CFD Letters www.cfdl.issres.net Vol. 4 (3) – September 2012

Lattice Boltzmann Numerical Scheme for Transient Hydrodynamics of Solid Particles in an Enclosure

Nor Azwadi Che Sidik[C], Leila Jahanshaloo

Faculty of Mechanical Engineering, Universiti Teknologi Malaysia
81310 UTM Johor Bahru, MALAYSIA

Received: 05/10/2011 – Revised 24/07/2012 – Accepted 22/10/2012

Abstract

In this paper, a Lagrangian–Lagrangian numerical simulation method for transient hydrodynamics of solid particles in an enclosure is presented. In this numerical scheme, we solve the fluid phase using a mesoscale method of lattice Boltzmann scheme. The particle motion is governed by Newton's law thus following the Lagrangian approach. The dynamics of solid particles in a lid-driven cavity was investigated at a wide range of Reynolds numbers. The results of this study suggest that the particle trajectories are critically dependent on the magnitude of Reynolds Numbers and the vortex behavior in the cavity. Comparisons with other previous studies demonstrate the application diversity of the present scheme.

Keywords: Fluid-solid interaction, Lattice Boltzmann method, Lid-driven cavity, Solid particles

1. Introduction

The phenomenon of multiphase flow can be seen not only in daily life situations but also in almost all engineering applications. The importance in understanding this problem results in many technical papers appearing in recent years discussing its impact on engineering. Because of the vast applications of the solid- liquid interaction in the industrial field, the progressing research in this area seems obligatory. Interestingly, this type of multiphase fluid flow plays an important role in the seeds drying technology, separation of grains, productions of milk powder, fluidized beds, coal combustion and many others.

It is believed that the main reason of lack of understanding on the fluid-solid interaction phenomenon is the complicated nature of the problem. The size of solid particles can be as big as grain seeds or as tiny as dust pollutants. Until the present day, most researchers rely on the computational, rather than the experimental approach to study the behavior of these particles in fluid flow. To the best of authors' knowledge, only Tsorng et al. [1] reported detailed experimental results on the behavior of solid particles in lid-driven cavity flow from micro to macro size of particles. Other experimental works are Adrian [2], Han et al. [3], Matas et al. [4], Ushijima and Tanaka [5], Ide and Ghil [6], Hu [7], Liao [8], etc.. However, according to these papers, high accurate laser equipments together with high-speed digital image capturing, and data interpretation

[C] Corresponding author: Nor Azwadi Che Sidik
Email: azwadi@fkm.utm.my Tel: +607-5534627 Fax: +607-5566159

PII: S2180-1363(12)43102-X

systems are required to obtain reliable experimental data. These high costs experimental devices will not be affordable if not supported by the research fund. However, the attitudes on solid–liquid systems have changed in the meantime, in parallel with the advancement in the global interests and technologies.

As an alternative approach, many researchers considered fully computational scheme in their investigations [9-11]. Kosinski et al. [12,13] provides a wide range of numerical results on this subject. From the behavior of one particle in a lid-driven cavity flow to thousands of particles in expansion horizontal pipe has been studied in their research works sheds new hope in understanding this problem. Kosinski et al. applied the combination of the continuum Navier-Stokes equations to predict fluid flow and the second Newton's law for solid particle flow. Their model predicts excellent results when compared to the experimental results, however, the complicated nature of Navier-Stokes equations demands high computational time in resolving fluid part. In contrast, particulate nature and local dynamics of Lattice Boltzmann Method (LBM) [14,15] make it a suitable tool for fluid-solid interaction prediction.

