TRANSIENT SEMI CIRCULAR LID DRIVEN CAVITY FLOW USING NON UNIFORM STRUCTURED GRID METHOD WITH UPWIND SCHEME

M.S. Idris[1*], N.M.M. Ammar[1],T.M.Y.S. Tuan Ya[1] and A. M. Amin[1]

[1]Faculty of Mechanical Engineering, Universiti Malaysia Pahang (Malaysia),
26600 Pekan, Pahang, Malaysia
*Email: idriss@ump.edu.my
Phone: +609-424-6233; Fax: +609-424-2202
Email: idriss@ump.edu.my

ABSTRACT

In this article, two dimensional lid driven cavity flow in semi circular cavity is simulated using non uniform finite different method with structured grid. Navier-Stokes and continuity equations are simplified using non dimensional streamfunction-vorticity approach. Reynolds number of 1000 is used and the vorticity and Streamfunction contour plot is monitored with convergence criteria of $1\text{x}10^{-7}$ set to both vorticity and streamfunction value. The result shows primary vortex move from the upper left cavity corner to the upper right corner while the magnitude of streamfunction grows at the primary vortex centre. The primary vortex size decrease steadily as the time increased. This phenomenon greatly affected by the increasing size of the secondary at the lower left. Slight changes of vortex size are observed as the flow achieving steady state condition. Validation of the simulation is conducted by comparing previous researches results based on the value of streamfunction and centre location of primary vortex.

***Keywords*:** Cavity flow, Finite different method, Semi-circular, Non uniform grid

INTRODUCTION

There are many applications of lid driven cavity flow considering fluid-particle interaction which is available in industrial application including fluidized beds, pneumatic or slurry transport of dusts, coal combustion, dust explosions and catalytic reactions (Kosinski et al., 2006). There are enormous numbers of articles regarding lid driven cavity flow and one of the most cited article by Ghia et. al. (1982). Patankar, (1980) has also shown great examples for simulation of lid driven cavity flow and have been considered for hundred papers as their main references. There are many other research papers and most of their intention is to use lid driven cavity problem as a test case for new scheme (Zin and Azwadi, 2010; Idris and Azwadi, 2010; Cheng and Hung, 2006; Barragy and Carey, 1997) rather than studying the behaviour of the flow pattern. In lid driven cavity flow, there are many variation of research that have been conducted and published compared to the original condition which is two dimensional square cavities and a driven top lid. The examples of the variation are two sided driven cavity (Albensoeder et. al., 2001) four sided driven cavity (Wahba, 2009), triangular cavity (Erturk and Gokcol, 2007), trapezium cavity (Zhang and Chai, 2010), convection mixed (Azwadi and Idris, 2010) and even semicircular cavity which quite recent for lid driven cavity flow. For semicircular cavity, there are only few research papers and basically the variation is the Reynolds number ranging from 100 to 6600. (Glowinski et. al., 2006

) used finite element method, (Mercan and Antalik, 2009) used skewed grid, (Yang et. al., 2012) using Lattice Boltzmann method and according to the results, it is found that streamline patterns are identical even though the method and scheme used is different. Non uniform grid with finite different method was never been tested before according to the authors knowledge and after thorough research. However, there are already a report based on this method but it is applied to elliptical shaped lid driven cavity flow (Idris and Irwan, 2012).

Thus, in this study non uniform grid and finite different method will be applied to semicircular cavity to obtain validation and once it is completed, the transient behaviour of streamline and vorticity contour will be studied until the steady state is achieved. The location of primary vortex centre will also be considered. The method to be used is the streamfunction vorticity approach with non uniform finite different method.

MATHEMATICAL MODEL

The following equations were used to obtain the simulation results which are the continuity equation and momentum equation in Cartesian coordinate. Cylindrical coordinates will not be applied as it will introduce complexity. Meanwhile, non uniform grid and special modification are used to facilitate curvature boundary at the bottom part of cavity. The streamfunction-vorticity approach is essential as the velocity, *U*, *V* and pressure, *P* is reduced to only two parameters which are streamfunction, *Ψ* and vorticity, *Ω*. These equations then reduced to dimensionless vorticity equation and vorticity transport equation.

