. 32, 93–136.
- Zdanski, P. S. B., M. A. Ortega, Jr. Fico, and G. C. R., Nide, (2003), 'Numerical Study of the Flow over Shallow Cavities', *Comput. & Fluids*, Vol. 32, 953–74.
- Schreiber, R. and H. B., Keller, (1983), 'Spurious Solutions in Driven Cavity Calculations', *J. Comput. and Phys.*, Vol. 49, 165–72.
- Mehta, U. B. and Z. Lavan, (1969), 'Flow in a Two-dimensional Channel with a Rectangular Cavity', *J. Appl. Mech.*, Vol. 36, 897–901.
- Shen, C. and J. M. Floryan, (1985), 'Low Reynolds Number Flow over Cavities', *Phys Fluids* Vol. 28, 3191–202.
- Pan, F. and A. Acrivos, (1967), 'Steady Flows in Rectangular Cavities', *J. Fluid Mechanics*, Vol. 28, 643–55.
- Shankar, P. N., (1993), 'The Eddy Structure in Stokes Flow in a cavity', *J. Fluid Mechanics*, Vol. 250, 371–83.
- Baloch, A., P. Townsend, and M. F. Webster, (1994), 'On Two- and Three-dimensional Expansion Flows', *Computers and Fluids*, Vol. 24, No. 8, 863–882.
- 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 Eng. Theory and Applications, NUMETA 87*, Vol. 2, T12/1–I, 1.
- Donea, J., (1984), 'A Taylor-Galerkin method for convective transport problems', *Int. J. Num. Meth. And Eng.*, Vol. 20, 101Pp.
- Van K. J., (1986), 'A Second-Order Accurate Pressure-Correction Scheme for Viscous Incompressible Flow', *SIAM J. Sci. Stat. Comput.*, Vol. 7(3), 870–872.