The LBM adopts the kinetic theory of gases, which considers the evolution of fluid based on the behaviour at molecular level [16,17].Accordingly, the LBM resolves the macroscale of fluid flow indirectly by solving the evolution equation of particle distribution function and models the propagation and collision of particle distribution, which are believed to be the fundamental behaviours at molecular level [18]. From this similarity between the mechanisms of the LBM and the behaviour of solid particles, it is considered that the LBM is the best choice to couple with the second Newton's law for the prediction of fluid-solid interaction. The emphasis is on the integration of the meso-scale of the LBM and the macro-scale of physical conditions. Other numerical issues related to the fluid solid simulation are also highlighted. There are some valuable LBM studies related to the solid fluid suspensions. Some interesting applications in treatment of fluid- particle interaction areas were carried out by Ladd [32,33] and Behrend [34]. In the model of Ladd , an approximation used to simulate the particles moving boundaries, and the distribution function fi is defined for grid points inside and outside the particle. In the suspensions of macroscopic particles (i.e. larger than 10μm), where the viscous forces alone are important, the fluctuation is ignored in lattice Boltzmann method. The particles fluctuation effects also were studied in Brownian motion by Ladd [35] and Duffty and Ernst [36]. Close quantitative agreement is found between experiments and the mentioned studies results. Therefore, the objectives of this study are coupling the techniques of the LBM formulation and solid particle dynamics (Lagrangian-Lagrangian), to enhance our fundamental physical understanding of fluid-solid interaction for two phase flow problems on a generic level.

2. Mathematical Modelling

2.1. *The Lattice Boltzmann Method*

Recently, there are a lot of researches applying the Lattice Boltzmann Method (LBM) to study various types of fluid flow problems [19-22]. They have demonstrated that the LBM is a powerful numerical tool in solving fluid flow parameters. The LBM originates from the kinetic Boltzmann equation derived by Ludwig Boltzmann (1844-1906) in 1988. It considers a fluid as an ensemble of artificial particles and explores the mesoscopic features of the fluid by using the propagation and collision effects among these particles. The LBM discretizes the whole flow region into a number of grids and numerically solves the simplified Boltzmann equation on the regular lattices [23]. The solution of the lattice Boltzmann equation converges to the Navier-Stokes solution in continuum limit up to second order accuracy in space and time [24]. This method bridges the gap between the mesoscopic world and the macroscopic phenomena. The LBM has emerged as a versatile numerical method for simulating various types of fluid flow problems including turbulent [25], multiphase [26], magnetohydrodynamics [27], flow in porous media [28], microchannel flow [29], etc.

The starting point for lattice Boltzmann simulations are the evolution equation of particle distribution function *f* which can be written as

$$f_i(\mathbf{x}+\mathbf{c}_i\Delta t, t+\Delta t) - f_i(\mathbf{x}, t) = -\frac{f_i - f_i^{eq}}{\tau} \tag{1}$$

where, f_i^{eq} is the equilibrium distribution function. c_i is the lattice velocity and i is the lattice direction, Δt is the time interval, τ is the relaxation time of the particle distribution function, respectively. The magnitude of c_i is set up In the LBM. So in each time step Δt, the distribution function propagates in a distance of lattice nodes spacing Δx. This ensures that the distribution function arrives exactly at the lattice nodes after Δt and collides simultaneously. The macroscopic variables such as the density ρ, fluid velocity u and temperature T can be computed in terms of the particle distribution function as

$$\rho = \int f d\mathbf{c}, \ \rho\mathbf{u} = \int \mathbf{c} f d\mathbf{c} \tag{2}$$

To simulate the fluid flow in a system, one often uses the D2Q9 model [30] with nine velocities assigned on a two-dimensional square lattice. These velocities include eight moving velocities along the links connecting the lattice nodes of the square lattice and a zero velocity for the rest particle. The rest particles are defined by the distribution functions f_0, the particles moving in the orthogonal direction by the function f_i (i = 1,2,3,4) and the particles moving in the diagonal directions by the function f_i (i = 5,6,7,8). The equilibrium distribution functions f_i^{eq} are given as

$$f_i^{eq} = \rho\omega_i\left[1 + 3\mathbf{c}_i\cdot\mathbf{u} + 4.5(\mathbf{c}_i\cdot\mathbf{u})^2 - 1.5\mathbf{u}^2\right] \tag{3}$$

where ω is the weight function and depends on the direction of the lattice velocity.

Through the multiscaling expansion, the mass and momentum equations can be derived for the D2Q9 model of the evolution equation of the density distribution function. Detailed derivation can be found in [31].

