Advanced Powder Technology 25 (2014) 1860–1871

Contents lists available at ScienceDirect

Advanced Powder Technology

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

Original Research Paper

A new relation of drag force for high Stokes number monodisperse spheres by direct numerical simulation

Ali Abbas Zaidi, Takuya Tsuji, Toshitsugu Tanaka *

Department of Mechanical Engineering, Osaka University, Suita 565-0871, Japan

ARTICLE INFO

Article history:
Received 7 May 2014
Received in revised form 3 July 2014
Accepted 31 July 2014
Available online 19 August 2014

Keywords:
Drag force law
Gas–solid flows
Multiphase flows
Immersed Boundary Method
Direct numerical simulation

ABSTRACT

Immersed Boundary Method simulations are used to propose a drag correlation for randomly arranged monodisperse spheres. The solid volume fraction is varied from 0.05 to 0.5 and Reynolds number is varied from 0.01 to 1000. The particles have constant relative velocity to fluid and thus the present results are applicable to high Stokes number gas–solid flows. Average drag force acting on particles in both regular and random arrangements is compared with literature and good agreement is observed. At the end a new drag relation for monodisperse spheres is proposed and the physics behind its development is explained.

1. Introduction

Gas–solid flows are described by flow of gases with solid particles. Scientists and engineers have been interested in such flows for many years. Furthermore, gas–solid flows found applications in many industrial processes such as pneumatic transport, particulate pollution control, combustion of pulverized coal, drying of food products, sand blasting, plasma-arc coating and fluidized bed processes. To improve the efficiency and performance of these processes the understanding of the physics and dynamics of gas–solid flows are required, however, we still have to rely on empirical correlations or models especially for the moderate and high particle Reynolds number cases and further investigation is required for these cases.

Computer simulations can be useful for the understanding of such flows and design and scale-up of such processes. For the wide use of computer simulations for designing systems, appropriate models are necessary and the methods are desired to be capable of simulating large size systems and should have small computational time. These methods include Eulerian-Eulerian methods based on two-fluid models [1,2] and Eulerian–Lagrangian methods based on discrete particle models [3–7]. The two-fluid model assumes both the gas and the solid phases as inter-penetrating continua and is especially useful and computationally cost efficient when the volume fractions of the phases are comparable, or when the interaction within and between the phases plays a significant role in determining the hydrodynamics of the system. On the other hand, in discrete particle models, locally averaged fluid equations are solved by continuum approach but particle-particle interactions are solved by discrete element methods [8] or by hard-sphere models. These methods based on discrete particle models are computationally more expensive for dense solid flows, however require less assumptions and the particle properties like particle size and density distribution can be directly taken into account in the simulation.

In either of the above two approaches, flow domain is divided into cells for fluid flow calculation, the size of which is smaller than the mesoscopic structures like bubbles or clusters of particles but larger than the particle size. The model equations rely on various constitutive relations to account for the many unknown terms emerging from averaging—fluid-particle drag, added-mass, lift, history force, and particle and fluid phase stresses. Among all these terms, the fluid-particle drag is particularly important for gas–solid suspensions. It is usually the primary force to suspend and transport the particles and have significant influence on the results. For example Benyahia [9], Gomez and Milioli [10], Heynderickx et al. [11] and Wang et al. [12] compared solid-volume-fraction variations in the riser of circulating fluidized bed and observed that the drag laws based on empirical relations give more homogenous structures as compared to experiments. Du et al. [13] compared voidage profiles, particle velocity profiles and solid flow patterns in spouted beds and observed different results by different drag relations. Bokkers et al. [14] analyzed the effect of drag law on

* Corresponding author. Tel./fax: +81 06 6879 7316.
E-mail address: tanaka@mech.eng.osaka-u.ac.jp (T. Tanaka).

http://dx.doi.org/10.1016/j.apt.2014.07.019

bubble formation. Leboreiro et al. [15] simulated segregation in gas–solid fluidized bed and observed the equations obtained from direct numerical simulation gave better results.

In literature from theoretical viewpoint, little is known about the drag force on particles. Most of the studies are limited to low Reynolds number (Re) and solid volume fractions (φ). Some of the examples of theoretical works are the work of Hasimoto [16], Sangani and Acrivos [17], Kim and Russel [18] and Goldstein [19]. However, practical applications of gas–solid flow involve high Reynolds number thus one has to rely on empirical relations. For example, Ergun [20] proposed a relation for pressure drop in packed bed based on his experiments, Richardson and Zaki [21] based on their sedimentation experiments proposed a relation for the determination of hindered settling velocity at different solid volume fractions. Wen and Yu [22] also conducted a series of fluidization experiments and proposed a drag relation. Based on these empirical relations some modifications can also be found in literature. For example, one of the most widely used relations in chemical engineering is the relation proposed by Gidaspow [23]. He combined Ergun [20] and Wen and Yu [22] relation. He suggested the use of Ergun [20] relation for $\varphi > 0.2$ and Wen and Yu [22] relation for $\varphi < 0.2$. One of the shortcoming of this relation is at $\varphi = 0.2$ there is a discontinuity in the calculation of drag force and this discontinuity increases with the Reynolds number. Di Felice [24] suggested that the exponent in Wen and Yu [22] equation should not be constant but a function of Reynolds number. Syamlal et al. [25] proposed their drag relation by converting terminal velocity correlations by Richardson and Zaki [21].

