

Numerical Simulations of Shear Driven Square and Triangular Cavity by Using Lattice Boltzmann Scheme

A. M. Fudhail, N. A. C. Sidik, M. Z. M. Rody, H. M. Zahir, and M.T. Musthafah

Abstract—In this paper, fluid flow patterns of steady incompressible flow inside shear driven cavity are studied. The numerical simulations are conducted by using lattice Boltzmann method (LBM) for different Reynolds numbers. In order to simulate the flow, derivation of macroscopic hydrodynamics equations from the continuous Boltzmann equation need to be performed. Then, the numerical results of shear-driven flow inside square and triangular cavity are compared with results found in literature review. Present study found that flow patterns are affected by the geometry of the cavity and the Reynolds numbers used.

Keywords—Lattice Boltzmann method, shear driven cavity, square cavity, triangular cavity.

I. INTRODUCTION

COMPUTATIONAL Fluid Dynamics (CFD) has been extensively used for the analysis of system pertaining to engineering field like fluid flows, heat transfer, chemical reaction, evaporation, condensation [1].

Over the years, fluid flow behaviors inside lid driven cavities have drawn many interested researchers and scientists. Applications of lid driven cavities are in material processing, dynamics of lakes, metal casting, galvanizing and etc [2]. Numerous studies have been carried out on flow patterns inside a cavity. Excellent reviews on lid driven square cavity were done in [3]-[5]. On the other hand, the numerical simulation of driven triangular cavity flow had been conducted in [6]-[8]. These researchers used the conventional CFD method which is by solving the 2-D Navier Stokes equation.

In recent years, there has been a rapid progress in developing the lattice Boltzmann method (LBM) as an efficient alternative way to the conventional CFD methods.

A. M. Fudhail is with Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, 76109 Melaka, Malaysia (phone: +606-3316575; fax: +606-3316511; e-mail: fudhail@utem.edu.my)

N. A. C. Azwadi is with Universiti Teknologi Malaysia, . He is with the Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 83100 Johor.Malaysia (e-mail: azwadi@fkm.utm.my).

M. Z. M. Rody is with Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, 76100 Melaka, Malaysia (e-mail: rody@utem.edu.my)

H. M. Zahir is with Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, 76100 Melaka, Malaysia (e-mail: zahir@utem.edu.my)

M. T. Musthafah is with Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, 76100 Melaka, Malaysia (e-mail: musthafah@utem.edu.my).

The main advantage of LBM is its flexibility in terms of programming and better accuracy in dealing with complicated boundary of geometries [9]. In addition to that, the LBM is also better than the classical CFD in the range of small to moderate Reynolds numbers if dealing with flows in complex geometries [10].

II. LATTICE BOLTZMANN SCHEME

The basic idea of Boltzmann work is that a gas is composed of interacting particles that can be explained by classical mechanics [11]. The mechanics can be very simple where it contains streaming in space and billiard-like collisions interactions [11]. In addition to that, since there are many particles involves, a statistical treatment is needed and is more suitable.

The statistical treatment of a system can be represented in terms of distribution function. This distribution functions $f(x, c, t)dx dc$ is the number of particles which its positions and velocities are dx and dc at time t respectively. Each particle would move from x to $x + c\Delta t$ if there is no collision occurs. Each particle velocity would change from $c + a\Delta t$ in which a is the acceleration due to external forces on a particle at x with a velocity of c . No collision means there is conservation of molecules which can be represented in equations of [12] :

$$f(x + c\Delta t, c + a\Delta t, t + \Delta t)dxdc - f(x, c, t)dxdc = 0 \quad (1)$$

However, if there is collision occurs, the equation represent this particular case is as follow:

$$f(x + c\Delta t, c + a\Delta t, t + \Delta t)dxdc - f(x, c, t)dxdc = \Omega(f)dxdc \quad (2)$$

where $f(x, c, t)$ is the single particle distribution function with discrete velocity of c and $\Omega(f)dxdc$ is the Boltzmann collision operator. It is from this equation that Bhatnagar-Gross-Krook (BGK) collision model was developed and further derived to become BGK - LBM equation.

