2nd International Conference on the Material Point Method for Modelling Soil-Water-Structure Interaction

Anura3D
MPM Research Community

Paper number: mpm2019.82
www.anura3d.com
www.mpm2019.eu

A coupled CFD-DEM framework to model submerged granular flow

Thien Q. Phan[1], Ha H. Bui[1,*], Giang D. Nguyen[2], Abdelmalek Bouazza[1]
[1]Department of Civil Engineering, Monash University, VIC, Australia
[2]School of Civil, Environmental and Mining Engineering, The University of Adelaide, Australia

*E-mail: ha.bui@monash.edu

ABSTRACT

This paper presents an implementation of 4-way coupling model between computational fluid dynamics (CFD) and discrete element method (DEM) in OpenFOAM to simulate fully submerged granular materials. The proposed CFD-DEM computational framework is first verified against analytical solutions for a single particle sedimentation in fluid. It is then applied to simulate submerged 3D granular flows and numerical results are qualitatively compared with experimental data available in the literature. Good agreements between the simulation and experiment suggest that the coupled CFD-DEM framework could be used to model the multiphase system consisting of a fluid and solid grains.

KEY WORDS: CFD/DEM coupling, submerged granular flow, phase interaction.

INTRODUCTION

Submerged granular flow is a flow of mass composed of granular material surrounded by ambient fluid. This highly inhomogeneous rheological flow intervenes in a wide variety of geomorphological and industrial processes (Koivisto et al., 2017). For risk assessments, understanding the dynamics of this flow and predicting the final morphology, runout distances, as well as the generation and propagation of the wave of fluid, is important. However, it is still questionable to justify the mechanical and hydraulic behaviour of submerged granular flows due to the complex phase interaction between granular and viscous fluid.

In the past few decades, there have been a large number of experimental studies of submerged granular flows. For instance, Rondon et al. (2011) reported that the morphology of the gravity deposit of submerged granular collapse is mainly controlled by the initial volume fraction of the granular mass. This is different from the case of dry granular column collapse, which typically depends on the initial aspect ratio of the granular column (Xiao and Sun, 2011). Experimental works have also shown that there are two distinct regimes of submerged granular flow corresponding to initially loose and dense packings and are strongly affected by the dilatancy and contractive behaviours (Rondon et al., 2011). Nevertheless, experimental methods cannot comprehensibly provide the micro mechanism underpin the nature of the rheological flow. Therefore, numerical methods are often considered as an alternative approach to the experiment to provide more insightful information into the detailed mechanisms of granular flows.

There have been several promising numerical approaches developed to characterise the mechanical and hydraulic behaviour of the submerged granular flows (Lube et al., 2005). Among these approaches, Eulerian-Lagrangian method is considered as one of the most widely used methods to simulate the multiphase system consisting of fluid and dispersed solid phases. Within these approaches, the solid particle phase is usually treated as discrete phase and solved by DEM, while the fluid phase is described within the CFD framework by either continuum mesh-free or mesh-based methods, such as Smooth Particle Hydrodynamics (SPH) (Bui and Nguyen, 2017), lattice Boltzmann method (LBM) (Han and Cundall, 2013), Finite Volume Method (FVM) (Chen et al., 2011), Finite Difference Method (FDM), Direct Numerical Simulation (DNS) (Zhou et al., 2010). Among these alternatives, the FVM/DEM coupling is a common approach used in CFD and proves to be advantageous over the others in term of computational efficiency and robustness. The coupling can be categorised as one-ways (fluid-to-particle action only), two-way force coupling (fluid-particle mutual interactions) and four-way coupling (fluid-particle interactions and particle-particle collisions with influence of void ratio effect) (Xiao et al., 2013). Since the considered granular flows are dense, the CFD-DEM approach with four-way coupling is necessary.

In this paper, a four-way coupled CFD-DEM framework will be presented to model submerged granular flow. The fluid is treated as a continuous phase and described in the Eulerian frame of reference using FVM, while solid particles are treated as discrete phase and expressed in the Lagrangian frame of reference with DEM. The paper first provides a compact overview of submerged granular flow and numerical methods to handle this type of flow in the introduction; in the next section, the brief of mathematical models for fluid phase and particle phase is presented followed by the discussion on the results of numerical validation and application using the implemented framework; the last section comes with the conclusion of this paper.

