2D DEEP CAVITY FLOW, LID DRIVEN USING CONVENTIONAL METHOD WITH NEW PERSPECTIVE

M.S. Idris and N.M.N. M. Ammar

Faculty of Mechanical Engineering
Universiti Malaysia Pahang, Pekan, Pahang, Malaysia
E-mail: idriss@ump.edu.my; ammar@ump.edu.my

ABSTRACT

In this study, conventional FDM with new perspective is applied for simulation of lid-driven flow in a 2-D, rectangular, deep cavity. The code for deep cavity is presented using rectangular cavity 100 x 300, 100 x 200 and 100 x100 with depth ratio of 3,2 and 1 respectively. Results are presented in streamlines pattern for deep cavity flow at steady state for Reynolds numbers of 100, 400 and 1000. Several features of the flow, such as the streamlines pattern, contour of stream function and midsection velocity profile are investigated. Result for deep cavities under steady state shows that the vortex pattern at the bottom enlarge, which thencombine to form a second primary-eddy as the cavity depth-ratio is increased. As the Reynolds number increase, the form of primary vortex and the second primary-vortex then formed through afast transition of an unsteady wall-eddy. Midsection velocity also yields high as Reynolds number increase. The predicted result from FDM with new perspective gives more improvement results as grid mesh increases.

Keywords: Cavity flow, Deep Cavity, lid driven Finite Difference, FDM.

INTRODUCTION

Cavity flow simulation was introduced in early 1980(Ghia et al., 1982; Azwadi and Tanahashi, 2008). During those days, performance of a computer to do simulations was not as superb as it is nowadays. As the performance of the technology of hardware as well as software improves, simulation has become a simple task. However, this technology is only applied for general-purpose engineering software for example FLUENT ©. In this paper, cavity flows were simulated using DNS (Direct Numerical Simulation) which is very unpopular for industrial application.

Nevertheless, DNS is very useful for researching. In this research, a conventional Finite Difference Method (FDM)was implemented to simulate the nature of deep cavity flow with Depth Ratio (D) of 3, 2 and 1.Research publication in the analysis of deep cavity flow has been very little. Yet, in terms of results for deep cavity, the outcome is very good. Results obtain in the simulation using the Lattice Boltzmann method is quite different from the results obtain in thisanalysis (Barragy and Carrey, 1997).

The main concern of this research is to investigate and determine the streamline pattern of deep cavity flow at steady state. It is an uncompleted task if the midsection velocity profile were not included in this research. Another purpose of this research is to improve the understanding of cavity flow especially the connection of vortex, streamfunction contour and streamlines.

GOVERNING EQUATIONS

Basically, the governing equations for 2D deep cavity flow are developed from Navier-Stokes equation and continuity equation. After the implementation of vorticity equation, two main equation which are literally derived from Navier-Stokes equation and continuity equation are as followed (Idris and Azwadi, 2010).

$$\frac{\partial \Omega}{\partial T} + U\frac{\partial \Omega}{\partial X} + V\frac{\partial \Omega}{\partial Y} = \frac{1}{Re}\left(\frac{\partial^2 \Omega}{\partial X^2} + \frac{\partial^2 \Omega}{\partial Y^2}\right) \tag{1}$$

$$\frac{\partial^2 \Psi}{\partial X^2} + \frac{\partial^2 \Psi}{\partial Y^2} = -\Omega \tag{2}$$

The first equation is the vorticity transport equation and the latter equation is the vorticity equation. The prescribed equations are in dimensionless form Tannehil et al. (1984). The Ω represent the vorticity and Ψ is stream function. Re is the Reynolds number of the flow in the cavity and T is the time. Meanwhile to provide midsection velocity profile, the following equation is introduced with U is the velocity in x-direction (horizontal) while V is the velocity in y-direction (vertical).