In general, the discretised version of BGK LBM equation can be written as follow:

$$f_i(\mathbf{x} + \mathbf{c}_i\Delta t, t + \Delta t) - f_i(\mathbf{x}, t) = -\Delta t \left(\frac{f_i - f_i^{eq}}{\tau_f} \right) \quad (3)$$

Where f_i^{eq} is equilibrium distribution function and τ_f is the time relaxation.

The Lattice Boltzmann model with BGK collision operator or BGK model in short, is the classical LB fluid model. This model is most often used to solve the incompressible Navier-Stokes equations.

III. NUMERICAL PROCEDURE

In this study, apart from square cavity geometry, two different types of triangle geometries were also considered. The first type is an isosceles triangle with the 90° being at the top right corner. On the other hand, the second type is an isosceles triangle with 90° being at the top left corner. The value of Reynolds (Re) numbers is varied for each case of cavity geometry.

The simulations of flow of lid driven square cavity are conducted for different range of Reynolds (Re) numbers. The top wall velocity, U was maintained at 0.1 lattice unit per second (lu/s) while the velocity of other three walls which is right, left and below was set to 0 lu/s. The fluid temperature is maintained to be constant (isothermal). The Re numbers are varied from 7500, 9000, and 12500.

On the other hand, the simulation of flow in shear driven triangular cavity is performed for Re number of 1000, 1500, and 2000. The rest of the parameters are equal to the square cavity.

IV. SIMULATION RESULTS

A. Shear Driven Square Cavity

The graph of velocity profile at mid height of each x and y axis were plotted for Reynolds number of 7500. The graph obtained in Fig. 1 for Re number of 7500 were compared with the numerical results in [3]. Good agreements between LBM and in [3] have been found for Re number equal to 7500.

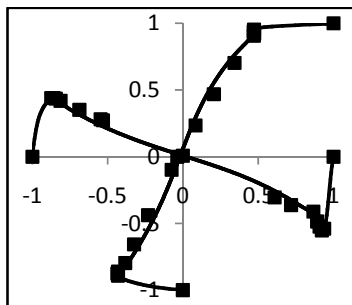


Fig. 1 Velocity profile at mid-height (x -velocity, U_x & y -velocity, U_y) of cavity for Re number 7500. Line-LBM and symbol-Ghia et al[3]

Apart from the numerical results, the streamline patterns were also shown for each Re number. These patterns are plotted when steady state solution is achieved. The streamline patterns are shown as in Fig. 2 to Fig 4. The figures depict that there is addition in terms of the number of secondary vortex when the Re number is increased. As shown in the Fig. 4, the number of secondary vortex is increase from 3 to 4 when the Reynolds number is increased from 9000 to 12500.

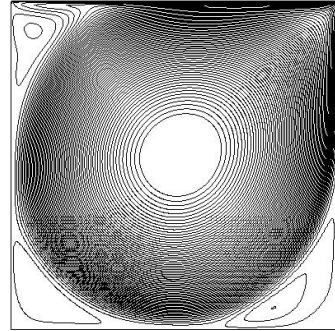


Fig. 2 Streamline patterns for Re number of 7500

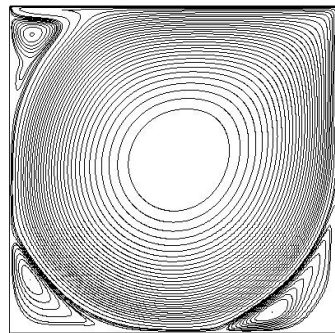


Fig. 3 Streamline patterns for Re number of 9000

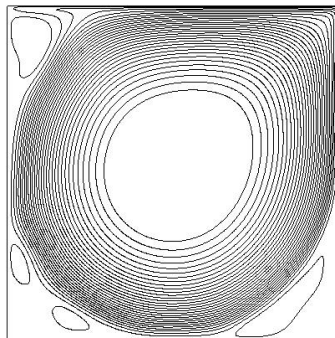


Fig. 4 Streamline patterns for Re number of 12500

The location of the primary vortex is also plotted on a graph as shown in Fig. 5. From the graph, it shows that the primary vortex moves downstream as Re number increases.

