Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 3, No. 1. Pages 17-26, 2014

Lagrangian Grid LBM to Predict Solid Particles' Dynamics immersed in Fluid in a Cavity

A. Ismail, L. Jahanshaloo[*,a] *and A. Fazeli*[b]

Department of Thermo-Fluids, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia (UTM), 81310 Skudai, Johor, Malaysia.
[*,a]*leilajahanshaloo@gmail.com*, [b]*fazeli.1985@gmail.com*

Abstract – *In this article, the lattice Boltzmann method (LBM) to predict moving solid particles in a fluid flow is presented. The scheme uses a uniform Eulerian grid for the flow domain and a Lagrangian grid to trace the dynamics of solid particles. The solid particles in a cavity located on the floor of a straight channel were simulated at two difference aspect ratios and a wide range of laminar Reynolds numbers. Two different shapes of cavity were selected to investigate their effect on the efficiency of the solid particles removal. The current study discovered that the rate of the particle removal depends significantly on the Reynolds number of the flow and the shape of the cavity. A fair agreement with the results obtained from other methods emphasizes the capability of the current scheme in predicting particulate problems.*

Keywords: Lattice-Boltzmann method, Particulate flow, Drag force, Channel flow

1.0 INTRODUCTION

Analysis on fluid flow of a system has been widely studied due to its huge involvement in the engineering field. Studies on the fluid flow are not limited only to a single-phase flow, but it has been extended to multiphase flow, which also involves the fluid-solid interaction. The fluid-solid interaction has become a topic of interest for the research in the past decade. The phenomenon of this flow can be seen in almost all engineering applications and plays an important role in the industrial manufacturing process, wastewater treatment, seeds drying technology, separation of grains and many others.

An experimental study has been reported by Tsorng et al. [1] focusing on behaviour of solid particles in the lid-driven cavity flow from micro to macro size particles. The nature of the problem itself created complexity for the experimental work. The particles' size can be in any range, to be as tiny as dust or as big as grain seeds. It has been also reported that in order to conduct such experimental works, high-cost devices are essential. According to previous experimental works, high accuracy laser equipment, high-speed digital image capture and data interpretation system are required to obtain reliable experimental data.

In early years, such problems used to be solved either by Finite Difference Method, Finite Element Method or Finite Volume Method. The combination of continuum Navier-Stokes equation for fluid flow prediction and the Newton's second law for solid particle are applied to solve the fluid-solid flow matters, which have been already reported by Kosinksi et al. [2], Azwadi et al. [3] and Fang et al. [4, 5]. However, these methods face difficulties to perform

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 3, No. 1. Pages 17-26, 2014

simulation accurately. The complicated nature of Navier-Stokes equation demands high computational time in resolving fluid part. Thus, lattice Boltzmann method is found to be a suitable numerical scheme for fluid-solid interaction problems due to its computational efficiency and parallel scalability [6]. In this case, instead of solving Navier-Stokes equation, LBM is proposed as an alternative numerical method to solve the fluid part. LBM foundation adopts the kinetic theory of gases that considers the evolution of fluid behavior at mesoscopic level [7, 8]. Accordingly, LBM solves the macroscale of fluid flow indirectly by solving the evolution equation of the particle distribution function and models the propagation and collision of particles' distribution, which are believed to be the fundamental behaviours at molecular level [9]. The similarity between the mechanisms of LBM and the behaviour of solid particle are the reason for LBM to be considered and proposed as the method for solving the fluid-solid interaction. This study summarizes the coupling technique of the LBM formulation and solid particle dynamics.

Two major objectives for this study are first, to propose an alternative numerical method for the fluid-solid interaction problem by using Lagrangian-Lagrangian approach and second, to determine the most influence factor on the particle trajectory and removal rate for two different cavity shapes. Variable parameters considered in this study are Reynolds number and cavity's aspect ratio.

2.0 MATHEMATICAL MODEL

In this study, the flow is considered laminar, incompressible and has a parabolic inlet velocity profile. Since the objective of the study is to solve the fluid-solid interaction flow problem, the numerical simulation is done for two parts, which are the fluid flow and solid particle. For fluid flow simulation, D2Q9 lattice Boltzmann method is adopted.

The starting point of LBM is the evolution equation of single-particle distribution-function 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 functions, $\mathbf{c}_i$ is the lattice velocity, i is the lattice direction, Δt is the time interval, and τ is the relaxation time of the particle distribution function.