$$U = \frac{\partial \Psi}{\partial Y} \tag{3}$$

$$V = -\frac{\partial \Psi}{\partial X} \tag{4}$$

GRID GENERATION

Grid generation is important for any kind of simulation. For instance, Finite Element Method usesmesh and shape function that is quite difficult to transfer into numerical codes. Finite Difference however is rather easy whereas it only utilize the grid nodes. In this analysis, three different grid were used which are 100x300,100x200 and 100x100 with each grid responsible for Depth ratio of 3, 2 and 1. Figure 1 below shows the grid.
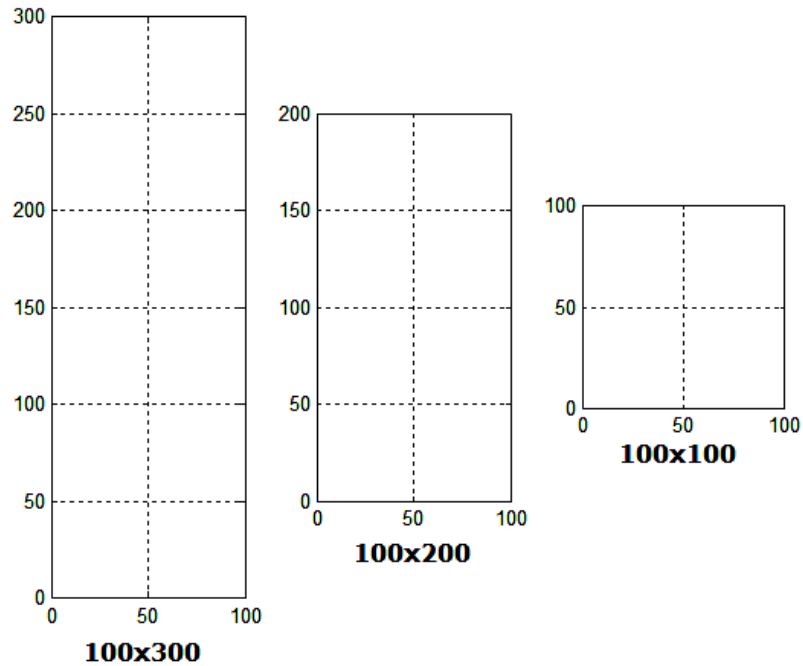
Figure 1: Grid generation for 2D Deep cavity flow

DEEP CAVITY FLOW

The simulation is based on dimensionless parameters. Thus, for the width and the velocity of the top lid is set at unity. The other three walls are at stationary where the streamfunction at this point is zero. For this flow, it is considered as an adiabatic process and the fluid is incompressible anywhere in the cavity. Figure 2 shows the general structure of lid driven deep cavity flow at steady state. Vortex 'a' is the primary vortex and usually only one primary vortex in the cavity flow. Secondary vortex usually located in the middle section of the cavity represented by vortex 'b'. This kind of vortex can be more than one and the direction of rotation is different. If the first secondary vortex is rotating counter clockwise, the next secondary vortex should be rotating in the opposite direction. The last but not least is the tertiary vortex 'c'. Usually two or more of this vortex appear in deep cavity flow. This vortex situated at the bottom corner and the region is small and sometimes it is not so obvious. Reynolds number of 100, 400 and 1000 is used to simulate the flow with dimensionless time increment of (ΔT) of 0.001.
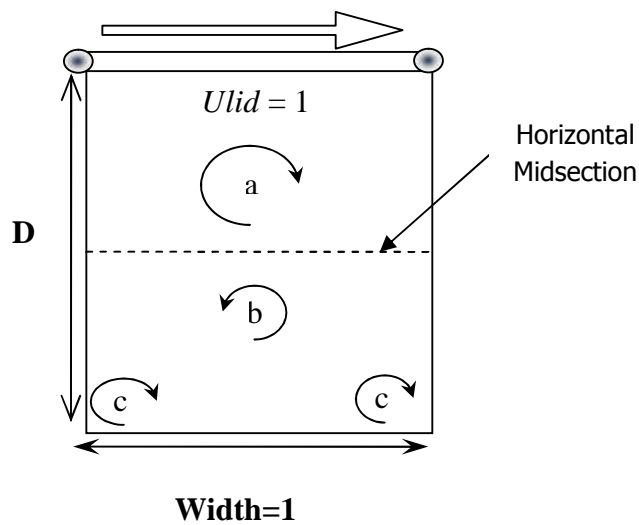
Figure 2: 2D Deep cavity flows with vortex

