International Journal of Heat and Mass Transfer 126 (2018) 452–465

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

An efficient multi-grid finite element fictitious boundary method for particulate flows with thermal convection

Khuram Walayat [a,b], Zekun Wang [a,b], Kamran Usman [c], Moubin Liu [a,b,*]

[a] BIC-ESAT, College of Engineering, Peking University, Beijing 100871, China
[b] StateKey Laboratory for Turbulence and Complex Systems, Department of Mechanics and Engineering Science, Peking University, Beijing 100871, China
[c] Department of Mathematics, Faculty of Basic and Applied Sciences, Air University, Islamabad 44000, Pakistan

A R T I C L E I N F O

Article history:
Received 16 October 2017
Received in revised form 30 March 2018
Accepted 1 May 2018

Keywords:
Direct numerical simulation
Fictitious boundary method
Multigrid
Particulate flows
Thermal convection
Sedimentation

A B S T R A C T

This paper presents a direct numerical simulation (DNS) technique, the Finite Element Fictitious Boundary Method (FEM-FBM) for the simulation of fluid-solid two-phase flows with heat transfer. The heat transfer equation is introduced to study thermal convection in fluid-solid two-phase flows. The Boussinesq approximation is considered for the coupling of momentum and temperature flow fields. Multi-grid finite element solver is used to compute flow equations of mass, momentum, and energy on a fixed Eulerian mesh which is independent of time and the solid particles are allowed to move freely in the whole computational domain. Fictitious boundary method (FBM) is used to treat the particles inside the fluid, FBM takes account of the thermal and momentum interaction between the fluid and the particles. The accuracy and stability of presented method are validated by comparing our test cases results with the results reported in the available literature. Numerical tests are performed to show that this method is potentially powerful and provides an efficient approach to simulate complex thermal convective particulate flows with a large number of particles.

1. Introduction

Particulate flows or motion of solid particles in fluids have a wide range of industrial applications, such as fluidized suspensions, lubricated transport, sedimentation, hydraulic fracturing of reservoirs, slurry flow, paper pulp, food products etc. These types of flows are common in many natural processes such as sand or dust particles in air blown by the wind, ocean current interaction with offshore structures, lava flow and sedimentation in estuary etc. Particulate flows in biological processes have been a subject of great importance with research contributions coming from the field of biology, chemistry, physics, engineering, and mathematics. The sedimentation of suspended particles has a great importance in the chemical, petroleum, paper pulp, wastewater, food, pharmaceutical, ceramic and other industries as a way of separating particles from the fluid as well as separating particles of different types settling with different speeds. Here we are particularly interested in the gravitational settling of particles with convection heat transfer.

Particulate flows are quite complex and hard to simulate numerically because frequent generation and deformation of the computational grid are required in many cases when the particle boundaries are complex and moving with time. The problem becomes more complex in the case with a large number of particles due to fluid-particle interaction as well as due to particle-particle and particle-wall collisions. Rapid advancements in computational power make the direct numerical simulation (DNS), an important and practical tool to study particulate flow mechanism. It treats the fluid and solid objects separately. The DNS approach is based on Navier–Stokes equation for the fluid and equations of rigid body motion for particles. A variety of DNS numerical schemes have been proposed over the past decade to simulate fluid-particle flow problems. These methods are broadly classified into two types, one is based on Lagrangian approach while other is a Eulerian approach. In Lagrangian approach, the mesh moves and follows the moving boundaries of the particles in the fluid. Since the motion of the mesh can be defined arbitrarily within the fluid, therefore this approach is usually called Arbitrary Lagrangian Eulerian (ALE). Hu et al. [14,15], Maury [12,13] and Feng et al. [16] have used the ALE method to study particulate flows. ALE method normally requires generating a new mesh at every time step in the case of moving particles, so it is computationally expensive especially for the simulations of problems with large number of

* Corresponding author at: BIC-ESAT, College of Engineering, Peking University, Beijing 100871, China.
E-mail address: mbliu@pku.edu.cn (M. Liu).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.05.007
0017-9310/

particles. Whereas the Eulerian approach is more efficient than the Lagrangian one but the resulting accuracy is often not as clear as in case of Lagrangian. Therefore, the overall objective is to deal with the moving boundaries in the fluid, improve the accuracy of the numerical approximation and reduce the computational cost. Eulerian methods do not require re-meshing, a fixed Cartesian mesh is required which covers the whole computational domain comprising of both particles and fluid. Peskin [17] introduced immersed boundary method (IBM), based on a Eulerian approach to study fluid-solid interaction problems. Similar to IBM, Glowinski et al. [7,18,19] developed a finite element fictitious domain method to simulate fluid-particle flow problems. Turek et al. [1,2,3,10] presented a multigrid finite element fictitious boundary method (FBM) for the simulation of particulate flows. In recent past, many hybrid methods have been developed to simulate sedimentation of suspended particles in fluid, For example, lattice Boltzmann method (LBM) is coupled with other numerical methods, such as IBM [27], Finite Element Method (FEM) [28] and Discrete Element Method (DEM) [29]. Sun et al. [31] and Hager et al. [30] used another hybrid method CFD-DEM, in which computational fluid dynamics (CFD) is coupled with DEM for the study of sediment transport.

