Jurnal Mekanikal
June 2008, No. 25, 1 - 8

NUMERICAL INVESTIGATION OF LID-DRIVEN CAVITY FLOW BASED ON TWO DIFFERENT METHODS: LATTICE BOLTZMANN AND SPLITTING METHODS

Nor Azwadi Che Sidik[1*], Kahar Osman[1], Ahmad Zahran Khudzairi[1], Zamani Ngali[2]

[1]Faculty of Mechanical Engineering,
Universiti Teknologi Malaysia,
81310 UTM, Skudai, Johor

[2]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia,
84000 Batu Pahat, Johor

ABSTRACT

Solutions to the Navier Stokes equations have been pursued by many researchers. One of the recent methods is lattice Boltzmann method, which evolves from Lattice Gas Automata, simulates fluid flows by tracking the evolution of the single particle distribution. Another method to solve fluid flow problems is by splitting the Navier Stokes equations into linear and non-linear forms, also known as splitting method. In this study, results from uniform and stretched form of splitting method are compared with results from lattice Boltzmann method. Lid-driven cavity problem at various Reynold numbers is used as a numerical test case.

Keywords: *Lattice Boltzmann method, splitting method, distribution function, SIMPLE, lid-driven cavity problem*

1.0 INTRODUCTION

The application of Navier-Stokes equation in solving fluid flow has evolved in the past few decades with numerical method as one of the most adopted techniques. In the traditional two dimensional solution of viscous incompressible flow, one of the most popular velocity-pressure coupling methods is SIMPLE (Semi-Implicit Method for Pressure-Linked Equation).

SIMPLE technique involves the convergence iteration to determine the pressure values for every main velocity-time iteration. As an alternative, Karniadakis et al. [1] introduced a new formulation for high-order time-accurate splitting scheme for the solution of the incompressible Navier-Stokes equations.

Principally, flow problems where large gradients are concentrated in specific regions require refinement of resolutions on those regions. Instead of using uniform, high resolution grid distribution in the physical domain, grid points may

[*] Corresponding author: E-mail: azwadi@fkm.utm.my

be clustered in the regions of high flow gradients and broaden at other regions. Stretched coordinate could demonstrate these advantages with direct usage of mathematical models of Navier-Stokes solution derived in Cartesian coordinates with minimum verifications of the discretization methods.

This work is meant to bring together the advantage of Splitting method as pressure-velocity solver of higher efficiency with the advantage of consuming stretched grid which produces more accurate results in relatively equal number of grid points as compared to Cartesian grid.

Lattice Boltzmann method (LBM), a numerical method based on particle distribution function has been demonstrated to be a very effective numerical tool for a broad variety of complex fluid flow phenomena that are problematic for conventional methods [2][3]. Compared with traditional computational fluid dynamics, LBM algorithms are much easier to be implemented especially in complex geometries and multi-component flows. Historically, LBM was derived from the lattice gas (LG) automata. It utilizes the particle distribution function to describe collective behaviors of fluid molecules. The macroscopic quantities such as density and velocity are then obtained through moment integrations of the distribution function.

In this paper, rephase carry out the simulation of lid-driven cavity flow using the two approaches mentioned above. It should be stated clearly that the objective of this work is not to make a comparison of both numerical algorithms with respect to computational efficiency in terms of CPU time. This was already a topic of our previous work [4], which demonstrated that the splitting method is more efficient for high Reynolds number simulation. Therefore, besides the physics of the flow in a square cavity, this paper focuses on the comparison of accuracy of both methods.

2.0 DESCRIPTION OF NUMERICAL METHODS

In the following section, a brief introduction to the two different numerical methods is given. For a more detailed description, the literature cited may be referred.