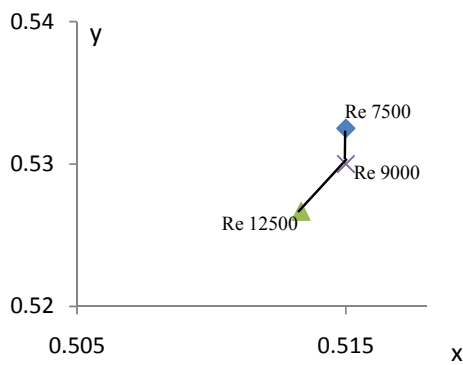
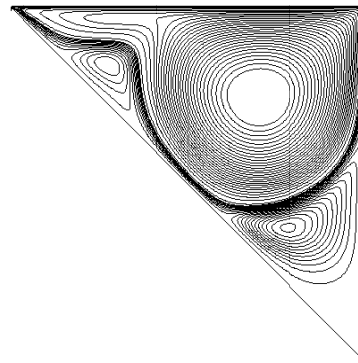


Fig. 5 Graph of the location of primary vortex for respective Re numbers



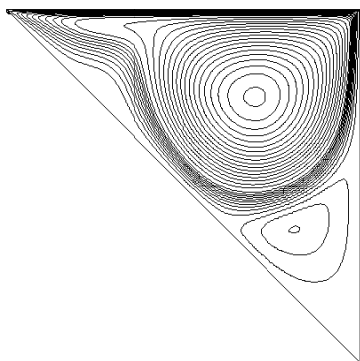
(c) Re 2000

Fig. 6 (a)-(c) Streamline patterns for isosceles triangle with 90° at the top right corner

B. Shear Driven Triangular Cavity

Fig. 6 (a) to (c) shows the streamline patterns for isosceles triangle with 90° at the top right corner. From the figures shown, there are two significant features revealed by the streamline contours. The first feature is that the number of vortices is increased when the Re numbers are increased. As we can see in Fig. 6 (b), the number of vortex is increased from previous which are two to three when the Re number is 1500.

On the other hand, the second significant feature is that the centre of the primary vortex moves upstream to the right as Reynolds number is increased. This can be confirmed by plotting the location of the primary vortex on a graph for the corresponding Reynolds number. The graph is shown in Fig. 7.



(a) Re 1000

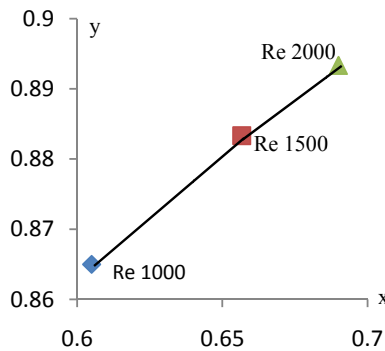
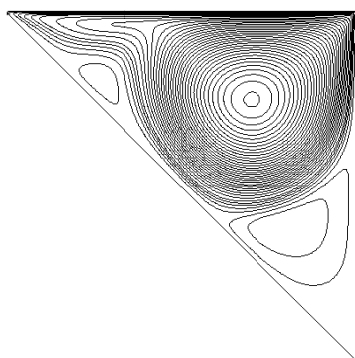
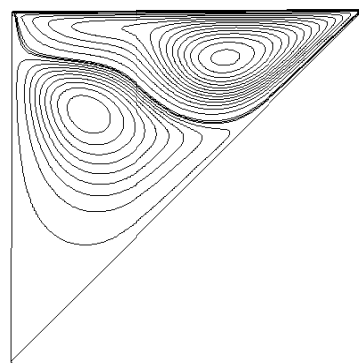


Fig. 7 Graph of the location of primary vortex for isosceles triangle with 90° at the top right corner.