RESULTS AND DICUSSION

The simulation produces three kinds of result which are the streamlines pattern, contour of streamfunction and midsection velocity profile.
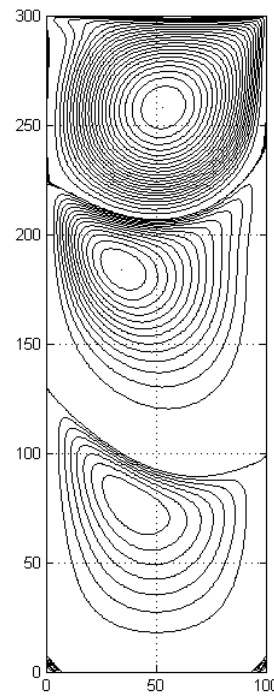
Streamline Pattern and Contour of Stream Function

Accordingto figure 3, the upper figure shows the stream function contour and the lower shows the streamline pattern. As Reynolds number increase, the number of secondary vortex increase to two. For Re = 400, the upper secondary vortex is much bigger from the lower one but for Re = 1000, both secondary vortices shows an equal sense of balance in term of size. For the stream function contour, it is shown that the stream function contour is projected onto the 2D plane and it generates streamlines pattern. The local maximum magnitude of streamfunction (negative peak of the contour) is located at the center of the primary vortex which is shown on the streamline pattern regardless of negative sign. The higher the magnitude of the streamfunction, the power of the vortex is more intense. This is the connection among the center of the vortex, streamline pattern and the contour of stream function.
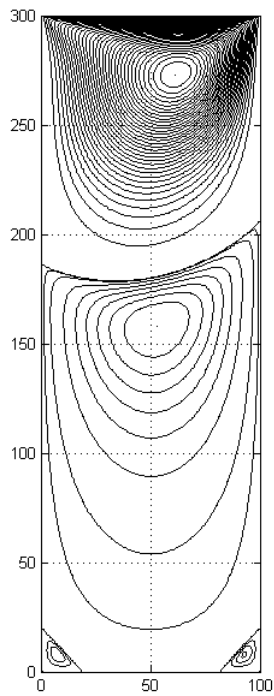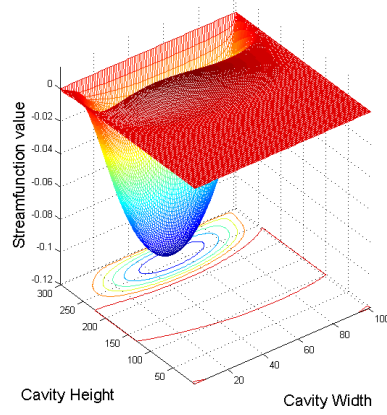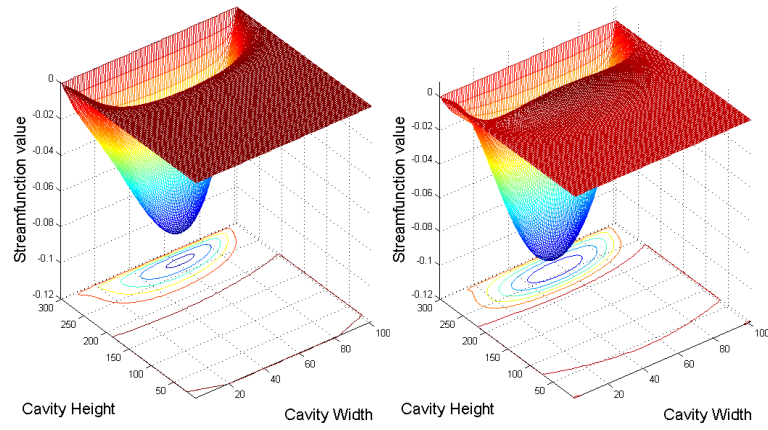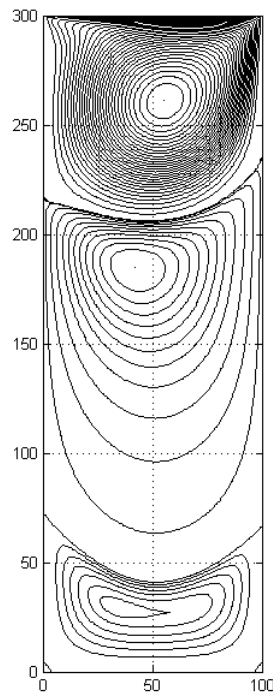
(a) Re = 100   (b) Re = 400   (c) Re = 1000

