Available online at www.sciencedirect.com
ScienceDirect
Powder Technology 184 (2008) 232–240
POWDER TECHNOLOGY
www.elsevier.com/locate/powtec

Motion of particle agglomerate involving interparticle force in dilute suspension

Shintaro Takeuchi [a,*], Isao Morita [b], Takeo Kajishima [b]

[a] Department of Mechanical Engineering, The University of Tokyo, 7-3-1 Hongo, Bunkyo, Tokyo 113-8656 Japan
[b] Department of Mechanical Engineering, Osaka University, 2-1 Yamada-oka, Suita-city, Osaka 565-0871 Japan

Available online 3 December 2007

Abstract

Particle assembly process and settling behaviours in a fluid are studied with a small-population particle system involving an interparticle force. Particle motions are solved individually by the equation of linear and angular momentums, and fluid phase is solved by direct numerical simulation resolving from the boundary layer of the particle surface to the far wake. Interaction between the solid and fluid phases is treated by an immersed boundary method proposed by the present authors (Kajishima et al., *JSME Int. J. Ser. B* 44-4 (2001) 526; *Int. J. Heat and Fluid Flow* 23 (5) (2002) 639). The particles form agglomerate due to the interparticle force. The descent velocities of the agglomerates of various sizes and interparticle strengths are higher than that of a single particle. By varying the parameters of the interparticle forces, transitions in shape and descent velocity of the agglomerates are observed depending on the balance of interparticle and hydrodynamic forces. The following correlations are found: fluid drag per unit area of the agglomerate with respect to the area of the convex closure of the agglomerate projected on a plane normal to the descent direction, drag coefficients of the agglomerate with respect to the squared agglomerate Reynolds number defined with descent velocity and equivalent diameter of the convex closure.

Keywords: Particle agglomerate; Particle structure; Interparticle force; Transition; Direct numerical simulation (DNS); Immersed boundary method

1. Introduction

Particle-induced flows encountered in environmental problems, industrial applications and biological flows often exhibit various flow characteristics over a wide range of time and spatial scales. Examples include transportation of aerosols, cloud formation in the atmosphere, transition of particle flow patterns in a fluidized bed, particle flocculation in a water purification system and aggregation of platelet under shear stress. Along with the effects of particle-induced turbulence and fluid-particle interaction, interparticle force plays an important role in determining the characteristics of the flow. The effect of the interparticle force becomes more dominant with decreasing size of the particles [1], and the behaviour of the system could become harder to predict.

In fluidized beds, for example, classification of the flow patterns has been attempted based on particle properties such as size, density and cohesiveness [2]. Cohesive interparticle force is often a combined effect of liquid bridge, electrical force, van der Waals force, etc., and the cohesive force is recognised as an important factor for determining the characteristic behaviours of a fluidized bed [3–5]. However, the effects of the interparticle force on the bed behaviours have only been partially addressed (e.g. [6,7]). This is because there exists very complex relationship between the magnitude of interparticle forces and the macroscopic physical properties of the gas-particle flow system [7].

Another example of the effect of the interparticle force is flocculation and sedimentation of contaminant particles in a liquid. A theory incorporating the existing drag force models (Stokes, Allen and Newton) often over-/under-estimates the settling velocity [8]. Also, a reliable drag force model for a bed/cluster of floc is not established. One of the reasons for the uncertainties in predicting the settling velocities is that the

* Corresponding author.
E-mail address: shintaro.takeuchi@fel.t.u-tokyo.ac.jp (S. Takeuchi).

0032-5910/$ - see front matter © 2007 Elsevier B.V. All rights reserved.
doi:10.1016/j.powtec.2007.11.039

assumptions of the drag force models (a single spherical object fixed in a free space) may not be appropriate. In addition, as Davis and Stone [9] showed, the permeable effect of particles may not be negligible on the sedimentation process in general situations.

The above examples highlight the importance of fundamental understanding of the global effect of the interparticle force on the agglomeration/sedimentation/dispersion behaviours of the particles and the local flow patterns at the scale of the smallest flow structures.

In recent years, numerical simulation has been widely used for studying particle-induced flows, as it promises to be a useful tool for obtaining a wide range of flow properties, including particle behaviours and hydrodynamic forces acting on the particles, simultaneously and continuously at desired spatial and temporal scales (e.g. [10–12]). Numerical simulations are shown to be suitable for studying particle suspension flow with imposed interparticle force.

Discrete element method (DEM) coupled with computational fluid dynamics (CFD) [13,14] has emerged as a viable approach for studying gas-particle systems. The DEM-CFD method, when applied to fluidized beds, has successfully reproduced the particle behaviours and bed pressure drop [6,14,15] by solving the continuum and particulate phases with empirical formulae for fluid-particle interaction (e.g. [16–18]) at the individual particle level. However, it is difficult to obtain the detailed information of the fluid behaviours flowing through the bed of particles due to the insufficient resolution for the boundary layers around the particles. In addition, particle rotation due to hydrodynamic force plays an important role for a clustered behaviour of particles in a dilute suspension system [19,20]. In the DEM-CFD approach, no reliable model exists for momentum exchange between the fluid and particles' rotating degrees of freedom.