MATHEMATICAL MODELS

In CFD-DEM coupling, each particle is tracked in Lagrangian reference system while fluid phase is described by Eulerian specification. The motion of a solid particle can be described by Newton's Second law, which consists of the force and moment balance equations (Cundall and Strack, 1979):

$$m_i \frac{\mathrm{d}\overrightarrow{u_i}}{\mathrm{d}t} = \vec{f}_i^{f-p} + \sum_{j \in CL_i} \vec{f}_{ij}^{c} + \vec{f}_i^{g} \tag{1}$$

$$I_i \frac{\mathrm{d}\overrightarrow{\omega_i}}{\mathrm{d}t} = \vec{M}_i^{f-p} + \sum_{j \in CL_i} \vec{M}_{ij}^{c} \tag{2}$$

where m_i, I_i are the mass and moment of inertia of particle i; $\overrightarrow{u_i}, \overrightarrow{\omega_i}$ are the translational and angular velocities of particles i; $\vec{f}_i^{f-p}$ and $\vec{M}_i^{f-p}$ are the interaction force and moment between the fluid and solid particle i; $\vec{f}_{ij}^{c}$ and $\vec{M}_{ij}^{c}$ are the contact force and moment acting on particle i by particle j and/or wall; and $\vec{f}_i^{g}$ is the force due to gravity. Particle motion is driven by the interaction forces with surrounding fluid and neighbour particles, and effect of gravity. Force due to the fluid phase acting on an individual solid particles, or phase interaction force, is the summation of drag force, pressure gradient force, viscous force and lift force, which are obtained either by analytical or empirical expression in cooperated with kinetic and kinematic properties of fluid surrounding the particle (Gidaspow, 1994; Zhong et al., 2006; Crowe et al., 2011). The contact forces among particles can be described using the soft-sphere model or slide spring-dashpot model, in which the interaction between two overlap spherical particles is equivalent to a system consisting of a spring, a dashpot and a slider (Tsuji et al., 1992).
The motion of fluid phase is obtained by solving the volume averaged Navier-Stokes (NS) equation system for the incompressible fluid. It consists of mass and momentum conservations equations (Anderson and Jacson, 1967). Particularly, the volume fraction of fluid ε_f using centre volume method is deployed to take into account the volume occupied by the present of solid particles in fluid cell (Vångö et al., 2017):

$$\frac{\partial(\rho_f \varepsilon_f)}{\partial t} + \nabla \cdot (\rho_f \varepsilon_f \vec{u}_f) = 0 \tag{3}$$

$$\frac{\partial(\rho_f \varepsilon_f \vec{u}_f)}{\partial t} + \nabla \cdot (\rho_f \varepsilon_f \vec{u}_f \cdot \vec{u}_f) = -\varepsilon_f \nabla p - \varepsilon_f \nabla \cdot \overrightarrow{\tau_f} - \vec{F}^A + \rho_f \varepsilon_f \vec{g} \tag{4}$$