First, the magnitude of $\mathbf{c}_i$ is set up so that in each time step Δt, the distribution function propagates in a distance of lattice nodes spacing Δx. This will ensure that the distribution function arrives exactly at the lattice nodes after Δt and collides simultaneously. Then, the macroscopic properties of fluid such as the density ρ and velocity $\mathbf{u}$ can be computed in terms of the particle distribution functions as below:

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 I Vol. 3, No. 1. Pages 17-26, 2014

Density:

$$\rho = \sum_i f_i \tag{2}$$

Velocity:

$$\rho \mathbf{u} = \sum_i \mathbf{c}_i f_i \tag{3}$$

For the simulation, the D2Q9 model with nine lattice velocities assigned on a two-dimensional square lattice. The nine lattice velocities consist of eight moving velocities along the links connecting the lattice nodes of the square lattice and a zero velocity for the rest particle. The particle at rest is 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 $f_i(i = 5,6,7,8)$.

As for the equilibrium distribution functions, it is given as:

$$f_i^{eq} = \rho\omega_i\left[1 + 3\mathbf{c}_i \cdot \mathbf{u} + 4.5(\mathbf{c}_i \cdot \mathbf{u})^2 - 1.5\mathbf{u}^2\right] \tag{4}$$

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

The LBM relaxation time formulation can be related to the fluid viscosity in macroscopic scheme as follows:

$$\tau = 3\upsilon + 0.5 \tag{5}$$

In the present study, we assume that the presence of solid particle gives no effect to the fluid flow, but instead the fluid flow will cause the particles trajectory. For solid particle simulation, the second Newton's law is applied, where the equation of motion for solid particle is expressed as:

$$m_p \frac{d\mathbf{v}_p}{dt} = \mathbf{f}_p \tag{6}$$

where m_p, $\mathbf{v}_p$ and $\mathbf{f}_p$ are the mass of particle, velocity of particle and drag force acting on particle due to the surrounding fluid respectively. According to Kosinski et al. [5], the drag force can be written as follows:

$$\mathbf{f}_p = C_D A_p \rho \frac{|\mathbf{u} - \mathbf{v}_p|(\mathbf{u} - \mathbf{v}_p)}{2} \tag{7}$$

where A_p is the projected area of solid particle and C_D is the drag coefficient that is given by:

$$C_D = \frac{24}{\mathrm{Re}_p} \tag{8}$$

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 3, No. 1. Pages 17-26, 2014

The density of particle and fluid is assumed to be similar; hence, buoyancy force can be neglected. Particles are also assumed to be far enough from each other so that particles interaction can also be neglected.

Since we are coupling the mesoscopic unit for lattice Boltzmann formulation together with the macroscopic unit for solid particles, we need to understand the relationship between these two different scales of units. The Reynolds number of the particles must be set the same for both planes. The relation between LBM plane and real physical plane can be found using:

$$\mathrm{Re}_p = \frac{u_l d_l}{\upsilon_l} = \frac{u_r d_r}{\upsilon_r} \tag{9}$$

where u is the velocity, d is the diameter of solid particle, υ is the viscosity and subscripts l and r denote the variables in lattice units and real physical units respectively.

Actual time can be obtained by converting the lattice time to real physical time as follows:

$$t_r = \left(\frac{d_r}{d_l}\right)^2 \left(\frac{\upsilon_l}{\upsilon_r}\right) t_l \tag{10}$$

3.0 RESULTS AND DISCUSSION

In this section, the code was validated to characterize the flow field without the presence of solid particle. Then, the validation was done by considering 100 particles (with unity diameter) inside the cavity to compare the trajectory of particles with the experimental result. The geometry of the problem is set as shown in Fig. 1. Another cavity shape, which is triangular, was also considered in this study. The maximum inlet velocity was varied to give the Reynolds number as 50, 100 and 400. The cavity aspect ratios (AR=W/D) of 1 and 2 were considered in the analysis as well. A number of solid particles were initially filled in the cavity as illustrated in Fig. 1. The particles have the same size, and their diameter is smaller than the lattice units.