Figure 3: Contour of Streamfunction and Streamline Pattern for D = 3

Meanwhile, for simulation with Depth ratio of 2 gives different result shown in Figure 4. In this section, for every Reynolds number, each produces only one secondary vortex. For Re = 100, according to the streamline pattern, the size of the secondary vortex is smaller than the primeone, however for Re = 400 and Re = 1000, it is opposite. Still, the primary vortex magnitude is far greater than the secondary vortex regardless of vortex size as the differences of peak magnitude shown in the contour of streamfunction
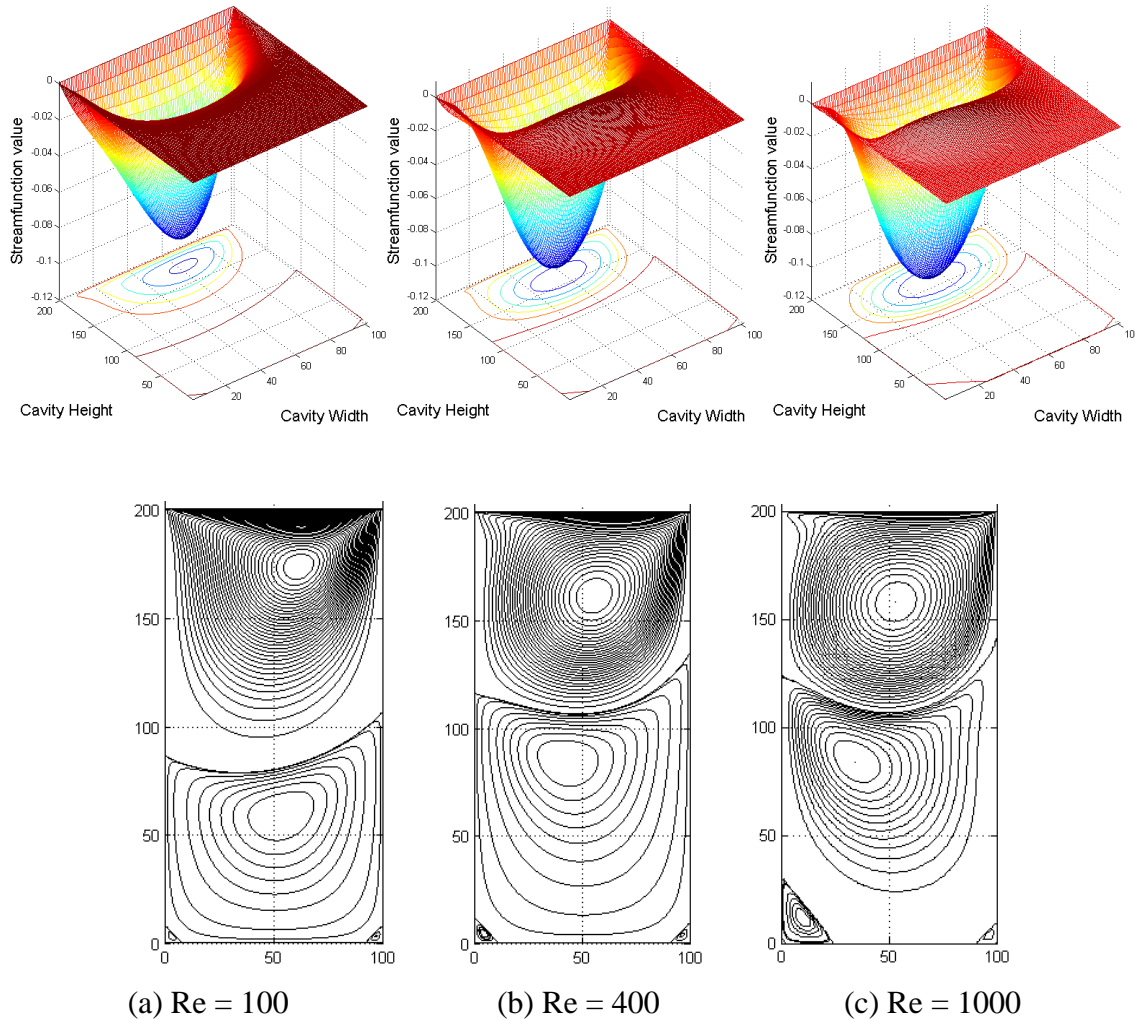