2.2. The flow of the particles

Numerical simulations of flow in monodisperse and bidisperse systems were studied by many researchers [37, 38]. The influence of the particle volume fraction and Reynolds numbers on the drag forces attracts attention in the presence of neighbouring particles in multi particles systems. Due to the results of these researches it should be noted that the correction of the drag forces is an important fact in multi particles systems, especially in dense suspension. In the first step of this investigation, we only consider one particle in a lid driven cavity and assume that the presence of solid particles has no effect on the fluid flow. The equation of motion for solid particles can be written as

$$m_p\frac{d\mathbf{v}_p}{dt} = \mathbf{f}_p \tag{4}$$

where m_p, $\mathbf{v}_p$ and $\mathbf{f}_p$ are the mass of particle, its velocity and drag force acting on particle due to surrounding fluid. According to Kosinski et al, the drag force can be written due to fundamental formula as follows

$$\mathbf{f}_p = C_D A_p \rho\frac{|\mathbf{u}-\mathbf{v}_p|(\mathbf{u}-\mathbf{v}_p)}{2} \tag{5}$$

where A_p is the projected area of solid particle in flow direction and C_D is the drag coefficient which is defined as

$$C_D = \frac{24}{\mathrm{Re}_p}(1+\frac{1}{6}\mathrm{Re}_p^{\frac{2}{3}}) \tag{6}$$

where Re_p is the Reynolds Number of solid particle.

In the second part we add 2000 particles by the diameters of 0.0005(m) inside the cavity which means the total packing fraction of less than 0.1, and the size of the particles are smaller than lattice unit. Therefore, the drag force correlation is not highly important and we can use the above formula for getting enough accurate results for multi particle's simulation. However, for the denser suspension, drag forces correction due to Beestra et al.[38] research is suggested.

Moreover, in the present analysis, since we are coupling the macroscopic unit for the solid particle and mesoscopic unit for lattice Boltzmann formulation, it is crucial to understand the relationship between these two different scales of units. Consider a solid particle in a system of fluid as shown above, the Reynolds Number of the particle must be set the same both in the lattice Boltzmann formulation and actual physical flow, that is,

$$Re_p = \frac{u_L d_L}{\upsilon_L} = \frac{u_r d_r}{\upsilon_r} \tag{7}$$

Here, the subscripts L and r denote the variables in lattice units and physical units, respectively. Hence, the actual time must be converted from lattice time t_L to physical time t_r as follows:

$$t_r = \left(\frac{d_r}{d_L}\right)^2 \left(\frac{\upsilon_L}{\upsilon_r}\right) t_L \tag{8}$$

3. Numerical Results

The code was first validated against other numerical solutions by comparing the trajectory of a particle with the surrounding fluid. In present study, the density of particle supposed to be same as the fluid so the particles assume buoyant and the buoyancy force is neglected. The main and noticeable force is counted in this research is the drag force which is acting on particles. For the simulation, the top lid is constantly moved. The data setting of Tsorng et al.[1] which assumed as the verification case of kosinski's et al.[13]. Research consisted of a cavity with side 10 cm, filled with a fluid of viscosity 37.2. The top lid is moving with a speed 17.5 cm/s. so that the resultant Reynolds Number is 470.

It is necessary to mention that the above value should be converted to lattice unit. By the help of real plan viscosity the time step and size step should be calculated for a suitable relaxation time in lattice Boltzmann method.

Figure 1 shows the comparison between the simulated particle's trajectory by the current Lagrangian-Lagrangian approach and the Eulerian-Lagrangian solution and the experimental result at the steady state position. As can be seen from the figure, except for a short interval of time around the starting point, the predicted orbits are quite similar in three methods.