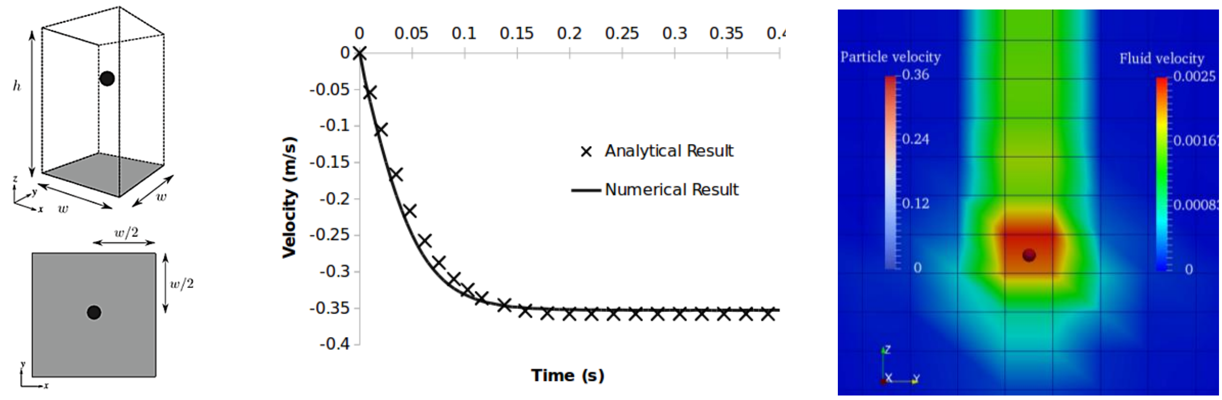
where $\rho_f, \vec{u}_f, p$ are fluid density, velocity and pressure, respectively; $\vec{g}$ is the gravitational acceleration. Momentum exchange from particles to fluid in the same fluid cell is calculated by Newton's third law and represented by volumetric average of all interaction forces from fluid to particles in that cell. The interaction forces are then simply separated to the fluid stress tensor by Jackson approach (Zhou et al., 2010). Therefore, total interaction force $\vec{F}^A$ can be estimated from drag force and lift force only.

NUMERICAL VALIDATION AND APPLICATION

Single particle sedimentation

For the purpose of validating the coupled CFD-DEM framework presented in the previous part of this paper, the sedimentation of a single aluminum particle in water is simulated and numerical results are compared with the analytical solutions.
Figure 1a shows the model geometry and initial setting conditions for the first test case. It consists of a 3 mm diameter spherical particle made of aluminum deposited in water, which is a Newtonian fluid with viscosity μ_f = 0.001 kg/ (m·s). The density of solid particle is ρ_p =2702 kg/m^3, which is approximately three times higher than the water density of ρ_f =996.51 kg/m^3. Initially, the particle is fixed in the water column of w = 0.2m in width and h=0.6m in height. This water column forms the simulation domain, which is discretised into the mesh of

(19×19×70) cells. The particle is located in the centre line at 0.8h from the bottom of the domain and then released to freely fall under the gravitational force.



(a) (b) (c)

Figure 1 Single particle sedimentation

Figure 1b shows a comparison between the numerical and analytical results for the evolution of the particle vertical velocity. After being released, the particle rapidly speeds up and reach a steady velocity of 0.35m/s after 0.15s. Compared to the analytical solution (Nouri et al., 2014), the result in this work shows a good agreement, suggesting that the coupled CFD-DEM model can approximately reproduce the sedimentation of a single particle in the fluid with a margin of error within 1%-7%. This implies that the interaction forces considered in the coupled CFD-DEM model, including gravity, buoyancy, drag and virtual mass effects, can qualitatively describe the motion of the solid particle in the fluid. However, as the fluid cell is considerably greater than the particle diameter, the velocity distribution of the fluid flow is smoothed out with the resolution, as shown in Figure 1c. Although this issue cannot be overcome by decreasing the mesh cell size because the unresolved methodrequires fluid cell volume must much larger than that of particle, the fluid flow can be approximated in the laminar regime. Nevertheless, the computational time required for the current CFD-DEM model is significantly less than those resolved methods, such as immersed boundary methods (Zhou et al., 2010), that require detail descriptions of particle boundaries. According to Sommerfeld (2000), there should be no limitation in the application of the Euler/Lagrange (i.e. unsolved methods) approach as long as all the required physics is modelled properly.