Continuity:

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

X-Momentum

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

Y-Momentum

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{\partial P}{\partial y}+\frac{\mu}{\rho}\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right] \quad (3)$$

Dimensionless Vorticity transport equation

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

Dimensionless Vorticity equation in streamfunction

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

Central finite difference for spatial derivative of non uniform grid and first order upwind scheme (explicit) for temporal derivative were implemented to provide simulation of transient behaviour. First and second order derivative of non uniform grid approximation were used and represent by the following equations (Idris and Irwan, 2012) which first discovered by (Veldman and Rinzema, 1992)

$$\frac{\partial f}{\partial x} \cong \frac{f(x_{i+1}) - f(x_{i-1})}{\Delta x_a + \Delta x_b} \tag{6}$$

$$\frac{\partial^2 f}{\partial x^2} \cong \frac{2f(x_{i+1})}{\Delta x_a(\Delta\Delta_a + \Delta x_b)} - \frac{2f(x_i)}{\Delta x_a \Delta x_b} + \frac{2f(x_{i-1})}{\Delta x_b(\Delta\Delta_a + \Delta x_b)} \tag{7}$$

and for the physical formulation, the collocated nodes of non-uniform grid size are presented in Figure 1.

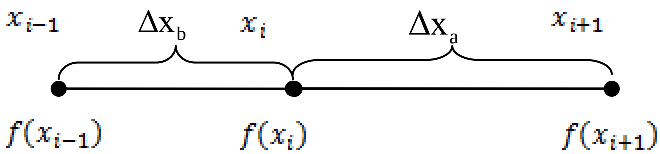

Figure 1. Non uniform grid for collocated node

The dimensionless variables is represent by the following equation

$$U = \frac{u}{U_{Top}}, \quad V = \frac{v}{U_{Top}}, \quad \Psi = \frac{\psi}{U_{top} D}, \quad \Omega = \frac{\omega D}{U_{top}}, \tag{8}$$

$$X = \frac{X}{W}, \quad Y = \frac{D}{W}, \quad T = \frac{U_{top} t}{W}, \quad Re = \frac{\rho U_{top} W}{\mu} \tag{9}$$

GRID GENERATION

The grid used in the simulation is 100 x 50. It is difficult to convey the idea of non uniform grid using the original grids thus in Figure 2, 12 x 6 grids are used to express the idea of providing semicircular cavity. Black dots represent the boundary, white dots represent non calculated or cavity outer part and dashed dots represent the internal nodes where numerical calculation take part or in other words the computational domain. From triangular cavity, it is possible to convert it into semicircular cavity by carefully adjust the grid size of individual rows and columns. The grids are arranged in a manner where at the circle centre, the gaps are bigger and at the end of the radius, the gaps are smaller. Normal finite difference equations is not suitable for this kind of boundary thus to satisfy the boundary, non uniform finite difference is used. For the test case physical system, it is shown in Figure 3 which *a* is the primary vortex and *b* for secondary vortex.