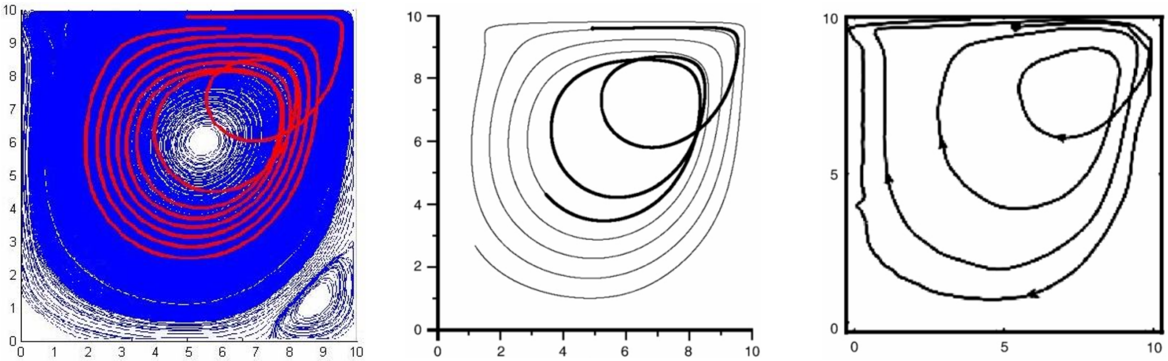
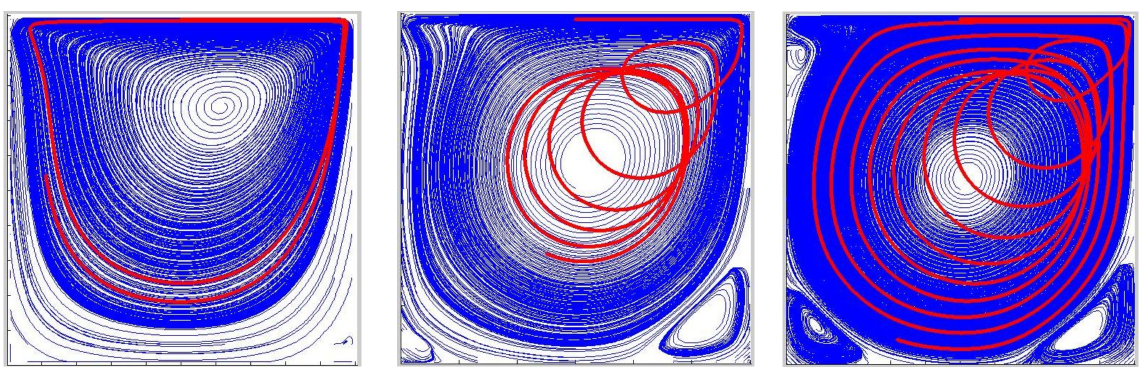


Figure 1. Comparison of particle's trajectory computed from current approach (left) and Eulerian-Eulerian scheme [13] (middle) and the experimental results of Tsorng et al. [1] (right) for *Re* = 470.

In the next calculations, the top lid velocity varied, which results in Reynolds Numbers 100 to 3200 but all the physical character such as fluid viscosity and particle density and size of cavity keep constant. Figure 2 shows the trajectory of a solid particle suspended in a square cavity. As can be seen from the figures, at low Reynolds Number (*Re* = 100), a vortex was formed near the top sliding wall and as the time evolved, it immediately moved downward into the cavity. However, this weak vortex was unable to drag the particle into it. As a result, the particle only spirals outwards of the centre of the cavity. However, for the simulation at higher Reynolds Number (*Re* = 1000 and 3200), a stronger vortex was initially developed below the top lid and then propagates to the right corner of the cavity together with the solid particle. As the vortex grows in size and strength, it trapped the particle and moved to the center of the cavity. This forced the particle to make few small spirals near the upper right of the cavity and then gradually spiral outwards as it was dragged by the vortex to the center of the cavity.



(a) (b) (c)

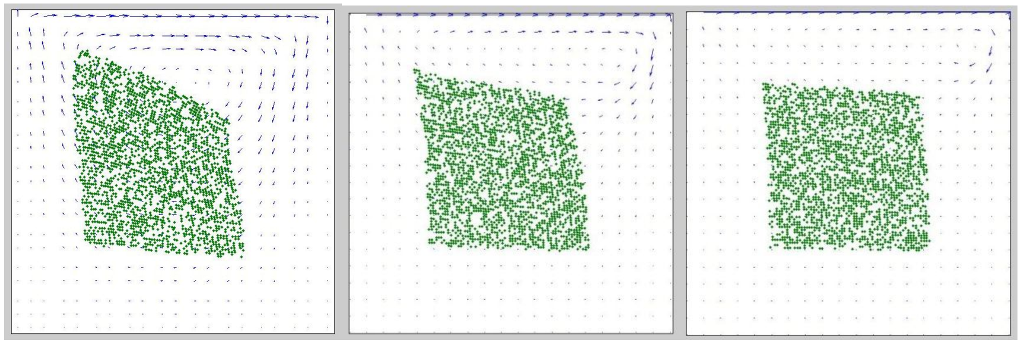
Figure 2. Particle's trajectory for (a) Re = 100, (b) Re = 1000 and (c) Re = 3200.

