Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **15**, No. 2, pp. 359-375

DOI: 10.4208/aamm.OA-2021-0064
April 2023

Floating Solid Particles Interacting with Tilted Square Obstacles in Fluid Affecting Drag Forces

K. Usman*, S. Jabeen and M. Shahid

Department of Mathematics, Air University, PAF Complex, Islamabad 44000, Pakistan

Received 3 March 2021; Accepted (in revised version) 28 September 2021

Abstract. We have analyzed the kinetics of solid circular particles interacting with fluid, outer boundary and internal square shaped obstacles tilted at a 45° angle. The effects on the motion of particle due to collision with obstacles and wall are inspected. An Eulerian approach is used to study the behavior of particle in the fixed computational mesh. The interactions between fluid, particles and obstacles have been carried out in the whole domain by using fictitious boundary method (FBM). In this work, the particulate flow simulations are computed by using finite element solver FEATFLOW. Numerical results are presented by assigning different alignments to the obstacles and varying their positions in the domain. Particle-wall, particle-particle and particle-obstacle collisions are treated by applying a modified collision model proposed by Glowinski et al. The rapid change in drag forces acting on obstacles due to nearby passing particles and its effect on the fluid motion has been investigated.

AMS subject classifications: 76M10

Key words: Particulate flow, direct numerical simulation, fictitious boundary method, finite element method, multigrid, sedimentation.

---

1 Introduction

Particulate flows [17,19] have a wide range of applications in natural and industrial processes. Some examples exhibiting such flows are wind blown small dust particles, sand flow, multiphase flow, filtration, melting, purification of petroleum fluid. Similarly, collisions of numerous particles with each other and their interactions with obstacles inside the boundary has a wide range of applications in industry. Due to continuous deformation and generation of computational grid, in Lagrangian approaches, the process of simulation becomes much more difficult when the particles have complex boundaries. The study of particle-particle and particle-wall collisions gets more complicated in fluid while simulating a large number of particles. When the particles surfaces reach other

*Corresponding author.
Email: kamran.usman@mail.au.edu.pk (K. Usman)

nearby surfaces, the gap between such surfaces need to be refined too much to avoid surface overlap but it increases the cost of simulation. A variety of numerical techniques are available to overcome the collisions and overlapping between particles.

For the purpose of simulating particulate flows, a lot of numerical techniques have been developed which include level-set methods [9, 28], penalty based method [26], discrete element models (DEM) [7, 8], population balance based models [17] and distributed Lagrange multiplier (DLM) fictitious domain methods [17, 29, 36–38]. A wide range of industrial problems have been examined using Discrete Element Methods (DEM) simulations [7, 8]. This method deals with solid particulate flows such that the interaction of particles with each other and with their environment is analyzed. Abbasi et al. [1] studied the flow around obstacles inside a channel taking one structure (obstacle) in wake of another using Lattice-Boltzmann technique. The mechanism depends upon the arrangement, gap/distance, shape and size of structures. To find the solution of 2D unsteady compressible Navier-Stokes equations, Inoue et al. [14] presented finite difference method for the analysis of two square cylinders placed on staggered positions. Wang et al. [42] considered computational mesh as a horizontal soap film tunnel and placed two circular cylinders in staggered positions to investigate the behavior. Space-time finite-element method has been implemented for analysis of the numerical simulations. Brady and Bossis [5] have proposed stokes flow assumption considering the stokesian dynamic techniques and have studied the particulate flow for the behavior of two cylinders. Solid particles are kept near each other in Lattice Boltzmann simulations [21] and stokes multiple simulations [5] to discuss the effect of lubrication forces. Chapman, Ogawa and Rao et al. [6, 22, 25] have investigated the flow mixtures including the rapid granular flow and slow granular flow. Recently, Sokolov and coauthors [28] have analyzed the results obtained from the solutions of partial differential equations using a level set method [9, 28]. To investigate particulate flows, mainly two different approaches for the computational grid are adopted namely the Eulerian approach, in which the computational mesh remains fixed and the Lagrangian approach, in which the mesh nodes move along with the motion of the particle boundary. Joseph and Glowinski proposed a DLM technique [11, 23, 27] based on the Eulerian approach. Eulerian approaches reduce the simulation cost but some times the accuracy of the resulting resolved flow fields is not as good as in Lagrangian approaches. The flow field around each particle, between the fluid-particle interface, has to be calculated with accuracy which needs either a very fine mesh in case of Eulerian approach or fine re-meshing of the domain near the interface in case of Lagrangian approach.

Fictitious Domain Methods (FDM) and Fictitious Boundary Methods (FBM) have become very popular over the last decade. In these methods, the governing Navier-Stokes equations are solved by introducing extra boundary conditions in them which arises due to the free moving particle boundaries and the whole computation is carried out on a FEM background grid [17, 29, 36–38]. Patankar et al. [13, 23] considered a stress field inside the particle domain and related this field with the DLM method. By this strategy they claimed that there is no need to calculate the translational and angular velocity for