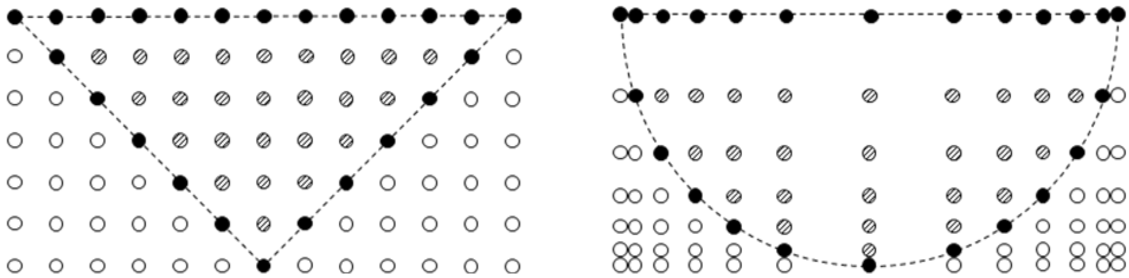

Figure 2. Grid of semicircle

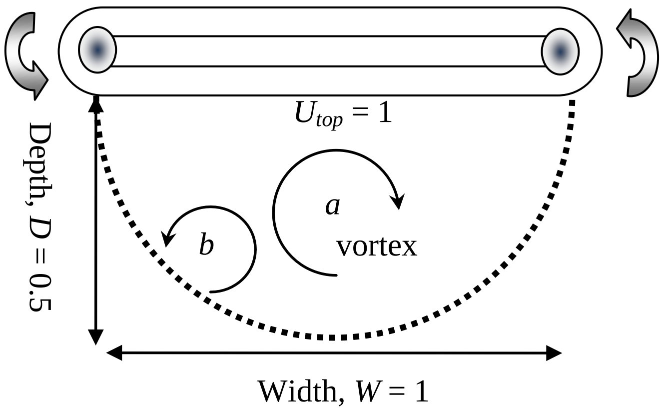

Figure 3. Semicircular lid driven cavity physical test case

The boundary conditions play a major role for the simulation as wrong interpretation of boundary condition will affect the final solution of simulation. For the boundary condition, again, non uniform grid method is applied especially for vorticity boundary condition and top lid *U* velocity.

Table 1. Boundary conditions for semicircular driven cavity flow

Boundary	***U***	***V***	***Ψ***	***Ω***
Top	1	0	0	$\frac{\partial^2\Psi}{\partial X^2}+\frac{\partial^2\Psi}{\partial Y^2}=-\Omega$
Curve	0	0	0	$\frac{\partial^2\Psi}{\partial X^2}+\frac{\partial^2\Psi}{\partial Y^2}=-\Omega$

VALIDATION

In this study, validation was conducted base on the steady state results for Reynolds number 1000 with (Glowinski et. al. 2006) both for highest value of streamfunction value and primary vortex centre location. Table 2 show the validation output and according to the results, it is less than 5% and it is acceptable for the current simulation.

Table 2. Boundary conditions for semicircular driven cavity flow

Result	$\boldsymbol{\Psi}$	$\%\Delta\boldsymbol{\Psi}$	$X_{location}$	$\%\Delta X_{location}$	$Y_{location}$	$\%\Delta Y_{location}$
(Glowinski et, al., 2006)	-0.07800		0.6156		-0.2042	
Present	-0.07586	2.7%	0.6243	1.4%	-0.1986	2.7%

RESULTS AND DISCUSSION

For Reynolds number 1000, to achieve steady state condition, more than 5 millions iteration is required which take about 24 hours using Intel Xeon processor, with two out of twelve cores operates. However in real application it is only about 25 second whereas the dimensionless time increment is only 5×10^{-6}. Small value of time increment is used to prevent error prone condition as advised by Courant Friedrich Lewy especially for explicit numerical methods. Only few results, out of 5 millions iteration are plotted and shown in Figure 5 which is the transient behaviour of streamline pattern. For both Figure 5 and 6, (a) is 0.000415 seconds, (b) is 0.18728 seconds, (c) is 0.78081 seconds, (d) 1.257 seconds, (e) 2.566 seconds, (f) is 5.239 seconds, (g) is 6.14 seconds, (h) is 7.2 seconds, (i) is 8.433 seconds and (j) is 27.7 seconds. At first, the vortex centre is not visible and as the time increase for about one second, the primary vortex centre appears at the top right corner of the cavity. The primary vortex centre then move slowly to the bottom and left section of cavity and the intensity of streamfunction increased. As the time achieving 5 seconds, sign of secondary vortex is appearing and it finally showing itself around 6 seconds of physical time. The secondary vortex steadily grows and occupied some of primary vortex area at the lower left bottom. When the time approaching 25 seconds, the size of secondary vortex gradually increased. As a consequence, the primary vortex decreases in size.