Submerged landslide

A submerged granular column collapse test conducted by Rondon et al. (2011) is simulated in this section to further demonstrate the applicability of the presented CFD-DEM approach. Figure 2 shows the initial setting conditions for the numerical simulation. The simulated granular sample has an initial aspect ratio of A=0.67, which is defined as the ratio of the initial height Hi =4.8 cm to the initial width Li = 6.0 cm of the soil column. The sample is generated using the expansion method (Lee et al., 2017) which uses 20000 glass bead particles to create an initial volume fraction of n = 0.60. These particles have a uniform diameter distribution of 1.2 mm, uniform mass density of 2500 kg/m^3 and contact friction angle between particles is 12^o.
As shown in Figure 3, the soil sample is initially placed in the bottom-left corner of the simulation domain which covers the box of 0.012cm × 0.3cm ×0.15cm. The domain is meshed into 2×84×42 cells to ensure that the cell volume is much larger than the particle volume but is still fine enough to capture essential mechanism of the fluid flow.Initially, the fluid with a density of 1000 kg/m^3 and a kinematic viscosity of 12 cps fills up the domain. The fluid can freely move in and out of the domain due to the pressure difference on the top boundary. The side walls and bottom boundaries are no-slip walls where the velocity of the fluid is zero and pressure is zero gradient. Front and back boundaries are assigned to be symmetry planes to emulate infinity depth in the 3D problem.

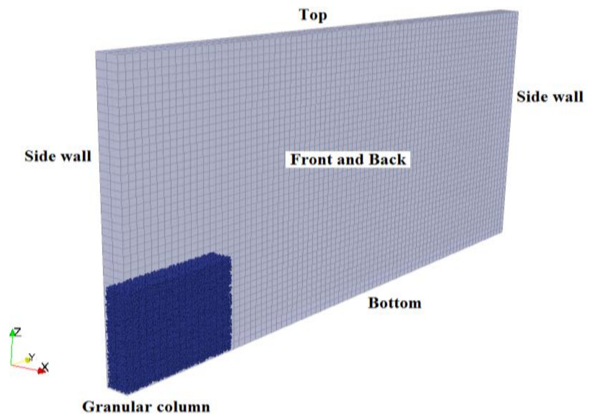
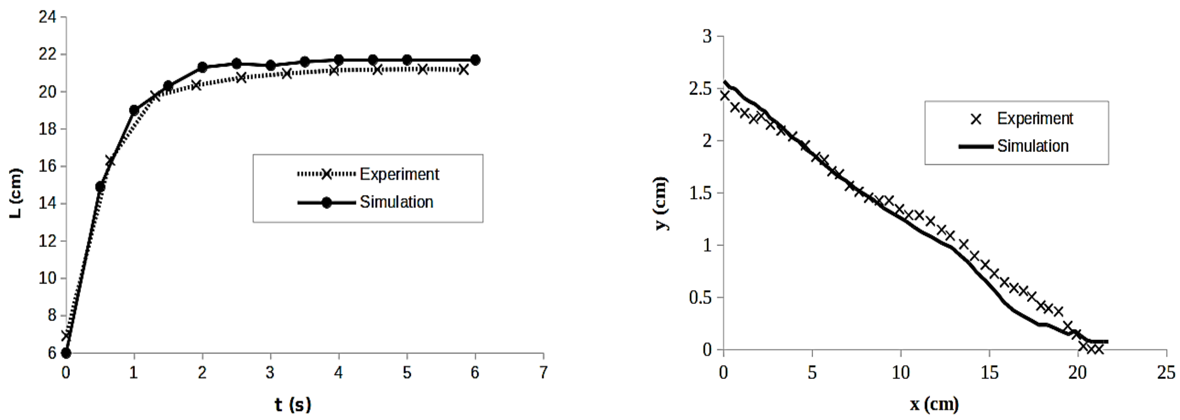


Figure 2 Simulation domain for submerged granular collapse

Figure 3 shows comparisons between the simulation and experiment for the time evolution of the runout distance and the final morphology of the granular column collapse. The runout distance in both experiment and simulation is measured by tracking the farthest particle that is still in contact with the main granular mass. Good agreement between the simulation and experiment is achieved with the margin of error within 5%, suggesting that the presented CFD-DEM model can qualitatively describe the behaviour of the multi-particle system in the fluid.