2.1 Splitting Method

The temporal integration of the Navier-Stokes system is achieved using a semi-implicit splitting method, similar to the method of Karniadakis [1], Osman [5] and others. Consider the Navier-Stokes expression below

$$\frac{\partial \vec{v}}{\partial t} + \vec{N}(\vec{v}) = -\nabla p + \frac{1}{R_e}\vec{L}(\vec{v}), \tag{1}$$

where $\vec{L}$ is the linear viscous term and $\vec{N}$ is the non-linear advective term,

$$\begin{aligned} \vec{L}(\vec{v}) &= \nabla^2 \vec{v}, \\ \vec{N}(\vec{v}) &= \vec{v} \cdot \nabla \vec{v}. \end{aligned} \tag{2}$$

Integrate the above equation over one time step, Δt,

$$\int_{t_k}^{t_{k+1}} \frac{\partial \vec{v}}{\partial t} dt + \int_{t_k}^{t_{k+1}} \vec{N}(\vec{v}) dt = -\int_{t_k}^{t_{k+1}} \nabla p dt + \int_{t_k}^{t_{k+1}} \frac{1}{R_e} \vec{L}(\vec{v}) dt, \tag{3}$$

where *k* is the time step.

The first term is easily evaluated without approximation. A semi-implicit method treats linear terms implicitly for stability, and non-linear term is achieved with the second-order Adams-Bashforth method. The pressure term is treated by reversing the order of integration and differentiation, and then introducing time-averaged pressure while the implicit treatment of the linear viscous term is achieved with the second-order Crank-Nicholson method. The combined difference equation becomes,

$$\vec{v}^{k+1} - \vec{v}^{k} + \left[\frac{3}{2} \vec{N}(\vec{v}^{k}) - \frac{1}{2} \vec{N}(\vec{v}^{k-1}) \right] \Delta t = -\nabla \overline{p}^{k+1} \Delta t + \frac{1}{2R_e} \left[\nabla^2 \vec{v}^{k+1} + \nabla^2 \vec{v}^{k} \right] \Delta t \tag{4}$$

The continuity equation is imposed at the leading time step,

$$\nabla \cdot \vec{v}^{k+1} = 0. \tag{5}$$

In splitting method, Equation (4) is integrated numerically in three parts for each time step, each stage addressing the three terms independently and take divergence of this equation and use the continuity equation to obtain the Poisson's equation for pressure,

$$\nabla^2 \overline{p}^{k+1} = \nabla \cdot \left(\frac{\hat{\vec{v}}}{\Delta t} \right), \tag{6}$$

where the nonlinear term is neglected. Take the normal component of Equation (4) to get,

$$\vec{k} \cdot \nabla \overline{p}^{k+1} = \vec{k} \cdot \vec{v}^{k} - \vec{k} \cdot \vec{v}^{k+1} - \vec{k} \cdot \left[\frac{3}{2} \vec{N}(\vec{v}^{k}) - \frac{1}{2} \vec{N}(\vec{v}^{k-1}) \right] \Delta t + \frac{1}{2R_e} \vec{k} \cdot \left[\nabla^2 \vec{v}^{k+1} + \nabla^2 \vec{v}^{k} \right] \Delta t \tag{7}$$

Karniadakis et al. [1] have shown that all the right hand side terms of above equation can be neglected for large Reynolds numbers, leaving,

$$\vec{n} \cdot \nabla \overline{p}^{k+1} = 0. \tag{8}$$

For that reason, Karniadakis et al. recommend higher order boundary conditions for a better approximation, especially for low Reynolds-number flows.

Regular Cartesian coordinates can be 'stretched' according to the specific requirement by the use of algebraic transformation technique. In generating grid coordinates for flow in a duct, [6] derived a set of algebraic expressions to transform points in computational Cartesian coordinates to physical stretched coordinates and vice versa.

For the case of square cavity flow, algebraic expressions are used to cluster grid points near solid boundaries and critical locations such as the corners of a cavity to provide adequate resolutions for the viscous boundary layer and secondary vortices. Since the transformation for flow in a duct was found to be in a single horizontal direction, modification is done for the cavity flow grid by first transforming the horizontal, x direction and then followed by transforming the vertical, y direction.

The algebraic formulation for transformation between physical and computational domain is shown as:

$$x = L\frac{(2\alpha+\beta)[(\beta+1)/(\beta-1)]^{(\eta_1-\alpha)/(1-\alpha)} + 2\alpha - \beta}{(2\alpha+1)\left\{1+[(\beta+1)/(\beta-1)]^{(\eta_1-\alpha)/(1-\alpha)}\right\}} \tag{9}$$

$$y = H\frac{(2\alpha+\beta)[(\beta+1)/(\beta-1)]^{(\eta_1-\alpha)/(1-\alpha)} + 2\alpha - \beta}{(2\alpha+1)\left\{1+[(\beta+1)/(\beta-1)]^{(\eta_1-\alpha)/(1-\alpha)}\right\}} \tag{10}$$

For a cavity of width L and height H, β is the clustering parameter, and α defines where the clustering takes place. When $\alpha = 0$ the clustering is at $x=L$ and $y=H$; whereas when $\alpha = 1/2$ clustering is distributed equally at the four sides of the cavity.