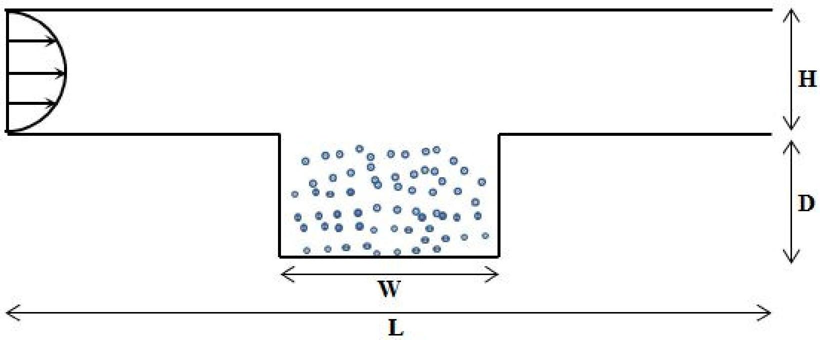

Figure 1: Illustrative the problem's geometry.

The results are discussed from two aspects; the flow development structure and the particle removal pattern. As demonstrated by Figs. 2-4, the main pattern of flow development is similar, which starts with the formation of vortex in the left-top corner of the cavity. Then, it moves to

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 3, No. 1. Pages 17-26, 2014

the centre of cavity. In higher Reynolds numbers, the vortex has the tendency to move to the right side and the circulating area of vortex is bigger.

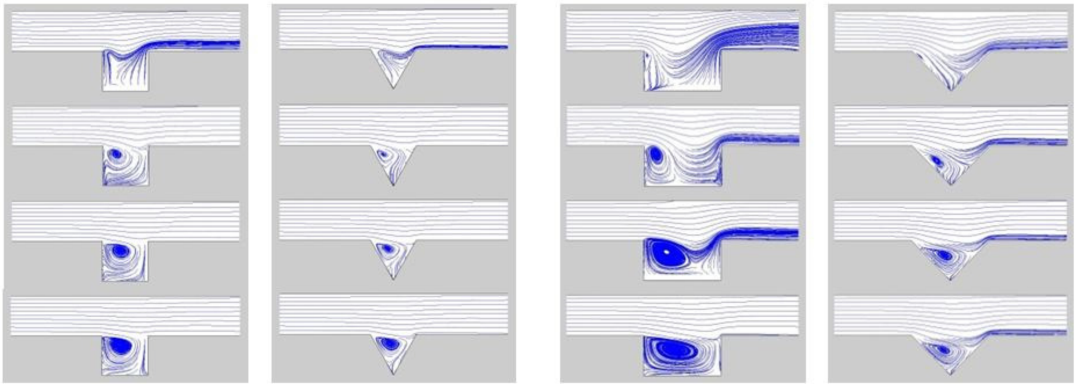

Figure 2: Snapshots of streamline plots for Re = 50 at AR = 1 (left) and AR = 2 (right).

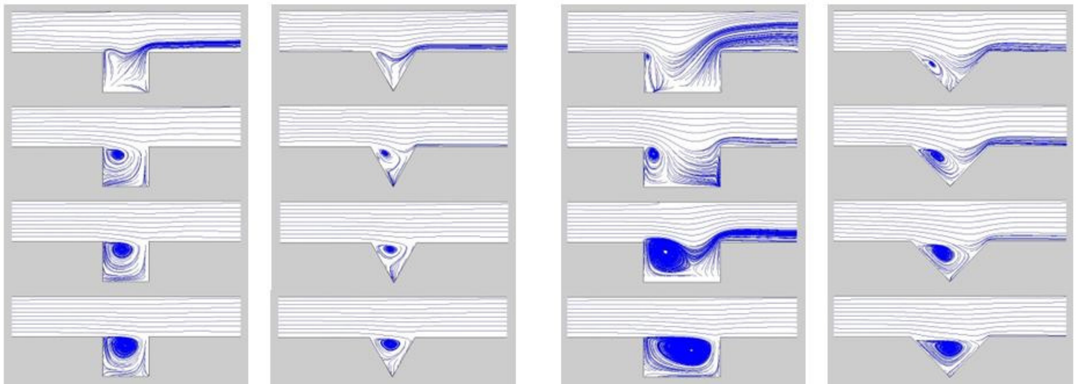

Figure 3: Snapshots of streamline plots for Re = 100 at AR = 1 (left) and AR = 2 (right).