Lastly, calculations were done to predict the dynamics of particles in a lid-driven cavity at Reynolds Numbers of 10, 400 and 1000. Two thousand particles were randomly located in the cavity in the range of 0.25 to 0.75 in both *x* and *y* direction.

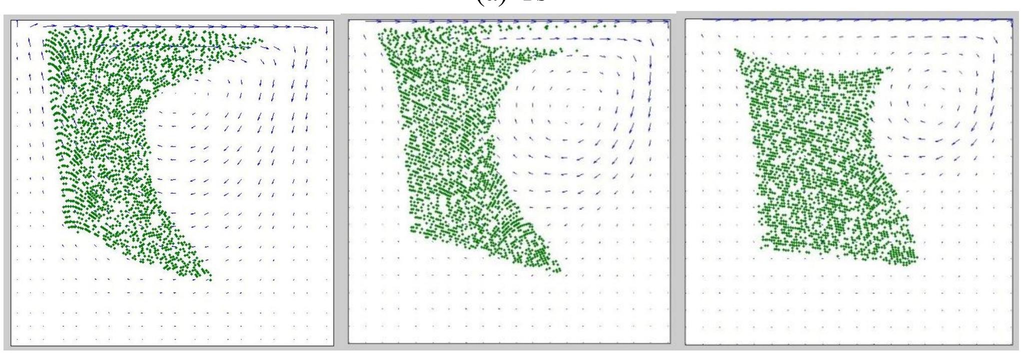
The snapshots of the transient hydrodynamics of particles are shown in Fig. 3. Surprisingly, the particles in the cavity with the lowest Reynolds number in the present study started to move earlier than those in the higher values of Reynolds Numbers. This can be explained by analysing the behaviour of the main vortex in the cavity for each Reynolds Number. For *Re* = 10, the main vortex immediately moves to the centre of the cavity and drags the particles into it. However, for the higher Reynolds Numbers, the main vortex initially moves to the right corner of the cavity before propagating to the centre where the particles are located. Then, the rotating fluid drags the particles into motion and is responsible for the drift of the particles.

The vortex strength also influences the dynamic behaviour of the particles. At low Reynolds Number (*Re* = 10), a weak vortex is formed and a gradual gradient of flow velocity from the vortex centre to the moving lid takes place. This makes the particles move in a bigger group along the flow streamline compared to the condition at high Reynolds Numbers. However, due to a comparatively weak vortex for *Re* = 10, the particles took longer time to circulate in the cavity compared to the predictions at *Re* = 400 and *Re* = 1000.

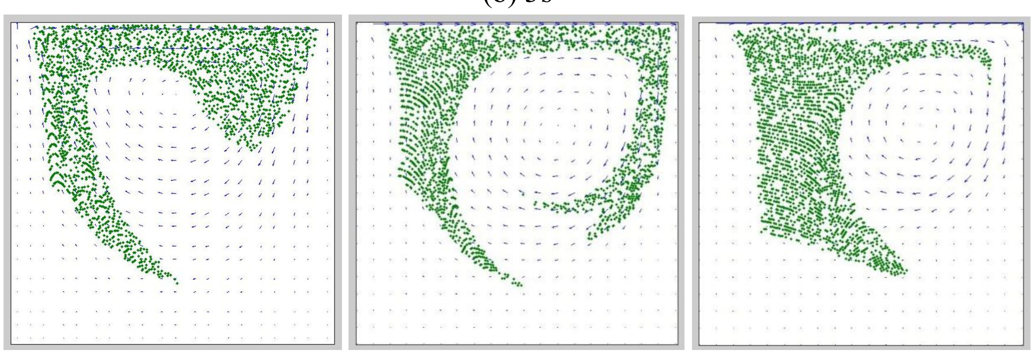
For all cases, due to the high inertia force acting on the moving particles, they are centrifuged outward and eventually all the particles propagate along the outer side of the vortex in the cavity.