2.2 Lattice Boltzmann Method (LBM)

The starting point for lattice Boltzmann simulation is the evolution equation for a set of distribution functions f_i which is discrete in both space and time

$$f_i(\mathbf{x}+\mathbf{e}_i, t+1) - f_i(\mathbf{x},t) = \frac{1}{\tau}\left[f_i(\mathbf{x},t) - f_i^{eq}(\mathbf{x},t)\right] \tag{11}$$

where $\mathbf{e}$ is the particle's velocity, τ is the relaxation time for the collision, f_i^{eq} is an equilibrium distribution function and i = 0, 1,…, 8 for two-dimensional nine-velocity model (D2Q9). The right hand side of Equation (1) is the collision term where the BGK approximation [7] has been applied. The discrete velocity is expressed as $\mathbf{e}_i = (0, 0)$ for $i = 0$, $\mathbf{e}_i = (\cos(i-1)\pi/4, \sin(i-1)\pi/4)$ for i = 1, 3, 5, 7 and $\mathbf{e}_i = 2^{1/2}(\cos(i-1)\pi/4, \sin(i-1)\pi/4)$ for i = 2, 4, 6, 8. Macroscopic density ρ and velocity $\mathbf{u}$ of the fluid are determined by the following velocity moments of the distribution function

$$\sum_i f_i^{eq} = \rho \tag{12}$$

$$\sum_i e_{i,\alpha} f_i^{eq} = \rho \mathbf{u}_\alpha \tag{13}$$

The equilibrium distribution function, f_i^{eq} is chosen such that the continuum macroscopic equations, approximated by evolution equation, correctly describe the hydrodynamics of the fluid. For D2Q9 model, f_i^{eq} is defined as

$$f_i^{eq} = \rho\omega_i \left[1 + 3\frac{\mathbf{e}_i \cdot \mathbf{u}}{c^2} + 9\frac{(\mathbf{e}_i \cdot \mathbf{u})^2}{2c^4} - \frac{3\mathbf{u}^2}{2c^2} \right] \tag{14}$$

where $c = (3RT)^{1/2}$ and the weights are $\omega_0 = 4/9$, $\omega_{1,3,5,7} = 1/9$ and $\omega_{2,4,6,8} = 1/36$. Through multiscaling expansion, the mass and momentum equations can be derived from D2Q9 model as below

$$\nabla \cdot \mathbf{u} = 0 \tag{15}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}\nabla \cdot \mathbf{u} = \frac{1}{\rho}\nabla p + \upsilon \nabla^2 \mathbf{u} \tag{16}$$

The viscosity, υ can be related to the time relation in lattice Boltzmann equation as below

$$\tau = 3\upsilon + \frac{1}{2} \tag{17}$$

The algorithm in LBM generally consists of two steps: (a) streaming (left hand side of Equation (11)), where the distribution function moves to the nearest nodes in the direction of its velocity, and (b) collision (right hand side of Equation (11)), which occurs when particle distribution function arrives at a node. In LBM, the collision is set to occur at the same time at every lattice node. Therefore, the simulation using LBM is restricted to uniform mesh only.

One of the important and crucial issues in lattice Boltzmann simulation of flow is accurate modeling of boundary condition. Boundary conditions in LBM were originally taken from the LG method, known as the bounce back scheme [8]. This type of boundary condition was used at walls to obtain no-slip velocity condition. The bounce back scheme means that when a particle distribution streams to a wall node, the particle distribution scatters back to the node it has come from. The easy implementation of this no-slip velocity condition by the bounce back boundary scheme is another advantage of LBM for simulating fluid flows in complicated geometries.