Although, heat transfer in particulate flows is involved in many industrial applications, yet only a few results are found in the literature. Among the reported studies, Kim et al. [35] proposed an Immersed Boundary Method (IBM) in context of a Finite Volume (FV) fixed grid scheme for the solution of heat transfer in complex two-dimensional geometries without considering the solid-phase motion. Pacheco et al. [25] also presented FV-IB method with a non-staggered grid technique for heat transfer and fluid flow around fixed particles. Demirdžić et al. [24] used finite volume non-orthogonal boundary-fitted grids to study the thermal convection around fixed solid-phase. Gan et al. [20,21] used ALE scheme for the simulations of two-dimensional particle sedimentation problem with heat transfer between particles and the surrounding fluid. Yu et al. [22] employed fictitious domain method to solve particulate flow problem and extended it to study heat convection at the interface between the fluid and solid particles. Feng et al. [11,34,37] developed finite volume IBM method for heat transfer between the fluid and moving solid particles in particle-laden flows. The numerical study of thermal convection in particulate flows has been attempted by some other researchers as well [36,38,39].

In the present work, we apply the multi-grid FEM fictitious boundary method [1,2,3,10,33] to simulate particle sedimentation problems and extend this method to study heat transfer in solid-liquid two-phase flows. The considerable advantage of multi-grid FEM fictitious boundary method which makes it an efficient scheme is that it is based on a fixed FEM background grid which is independent of flow features, hence re-meshing is not required with the edge of high convergence rate of the multi-grid solver. By applying boundary conditions at the interface between fluid and particles which become an additional constraint to the governing equations, so the fluid domain can be extended into the whole domain which covers both fluid and particles.

This paper is organized as: in the second section, we present the mathematical modeling and governing equations of the problem and discussed fluid-particle interaction. The computational scheme is described in the third section. The fourth section consists of numerical experiments to check the validation of our method and test its efficiency and potential to simulate real particulate flows. The conclusion of the presented study is given in the final section.

2. Mathematical modeling

We consider the flow of N number of solid particles of mass $M_i (i = 1, 2, \ldots, N)$ in an incompressible Newtonian fluid, as shown in Fig. 2.1. The density of the fluid is ρ_f and its viscosity is ν. $\Omega_f(t)$ and $\Omega_i(t)$ denotes the domain occupied by the fluid and the ith particle at time t respectively.

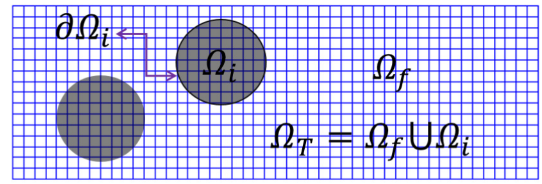

Fig. 2.1. Rigid particle and fluid.

where Ω_T is the total domain and is given as,

$$\Omega_T = \Omega_f(t) \cup \Omega_i(t) \quad \forall\, i \in \ (1, 2, \ldots, N) \tag{2.1}$$

Ω_T as an entire computational domain is independent of t. As Ω_f and Ω_i are always depended on time t we denote $\Omega_f(t) = \Omega_f$ and $\Omega_i(t) = \Omega_i$ dropping t in the notations. Where $\partial\Omega_i$ represents the boundary of the ith particle.

2.1. Fluid flow model

The motion of an incompressible fluid with density ρ_f is governed by the equations of continuity, momentum, and energy in the domain Ω_f as,

$$\begin{aligned} &\nabla \cdot \boldsymbol{u} = 0 \quad (a)\\ &\rho_f \left[\frac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u} \cdot \nabla \boldsymbol{u} \right] - \nabla \cdot \boldsymbol{\sigma} = f \quad (b)\\ &\rho_f c_f \left[\frac{\partial T}{\partial t} + \boldsymbol{u} \cdot \nabla T \right] = k_f \nabla^2 T \quad \forall\, X\ \in \Omega_T \quad (c) \end{aligned} \tag{2.2}$$

σ is the total stress tensor in the fluid phase defined as,

$$\boldsymbol{\sigma} = -p\boldsymbol{I} + \mu_f [\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T]. \tag{2.3}$$

where $\boldsymbol{u}$ is the fluid velocity, p is the pressure, μ_f is the coefficient of viscosity, f is the source term and $\boldsymbol{I}$ is the identity tensor. T is the temperature, c_f is specific heat and k_f is the thermal conductivity of the fluid.

2.2. Particle motion model

The rigid particles are free and allowed to move in the fluid domain. The particles have both translational and rotational motion under the action of gravity, forces due to fluid called hydrodynamic forces and collision forces due to interactions between particle-particle or particle-wall. The motion of solid particles is governed by the Newton-Euler equations [2,9], i.e. if $\boldsymbol{U}_i$ and $\boldsymbol{\omega}_i$ are the translational and angular velocities of the ith particle respectively, then the particle must satisfy the following equations,

$$m_i \frac{d\boldsymbol{U}_i}{dt} = (\Delta m_i)\boldsymbol{g} + \boldsymbol{F}_i + \boldsymbol{F}_i', \quad I_i \frac{d\boldsymbol{\omega}_i}{dt} + \boldsymbol{\omega}_i \times (I_i \boldsymbol{\omega}_i) = \boldsymbol{\tau}_i, \tag{2.4}$$

where M_i is the mass of the ith particle and if M_f is the mass of fluid occupying the same volume as M_i then ΔM_i is given by the mass difference between the mass of the ith particle M_i and the mass of the fluid M_f,

$$\Delta M_i = M_i - M_f, \tag{2.5}$$

$\boldsymbol{F}_i$ represents resultant hydrodynamic i.e. drag and lift forces acting on the ith particle, $\boldsymbol{F}_i'$ are the collision forces acting on the ith particle due to particle-particle and particle-wall collision, $\boldsymbol{I}_i$