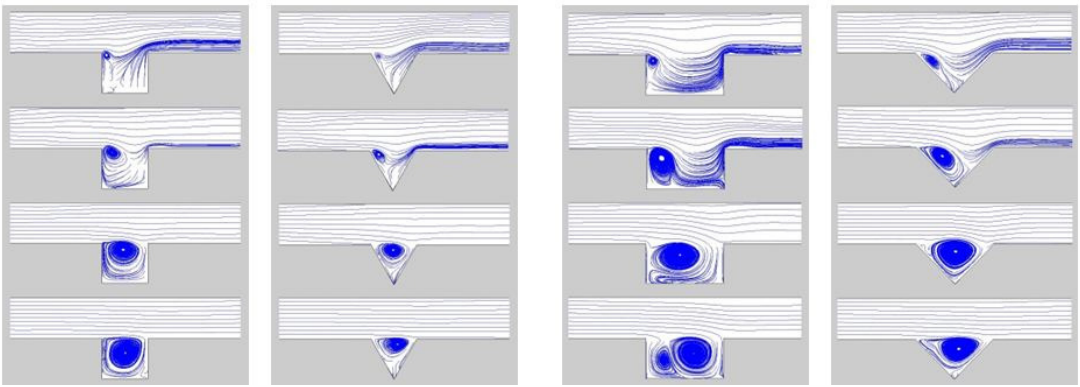

Figure 4: Snapshots of streamline plots for Re = 400 at AR = 1 (left) and AR = 2 (right).

For AR=1, there is no noticeable difference of the flow development in different Reynolds numbers. However, for AR=2, the vortex can be seen to grow through the cavity and as it reaches the steady state, it covers three third of the cavity. With the increase in Reynolds

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 I Vol. 3, No. 1. Pages 17-26, 2014

number, the primary vortex moves towards the centre of cavity and increases the circulation area.

Figs. 5-7 show the particles movement pattern for two different cavity shapes with aspect ratios of 1 and 2. From the figures, the particles movement follows the flow structure. During the start-up, flow penetration and acceleration forced particles to leave the cavity. The highest rate of removal occurs during the early penetration of fluid flow into cavity. This is due to the early development of vortex, which penetrated into the cavity and brought the particles out from the cavity. Recirculation area is formed in the cavity and captured part of the particles located in the cavity. For longer time, this area became stronger and trapped the remaining particles until a steady state is achieved. From the results, it can be seen that Reynolds number, aspect ratio and cavity shape have significant effects on the particles removal percentage.

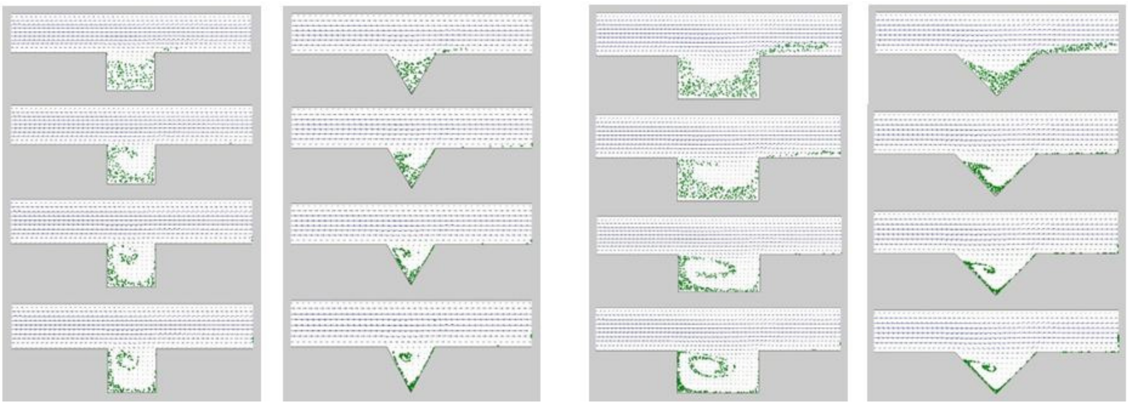

Figure 5: Snapshots of particle removal process at Re = 50 for AR = 1 (left) and AR = 2 (right).

