Advanced Powder Technology 24 (2013) 565–574

Contents lists available at SciVerse ScienceDirect

Advanced Powder Technology

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

Translated paper; SPTJ Best Paper Award 2011

Effect of a wall on flow with dense particles ☆

Takuya Tsuji *, Eiji Narita, Toshitsugu Tanaka

Department of Mechanical Engineering, Graduate School of Engineering, 2-1 Yamada-Oka, Suita, Osaka 565-0871, Japan

ARTICLE INFO

Article history:
Received 6 November 2012
Accepted 12 November 2012
Available online 30 November 2012

Keywords:
Wall effect
Gas–solid flow
Fluidized bed
Direct numerical simulation
Immersed boundary method

ABSTRACT

The behavior of dense gas–solid flows in engineering applications such as fluidized beds and pneumatic conveyers is highly complex and a reliable numerical model is required. Such flows are usually within solid walls that considerably affect the flow fields, and it is important to correctly include this effect in numerical models to improve their prediction capability. The observation of microscopic flows near walls can enhance our understanding of the flow behavior and assist in improving models. In this study, direct simulations are performed to investigate the effect of a wall on flow fields at a microscopic level. The effects of the bulk void fraction, particle Reynolds number, and particle diameter are investigated. The prediction performances of existing correlation equations usually used in mesoscopic model calculations are also investigated. It is found that the Ergun and Beetstra equations produce large discrepancies in the region within a distance equal to the particle diameter from the wall.

1. Introduction

Dense particles coexist with a gas in industrial equipment such as fluidized beds and pneumatic conveying systems. The flow behavior of the gas–particle mixture is significantly influenced by the interactions between the particles, between the particles and the wall and between the particles and the gas. The spontaneous formation of mesoscopic heterogeneous structures inside the equipment, such as bubbles that are far larger than the particles, makes the flow more complex. A better understanding of the flow behavior and design optimization is therefore desired. This has, however, not been successfully achieved owing to observation difficulties in experiments. Against this background, the development of numerical models has been extensively pursued until the present time. The majority of these models are mesoscopic in the sense that the size of the computational cells of the fluid motion is small enough to resolve characteristic mesoscopic structures but too large to directly capture the microscopic phenomena on the particle level [1–3]. The models have been applied to several flow problems, and the reproduction of size and frequency with which bubbles appear in fluidized beds have been successfully demonstrated.

In practical engineering situations, solid walls that significantly influence the behavior of both phases are very common. It is important to correctly express the influence of the solid walls whenever a numerical simulation is performed. In mesoscopic model calculations, the momentum exchange between solid particles and the gas in a computational cell is generally determined by using empirical equations that are a function of the void fraction and the particle Reynolds number. In the majority of existing models, empirical equations such as those of Ergun [4], Di Felice [5], and Beetstra et al. [6] are used without any modifications, including in computational cells near a solid wall. The existence of a wall directly exerts viscous friction on nearby fluids and affects the arrangement of nearby particles. As a result, the anisotropy of the flow in the vicinity of solid walls is increased. In the aforementioned empirical equations, the effect of the wall is not considered and it is questionable to use these equations for computational cells near solid walls. Moreover, the problem is one that has been recognized by researchers since early times and was mentioned by Beetstra et al. [6]. When microscopic flows are directly considered, fluid motions should obey the Navier–Stokes and continuity equations, and it is appropriate to use the no-slip boundary condition on the surface of solid walls. However, in mesoscopic model calculations based on locally phase averaged equations, microscopic flows occurring on a subgrid scale are averaged and it is questionable to use the no-slip boundary condition on the surface of solid walls on the mesoscopic cell scale. Currently, different boundary conditions are used depending on a study, and a general consensus is yet to be achieved.

For further improvement of mesoscopic models, it is important to adopt a proper drag correlation in computational cells near a solid wall and a proper boundary condition on the wall. This cannot be accomplished without a thorough understanding of the behavior of microscopic flows near such walls, which never be captured by the mesoscopic model calculations. Although the effects of a solid wall on a flow that contains dense particles have been investigated [7–9], it is still difficult to directly observe microscopic flows

☆ Japanese version published in JSPTJ, Vol. 48 No. 12 (2011); English version for APT received on November 6, 2012.

* Corresponding author. Tel./fax: +81 6 6879 7317.
E-mail address: tak@mech.eng.osaka-u.ac.jp (T. Tsuji).

0921-8831/$ - see front matter © 2012 The Society of Powder Technology Japan. Published by Elsevier B.V. and The Society of Powder Technology Japan. All rights reserved.
http://dx.doi.org/10.1016/j.apt.2012.11.006

