ARTICLE IN PRESS

International Journal of Heat and Fluid Flow xxx (2014) xxx–xxx

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Direct numerical simulation of finite sized particles settling for high Reynolds number and dilute suspension

Ali Abbas Zaidi *, Takuya Tsuji, Toshitsugu Tanaka

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

ARTICLE INFO

Article history:
Received 24 February 2014
Received in revised form 5 September 2014
Accepted 8 September 2014
Available online xxxx

Keywords:
Finite size particle clustering
Particle induced turbulence
Particle microstructures in settling
Direct numerical simulations

ABSTRACT

Finite sized particles settling under the action of gravity are investigated by immersed boundary method for dilute suspensions and low to high range of Reynolds number. The Reynolds number based on the terminal velocity of a single particle is varied from 1 to 300 and the solid volume fraction (φ) is varied from 0.005 to 0.05. The studied range of Reynolds number corresponds from streamlined flow to downstream vortex shedding flow for single particle. For $\varphi = 0.005$, 0.01 and high Reynolds number ($Re \geqq 175$), settling particles clusters due to the entrapment in high fluid shear regions. However for $Re = 50$ and 100, particles form separated pairs due to relatively weak strength of wakes that promote to drafting–kissing and tumbling scenario. These particle structures at high Reynolds number changes the settling behaviors of particles e.g. increase in the settling velocity and fluid velocity fluctuations. For $\varphi = 0.03, 0.05$, hindered settling effects dominate and reduce the effects of particles clustering due to Reynolds number.

1. Introduction

In nature and industrial applications, particles affect the rate of heat and mass transfer and flow structures (Hughmark, 1980). One particular class of these processes is sedimentation. In sedimentation, the particles are usually solid and the fluid can be liquid or gas. Some of the applications of sedimentation are: separating dirt and debris from lakes and water reservoirs, solute from solvents, dust or product particles from air streams and removal of contaminants in ground water etc. (Ji et al., 2009; O'Melia, 1998). For understanding of the underlying physics, further investigations are required especially for moderate and high ranges of Reynolds number. Some of the difficulties associated with its understanding are the inter-particle hydrodynamic interactions and constantly changing suspension microstructures.

Particle resolved direct numerical simulation (DNS) can improve our understanding of sedimentation. Experimentally, it is difficult to track large number of particles (orders of 10^2), visualization of flow structures and calculation of fluid force on particles. In DNS the governing fluid equations of continuity and Navier–Stokes are directly solved without the use of any empirical relations. Thus by careful selection of grid in DNS, reliable and accurate results can be obtained. For particle laden flows, DNS can be broadly classified as body fitted (Bagchi and Balachandar, 2003; Hu et al., 2001) and non-body fitted grid type (Kajishima et al., 2001; Ladd, 1994; Uhlmann, 2005). In the current article, immersed boundary method (IBM) which is a type of non-body fitted grid DNS, is used for performing particle settling simulations.

The flow structures around particles depend on the settling Reynolds number and the inter-particle distance. In the simplest case i.e. flow around a single fixed sphere in a uniform stream is investigated both experimentally (Achenbac, 1974; Sakamoto and Haniu, 1990) and numerically (Johnson and Patel, 1999; Shirayama, 1992). It is observed that the flow patterns around the sphere change with the increase of Reynolds number. According to references mentioned, for low Reynolds number i.e. about $Re \leqslant 2$, the flow around the sphere remains streamlined. This streamlined flow is replaced by steady and axisymmetric vortex flow for $Re > 2$ and $Re \leqslant 210$. Further increase in Reynolds number up to about 270 changes the axisymmetric characteristics of the downstream wakes to plane-symmetric. For $Re > 270$, unsteady vortex shedding starts and further increase of Reynolds number makes the period and shape of shed vortices more and more random.

For understanding the effect of inter-particle distance and alignment of particles on flow structures and drag on particles, (Tsuji et al., 2003) investigated the effect of Reynolds numbers on fixed particle pairs in uniform fluid stream using DNS. According to them, drag force on particle pairs is attenuated when the particles are aligned in stream-wise direction and drag force is augmented in side-by-side arrangements of particles. Furthermore,

* Corresponding author. Tel./fax: +81 80 3824 9136.
E-mail address: ali@cf.mech.eng.osaka-u.ac.jp (A.A. Zaidi).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.09.007
0142-727X/

Please cite this article in press as: Zaidi, A.A., et al. Direct numerical simulation of finite sized particles settling for high Reynolds number and dilute suspension. Int. J. Heat Fluid Flow (2014), http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.09.007

