Prediction of Dynamics of Solid Particles using Lattice Boltzmann Method

Nor Azwadi C. Sidik$^1$, Leila Jahanshaloo$^2$

Abstract – Research on numerical schemes on fluid-solid interaction has been quite intensive in the past decade. The difficulties associated with accurate predictions of the interaction at specific spatial and temporal levels. Traditional computational fluid dynamics schemes are struggling to predict at high level of accuracy for this type of problem. Hence, in the present study, an alternative numerical scheme was proposed to predict the fluid flow and coupled with a Lagrangian scheme on the prediction of solid phase. The dynamics of solid particles in an enclosure and a channel with a cavity were investigated at a wide range of Reynolds numbers. The results show that the particle trajectories are critically dependence on the magnitude of Reynolds numbers and the vortex behavior in the cavity. Good comparisons with the previous studies demonstrate the multidisciplinary applications of the present scheme.

Keywords: Fluid-solid interaction, Lattice Boltzmann method, Cavity flow, Solid particle

I. Introduction

The phenomenon of multiphase flow can be seen not only in daily live situations but also in almost all engineering applications. The importance in understanding this problem results in many technical papers appear in recent years discussing its impacts on engineering. Among the researches pertinent to this problem, comparatively very few researchers devoted their study on the interaction between solid particles and fluid flow. 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 grains seed or very tiny such as dust pollutant. Till present day, most researchers rely on computational rather than experimental approach to study the behavior of these particles in fluid flow. To the best of authors’ knowledge, only [1] reported details experimental results on the behavior of solid particles in lid-driven cavity flow from micro to macro size of particles. Other experimental works are [2]-[8]. However, according to these papers, high accuracy of laser equipment together with high-speed digital image capture and data interpretation system is required to obtain reliable experimental data. Such these high cost experimental devices will not be affordable if not supported by research fund.

As an alternative approach, many researchers considered fully computational scheme in their investigations. [9]-[10] provides extensive numerical results on the 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. [9] applied the combination of continuum Navier-Stokes equations to predict fluid flow and second Newton’s law for solid particle. As their model predicts excellent results when compared to the experimental results, however, the complicated nature of Navier-Stokes equation demands high computational time in resolving fluid part. In contrast, the mathematical foundation of lattice Boltzmann method (LBM) [10] makes it a suitable tool for fluid-solid interaction prediction.

LBM foundation adopts the kinetic theory of gases which considers the evolution of fluid based on the behavior at molecular level [11]-[12]. Accordingly, 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 behaviors at molecular level [13]. From this similarity between the mechanisms of LBM and the behavior of solid particle, it is considered that the LBM is the best choice to couple with the second Newton’s law for prediction of fluid-solid interaction. The emphasis is on the integration of the meso-scale of LBM and the macro-scale of physical condition. Other numerical issues related to the fluid solid simulation are also highlighted. Therefore, the objectives of this study are coupling the technique of the LBM formulation and solid particle dynamics (Lagrangian-Lagrangian), and to do improvement on this fundamental knowledge on the fluid-solid interaction.

II. The Lattice Boltzmann Method

Recently, there are a lot of researches applying the lattice Boltzmann method (LBM) to study various types of fluid flow problem [14]-[17]. They have demonstrated that the LBM is a powerful numerical tool in solving fluid flow parameters. The LBM originate from the kinetic Boltzmann equation derived by Ludwig Boltzmann (1844-1906) in 1888. It considers a fluid of artificial particles and explores the mesoscopic features of the fluid by using the propagation and collision effects among these particles. LBM discretizes the whole flow region into a number of grids and numerically solves the simplified Boltzmann equation on the regular lattices [18]. The solution to the lattice Boltzmann equation converged to the Navier-Stokes solution in continuum limit up to second order accuracy in space and time [19]. This method bridges the gap between the mesoscopic world and the macroscopic phenomena. LBM has emerged as a versatile numerical method for simulating various types of fluid flow problem including turbulent [20], multiphase [21], magneto hydrodynamics [22], microchannel flow [23], etc.

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

\[
f_i(x+c_i \Delta t, t+\Delta t) - f_i(x, t) = -\frac{f_i - f_{eq}^i}{\tau}
\]

(1)

where, \( f_{eq}^i \) is the equilibrium distribution function. \( c_i \) is the lattice velocity and \( i \) is the lattice direction, \( \Delta t \) is the time interval, \( \tau \) is the relaxation times of the particle distribution functions, respectively. The macroscopic variables such as the density and fluid velocity \( u \) can be computed in terms of the particle distribution functions as

\[
\rho = \int \rho_i c_i d\xi, \quad \rho u = \int \rho_i c_i u d\xi
\]

(2)

To simulate the fluid flow in a system, one often uses the D2Q9 model [24] 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 equilibrium distribution functions is given as

\[
f_{eq}^i = \rho \omega_i \left[ 1 + 3c_i \cdot u + 4.5(c_i \cdot u)^2 - 1.5u^2 \right]
\]

(3)

where \( \omega \) is the weight function and depends on the direction of the lattice velocity.

Through the multiscale expansion, the mass and momentum equations can be derived for the D2Q9 model of the evolution equation of the density distribution function. Details derivation can be found in [25].