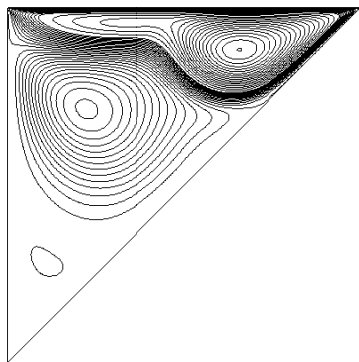


(b) Re 1500

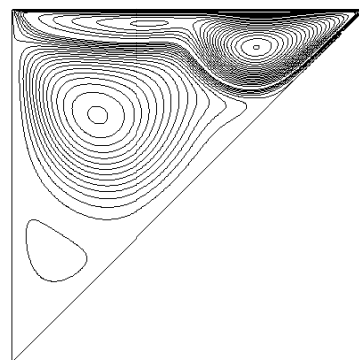
On the other hand, Fig. 8 (a) to (c) shows the streamline contours for the isosceles triangle with 90° at the top left corner. From the figures, it is noticed that the primary vortex moves upstream to the right as Reynolds (Re) number is increased. This is contrary to the results obtained for isosceles triangle with 90° at the top right corner.



(a) Re 1000



(b) Re 1500



(c) Re 2000

Fig. 8 (a)-(c) Streamline patterns for isosceles triangle with 90° at the top left corner.

From these figures, it shows that the number of vortex also increases when the Re number is higher. For instance, when the Re number is increased to 1500, the third vortex emerges in the streamline pattern. It is also noticed that the streamline contour is significantly different from the results for isosceles triangle with 90° at the top right corner although Re number is similar in each cases. Apart from that, as the Re number increases, the primary vortex moves downstream as proven in the graph in Fig. 9.

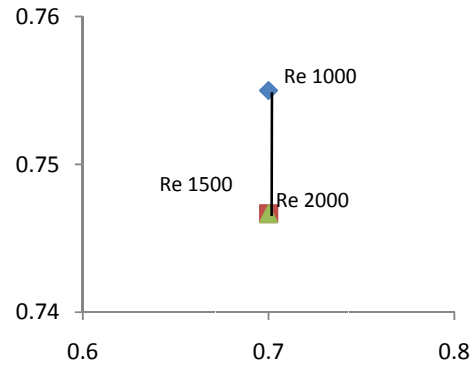


Fig. 9 Graph of the location of primary vortex for isosceles triangle with 90° at the top left corner.

V. CONCLUSION

The capability of LBM to simulate flow in shear driven square and triangular cavity has been demonstrated successfully. Numerical results that are obtained for Re number of 7500 for square cavity are in good agreements with the references.

Apart from that, the number of secondary vortex is affected by the increase of Reynolds numbers for shear driven cavity. In addition to that, the location of the centre of the primary vortex in driven cavity flow is also affected by the Reynolds numbers.

Last but not least, the streamline contours of flow in a cavity are depends on the geometry of the cavity itself.

ACKNOWLEDGMENT

The authors would like to thank the Universiti Teknikal Malaysia Melaka (UTeM), Universiti Teknologi Malaysia and government of Malaysia for supporting this research activity.

REFERENCES

- [1] C.S. N. Azwadi and A.M. Fudhail, "Mesoscale Numerical Prediction of Fluid Flow in a Shear Driven Cavity", unpublished.
- [2] M.A.R. Sharif, "Laminar mixed convection in shallow inclined driven cavities with hot moving lid on top and cooled from bottom", *Applied Thermal Engineering.*, vol. 27, 2006, pp. 1036-1042.
- [3] U. Ghia., K.N. Ghia and C.Y. Shin, "High Re Solutions for incompressible Flow using the Navier-Stokes Equations and a Multigrid Method", *Journal of Computational Physics.*, vol. 48, 1982, pp. 387-411.
- [4] E. Erturk., T.C. Corke and O. Gokcol, "Numerical Solutions of 2-D Steady Incompressible Driven Cavity Flow at High Reynolds Numbers", *International Journal for Numerical Methods in Fluids.*, vol. 48, 2005, pp. 747-774.
- [5] S. Albensoeder and H.C. Kuhlmann, "Accurate three dimensional lid-driven cavity flow", *Journal of Computational Physics.*, vol. 206, 2005, pp. 536-558.
- [6] E. Erturk and O. Gokcol, "Fine grid numerical solutions of triangular cavity flow", *The European Physical Journal Applied Physics*, Vol. 38, 2007, pp. 97-105.