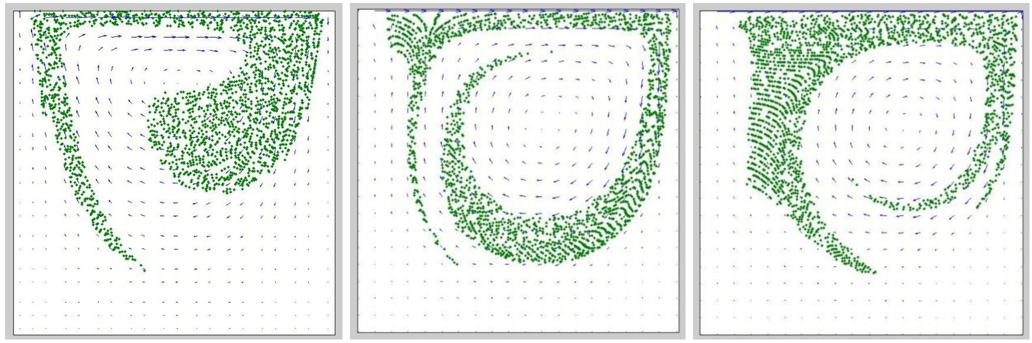
(a) 1s



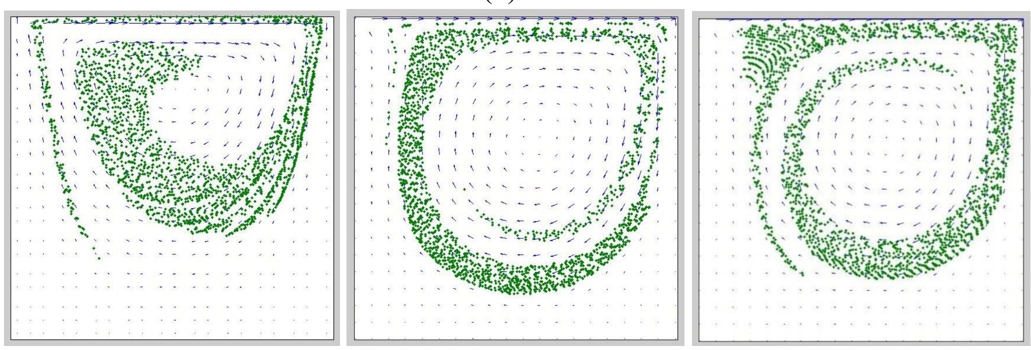
(b) 3s



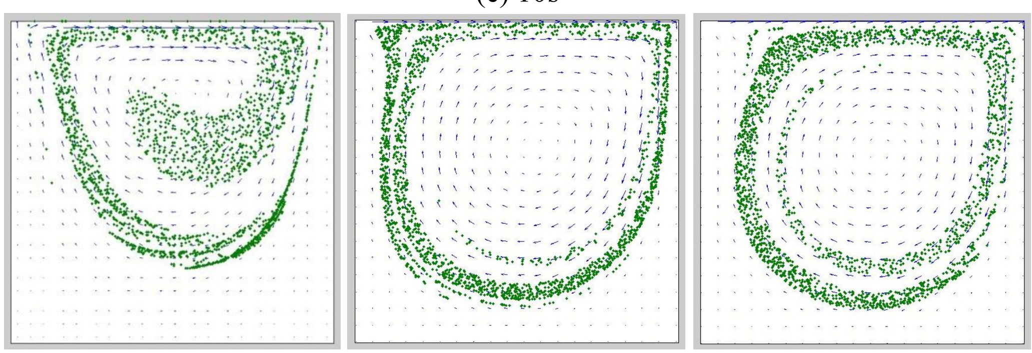
(c) 5s



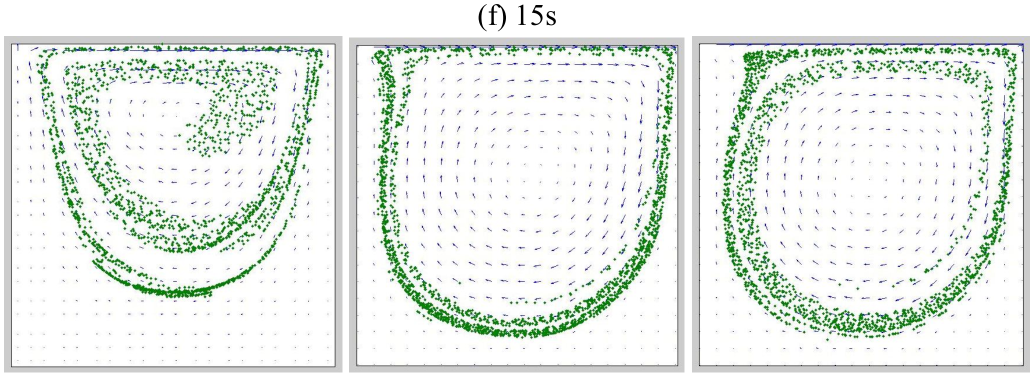
(d) 7s



(e) 10s



(f) 15s



(g) 20s

Figure 3. Snapshots of particles' positions *Re*=10 (left), *Re*=400 (Center) and *Re*=1000 (right), at time (a) 1s, (b) 3s, (c) 5s, (d) 7s, (e) 10s, (f) 15s and (g) 20s.