Nomenclature

a	particle radius (m)
d_p	particle diameter (m)
F_D	fluid drag force (N)
F_L	fluid lift force (N)
$\boldsymbol{f}_p$	interaction force (m/s^2)
$\boldsymbol{G}_p$	external force acting on a particle (N)
$\boldsymbol{g}$	acceleration due to gravity (m/s^2)
$\boldsymbol{I}$	unit tensor (–)
$\boldsymbol{I}_p$	moment of inertia (kg m^2)
L	streamwise domain size (m)
$L_{x,\,y,\,z}$	domain size in x, y, and z directions (m)
$\boldsymbol{M}_p$	external moment acting on a particle (N m)
m_p	particle mass (kg)
N_p	number of particles in the computational domain (–)
$\boldsymbol{n}$	vector normal to the surface of a particle (–)
$n_{x,\,y,\,z}$	number of grid points in x, y, and z directions (–)
p	fluid pressure (Pa)
Re_p	particle Reynolds number (–)
$\boldsymbol{r}$	relative vector from the center of a particle (–)
S_p	surface area of a particle (m^2)
t	time (s)
U	inflow velocity (m/s)
$\boldsymbol{U}_p$	translating velocity of a particle (m/s)
$\boldsymbol{u}$	coupling velocity weighted by particle volume fraction (m/s)
u_0	superficial velocity (m/s)
$\boldsymbol{u}_f$	fluid velocity (m/s)
$\boldsymbol{u}_p$	particle velocity (m/s)
u_y	streamwise fluid velocity (m/s)
$\tilde{\boldsymbol{u}}$	velocity predicted as a fluid (m/s)
V	volume of computational domain (m^3)
V_p	volume of a particle (m^3)
α	volume fraction of particle in each computational cell (–)
Δp	pressure drop (Pa)
$\Delta p_{\mathrm{Beetstra}}$	pressure drop predicted by Beetstra equation (Pa)
Δp_{DF}	pressure difference imposed as a driving force (Pa)
$\Delta p_{\mathrm{Ergun}}$	pressure drop predicted by Ergun equation (Pa)
Δt	temporal increment (s)
Δx	computational cell size (m)
ε	void fraction in whole region (–)
ρ_f	density of fluid (kg/m^3)
μ_f	viscosity of fluid (Pa s)
ν_f	kinetic viscosity of fluid (m^2/s)
$\boldsymbol{\tau}$	stress tensor (Pa)
$\boldsymbol{\Omega}_p$	rotational velocity of a particle (rad/s)
ψ	nondimensional pressure drop (–)
$\langle\,\rangle$	spatial average in whole region
$\langle\,\rangle_{yz}$	spatial average in y–z plane
—	temporal average

on a particle scale by experimental methods. In the present study, the body-force-type immersed boundary method (IBM) is used to conduct microscopic numerical simulations of flows that contain dense solid particles near a solid wall, and the effects of the wall on the flow behavior are investigated in detail. In addition, the data obtained from the microscopic simulations are used to discuss the validity of the empirical drag equations popularly used in mesoscopic model calculations. In this paper, which is the first report, we restrict our discussions to the case of a fixed particle.

2. Calculation methods

2.1. Governing equations

It is assumed that the fluid is incompressible and Newtonian and the particles are rigid spheres. The governing equations of the fluid flow are the continuity equation and Navier–Stokes equation, respectively, expressed below:

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

$$\rho_f \frac{D\boldsymbol{u}_f}{Dt} = \nabla \cdot \boldsymbol{\tau} + \rho_f \boldsymbol{g} \tag{2}$$

where $\boldsymbol{u}_f$ and ρ_f are respectively the fluid velocity and density, $\boldsymbol{g}$ is the acceleration due to gravity, p is the pressure, μ_f is the fluid viscosity, and $\boldsymbol{\tau}$ is the stress tensor given by

$$\boldsymbol{\tau} = -p\boldsymbol{I} + \mu_f[\nabla \boldsymbol{u}_f + (\nabla \boldsymbol{u}_f)^T] \tag{3}$$

The particles are tracked individually in a Lagrangian manner by using the equations of translational and rotational motions:

$$\frac{d(m_p \boldsymbol{U}_p)}{dt} = \int_{S_p} \boldsymbol{\tau} \cdot \boldsymbol{n}\, dS + \boldsymbol{G}_p \tag{4}$$

$$\frac{d(\boldsymbol{I}_p \boldsymbol{\Omega}_p)}{dt} = \int_{S_p} \boldsymbol{r} \times (\boldsymbol{\tau} \cdot \boldsymbol{n})\, dS + \boldsymbol{M}_p \tag{5}$$

where $\boldsymbol{U}_p$ and $\boldsymbol{\Omega}_p$ respectively represent the translational and rotational velocities of the particle, m_p is the mass of the particle, $\boldsymbol{I}_p$ is the moment of inertia given by $\boldsymbol{I}_p = (2/5)a^2 m_p \boldsymbol{I}$ for a particle with radius a, $\boldsymbol{G}_p$ and $\boldsymbol{M}_p$ are respectively the external force and moment, and $\boldsymbol{r}$ is the relative position vector from the center of a particle.

2.2. Body-force-type IBM [10]

Kajishima et al. developed the body-force-type IBM for particle–fluid systems. Here, the assumption of a uniform Cartesian grid is used to simplify the explanation. The presence of particles is represented by the solid volume fraction of particles in each computational cell α. Kajishima et al. [10] introduced the following coupling velocity weighted by the solid volume fraction of particles:

$$\boldsymbol{u} = \alpha \boldsymbol{u}_p + (1-\alpha)\boldsymbol{u}_f \tag{6}$$

where $\boldsymbol{u}_p$ is the particle velocity given by $\boldsymbol{u}_p = \boldsymbol{U}_p + \boldsymbol{r} \times \boldsymbol{\Omega}_p$. The particles are solid and no-slip and no-permeability conditions are imposed on their interfaces. Hence, the continuity restriction also applies to $\boldsymbol{u}$:

$$\nabla \cdot \boldsymbol{u} = 0 \tag{7}$$

The following equation for $\boldsymbol{u}$ is introduced:

$$\frac{\partial \boldsymbol{u}}{\partial t} = -\frac{1}{\rho_f}\nabla p + \boldsymbol{H} + \boldsymbol{f}_p \tag{8}$$

where

$$\boldsymbol{H} = -\boldsymbol{u}\nabla \boldsymbol{u} + \nu_f \nabla \cdot [\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T] + \boldsymbol{g} \tag{9}$$

and ν_f is the kinematic viscosity of the fluid. In Eq. (8), $\boldsymbol{f}_p$ is the interaction force term that enforces all the predicted velocity fields (including inside the particles) to the particle–fluid coupling velocity field. This is explained in terms of the time-marching procedure. If Eq. (8) marches in time with the explicit Euler method, it would be