Nomenclature

Ar	Archimedes number
d_p	particle diameter (m)
E	energy of wave (Watt-cm)
f	friction coefficient
$\boldsymbol{F}$	force on particle–particle collision (N)
$\boldsymbol{F_f}$	fluid force (N)
$\boldsymbol{f_p}$	forcing term
Fr	Froude number
g	gravity (m/s^2)
$g(r)$	radial distribution function at an inter-particle distance r
$\boldsymbol{G}$	relative particle velocity vector between colliding particles (m/s)
GB	Gega byte
K	spring constant (N/m)
k_1	multiplication factor
k	wave number (cm^{-1})
L	length of the computational domain in the gravity direction (m)
$\boldsymbol{M_f}$	fluid moment (N-m)
n	time step
$\boldsymbol{n}$	normal direction
n_e	power law exponent
$n(r)$	number of particles in the shell of radius r
np	number of particles
p	fluid pressure (N/m^2)
$\boldsymbol{r}$	unit vector from the center of rotation to the surface
$\boldsymbol{r_i}$	position vector of particle i
$\boldsymbol{r_j}$	position vector of particle j
$r_{i,j}$	distance between particle i and j (m)
r_s	shell radius
$\boldsymbol{t}$	tangential direction
t	time (s)
TB	tera byte
t'	Stokes time
Re	terminal Reynolds number
Re_p	average settling Reynolds number
$\boldsymbol{u}$	fluid-particle volume-weighted velocity (m/s)
$\tilde{\boldsymbol{u}}$	predicted velocity (m/s)
$\boldsymbol{u_f}$	fluid velocity (m/s)
$\boldsymbol{u_p}$	velocity inside the solid particle (m/s)
U_s	terminal velocity of particle (m/s)
$\dot{U}_{rms}$	root mean square velocity fluctuations (m/s)
v_p	velocity of particle center (m/s)
$\boldsymbol{V_p}$	average particle velocity (m/s)
$Vp1$	volume of cube with sides equal to the sphere diameter (m^3)
ω_p	angular velocity of particle rotation (rad)
x, y, z	cartesian coordinate directions
Greek symbols	
α	volume fraction of particles inside the grid cell
ρ	fluid density (m^3)
ρ_p	particle density (m^3)
ρ^n	number density (m^{-3})
ν	kinematic viscosity (m^2/s)
μ	viscosity (N-s/m^2)
δ	particle overlap during collision
η	damping coefficient
φ	solid volume fraction
ΔT	time step
$\triangle r$	shell radius
$\triangle x$	mesh size
$\langle\rangle_t$	time averaging
$\langle\rangle_{np}$	particle averaging
∇	gradient
Subscript	
0	previous time step
n	normal direction
t	tangential direction

they also observed that when the inter-particle distance between two particles is less than 0.1 times the particle diameter, unsteady vortex shedding starts for $Re < 200$.

For low Reynolds number ($Re < 1$) settling process, there are reasonable number of experimental (Nicolai and Guazzelli, 1995; Nicolai et al., 1995; Segre, 2002), theoretical (Brenner, 1999; Caflisch and Luke, 1985) and numerical studies (Ladd, 1996; Padding and Louis, 2004; Tee et al., 2007) available. However for moderate and high Reynolds number, studies using DNS are few. This is because moderate and high Reynolds number DNS need fine grid to resolve the small scale fluid structures and thus increases the computational cost. According to authors knowledge, we only know three studies for moderate and high Reynolds number (Climent and Maxey, 2003; Yin and Koch, 2007, 2008; Doychev and Uhlmann, 2013; Kajishima, 2004; Kajishima and Takiguchi, 2002).

The objectives of this article are to study the effects of Reynolds number and solid volume fraction on average settling velocity, velocity fluctuations and particles settling structures for dilute suspensions. The range of Reynolds number studied in this article varies from 1 to 300 and solid volume fraction varies from 0.005 to 0.05. The novelty of this article is to study the diminishing effects of Reynolds number with the increase in solid volume fraction.

The organization of this paper is as follows. The numerical methods which are used in the simulations are given in Section 2. Simulation setup is given in Section 3. The obtained results and their discussions are presented in Section 4. Finally the conclusions are given in Section 5.

2. Numerical scheme

2.1. Immersed boundary method (IBM)

The fluid solver used in the present paper is body force type IBM proposed by Kajishima et al. (2001). In this article, we will only highlight the main points; details of this method can be found in the reference mentioned. In IBM, grid size is smaller than the size of particles and the fluid flow calculations are performed by assuming that the fluid occupies the entire flow field. The effect of particles is expressed by a body force into the momentum equation of fluid to constrain the no slip boundary condition at the nodes inside the particles. The equations of continuity and incompressible Navier–Stokes equation without gravity effects is given by:

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

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

For numerical integration, fluid-particle volume-weighted velocity ($\boldsymbol{u}$) is defined which is given by:

Please cite this article in press as: Zaidi, A.A., et al. Direct numerical simulation of finite sized particles settling for high Reynolds number and dilute suspension. Int. J. Heat Fluid Flow (2014), http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.09.007