INTERNATIONAL JOURNAL OF
NUMERICAL ANALYSIS AND MODELING
Volume 5, Supp , Pages 1–16

A FORMULATION FOR FULLY RESOLVED SIMULATION (FRS) OF PARTICLE-TURBULENCE INTERACTIONS IN TWO-PHASE FLOWS

SOURABH V. APTE AND NEELESH A. PATANKAR

Abstract. A numerical formulation for fully resolved simulations of freely moving rigid particles in turbulent flows is presented. This work builds upon the fictitious-domain based approach for fast computation of fluid-rigid particle motion by Sharma & Patankar ([1] Ref. J. Compt. Phy., (205), 2005). The approach avoids explicit calculation of distributed Lagrange multipliers to impose rigid body motion and reduces the computational overhead due to the particle-phase. Implementation of the numerical algorithm in co-located, finite-volume-based, energy conserving fractional-step schemes on structured, Cartesian grids is presented. The numerical approach is first validated for flow over a fixed sphere at various Reynolds numbers and flow generated by a freely falling sphere under gravity. Grid and time-step convergence studies are performed to evaluate the accuracy of the approach. Finally, simulation of 125 cubical particles in a decaying isotropic turbulent flow is performed to study the feasibility of simulations of turbulent flows in the presence of freely moving, arbitrary-shaped rigid particles.

Key Words. DNS, particle-turbulence interactions, point-particle, fully resolved particles.

1. Introduction

Many problems in nature and engineering involve two-phase flows where solid particles of arbitrary shape and sizes are dispersed in an ambient fluid (gas or liquid) undergoing time dependent and often turbulent motion. Examples include sediment transport in rivers, fluidized beds, coal-based oxy-fuel combustion chambers, biomass gasifiers, among others. These applications involve common physical phenomena, at disparate length and time scales, of mass, momentum, and energy transport across the interface between the dispersed particles and a continuum fluid.

Numerical simulations of these flows commonly employ Lagrangian description for the dispersed phase and Eulerian formulation for the carrier phase. Depending on the volumetric loading of the dispersed phase two regimes are identified: dilute ($d_p << \ell$) and dense ($d_p \approx \ell$), where d_p is the characteristic length scale of the particle (e.g. diameter), and ℓ the inter-particle distance. Furthermore, the grid resolution (Δ) used for solution of the carrier phase could be such that the particles are 'subgrid' ($d_p << \Delta$), 'partially resolved' ($d_p \sim \Delta$), or 'fully resolved' ($\Delta < d_p$). In addition, these regimes may occur in the same problem and are dependent on the particle size as well as the grid resolution. Clearly, multiscale numerical approaches are necessary to simulate various regimes of the flow.

Received by the editors September 30, 2007 and, in revised form, May 7, 2008.
2000 *Mathematics Subject Classification.* 35R35, 49J40, 60G40.

The standard approach to simulate turbulent particle-laden flows uses direct numerical simulation (DNS) [2, 3, 4], large-eddy simulation (LES) [5, 6, 7, 8] or Reynolds-Averaged Navier Stokes (RANS) approach [9] for the carrier phase whereas the motion of the dispersed phase is modeled. In all these approaches, the particles are assumed 'point-sources' compared to the grid resolutions used (so $d_p < \eta$, the Kolmogorov length scale, for DNS whereas $d_p < \Delta$, the grid size, in LES or RANS). The fluid volume displaced by the particles are presumed negligible. Recently, an improved approach, based on mixture theory and considering the volumetric displacements of the fluid by the particles was developed for DNS/LES of particle-laden flows [10]. However, both of these approaches model the interactions between the fluid and the particles, use drag and lift correlations to approximate the drag and lift forces on the particles. The accuracy of these approaches in capturing the complex particle-fluid interactions in turbulent flows depends on the validity of simplified drag and lift laws.

Fully resolved simulation (FRS) of these flows require grid resolutions finer than the characteristic size of particles. In this approach, all scales associated with the particle motion are resolved and the drag/lift forces on the particles are *directly* evaluated rather than modeled. Considerable work has been done on fully resolved simulations of particles in laminar flows. Arbitrary Lagrangian-Eulerian (ALE) method [11], distributed Lagrange multiplier/fictious domain (DLM) based methods [12, 13, 14], Lattice Boltzmann (LBM) [15], and Immersed Boundary (IBM) based methods [16, 17, 18] have been proposed and used. These methods have been applied to simulate a modest number of particles (around 1000s) at low Reynolds numbers. In spite of several different numerical schemes, full three-dimensional direct simulations of two-phase turbulent flows in realistic configurations are rare. There are only few three-dimensional turbulent flow studies in canonical configurations on fully resolved rigid particles [19, 20] . There appears to be no reported study of fully resolved moving particles in complex geometries. In the present work, a fictitious domain based approach for motion of arbitrary rigid particles is implemented in a structured finite-volume solver capable of simulating turbulent flows. The approach is based on an efficient numerical algorithm proposed by Patankar [21] to constrain the flow field inside the particle to a rigid body motion. This facilitates simulation of large-number of particles by reducing the overhead associated with the computation of particle motion.

The paper is arranged as follows. A mathematical formulation of the basic scheme is described briefly. Numerical implementation of the scheme in a co-located grid, finite volume framework is provided next. The numerical scheme is validated for flow over a fixed sphere at different Reynolds numbers and a freely falling sphere under gravity. Simulation of 125 cubic particles in an isotropic turbulent flow is then performed to show the feasibility of the approach to capture multiscale interactions between the particles and unsteady turbulent flows.

2. Mathematical Formulation

Let Γ be the computational domain which includes both the fluid (Γ_F) and the particle ($\Gamma_P(t)$) domains. Let the fluid boundary not shared with the particle be denoted by $\mathcal{B}$ and have a Dirichlet condition. For the present work, focus is placed on flows in closed domains and thus number of particles inside the computational domain remains fixed. Further evaluation of generalized boundary conditions is necessary for inflow-outflow systems where number of particles inside the domain may vary with time. For simplicity, let there be a single particle in the domain