(a) Re = 100   (b) Re = 400   (c) Re = 1000

Figure 4: Contour of Streamfunction and Streamline Pattern for D = 2

On the other hand, for depth ratio of D = 1 which was greatly analyzed by various researchers all over the world show no secondary vortex. Only primary vortex and two tertiary vortices at the bottom corner were shown on figure 5. The only attention from the streamline pattern is as the Reynolds number increase from 100 to 1000, the tertiary vortex increase as well especially at the bottom right corner. For the streamfunction contours, the peak settled near the center of the cavity and applies to every Reynolds number for this particular case.
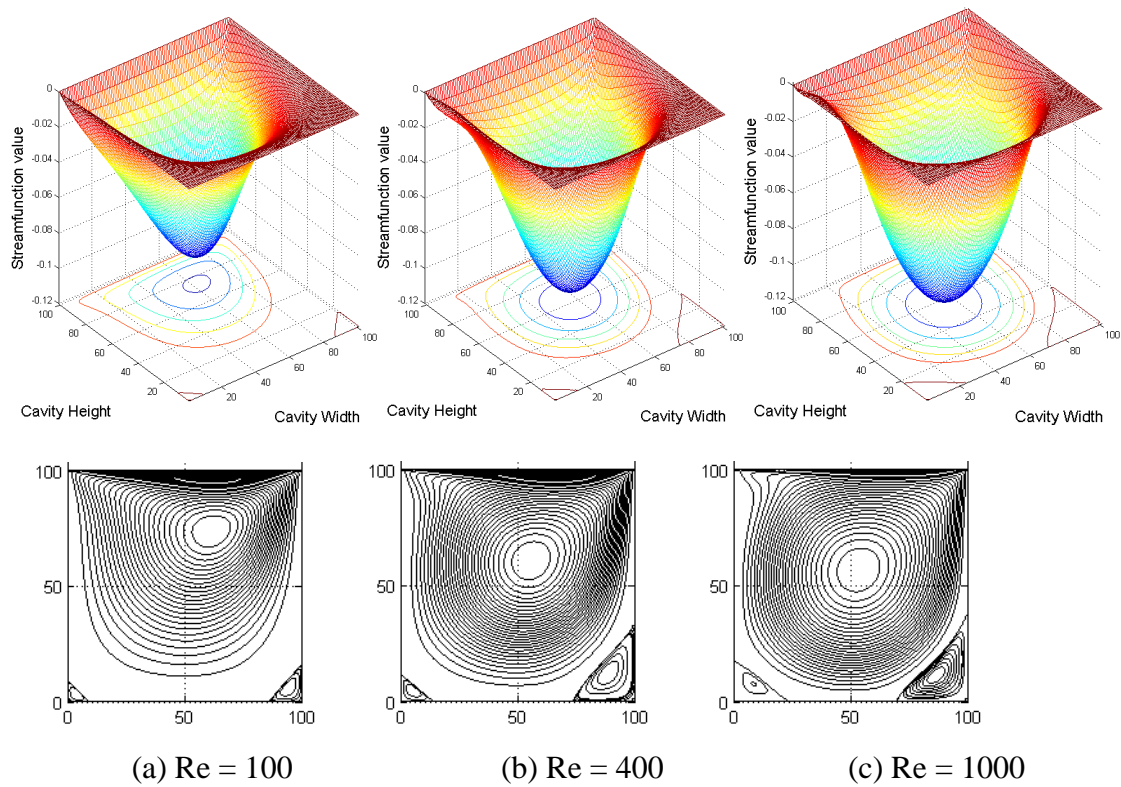
(a) Re = 100   (b) Re = 400   (c) Re = 1000