4. Conclusion

Numerical computations of solid particles in a lid-driven cavity flow were performed using the lattice Boltzmann mesoscale method and the Newton's second law (Lagrangian-Lagrangian scheme). Results of the present computations show that, almost all the physical detail of this transient flow at wide range of Reynolds Numbers are reproduced by the current scheme. The computed particle's trajectories clearly indicate the influence of vortex structure on the dynamics of particle in the cavity. Thise demonstrates the capability and the application diversity of the present numerical scheme. Future efforts need to extend the current formulation for investigation at various types of solid fluid flow related to real engineering problems.

Acknowledgement

The authors wish to acknowledge Universiti Teknologi Malaysia and the Malaysia Government for supporting this research.

References

[1] S.J. Tsorng, H. Capart, J.S. Lai, L.D. Young, Three-dimensional tracking of the long time trajectories of suspended particles in a lid-driven cavity flow, Exp. Fluids 40 (2006) 314-328.

[2] R.J. Adrian, Particle-imaging techniques for experimental fluid mechanics, Ann. Rev. Fluid Mech. 23 (1991) 261-304.

[3] M. Han, C. Kim, M. Kim, S. Lee, Particle migration in tube flow of suspensions, J. Rheology 43 (1991) 1157-1174.

[4] J.P. Matas, J.F. Morris, E. Guazzelli, Inertial migration of rigid spherical particles in Poiseuille flow, J. Fluid Mech. 515 (2004) 171-195.

[5] S. Ushijima, N. Tanaka, Three-dimensional particle tracking velocimetry with laser-light sheet scannings, J. Fluid Eng. 118 (1996) 352-357.

[6] K. Ide, M. Ghil, Extended Kalman filtering for vortex systems. Methodology and point votices, Dyn. Atm. Oceans 27 (1997) 301-332.

[7] C.C Hu, Analysis of motility for aquatic sperm in videomicroscopy, Ph.D Thesis, National Taiwan University 2003.

[8] J.I. Liao, Trajectory and velocity determination by smoother, Ph.D Thesis, National Taiwan University 2002.

[9] S.R. Idelsohn, E. Oñate, F. Del Pin, A Lagrangian meshless finite element method applied to fluid–structure interaction problems, Comp. Struc. 81 (2003) 655-671.

[10] H. Ramkissoon, K. Rahaman, Wall effects on a spherical particle, Intl. J. Eng. Sci. 41 (2003) 283-290.

[11] C. Antoci, M. Gallati, S. Sibilla, Numerical simulation of fluid–structure interaction by SPH, Comp. Struc. 85 (2007) 879-890.

[12] P. Kosinski, C.G. Ilea, A.C. Hoffmann, Three-dimensional of a dust lifting process with varying parameters, J. Multiphase Flow 34 (2008) 869-878.

[13] P. Kosinki, A. Kosinska, A.C. Hoffmann, Simulation of solid particles behavior in a driven cavity flow, Pow. Tech. 191 (2009) 327-339.

[14] C.S. Nor Azwadi, A.R. Mohd Rosdzimin, Cubic interpolated pseudo particle (CIP)-thermal BGK lattice Boltzmann numerical scheme for solving incompressible thermal fluid flow problem, Malay, J. Math. Sci. 3 (2009) 183-202.

[15] M.A. Mussa, S. Abdullah, C.S. Nor Azwadi, N. Muhamad, Simulation of natural convection heat transfer in an enclosure by the lattice-Boltzmann method, Comp. Fluids 44 (2011) 162-168.

[16] S.Chen, G. Doolen, Lattice Boltzmann Method for fluid flows, Ann. Rev. Fluid Mech. 30 (1998) 329-364.
[17] N.S. Martys, H. Chen, Simulation of multicomponent fluids in complex three-dimensional geometries by the lattice Boltzmann method, Phys. Rev. 53 (1996) 743-750.