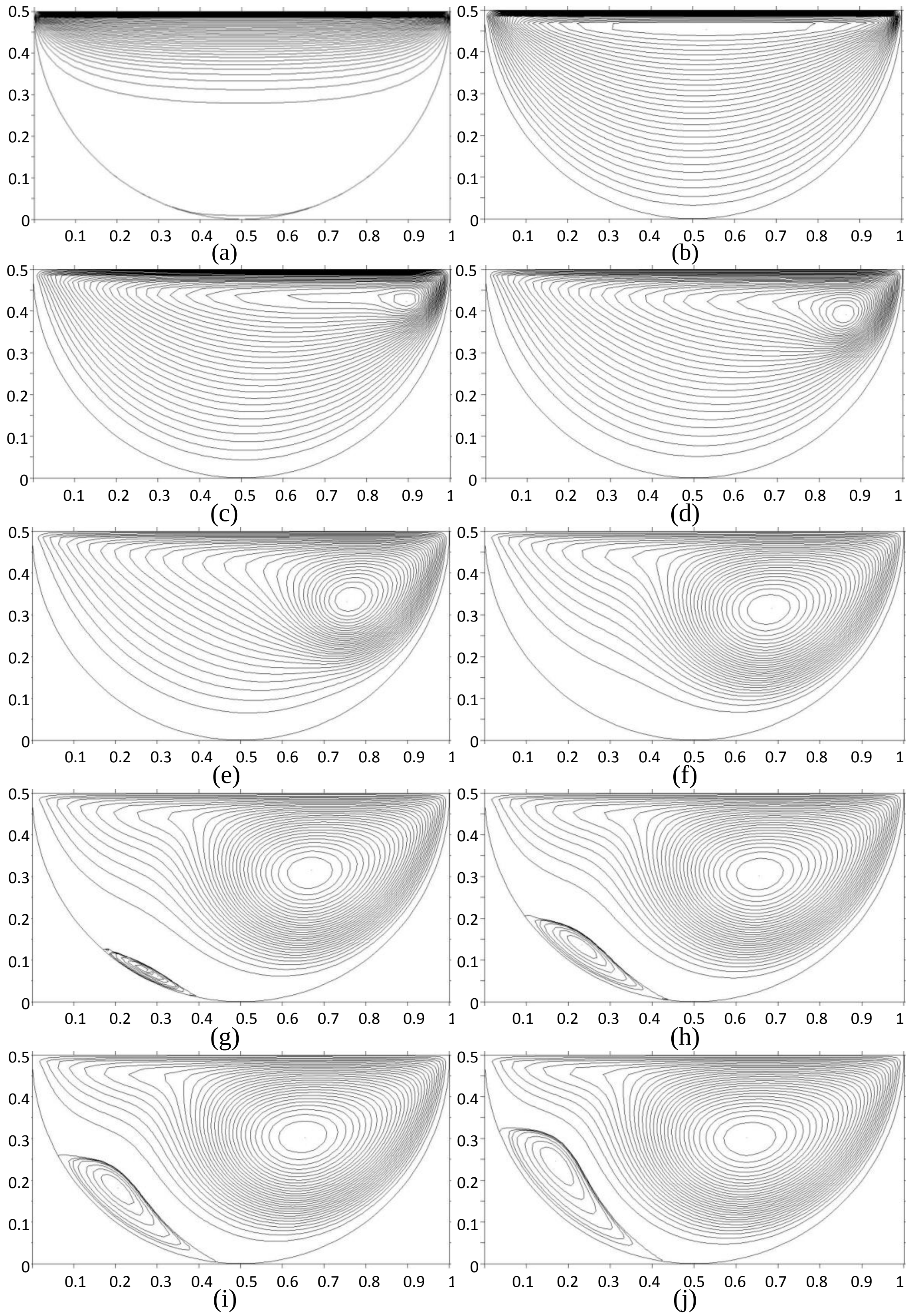

Figure 4. Transition Streamlines pattern for semicircular cavity flow for Reynolds Number 1000 from (a) to (j)

Meanwhile for the vorticity contours shown in Figure 6, rapid changes of pattern appear for physical time between 1 to 8 seconds.