Figure 5: Contour of Streamfunction and Streamline Pattern for D = 2

Midsection Velocity Profiles

Corresponded to figure 6, velocity profile of horizontal and vertical of cavity for every Reynolds number and Depth ratio is presented. Each graph in figure 6 contains three curves correspond to three Reynolds number as prescribed earlier in this analysis. For each depth ratio, two graphs were plotted which are for horizontal midsection and vertical midsection. For Depth ratio, D = 3, the vertical midsection velocity profile show significant changes of velocity between 2.0 and 3.0 depth from below. This is due to the primary vortex is located in the region. In the meantime, the velocity profile for horizontal midsection show puny changes because, this is the region of secondary vortex regardless of Reynolds number. For Depth ratio, D = 2, the velocity of vertical midsection also show significant result at the upper half of the cavity also because of primary vortex. However, for Re = 100, the curve is slightly higher from the rest as a result of higher location of primary vortex center. At the same time, the horizontal midsection velocity profile show significant result for Re = 400 and 1000. This is mostly caused by the yield of secondary vortex. For Depth ratio, D =1, The midsection velocity profile is same as previous researches [1] and there is no need to further explain of the curves as it is solely govern by the primary vortex .
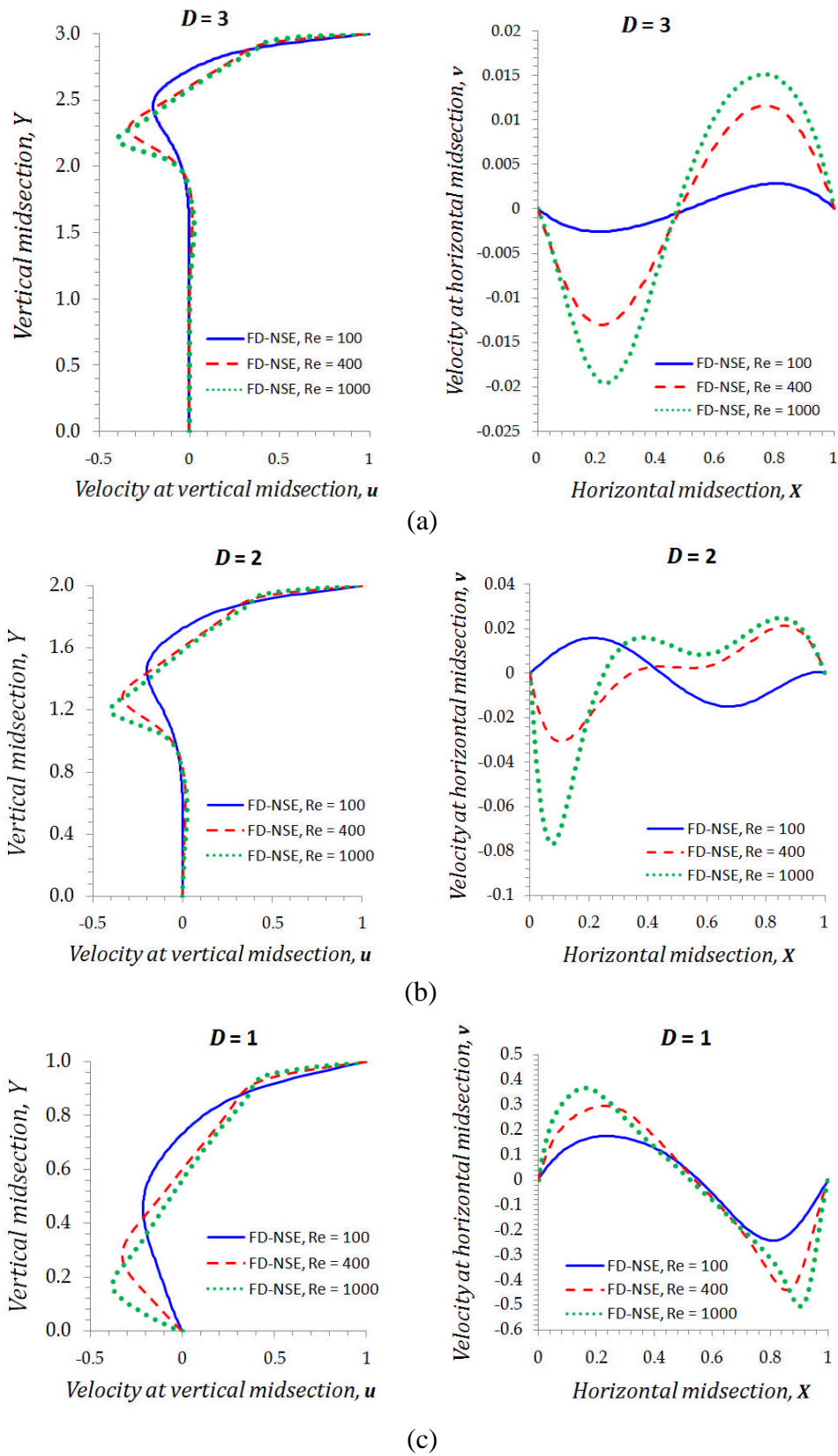
Figure 6: Midsection velocity profile for Depth ratio of (a) 3, (b) 2 and (c) 1.