The present authors have developed a technique of direct numerical simulation (DNS) of particle-laden flow [19–21]. DNS employs no assumptions for solving the fluid motions, and it resolves the flow field from the boundary layer of the particle surface to the far wake at the scale of the smallest vortex size. In our method, particle behaviours are solved individually by the equation of linear and angular momentums, and momentum exchange between the fluid and solid phases is solved by our immersed boundary method [19,20] including the rotating degrees of freedom of non-spherical particles [21]. This interphase coupling method guarantees the conservation of momentums, and it was proved to be very efficient in computation time [21]. The usefulness of our DNS approach for studying the fundamentals of the particle-induced flow has been demonstrated through the analysis of the clustering process of $O(10^3)$ spherical particles in a turbulent flow [20] and sedimentation process including interparticle collisions in a 10^5-particle system [22].

The present work aims to obtain an improved physical understanding of the interaction between fluid and particles with an imposed interparticle force in three-dimensions by DNS. A simple interparticle force characterised by two parameters, magnitude and effective distance, is employed to investigate the independent effect of each parameter on the fluid-particle system. In this work, we look at the transition of the shape of particle agglomerate by varying the above two parameters, together with the drag force and flow patterns past an agglomerate.

2. Numerical methods and conditions

2.1. Basic equations

The governing equations for the fluid flow are the equation of continuity and Navier–Stokes (N–S) equations:

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

$$\frac{\partial \boldsymbol{u}_f}{\partial t} + \boldsymbol{u}_f \cdot \nabla \boldsymbol{u}_f = -\frac{1}{\rho_f}\nabla p + \nu_f \nabla \cdot \left[\nabla \boldsymbol{u}_f + \left(\nabla \boldsymbol{u}_f\right)^T\right], \tag{2}$$

where $\boldsymbol{u}_f$ is fluid velocity, t time and p pressure. Density and kinematic viscosity of the fluid, ρ_f and ν_f, are constant throughout the present work. Cartesian grid is uniformly distributed with grid spacing Δ over the computational domain.

The motion of a solid object is simulated by solving the equations of linear and angular momentums as follows:

$$\frac{d\left(m_p \boldsymbol{v}_t\right)}{dt} = \int_{S_p} \boldsymbol{\tau} \cdot \boldsymbol{n} dS + \boldsymbol{G}_p, \tag{3}$$

$$\frac{d\left(\boldsymbol{I}_p \cdot \boldsymbol{\omega}_p\right)}{dt} = \int_{S_p} \boldsymbol{r} \times (\boldsymbol{\tau} \cdot \boldsymbol{n}) dS + \boldsymbol{N}_p, \tag{4}$$

where m_p is mass of the object, $\boldsymbol{v}_t$ translating velocity of the object, $\boldsymbol{\tau}$ stress tensor of the fluid, $\boldsymbol{I}_p$ inertia tensor, $\boldsymbol{\omega}_p$ angular velocity, $\boldsymbol{G}_p$ external force, $\boldsymbol{N}_p$ external torque, S_p object surface, $\boldsymbol{n}$ unit vector in the normal outward direction at the surface and $\boldsymbol{r}$ relative vector from the gravity centre to a point in the integral region.

2.2. Fluid–solid interaction model

Momentum exchange at the fluid–solid interface, where a cell is partially occupied by a solid particle (intruder), is solved by the immersed boundary method of Kajishima and the coworkers [19,20]. This is briefly described below.

A velocity field $\boldsymbol{w}$ is established through volume-averaging the local fluid velocity $\boldsymbol{u}_f$ and local particle velocity $\boldsymbol{u}_p$ in a cell, i.e.,

$$\boldsymbol{w} = (1-\alpha)\boldsymbol{u}_f + \alpha \boldsymbol{u}_p, \tag{5}$$

where α $(0 \le \alpha \le 1)$ is the local solid volume fraction in the cell. The particle velocity u_p can be decomposed into translating and rotating components as $\boldsymbol{u}_p = \boldsymbol{v}_t + \boldsymbol{r} \times \boldsymbol{\omega}_p$. The velocity field $\boldsymbol{w}$ is assumed` to obey the following N–S equation:

$$\frac{\partial \boldsymbol{w}}{\partial t} = -\nabla P + \boldsymbol{H}_w + \boldsymbol{f}_p, \tag{6}$$

where $P = p/\rho_f$ and $\boldsymbol{H}_w$ contains convective and viscous terms as follows:

$$\boldsymbol{H}_w = -\boldsymbol{w} \cdot \nabla \boldsymbol{w} + \nu_f \nabla \cdot \left[\nabla \boldsymbol{w} + (\nabla \boldsymbol{w})^T\right]. \tag{7}$$