(a) (b)

Figure 3 Result comparison: (a) Time evolution of the front position; (b) Final morphology

Figure 4 shows the time evolution of granular morphology and velocity distribution of fluid and particles. After releasing the constraint on the right boundary, owing to the relatively low initial aspect ratio, the shear failure starts to happen as the granular material propagates in both vertical and horizontal directions. The rapid motion of granular mass at this stage generates fluid vortices, which move with the granular flow front. The vortices develop as the granular mass spreads and later disperse as the front slows down and reach the furthest runout distance as shown in Figure 4d and 4e. The formation and propagation of the vortex distribute the kinetic energy from the rapid shear at the beginning of the spreading process to the motion of particle flow afterward. As a result, the transformation of particle flow in submerged case is much slower than in dry case. It is largely attributed to the fact that different from the dry case where inertial is mainly responsible for the collapse of the granular column, in submerged case, the dynamics of granular flow is predominantly controlled by the fluid viscosity and pore water pressure (Kumar et al., 2017).

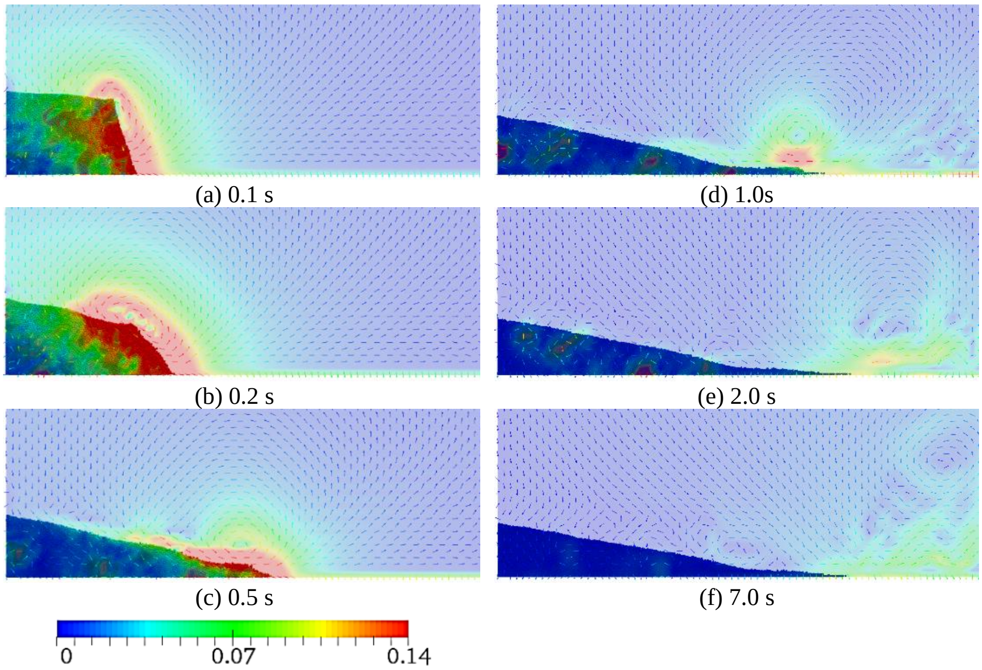


Figure 4. Velocity distributionover time

Nevertheless, the response of pore water pressure predicted by the implemented CFD-DEM model shows highly fluctuation. This can be attributed to numerical noise introduced by the void fraction calculation using the centre void fraction method (Vångö et al., 2017). However, the morphology, as well as the evolution of the front of the granular mass, is well captured. This suggests that the implemented coupled CFD-DEM framework can capture essential mechanisms of the multiphase-flow consisting of a fluid and dispersed discrete particles, thereby applicable to a wide range of applications in engineering.

CONCLUSION

In this study, a framework of coupled CFD-DEM is implemented in OpenFOAM and validated against analytical solution and experimental data available in the literature. The results in the case of single particle settling in Newtonian fluid shows a good agreement with analytical result, suggesting that the implemented model can capture essential interaction forces between the fluid and solid particle. The model is then employed to predict the morphology and the motion of the granular column collapse in water and good agreement between the simulation and experiment is achieved. This suggests that the current CFD-DEM model can be further applied to study the behaviour of multi-particle systems consisting of the continuum fluid and dispersed solid particles phases.

ACKNOWLEDGEMENT

Funding support from the Australian Research Council via projects DP160100775 (Ha H. Bui), FT140100408 (Giang D. Nguyen), DP170103793 (Nguyen, Bui) and DP190102779 (Bui & Nguyen) is gratefully acknowledged.