CONCLUSION

It can be concluded that the simulation of lid driven cavity is mostly successful as the streamline pattern, streamfunction contours and midsection velocity profile were gained and properly analysed. On top of that, further understanding of vortex, streamline and streamfunction is achieved as these three things are being connected by the figures and further explanations.The arrangement of the primary vortex under the top lid is severely affected as Reynolds number increase, but shape is not as much as the secondary eddy due to cavity-depth. The flow-structure near the end bottom-wall reaches the theoretical limiting case of creeping flow, as the cavity-depth increases. Future efforts need to extend these simulations to fully three-dimensional flow, to investigate the effects of boundary conditions in the lateral direction on the flow structure.It is highly recommended that the simulation of deep lid driven cavity flow use higher number of grid and more efficient method in the future.

ACKNOWLEDGMENTS

REFERENCES

Azwadi, C.S.N. and Tanahashi, T. 2008. Simplified Finite Difference thermal Lattice Boltzmann Method.International Journal of Modern Physics. 22: 3865-3876.

Barragy, E. and Carrey G.F. 1997. Stream function Driven Cavity Solution Using p Finite. Computers and Fluids. 26(7): 453-468.

Ghia, U.; Ghia, K.N. and Shin C.T. 1982. High-Re solutions for incompressible flow using the Navier–Stokes equations and a multigrid method.Journal of Computational Physics. 48: 367-411.

M.S.Idris and C.S.N.Azwadi, "Numerical investigation of 2D lid driven cavity flow by the non uniform finite different method" Proceeding, National Conference in Mechanical Engineering for Research and Postgraduate Studies, 26 May, UMP Kuantan.

Migeon, C.; Texier, A. and Pineau, G. 2000. Effects of lid-driven cavity shape on the flow establishment. Journal of Fluids and Structures.14: 469-488.

Tannehil, J.C.; Anderson, D.A. and Pletcher, R.H. 1984.2nd. Computational Fluid Mechanics and Heat Transfer. New York:Taylor& Francis.