- [7] M. Li and T. Tang, "Steady Viscous Flow in a Triangular Cavity by Efficient Numerical Techniques". *Computers Mathematics Application*, Vol. 31, No 10, 1996, pp. 55-65.
- [8] C.J. Ribbens, L.T. Watson, and C.Y. Wang, "Steady Viscous Flow in a Triangular Cavity", *Journal of Computational Physics*, Vol. 112. 1992.
- [9] G. Poutrelli, S. Ubertin and S. Succi, "The unstructured lattice Boltzmann Method for Non-Newtonian Fluid, vol. 06, 2009.
- [10] O. Filippova and D. Hanel, "Grid Refinement for Lattice-BGK Models", *Journal of Computational Physics*, vol. 147, 1998, pp. 219-228.
- [11] Sukop, M.C. and Thorne, D.T., *Lattice Boltzmann Modeling: An introduction for geoscientists and engineers*, Springer, New York, 2006.
- [12] C.S. N. Azwadi, "The Development of New Thermal Lattice Boltzmann Models for the Simulation of Thermal Fluid Flow Problems. Doctor of Philosophy, Keio University, Japan. 2007.
- [13] C.S. N. Azwadi and T. Tanahashi, "Simplified Thermal Lattice Boltzmann in Incompressible Limit", *International Journal of Modern Physics*, B 20, 2006, pp. 2437-2449.
- [14] C. H. Bruneau, and M. Saad, "The 2D Lid Driven Cavity Problem Revisited ", *Computers and Fluids.*, vol. 35, 2006, pp 326-348.

Fudhail Abdul Munir received the Bachelor of Engineering (2006) in mechanical engineering from International Islamic University of Malaysia. He then obtained his Master degree in Mechanical Engineering from Universiti Teknologi Malaysia in 2009. He is a lecturer attached to Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, Melaka, Malaysia. His current interest is Computational Fluid Dynamics (CFD), heat transfer, vehicle power train and vehicle aerodynamics.

Nor Azwadi Che Sidik received the Bachelor of Science (2001), M. Eng (2003) and Ph.D (2007) degrees in mechanical engineering from Keo University of Japan. He is a Senior Lecturer attached to Faculty of Mechanical Universiti Teknologi Malaysia Johor Malaysia. His current interest is Computational Fluid Dynamics (CFD), heat transfer, fluid structure interaction and nano/micro fluids.

Mohd Rody Mohd Zin received the Bachelor of Engineering (2006) in mechanical engineering from Universiti Teknikal Malaysia Melaka, Malaysia. He then obtained his Master degree in Mechanical Engineering from Universiti Teknologi Malaysia in 2009. He is a lecturer attached to Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, Melaka, Malaysia. His current interest is Computational Fluid Dynamics (CFD) multi phase flow and fluid structure interaction.

Muhammad Zahir Hassan received the Bachelor of Science (2001) in mechanical engineering. He then obtained his Master degree in Mechanical Engineering from Coventry University in 2004. He received his PhD from University of Leed, United Kingdom in 2009. He is a lecturer attached to Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, Melaka, Malaysia. His current interest is vehicle brake system, Computational Fluid Dynamics (CFD), and vehicle power train.

Musthafah Mohd Tahir received the Bachelor of Engineering (1991) in mechanical engineering. He then obtained his Master degree in Mechanical Engineering in 1997. He received his PhD from Nagaoka University of Technology, Japan in 2010. Currently, he is a lecturer attached to Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, Melaka, Malaysia. His current interest is vehicle internal combustion engine, Computational Fluid Dynamics (CFD), and vehicle power train.