With the advancement of computing power the direct numerical simulation (DNS) has become possible. The advantages of DNS are better control of the setup like particle size and shape without the need of closure relations. Furthermore, simulations can also give better insight and understanding of the physics of flow. Some of the early works for the calculation of drag force on porous media and cylinders are made by Koch and Ladd [26], Andrade et al. [27] and Rojas and Koplik [28]. Later, three dimensional simulations are performed by Hill et al. [29,30] for drag force calculation for ordered and random arrangements of monodisperse spheres for Reynolds number up to 100. Van der Hoef et al. [31] extended the work for poly-disperse spheres and Stokes flow condition. Beetstra et al. [32] simulated flow through poly-disperse spheres for Reynolds number up to 1000. Yin and Sundaresan [33,34] and Holloway et al. [35] performed simulations for poly-disperse spheres with relative motion for Stokes and moderate Reynolds number flow. All these simulations are based on the SUSP3D code based on lattice Boltzmann method (LBM) developed by Ladd [36,37]. Recently Tenneti et al. [38] proposed a new drag relation for monodisperse spheres using Immersed Boundary Method (IBM) with much refined grid resolution up to $Re = 300$. Their work raised some questions for the drag relations proposed by LBM. For example, their relation showed large deviation from the relation proposed by Beetstra et al. [32] (about 38% over-estimation) at $Re = 300$. Furthermore, they also mentioned that the simulation setup used by Holloway et al. [35] do not satisfy the Galilean invariance of particles with relative motion and may results an error of about 10%. Thus in literature there is still ambiguity in the development of drag relation by direct numerical simulation even for monodisperse spheres and there is still space for further research and investigations. The main aim of this paper is to propose a drag relation for randomly arranged monodisperse spheres. The new findings in this paper are the drag relation for extended range of Reynolds number ($Re \leqslant 1000$) and solid volume fractions for monodisperse spheres and explanation of physics of the correlation for low and high solid volume fractions.

The sequence of the present paper which we will follow: After introduction we will give a brief overview of the formulation used in the present simulations which will be followed by simulation setup and testing of parameters of simulation setup. Then we will discuss about the benchmarking of our simulation results with other researchers. Later we curve fit our simulation data and propose a new drag correlation. At the end we will conclude the paper. Because of the frequent use of some of the references in this paper from now on we will use HEL for Hill et al. [29,30], BEL for Beetstra et al. [32] and TEL for Tenneti et al. [38].

2. Formulation

For drag force calculation, we have used Immersed Boundary Method (IBM). IBM is a class of non-body conformal grid methods. Since the grid does not conform to the solid boundary, imposition of solid boundary requires modifications of fluid equations near the solid boundary. This can be done either by indirect or direct forcing [39]. In indirect forcing, the no-slip condition at solid surface is imposed by the use of smoothed distribution function on the fluid grid near the boundary. This leads to smoothing of the forcing function and spreading its effects over a band of fluid cells; thus diffusing the sharp representation of the solid surface. It is particularly undesirable for high Reynolds number flows. In direct forcing, computational stencil near the immersed boundary is directly modified by interpolation. Thus the sharp interface of the solid surface near the immersed boundary can be obtained. Furthermore, if the resolution of fluid grid is fine then the shape of solid surface can be reasonably reproduced.

In the present article, direct forcing type IBM is used. We will explain only the main points, detail of this method can be found in Kajishima et al. [40]. In the IBM the fluid can be liquid or gas but it is assumed to be incompressible and Newtonian. The solid spherical particle is assumed to be rigid. The grid size used for discretizing the computational domain is smaller than size of particles and fluid flow equations are solved by assuming that the fluid occupies the entire flow field and the effect of particles is expressed by a body force in the momentum equation which constrains the no slip boundary condition at the particles surface. The equations of continuity and incompressible Navier–Stokes equation without gravity effects is given by:

$$\nabla \cdot \boldsymbol{u}_f = 0 \tag{1}$$

$$\frac{\partial \boldsymbol{u}_f}{\partial t} + \boldsymbol{u}_f \cdot \nabla \boldsymbol{u}_f = \nu \nabla^2 \boldsymbol{u}_f - \frac{\nabla p}{\rho} \tag{2}$$

where $\boldsymbol{u}_f$ is the fluid velocity, ρ is density, ν is the kinematic viscosity and p is pressure. For improving the efficiency of numerical integration the following fluid-particle volume-weighted velocity ($\boldsymbol{u}$) is defined as:

$$\boldsymbol{u} = \alpha \boldsymbol{v}_p + (1 - \alpha)\boldsymbol{u}_f \tag{3}$$

where v_p is the particle velocity and α is the volume fraction of particle in a target cell. α takes the value zero for fluid and one for particle and in the range of zero to one at the particle interfacial cell.

Time-marching of momentum equation in IBM consist of two steps. In the first step the fluid velocity is predicted by using Eq. (4):

$$\tilde{\boldsymbol{u}} = \boldsymbol{u}_{n0} + t\left(-\frac{\nabla p}{\rho} - \boldsymbol{u} \cdot \nabla u + \nu \nabla^2 \boldsymbol{u}\right) \tag{4}$$

where $n0$ denotes the previous time step. In the second step this predicted velocity $\tilde{u}$ is used to calculate forcing term (f_p) in Eq. (5) for the calculation of fluid velocity at $n0 + 1$ time step. The forcing term is given by:

$$\boldsymbol{f}_p = \alpha(\boldsymbol{v}_p - \tilde{\boldsymbol{u}})/\Delta t \tag{5}$$