National Conference in Mechanical Engineering Research and Postgraduate Students (1st NCMER 2010)
26-27 MAY 2010, FKM Conference Hall, UMP, Kuantan, Pahang, Malaysia; pp. 91-98
ISBN: 978-967-5080-9501 (CD ROM); Editors: M.M. Rahman, M.M. Noor and K. Kadirgama

NUMERICAL INVESTIGATION OF 2D LID DRIVEN CAVITY FLOW EMPHASIZING FINITE DIFFERENCE METHOD OF NON UNIFORM MESHING

M. S. Idris[1] and C. S. N. Azwadi[2]

[1]Faculty of Mechanical Engineering, Universiti Malaysia Pahang,
26300 Kuantan, Pahang, Malaysia,
Email: idriss@ump.edu.my / defog171@yahoo.com
[2]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Skudai, Johor, Malaysia

ABSTRACT

In this paper, 2D lid driven square cavity flow is simulated using finite different method with non uniform meshing at Reynolds number 100, 400 and 1000. Special non uniform Finite different approximation is developed. 50×50 non uniform meshing was used to simulate the streamline patterns, centre of vortex and horizontal and vertical midsection velocity. Convincing results were obtained from the streamline pattern and centre of Vortices for verification of simulation

***Keywords**:* lid-driven, cavity flow, finite different, non uniform mesh

INTRODUCTION

In recent years, the attention of two dimensional lid driven cavity flows has been gradually reduced. Since the infamous discovery by Ghia et al. (Ghia et al, 1982) many research were done either using simulation or by experiment. The former technique however draws enormous interest from various researchers. As a result, many articles regarding the simulation were established and it was compared to the pioneer result of Ghia. Various authors have been validating their results with the pioneer results of Ghia and the outcomes are flawless even though the methods that were applied are different.

Numerous mathematical models have been applied to this flow problem including Finite different method, Finite element method (Barragy and Carrey, 1997), Finite volume method (Albensoeder et al., 2001), Galerkin spectral method (Auteri and Quartapelle, 1999) and Lattice Boltzmann method (Azwadi and Tanahashi, 2008). Finite different Cavity flow simulation using finite different method is widely applied in computer simulation which had established remarkable result. However, very few analyses utilized non uniform meshing especially finite difference method, particularly for the lid cavity flow.

The purpose of this study is to simulate 2D lid driven cavity flow using finite different method with non uniform meshing. The finite different method using non uniform meshing is slightly different from the uniform one. This is due to the spacing of each gap between the two nodes is different. Therefore the finite different approximation of the derivative for non uniform meshing is also different. In addition, the higher the order of the derivative yields more complicated finite different approximation.

GOVERNING EQUATIONS

For 2D lid driven cavity flow, there are few equations that incorporated in the mathematical modelling which are the continuity and the momentum equation for each respective axis. Note that the momentum equation is the incompressible Navier-Stokes equation for 2D. The continuity equation (1) and the momentum equations (2-3) are

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad (1)$$

The momentum equation in x and y direction

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \frac{\mu}{\rho}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \qquad (2)$$

$$\frac{\partial u}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \frac{\mu}{\rho}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \qquad (3)$$

u and v is the velocity in x and y direction, t is time, ρ is density and μ is viscosity. These three equation can be combined with the stream function and vorticity equation which resulting the vorticity transport equation and the vorticity equation. This method is known as Stream function – Vorticity approach (Tannehil, 1984). These equations are:

$$\frac{\partial \Omega}{\partial T} + U\frac{\partial \Omega}{\partial X} + V\frac{\partial \Omega}{\partial Y} = \frac{1}{Re}\left(\frac{\partial^2 \Omega}{\partial X^2} + \frac{\partial^2 \Omega}{\partial Y^2}\right) \qquad (4)$$

$$\frac{\partial^2 \Psi}{\partial X^2} + \frac{\partial^2 \Psi}{\partial Y^2} = -\Omega \qquad (5)$$

and two additional velocity equation;

$$U = \frac{\partial \Psi}{\partial Y} \qquad (6)$$

$$V = -\frac{\partial \Psi}{\partial X} \qquad (7)$$

Ω is vorticity and Ψ is stream function. These variables are in dimensionless form and independent of units.