3.0 NUMERICAL RESULTS

The lid-driven cavity flow has been used as a benchmark problem for many numerical methods due to its simple geometry and complicated flow behaviors. It is usually very difficult to capture the flow phenomena near the singular points at the corners of the cavity.

In this section, the lattice Boltzmann and splitting methods are applied to this lid-driven cavity flow of height *L*. The top plate moves from left to right along the *x* direction with a constant velocity *U* , and the other three walls are fixed. Results published by Ghia et al. [9] were used as the benchmark for accuracy. Only uniform grid was applied for the lattice Boltzmann computation while the uniform and stretched coordinates were applied for splitting method.

The accuracy of both methods at steady state conditions was first compared. The two velocity components, horizontal velocity *u* and vertical velocity, *v* along the vertical and horizontal lines through the cavity center together with the benchmark solution are shown in Figures 1-3. For 33 × 33 grid at *Re* = 100, all methods showed good comparison with those of Ghia et al. [9]. Looking closely, the stretched coordinate of splitting method seemed to agree very well, followed by splitting method using uniform coordinates and finally the least accurate is the lattice Boltzmann method. Increasing the Reynolds number to 400, both results produced by splitting method (uniform and stretched) began to show some discrepancies with those of Ghia et al. [9]. However, results by lattice Boltzmann method remained to agree well with those of Ghia et al. [9]. For higher *Re*, that is 1000, results by splitting method with Cartesian coordinates failed to accurately predict the flow behavior.

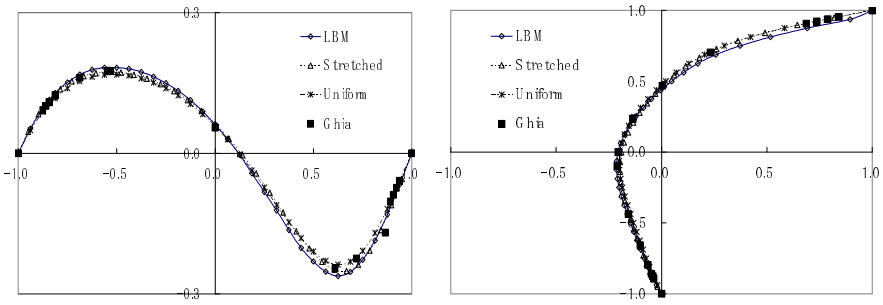


Figure 1: Vertical (left) and horizontal (right) velocity for *Re* = 100

Figures 4(a) – (c) show plots of stream lines for the Reynolds numbers considered simulated using lattice Boltzmann method. It is apparent that the flow structure is in good agreement with the previous work of Ghia et al. [9]. For low Re (*Re* = 100), the center of vortex is located at about one-third of the cavity depth from the top. As Re increase (*Re* = 400), the primary vortex moves towards the center of cavity and increasing circular. At *Re* = 1000, a pair of counterrotating eddies can be clearly seen at the lower corners of the cavity.