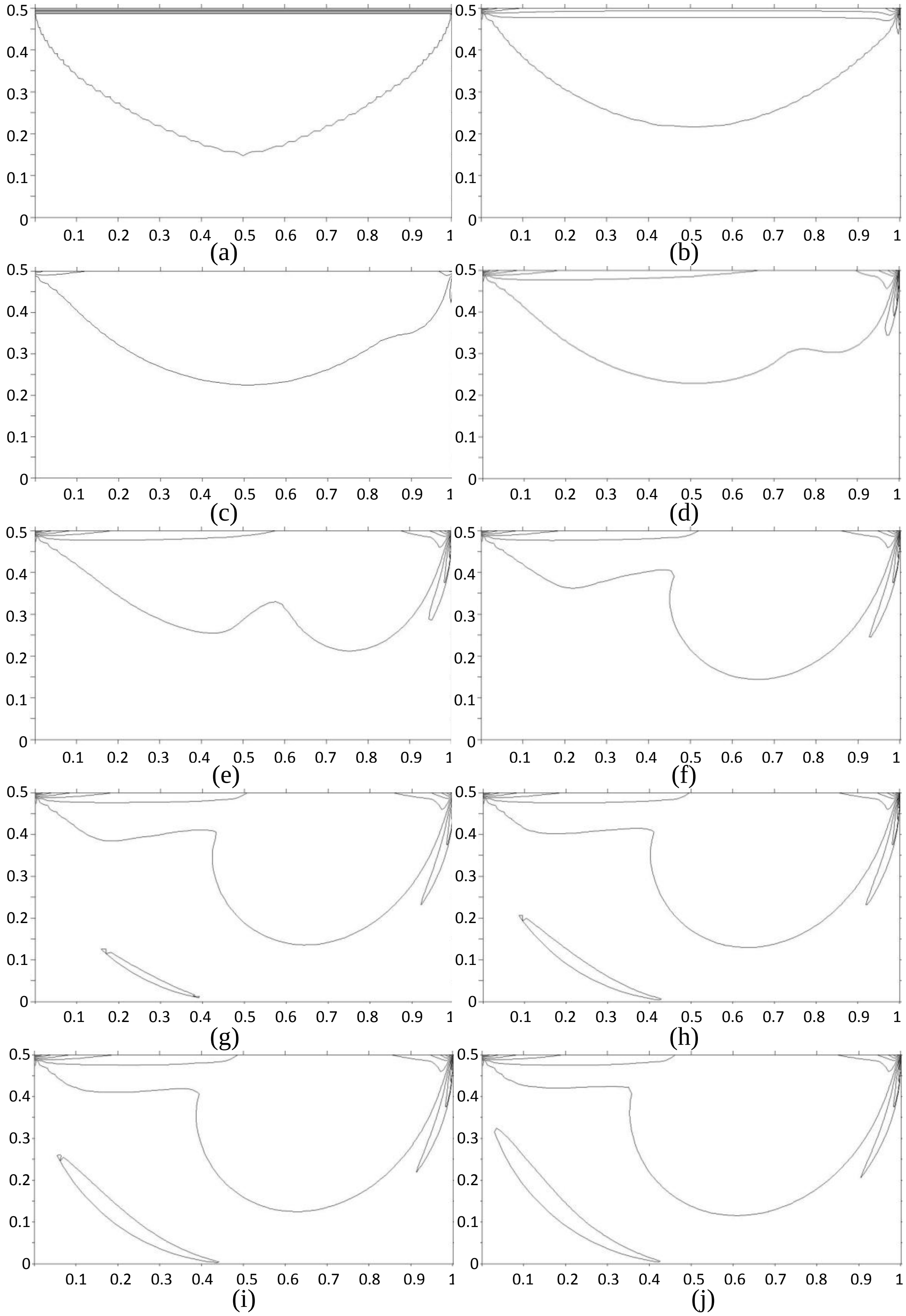

Figure 5. Transient vorticity contour for semicircular cavity flow for Reynolds Number 1000 from (a) to (j)

It is impossible to plots the centre of primary vortex for each iteration ranging from 1 to 5000000 iterations thus few selected results id plotted which is shown in Figure 6. *X*-axis represents the width of the cavity and *Y*-axis represents the height of cavity. The position of primary vortex centre actually moved from the middle top section and move toward the right cavity section. This condition only happens for physical time of around 1 second. As the time increased, the point moving downward and toward the middle section until it settle at 27.7 seconds at location (0.6243,0.3014) when the simulation had achieved the steady state.