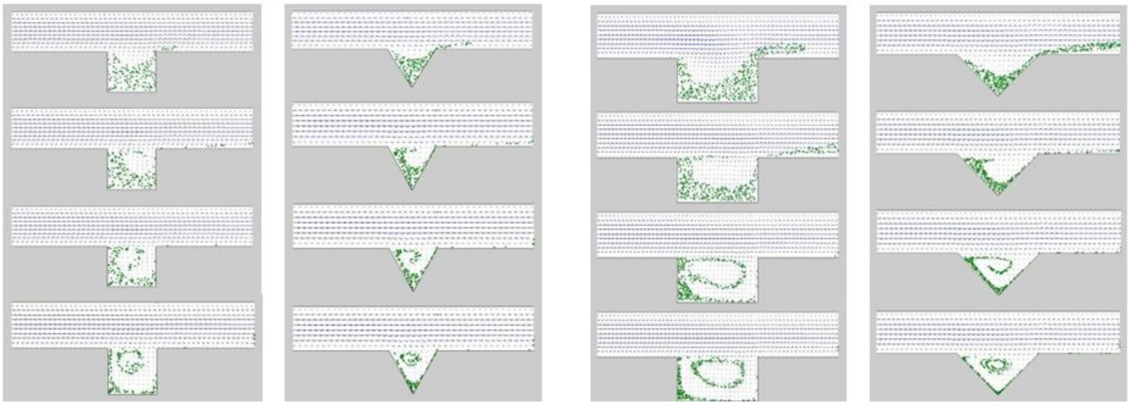

Figure 6: Snapshots of particle removal process at Re = 100 for AR = 1 (left) and AR = 2 (right).

The effect of Reynolds number on both cavity shapes can be seen from Figs. 8-9. As the Reynolds number increases, the percentage of particles removal also grows. This is due to higher speed of flow velocity inside cavity and deeper penetration of vortex in the cavity, which

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 I Vol. 3, No. 1. Pages 17-26, 2014

is resulted from higher Reynolds number. Hence, solid particles in the cavity are dragged and flushed out from cavity.

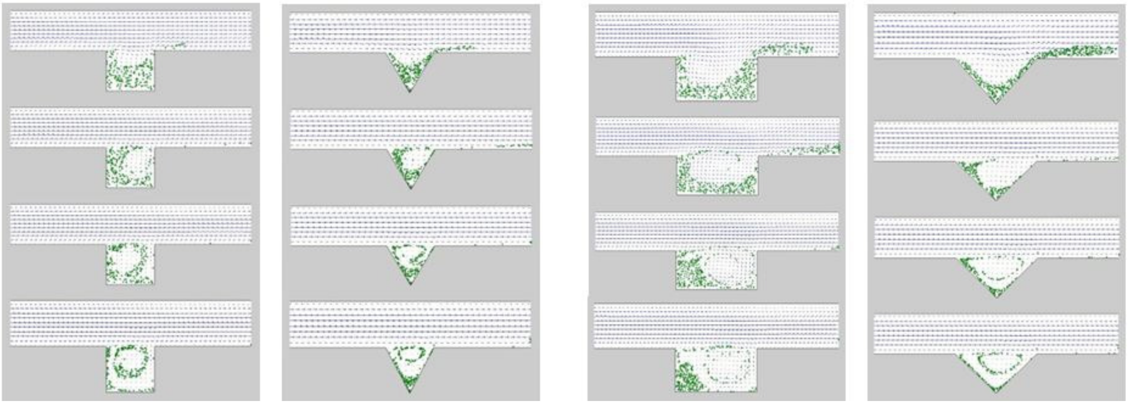

Figure 7: Snapshots of particle removal process at Re = 400 for AR = 1 (left) and AR = 2 (right).

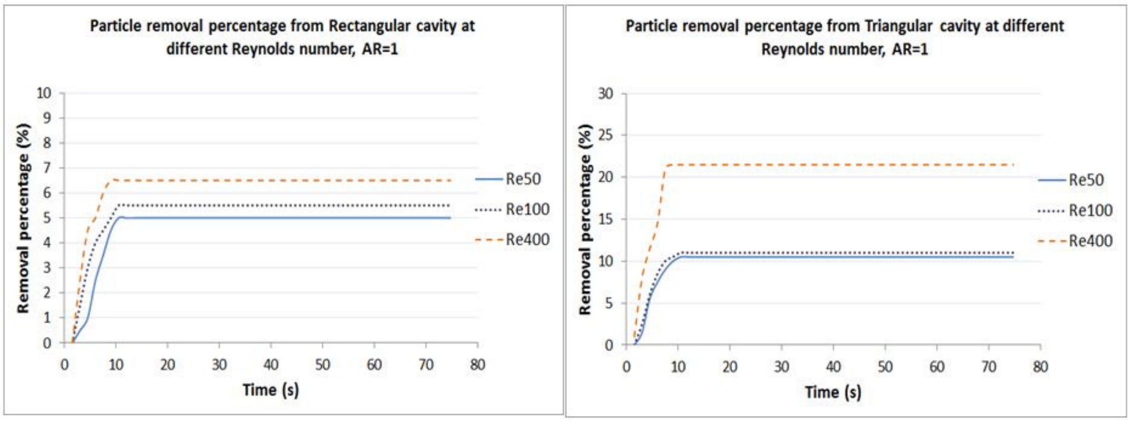

Figure 8: Time versus percentage of particle removal for AR = 1.