In the present investigation, we only consider one particle in a lid driven cavity and assume the presence of solid particle gives no effect to the fluid flow. The equation of motion for solid particle is written as

\[
m_p \frac{d\mathbf{v}}{dt} = \mathbf{f}_p
\]

(4)

where \( m, v \) and \( \mathbf{f} \) are the mass of particle, its velocity and drag force acting on particle due to surrounding fluid. Here, the drag force can be written as follow

\[
\mathbf{f}_p = C_D A_p \rho \frac{|\mathbf{u} - \mathbf{v}_p|(|\mathbf{u} - \mathbf{v}_p|)}{2}
\]

(5)

where \( A \) is the projected area of solid particle and \( C_D \) is the drag coefficient which is defined as

\[
C_D = \frac{24}{Re_p}
\]

(6)

where, \( Re_p \) is the Reynolds number of solid particle.
III. Problem Physics and Boundary Conditions

The physical domain of the problem is represented in Fig. 1. The top lid was constantly moved to the right direction at a constant velocity to give a value of Reynolds number [26].

![Fig. 1. Problem physics and boundary conditions](image)

A solid particle was located just touching the moving top lid of the cavity. In the present analysis, the computations are conducted on a two-dimensional plane as shown in Fig. 1. This two-dimensional approximation was undertaken based on a physical assumption that the behavior of the lid driven vortex is relatively unaffected by the three dimensionality of the flow.

In the present analysis, since we are coupling the macroscopic unit for 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 lattice Boltzmann formulation and actual physical flow, that is,

\[ \text{Re}_p = \frac{u_L d_L}{\nu_L} = \frac{u_r d_r}{\nu_r} \]  

(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 to physical time as follow

\[ t_r = \left( \frac{d_r}{d_L} \right)^2 \left( \frac{\nu_L}{\nu_r} \right) t_L \]  

(8)

IV. Results and Discussion

The code was first validated against other numerical solution by comparing the trajectory of a particle with approximately the same density with the surrounding fluid. For the simulation, the top lid is constantly moved so that the resultant Reynolds number is 470 as in Kosinki et al. [9].

Fig. 2 gives the comparison between the simulated particle’s trajectory simulated by the current Lagrangian-Lagrangian approach and the Eulerian-Lagrangian solution. 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 character.

In the next analysis, the transient hydrodynamic removal of solid particles from two different aspect ratio of a cavity on the floor of a channel is presented. In the simulation, the Reynolds number was set up at four different values (50, 100, 400 and 1000) which is based on the maximum inlet flow velocity and the channel height.

Fig. 3 shows how the process of solid particle removal for cavity aspect ratio of one and four. The solid particles are initially filled up the cavity with the same amount for these two aspect ratios. As demonstrated by the figures, the highest rate of removal occurs on the early penetration of fluid flow into the cavity. Then, when the recirculation areas are shaped in the cavity, they trapped some of the particles and remain in the cavity until steady state is achieved. The cavity ratio and Reynolds number also plays significant effect on the percentage of particle removal from the cavity. These are presented in Fig. 4, where the comparisons are made for all the cases.
As can be seen from Figs. 5 and 6, the percentage of removal clearly dependence on the Reynolds number for a square cavity. Higher value of Reynolds number results in higher speed of flow velocity inside the cavity and deeper penetration of the vortex into the cavity. This will drag the solid particles at the lower region inside the cavity and due to the inertia force, these particles are flushed out from the cavity to the downstream channel. However, even at the highest Reynolds number in the present study, the maximum percentage of removal is quite low which is around 14%.

For the simulation at higher aspect ratio, the graph in Fig. 6 indicates that the removal percentage has almost independence on the magnitude of Reynolds number. It is found that the percentage is around 60% when the flow achieves steady state condition. However, the slope of the graphs indicates that the removal rate is at their maximum value at the early phase of particles motion. Furthermore, higher Reynolds number leads to higher acceleration of fluid inside the cavity and contribute to higher removal rate of the solid particles.
For the both cases, we can see at steady state condition, the remaining particles are trapped near the boundaries of the cavity. These particles can be further removed by introducing the buoyancy force on the particles. This can be done by heating up the bottom cavity to induce diffusion effect on the fluid as proposed by Chilikuri and Middleman [27]. Further discussion on this subject will be our next research topic.

V. Conclusions

Numerical computations of transient hydrodynamics of solid particle were performed using the coupled lattice Boltzmann formulation and 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. These demonstrate the capability and the multidisciplinary applications 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.

Acknowledgements

This work was supported by Universiti Teknologi Malaysia and Ministry of Higher Education under Research University Grant 00H40.
References


Authors’ information

Nor Azwadi C. Sidik was born in Kelantan on 23rd September 1977. He received Ph.D degree (2010) from Keio University, Japan His current interest includes computational fluid dynamics, numerical methods and fluid structure interaction. Dr Nor Azwadi is a senior lecturer at Department of Thermofluid, Universiti Teknologi Malaysia, Malaysia.

Leila Jahanshaloo was born in Iran on 7th December 1979. She received his B.Sc. degree (2002) from Mazandaran University of Technology (MUT) Iran, M.Sc. degree (2010) from University Teknologi Malaysia in Mechanical Engineering. She is currently a Ph.D. candidate at the Department of Thermofluid in Universiti Teknologi Malaysia (UTM). Her current interest includes computational fluid dynamics, turbulent flows modeling and solid-fluid interaction analysis.