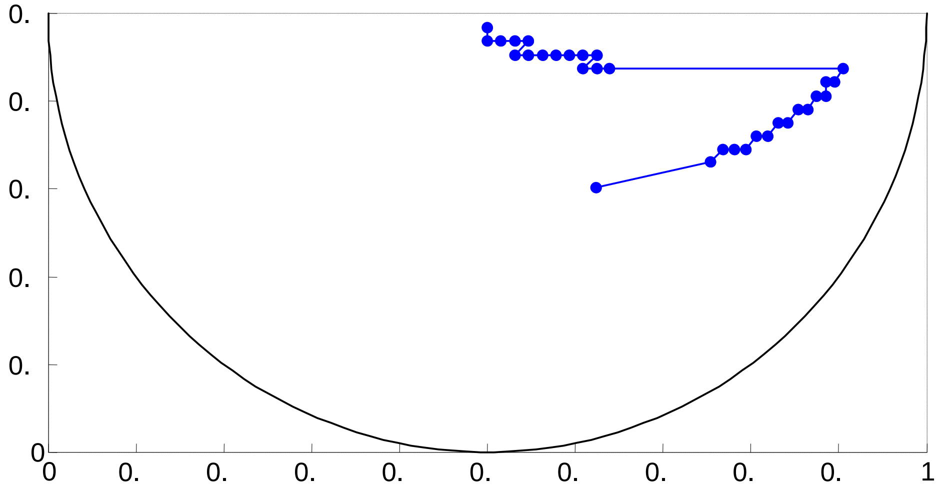

Figure 6. Transient plots of primary vortex center

CONCLUSION

It can be concluded that using non uniform grid, it is possible to simulate semi-circular lid driven cavity. As an addition, the streamline pattern shows secondary vortex appears at the later part of simulation which and vorticity contour show rapid changes at the same time. The primary vortex centre also show movement from the middle section and end up at the middle bottom of cavity.

ACKNOWLEDGEMENTS

The authors would like to thank the Faculty of Mechanical Engineering of Universiti Malaysia Pahang (UMP) and Universiti Malaysia Pahang for financial support under RDU110382.

REFERENCES

Albensoeder, S., Kuhlmann, H.C. and Rath, H.J. 2001. Multiplicity of steady two-dimensional flows in two-sided lid-driven cavities. Theoretical and Computational Fluid Dynamics, 14: 223-241

Azwadi, C.S.N. and Idris, M.S. 2010. Finite different and lattice boltzmann modelling for simulation of natural convection in a square cavity. International Journal of Mechanical and Materials Engineering, 5: 80-86.

Barragy, E. and Carey, G.F. 1997. Stream function-vorticity driven cavity solution using P finite elements. Computers and Fluids, 26: 453-468.

Cheng, M. and Hung, K.C. 2006. Vortex structure of steady flow in a rectangular

cavity. Computers and Fluids, 35: 1046-1062.
Erturk, E. and Gokcol, O. 2007. Fine grid numerical solutions of triangular cavity flow. Epj Applied Physics, 38: 97-105.
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: 387-411.
Glowinski, R., Guidoboni, G. and Pan, T.W. 2006. Wall-driven incompressible viscous flow in a two-dimensional semi-circular cavity. Journal of Computational Physics, 126: 76-91.
Idris, M. S. and Azwadi, C. S. N. 2010. Finite different and lattice boltzmann modelling for simulation of natural convection in a square cavity. International Journal of Mechanical and Materials Engineering, 5:80-86.
Idris, M. S. and Irwan, M. A. M. Irwan and Ammar, N. M. M. 2012. Steady state vortex structure of lid driven flow inside shallow semi ellipse cavity. Journal of Mechanical Engineering and Sciences, 2:206-216
Kosinski, P., Kosinska, A. and Hoffmann, A.C. 2009. Simulation of solid particles behaviour in a driven cavity flow. Powder Technology, 191: 327-339.
Mercan, H. and Atalik, K. 2009. Vortex Formation in lid-driven arc-shape cavity flows at high reynolds numbers. European Journal of Mechanics, B/Fluids, 28:61-71. 2012
Patankar, S. V. 1980.Numerical Heat Transfer and Fluid Flow, McGraw-Hill, New York,
Veldman, A.E.P. and Rinzema, K. 1992. Playing with nonuniform grids. Journal of Engineering Mathematics, 26: 119-130.
Wahba, E.M. 2009. Multiplicity of states for two-sided and four-sided lid driven cavity flows. Computers and Fluids, 38: 247-253.
Yang, F., Shi, X., Guo, X. and Sai, Q. 2012. MRT Lattice Boltzmann Schemes for High Reynolds Number Flow in Two-Dimensional Lid-Driven Semi-Circular Cavity International Conference on Future Energy, Environment, and Materials. Energy Procedia,16:639-644
Zhang, T., Shi, B. and Chai, Z. 2010. Lattice Boltzmann simulation of lid-driven flow in trapezoidal cavities. Computers and Fluids, 39: 1977-1989.
Zin, M. R. M. and Azwadi, C. S. N. 2010. An accurate numerical method to predict fluid flow in a shear driven cavity. International Review of Mechanical Engineering, 4: 719-725.