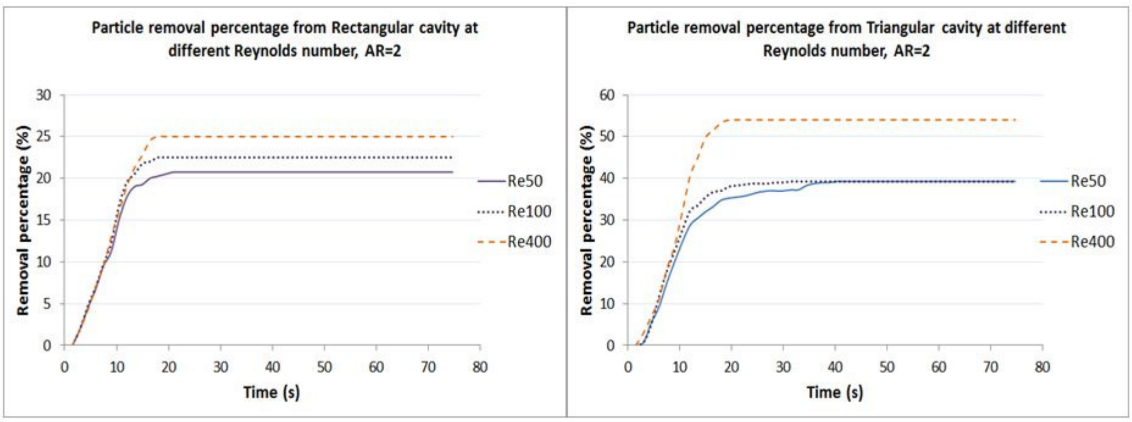

Figure 9: Time versus percentage of particle removal for AR = 2.

The comparisons of particles removal percentage for different cavity shapes are demonstrated in Figs. 10-12. Triangular cavity provides higher removal percentage compared to that of rectangular. This might be due to the shallow depth of the cavity. As Reynolds number increases, the vortex center moves to the upper side of the cavity in a triangular shape. However, in the rectangular cavity, it remains in the center. This vortex dynamics feature gives definite effect on the efficiency of particles removal. During the formulation of the vortex in the cavity, solid particles are trapped. As it moves to the upper side of the cavity, particles are brought together and flushed out from the cavity by centrifugal force. This is the reason that causes higher removal percentage in triangular cavity.

The influence of aspect ratio on particles removal percentage is demonstrated in Fig. 13. Removal percentage soars upward with the growth in aspect ratio. For similar aspect ratios, the final removal percentage is almost the same. The difference between the aspect ratio is the trend of particle removal. The flow profile in higher Reynolds numbers occurs in some steps. It starts with the development of vortices inside the cavity during the unsteady state. The drag forces of vortex sucks the particles inside and moves them to a different side of cavity. Some of the particles are totally released from the vortices. Due to this nature of flow profile, it can be seen that the increase of particles removal in Reynolds number 400 is more during unsteady state. In this Reynolds number, secondary vortices are also formed. This type of vortex has a dynamic behavior of moving along the cavity. While moving, the particles are also forced to move together and for those located in the outer layer of circulating area, it will leave the cavity in a long time after start-up. The moving nature of the higher Reynolds numbers cannot be seen in smaller aspect ratio.