NON UNIFORM FINITE DIFFERENT APPROXIMATION

Figure 1 illustrates the nodes that incorporated for the approximation of any first order and second order derivative. In this study, finite difference approximation is applied. From here, the approximation for the derivative of non uniform meshing can be derived as followed. For first order derivative, the approximation for central difference:

$$\frac{\partial f}{\partial x} \cong \frac{f(x_{i+1}) - f(x_{i-1})}{h_a + h_b} \tag{8}$$

As for the second order derivative, the central difference approximation is;

$$\frac{\partial^2 f}{\partial x^2} \cong \frac{2f(x_{i+1})}{h_a(h_b + h_a)} - \frac{2}{h_a h_b} f(x_i) + \frac{2f(x_{i-1})}{h_b(h_b + h_a)} \tag{9}$$

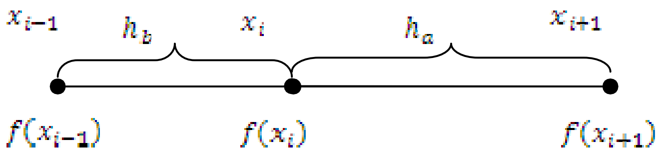


Figure 1: The nodes of non uniform meshing which the grid size is different for each grid

Equation (8) and (9) than applied to equation (4-7) to determine the value of stream function, vorticity and velocity of each node for the non uniform meshing.

GRID GENERATION

The size of each grid is based on the above mathematical expressions. These equations created so that the meshing of the centre is larger than the near-boundary area. Note that $maxi$ is the maximum grid point for the iteration. Each grid in the y direction also applies the similar method to determine grid size and also the value of y. Figure 2 shows exactly the grid size different of the middle area and the near-boundary area. The cavity structure is simulated using 50×50 grids as exposed in Figure 2. This type of non uniform meshing also acknowledged as stretched meshing.

$$h(i) = i \qquad 1 \le i \le fix(maxi/2) \tag{10}$$

$$h(i) = maxi - i \qquad fix(maxi/2) < i \le (maxi - 1) \tag{11}$$

$$x(1) = 0, \quad x(i+1) = \sum_{1}^{i} h(i) \Big/ \sum_{1}^{maxi-1} h(i) \tag{12}$$

The advantage of using non uniform meshing is that any vicinity is capable to be given more attention. It should be an easy task by increasing the number of grid or increasing the grid intensity. Meanwhile, for less important region, the number of grid can be reduced in order to improve the computational time. Figure 2 represents the cavity of the flow. According to the meshing, the grid spacing is smaller as the nodes nearing the boundary however as the nodes approaching the centre of the cavity, the grid spacing expands