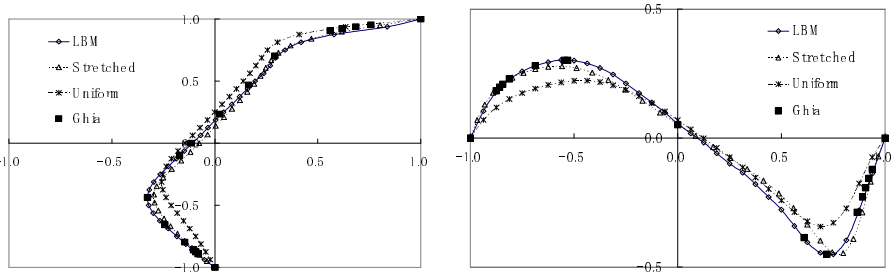


Figure 2: Vertical (left) and horizontal (right) velocity for *Re* = 400

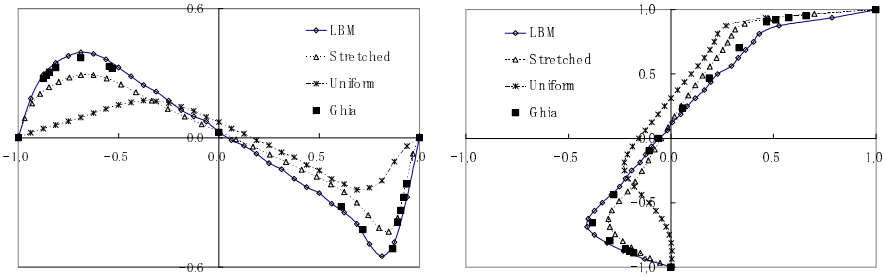
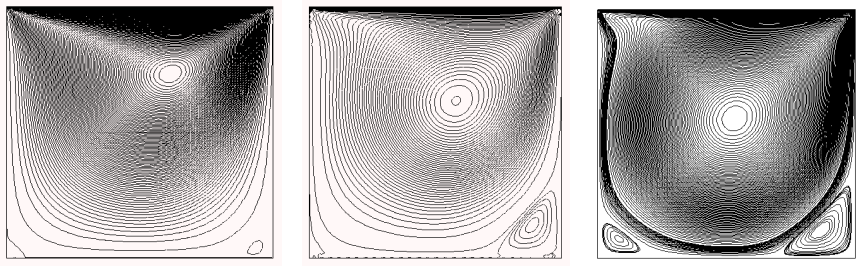


Figure 3: Vertical (left) and horizontal (right) velocity for *Re* = 1000



(a) (b) (c)

Figure 4: Streamline plots for (a) *Re* = 100, (b) *Re* = 400, (c) *Re* = 1000

4.0 CONCLUSIONS

Three methods were employed to solve the traditional lid-driven cavity problems with different force strength: Lattice Boltzmann, splitting method with Cartesian coordinates and splitting method with stretched coordinates. For low *Re*, splitting method with both uniform and stretched coordinates showed good results with relatively coarse mesh. At high *Re*, both splitting method with uniform and stretched coordinates failed to show results with acceptable accuracy. For the splitting methods, the stretched coordinate results were more accurate compared to those of Cartesian coordinates. Lattice Boltzmann results showed some discrepancies between those of Ghia et al. [9] for the computation at low Reynolds

numbers. However, lattice Boltzmann computation has shown an excellent stability and accuracy for the simulation at high Reynolds numbers.

ACKNOWLEDGEMENTS

The authors would like to acknowledge Universiti Teknologi Malaysia, Universiti Tun Hussein Onn and Malaysia Government for supporting this research.

REFERENCES

1. Karniadakis, G., Israeli, K., Orszag, S.,1991. High-Order Splitting Methods for the Incompressible Navier-Stokes Equations, *J. Comp. Phy. 97*, 414-443.
2. He, X., Luo, L.S., 1997. Lattice Boltzmann Model for the Incompressible Navier–Stokes Equation, *J. Stat. Phys. 88*, 927-944.
3. Sidik, N.A.C., Tanahashi, T., 2006. Simplified Thermal Lattice Boltzmann in Incompressible Limit, *Intl. J. Mod. Phys. C 20*, 2437-2449.
4. Ngali, M.Z., Sidik, N.A.C., Osman, K., Khudzairi, A.Z.M., 2007. Solution to Navier-Stokes Equation for Lid-driven Cavity Problem; Comparison between Lattice Boltzmann and Splitting Method, *Proceeding of Regional Conference on Engineering Mathematics, Mechanics, Manufacturing & Architecture*, Malaysia.
5. Osman, K., 2004. *Multiple Steady Solutions and Bifurcations in the Symmetric Driven Cavity*, Ph.D Thesis, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Malaysia.
6. Anderson, D.A., Tannehill, J.C., Pletcher, R.H., 1984. *Computational Fluid Mechanics and Heat Transfer.* Taylor and Francis.
7. Bhatnagar, P.L., Gross, E.P., Krook, M., 1954. Model for Collision Processes in Gasses, *Phys. Rev. 94*, 511-525.
8. Sidik, N.A.C., Tanahashi, T., 2006. Simplified Thermal Lattice Boltzmann in Incompressible Limit, *Proceeding of 11th Asian Congress of Fluid Mechanics*, Malaysia.
9. Ghia, U., Ghia, K.N., Shin, C.Y., 1982. High-Re solutions for Incompressible Flow using the Navier-Stokes Equations and a Multigrid Method, *J. Comp. Phys. 48*, 387-411.