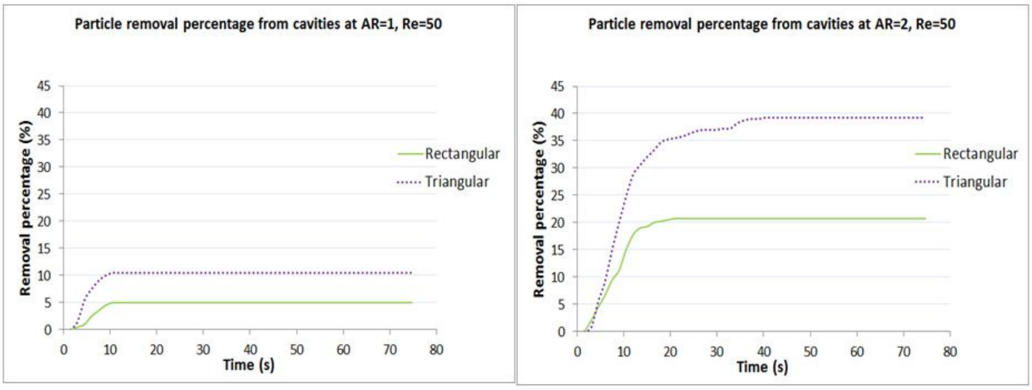

Figure 10: Time versus percentage of particle removal at Re = 50.

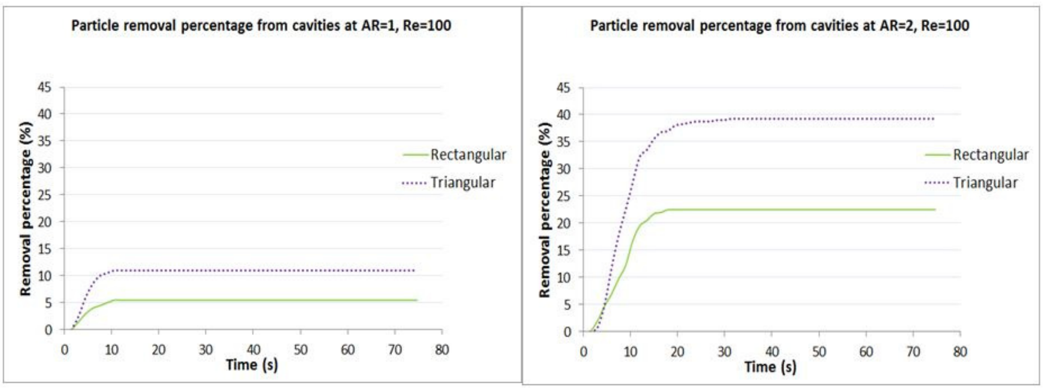

Figure 11: Time versus percentage of particle removal at Re = 100.

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 3, No. 1. Pages 17-26, 2014

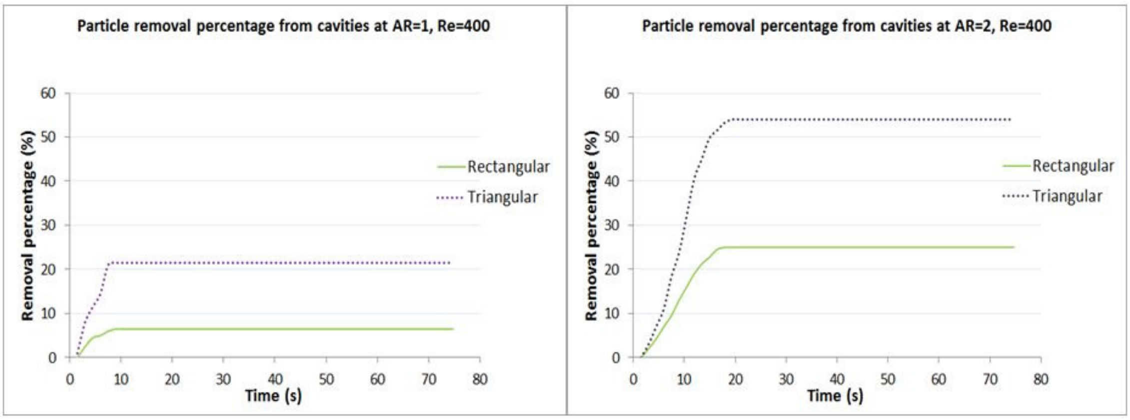

Figure 12: Time versus percentage of particle removal at Re = 400.