REFERENCES

Anderson, T.B. and Jackson, R., (1967). Fluid mechanical description of fluidized beds. Equations of motion. *Industrial & Engineering Chemistry Fundamentals, 6(4)*, pp.527-539.

Bui, H.H. and Nguyen, G.D., (2017). A coupled fluid-solid SPH approach to modelling flow through deformable porous media. *International Journal of Solids and Structures, 125*, pp.244-264.

Chen, F., Drumm, E.C. and Guiochon, G., (2011). Coupled discrete element and finite volume solution of two classical soil mechanics problems. *Computers and Geotechnics, 38(5)*, pp.638-647.

Cundall, P.A. and Strack, O.D., (1979). A discrete numerical model for granular assemblies. *Geotechnique, 29(1)*, pp.47-65.

Gidaspow, D., (1994). *Multiphase flow and fluidization: continuum and kinetic theory descriptions*. Academic press.

Han, Y. and Cundall, P.A., (2013). LBM–DEM modeling of fluid–solid interaction in porous media. *International Journal for Numerical and Analytical Methods in Geomechanics, 37(10)*, pp.1391-1407.

Hjelmfelt, A.T. and Mockros, L.F., (1966). Motion of discrete particles in a turbulent fluid. *Applied Scientific Research, 16(1)*,

pp.149-161.
Koivisto, J., Korhonen, M., Alava, M., Ortiz, C.P., Durian, D.J. and Puisto, A., (2017). *Friction controls even submerged granular flows. Soft matter, 13(41)*, pp.7657-7664.
Kumar, K., Delenne, J.Y. and Soga, K., (2017). Mechanics of granular column collapse in fluid at varying slope angles. *Journal of Hydrodynamics*, *29*(4), pp.529-541.
Lee, J.S., Tran, M.K. and Lee, C., (2012). Evolution of layered physical properties in soluble mixture: experimental and numerical approaches. *Engineering geology*, *143*, pp.37-42.
Lube, G., Huppert, H.E., Sparks, R.S.J. and Freundt, A., (2005). Collapses of two-dimensional granular columns. *Physical Review E, 72(4)*, p.041301.
Nouri, R., Ganji, D.D. and Hatami, M., (2014). Unsteady sedimentation analysis of spherical particles in Newtonian fluid media using analytical methods. *Propulsion and Power Research*, *3*(2), pp.96-105.
Rondon, L., Pouliquen, O. and Aussillous, P., (2011). Granular collapse in a fluid: role of the initial volume fraction. *Physics of Fluids, 23(7)*, p.073301.
Schwarzkopf, J.D., Sommerfeld, M., Crowe, C.T. and Tsuji, Y., (2011). *Multiphase flows with droplets and particles*. CRC press.
Sommerfeld, M., (2000). Theoretical and experimental modelling of particulate flows. *Lecture series*, *6*, pp.3-7.
Tsuji, Y., Tanaka, T. and Ishida, T., (1992). Lagrangian numerical simulation of plug flow of cohesionless particles in a horizontal pipe. *Powder technology, 71(3)*, pp.239-250.
Vångö, M., Pirker, S. and Lichtenegger, T., (2018). Unresolved CFD–DEM modeling of multiphase flow in densely packed particle beds. *Applied Mathematical Modelling*, *56*, pp.501-516.
Xiao, H. and Sun, J., (2011). Algorithms in a robust hybrid CFD-DEM solver for particle-laden flows. *Communications in Computational Physics, 9(2)*, pp.297-323.
Zhong, W., Xiong, Y., Yuan, Z. and Zhang, M., (2006). DEM simulation of gas–solid flow behaviors in spout-fluid bed. *Chemical Engineering Science*, *61*(5), pp.1571-1584.
Zhou, Z.Y., Kuang, S.B., Chu, K.W. and Yu, A.B., (2010). Discrete particle simulation of particle–fluid flow: model formulations and their applicability. *Journal of Fluid Mechanics, 661*, pp.482-510.