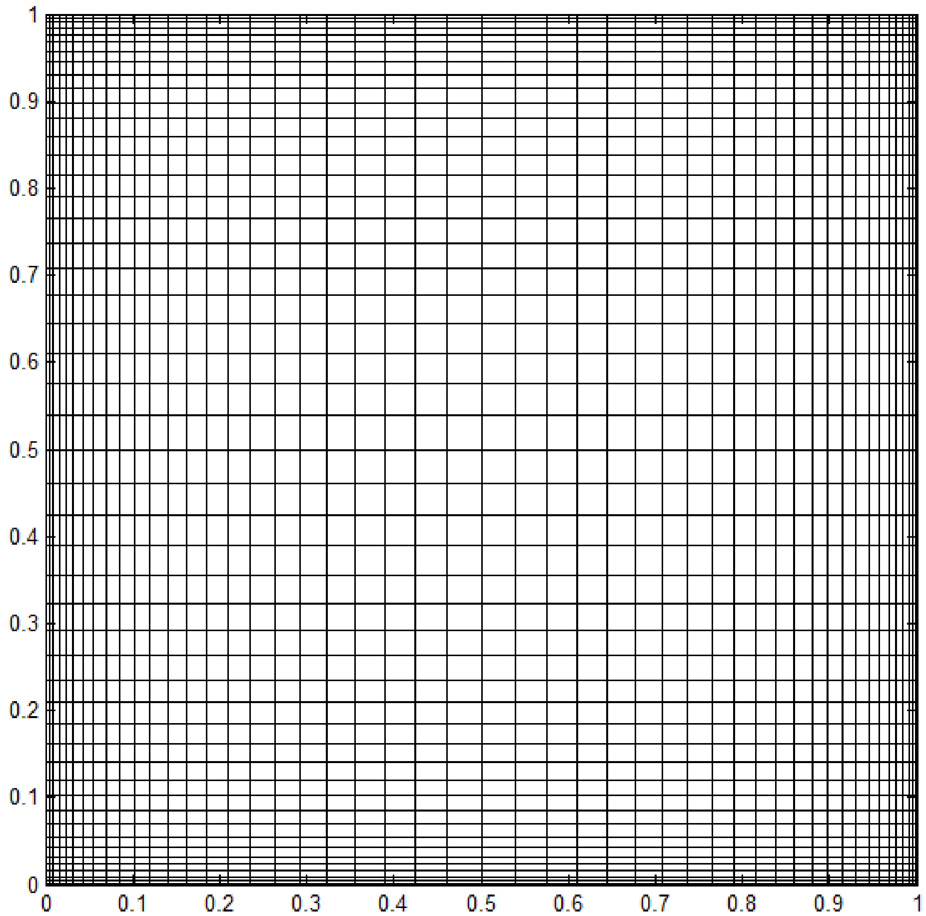


Figure 2: The non uniform meshing used for the flow simulation.

2D SQUARE LID DRIVEN CAVITY FLOW

The structure of 2D lid driven cavity flow is shown in Figure 3. There are four boundaries which are the bottom, left wall, right wall and top. The former three boundary is at stationary while the top or the lid moving with U = 1. Each of these boundaries has its own boundary condition and need treatment. The most important boundary condition is vorticity boundary condition. Previous study shows that vorticity boundary condition give significant influence on the stability of the simulation. (Weinan and Liu, 1996). On the other hand, the physical structure of the cavity itself is in dimensionless parameters where the width and the length is 1. For this simulation, all parameters are in dimensionless for, the flow is considered as incompressible, no heat generation or transfer. The Reynolds number is varied from 100, 400 and 1000. For this particular simulation, 50×50 grid and dimensionless time increment, T of 0.0005 were used.

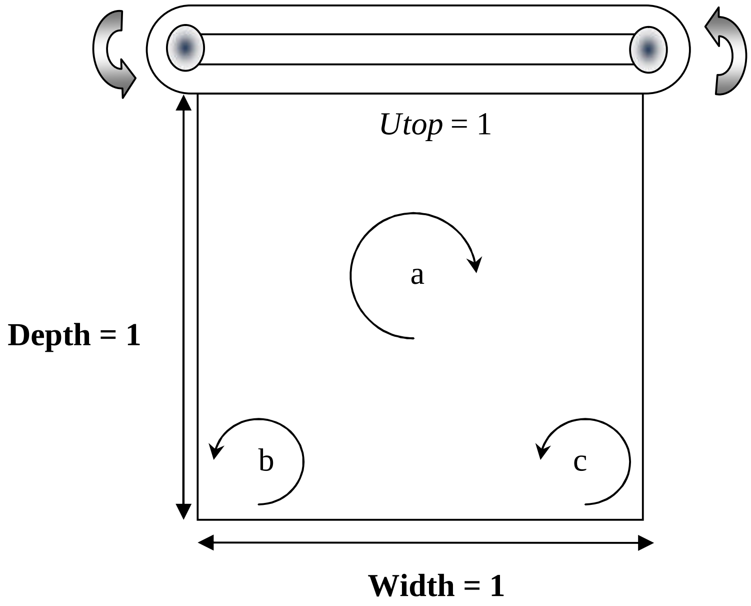


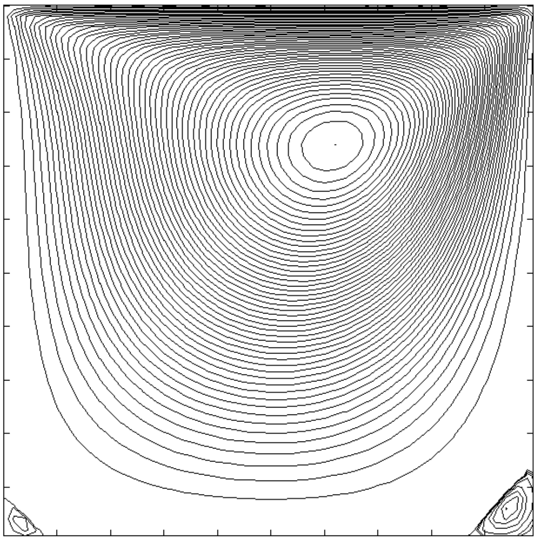
Figure 3: The cavity with driven lid with (a) primary vortex (b) downstream secondary vortex (c) upstream secondary vortex (Migeon et al., 2000)