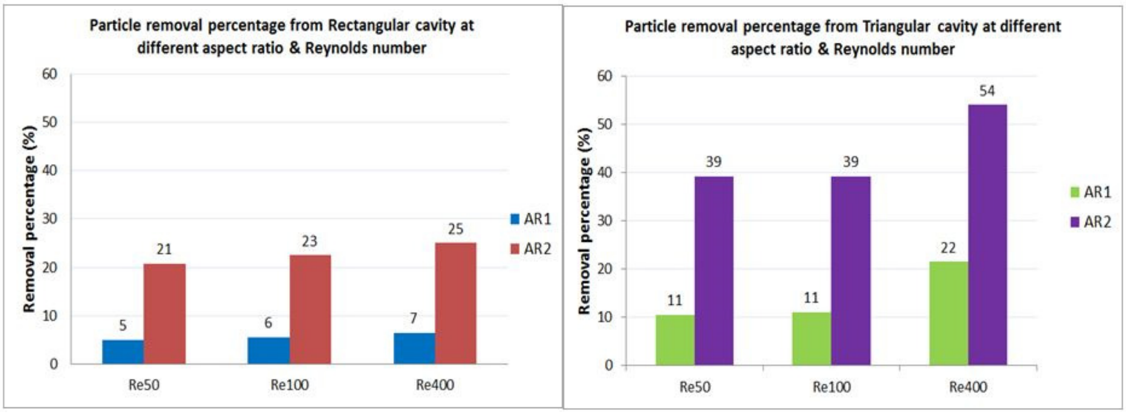

Figure 13: Time versus particle removal percentage for different aspect ratios and Reynolds numbers.

4.0 CONCLUSION

Numerical simulation of fluid-solid particle interaction for two different cavity shapes was studied by coupling LBM and Newton's second law (Lagrangian-Lagrangian scheme). Validation of the results was carried out for two situations. The first validation involved the flow field without the presence of solid particle, and the second one consisted particles inside the cavity to compare the trajectory of particles with the experimental results. The objectives of this study were to investigate the effects of Reynolds numbers, aspect ratios and cavity shapes on trajectory and removal efficiency of particles in the cavity. The results show that all the three parameters, to some degree, have influences on the rate of particles removal and flow structures. Higher Reynolds number exhibits higher rate of removal due to the higher speed of early penetration of flow within the cavity, especially in the triangular-shaped cavity. Due to moving nature of vortices, the effect of cavity aspect ratio seems to give more significant effectiveness of particles removal by comparing with variation of Reynolds number. As good agreements were obtained with the previous studies of experimental results and numerical simulations, it can be concluded that the present results support the idea that the lattice Boltzmann simulation scheme is a reliable numerical method that contains sufficient qualities to predict fluid-solid particle interaction.

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 I Vol. 3, No. 1. Pages 17-26, 2014

ACKNOWLEDGEMENT

The authors would like to thank the Universiti Teknologi Malaysia and Ministry of Education of Malaysia for supporting this research activity. This research was financially supported by the Research University Grant 06H23.

REFERENCES

[1] S.J. Tsorng, H. Capart, D.C. Lo, J.S. Lai, D.L. Young, Behaviour of macroscopic rigid spheres in lid-driven cavity flow, International Journal Multiphase Flow 34 (2008) 76-101.

[2] P. Kosinski, A. Kosinska, A.C. Hoffmann, Simulation of solid particles behaviour in a driven cavity flow, Powder Technology 191 (2009) 327-339.

[3] C.S.N. Azwadi, M.R. Attarzadeh, The use of cubic interpolation method for transient hydrodynamics of solid particles 51 (2012) 90-103.

[4] L.C. Fang, D. Nicolaou, J.W. Cleaver, Transient removal of a contaminated fluid from a cavity, International Journal Heat and Fluid Flow 20 (1999) 605-613.

[5] L.C. Fang, D. Nicolaou, J.W. Cleaver, Numerical simulation of time-dependent hydrodynamic removal of a contaminated fluid from a cavity, International Journal Numerical Method Fluids 42 (2003) 1087-1103.

[6] C.S.N. Azwadi, M.S. Idris, Mesoscale numerical approach to predict macroscale fluid flow problems, Journal of Applied Sciences 10 (2010) 1511-1524.

[7] S. Chen, D. Martinez, W.H. Mattaeus, Lattice Boltzmann magneto hydrodynamics, Physic of Plasmas 1 (1994) 1850-1860.

[8] N.S. Martys, H. Chen, Simulation of multicomponent fluids in complex three dimensional geometries by the lattice Boltzmann method, Physical Review A 53 (1996) 743-750.

[9] U. Frish, B. Hasslacher, Y. Pomeau, Lattice Gas Automata for the Navier-Stokes equation, Physic Review Letter 56 (1986) 1505-1509.