[18] U. Frish, B. Hasslacher, Y. Pomeau, Lattice Gas Automata for the Navier-Stokes equation, Phys. Rev. 56 (1986) 1505-1509.
[19] C.S Nor Azwadi, S. Syahrullail, Three-Dimension Double-Population Thermal Lattice BGK Model for Simulation of Natural Convection Heat Transfer in a Cubic Cavity, WSEAS Trans. Math. 8 (2009) 561-571.
[20] S. Hou, Q. Zou, S. Chen, G. Doolen, A.C. Cogley, Simulation of cavity flow by the lattice Boltzmann method, J. Comp. Phys. 118 (1995) 329-347.
[21] C.S. Nor Azwadi, T. Takahiko, Simplified Thermal Lattice Boltzmann in Incompressible Limit, Intl. J. Mod. Phys. B 20 (2006) 2437-2449.

[22] Y.W. Kwon, S. Hosoglu, Application of lattice Boltzmann method, finite element method, and cellular automata and their coupling to wave propagation problems, Comp. Struc. 86 (2008) 663-670.
[23] Y. Peng, C. Shu, Y.T. Chew, 3D incompressible thermal lattice Boltzmann model and its application to simulation natural convection in a cubic cavity, J. Comp. Phys. 193 (2003) 260-274.
[24] M. Krafczyk, J. Tölke, E. Rank, M. Schulz, Two-dimensional simulation of fluid–structure interaction using lattice Boltzmann methods, Comp. Struc. 79 (2001) 2031-2037.
[25] J.Jonas, B. Chopard, S. Succi, F. Toschi, Numerical analysis of the average flow field in a turbulent lattice Boltzmann simulation, Phys. A 362 (2006) 6-10.
[26] A. Alapati, S. Kang, Y.K. Suh, 3D lattice Boltzmann simulation of droplet formation in a cross-junction microchannel, 3rd IASME/WSEAS Intl. Conf. Cont. Mech (2008).
[27] G.V. Breyiannis, D. Valougeorgis, Lattice kinetic simulations in three-dimensional magneto hydrodynamics, Phys. Rev. E 69, 065702/1-065702/4 (2004).
[28] S.B. Edo, V. Maddalena, Lattice Boltzmann studies of fluid flow in porous media with realistic rock geometries, Comput. Math. App. 59 (2010) 2305-2314.
[29] Y. Zhang, R. Qin, D.R. Emerson, Lattice Boltzmann simulation of rarefied gas flow in microchannel, Phys. Rev. E 71, 047702/1-047702/4.
[30] X. He, L.S. Luo, Lattice Boltzmann model for the incompressible Navier-Stokes equation, Stat. Phys, 88 (1997) 927-944.
[31] C. Cercignani, The Boltzmann equation and its application in applied mathematical sciences, Springer 1998.
[32] A. J. C Ladd, Numerical simulations of particulate suspensions via a discretized Boltzmann equation, Part I. J. Fluid Mech. (1994), 271, 285–309.
[33] A. J. C Ladd, Numerical simulations of particulate suspensions via a discretized Boltzmann equation, Part II. J. Fluid Mech. (1994), 271, 311–339
[34] O. Behrend, Solid-fluid boundaries in particle suspension simulations via the lattice Boltzmann method, Phys. Rev. (1995), 52, 1164–75
[35] A. J. C Ladd, Short-time motion of colloidal particles numerical simulation via a fluctuating lattice-Boltzmann equation ,Phys. Rev. (1993), 70, 1339–1342
[36] J.W. Dufty, M.H. Ernst , Lattice Boltzmann Langevin equation, Fields Inst. Comm (1996), 6, 99–107
[37] M. A. van der Hoef, R. Beetstra, J. A. M. Kuipers, Lattice-Boltzmann simulations of low-Reynolds-number flow past mono- and bidisperse arrays of spheres, J. Fluid Mech. (2005), 528, 233–254.

[38] R. Beetstra, M. A. van der Hoef, J. A. M. Kuipers, Drag force of intermediate Reynolds number flow past mono- and bidisperse arrays of spheres, AIChE Journal (2007), 53, 489–501.