RESULTS AND DISCUSSION

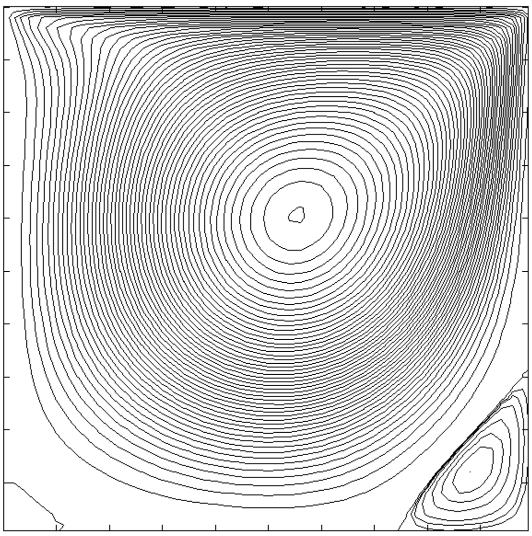
In Figure 4, the results of the simulation using non uniform mesh is presented. From the streamline pattern, there are three vortexes for each Reynolds number. As the Reynolds number increase, the secondary vortex size is also increase. Obviously we can see that the increasing size is very significant for the upstream vortex compare to the downstream vortex. Another major behaviour is the centre of the primary vortex approaching the centre of the cavity as the Reynolds number increase. As additional evidence, from Table 1, it can be noted that the value of the primary vortex centre confirmed such behaviour. Thus, the simulation results are validated as the same condition occurs in Ghia simulation results.

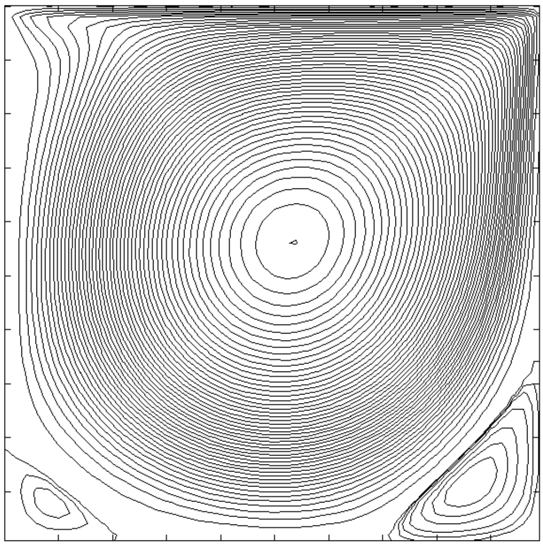
Table 1 Centre of primary vortex and secondary vortex

Re		Primary vortex	Secondary vortex (Downstream)	Secondary vortex (Upstream)
100	x	0.61077	0.032308	0.94462
	y	0.73692	0.032308	0.055385
400	x	0.53846	0.055385	0.88
	y	0.61077	0.043077	0.12
1000	x	0.53846	0.084615	0.86
	y	0.57538	0.084615	0.10154



(a) Re = 100





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

Figure 4: The streamline pattern of non uniform 2D lid driven cavity flow of Re 100, 400 and 1000 (50×50grid). For each streamline pattern, there are three vortex incorporated.

Figure 5 shows the plots of horizontal and vertical midsection velocity for Re 100, 400 and 100 and compared with Ghia. The results do not meet the expectation as the plots deviates against the Ghia plots. This is due to the number of grid used for this simulation is still low compared to Ghia. Thus, in order to replicate the results of Ghia, higher number of Grid point should be applied. However, there are some constraints for the simulation. The major constraint is the stability of the computational method. Due to the usage of Finite different approximation, as the number of grid increase, the time increment should be reduced. This is to compensate the CFL number in order to maintain the stability of the computation.

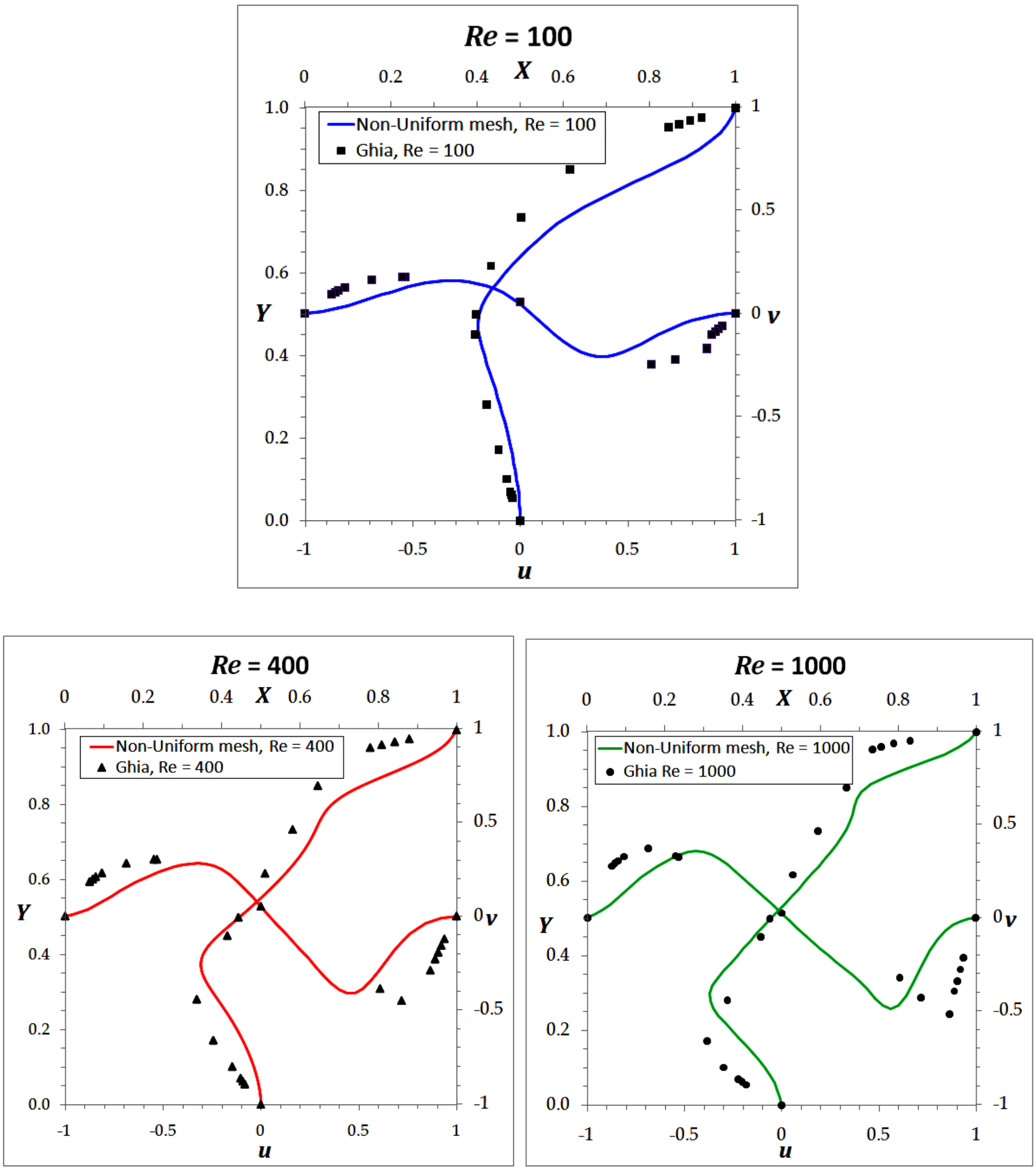


Figure 5: Profile of horizontal midsection velocity (u) and vertical midsection velocity (v) for Reynolds number 100, 400 and 1000 (50x50 grid) compare to Ghia (128x128 grid)

CONCLUSION

From the analysis, numerical simulation using Finite different method for non uniform meshing that applied in 2D lid driven cavity flow is successful. The result of square 2D lid driven cavity flow has good agreement with famous benchmark Ghia especially for the streamline pattern and the vortex center. However, the midsection velocity profile is still lacking and in order to improve the result, higher number of grid should be applied for the simulation. For future recommendation, in order to provide good simulation results, grid independence test should be done using higher number of grid. Also to increase the accuracy, Constrained Interpolated Profile or other kind of method can be applied.

ACKNOWLEDGMENTS

The authors would like to acknowledge Universiti Malaysia Pahang and Universiti Teknologi Malaysia for sponsoring this work.

REFERENCES

Albensoeder,S.; Kuhlmann,H.C. and Rath, H.J. 2001. Multiplicity of SteadyTwo-Dimensional Flows inTwo-Sided Lid-Driven Cavities. Theoretical Computational Fluid Dynamics. 14: 223–241.

Auteri, F. and Quartapelle, L. 1999. Galerkin Spectral Method for the Vorticity and Stream Function Equations. Journal of Computational Physics. 149:306–332.

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.

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.doi:10.1006/j#s.1999.0282

Tannehil, J.C.; Anderson, D.A. and Pletcher, R.H. 1984. 2nd. Computational Fluid Mechanics and Heat Transfer. New York:Taylor & Francis.

Weinan, E. and Liu, J.G. 1996. Vorticity Boundary Condition and Related Issues for Finite Difference Schemes. Journal of Computational Physics. 124:368–382.