Advanced Powder Technology xxx (2016) xxx–xxx

Contents lists available at ScienceDirect

Advanced Powder Technology

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

Original Research Paper

DNS of a turbulent flow past two fully resolved aligned spherical particles

Fan Wu, Kun Luo, Jianren Fan *

State Key Laboratory of Clean Energy Utilization, Zhejiang University, Hangzhou 310027, PR China

ARTICLE INFO

Article history:
Received 3 December 2015
Received in revised form 29 March 2016
Accepted 30 March 2016
Available online xxxx

Keywords:
Finite sized particle
Fully resolved simulation
Isotropic turbulence
Multiphase flow

ABSTRACT

Direct numerical simulation with fully resolved immersed boundary method is employed to study a turbulent flow past two aligned spherical particles with different distances. A homogeneous isotropic turbulence field is generated as the inflow boundary condition and is kept developing throughout the simulation to avoid periodicity. The particle radius is about 4.7–7.9 times the Kolmogorov length and the turbulent intensities are about 12.9%, 24% and 36%. Validation is made comparing with former researchers and the distance effect is studied by comparing the hydrodynamic forces, particle induced extra dissipation, turbulent kinetic energy, and the synchronous rate of the cross-stream forces. It is shown that, under a mean flow with turbulent intensities, with enlarging the particle distances, the wake shedding behind the upstream particle is longer; the magnitudes of the stream-wise hydrodynamic forces exerted on both particles are higher. Increased turbulent intensity of the inlet condition or increased particle Reynolds number contributes to the particle induced kinetic energy and the increase of Reynolds number diminishes the difference on the particle boundary at its lateral sides. The monotonic decrease of synchronous rate of the cross-stream forces is observed with increasing particle distances when $Re_p = 80$, while it fails under higher Reynolds number.

1. Introduction

Solid particle suspended in gas flow is a quite common situation in nature and various industrial applications such as transport of suspended sediments, fluidized beds, the formation and dynamics of sand dunes. The gas phase, in most cases, is turbulent. The existence of particles and their relative positions are known to strongly influence the performance of engineering devices. Therefore, a clear understanding of the complex force responses caused by particles to the turbulent flow is important in the research of particle-laden flows. In simulating this kind of gas–solid multiphase flow, point-particle model [1], which is only applicable to particles with sizes smaller than the Kolmogorov scale and under very dilute concentrations, has long been adopted by researchers. While much work [2–6] has been done with this model due to its simplicity in modeling and calculation, the absence of accounting the volume effect on the hydrodynamic interactions between the gas and solid phases makes it inappropriate for the situation where local concentration is high or the magnitude difference between Kolmogorov scale and particle diameter is small. For the same reason, neither can it be applied when relative positions of the particles is taken account of. On the other hand, particle's size effect cannot be reflected on the gas phase. The modulation of the solid phase on the gas phase hence cannot be well captured and the simulation is in a way simplified. This provides additional motivation for the present paper toward better understanding and accurate parameterization of the hydrodynamic forces on the particle in a turbulent flow.

Not surprisingly, despite its strong limitations, point particle model has been widely adopted in studying multiphase flows. Although large number of its adoptions in this field, it is reasonable that point-particle model has been unable to explain major experimental results such as the greatly increased dissipation of [7] or the strong dependence of the turbulence modulation on the ratio of the particle size to the integral scale of the turbulence [8]. In recent years, fully resolved numerical simulations of finite-sized particles in the presence of turbulence are just beginning to emerge and it is natural that some leading works starts from the situation where a single particle exists in the turbulence [9–11]. The fully resolved simulation of turbulent flow over particles that accurately accounts for the hydrodynamic forces between the solid and fluid phase, separation of the shear layers on the particles, and the shedding structures will faithfully provide references to the

* Corresponding author. Tel./fax: +86 571 87951764.
E-mail address: fanjr@zju.edu.cn (J. Fan).

http://dx.doi.org/10.1016/j.apt.2016.03.027
0921-8831/

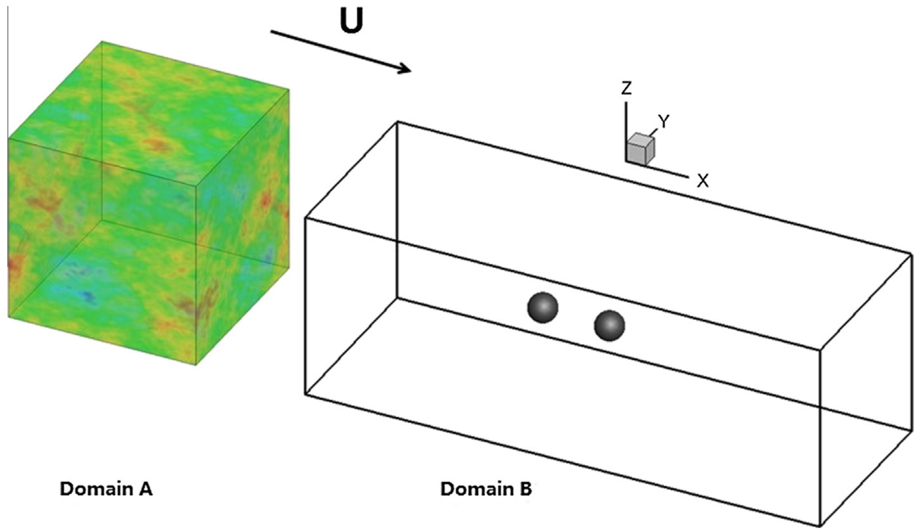

Fig. 1. Schematic of the simulation setup. A homogeneous isotropic turbulent field generated in the domain A is convected with a mean velocity.

Table 1
Characteristics of the homogeneous isotropic turbulence generated in the domain A: u'/U: Turbulence intensity; Re_λ: Taylor-microscale turbulence Reynolds number; η: Kolmogorov scale.

Re_λ	a/η	u'/U (%)
3	4.7	12.9
5.4	6.4	24
8.1	7.9	36

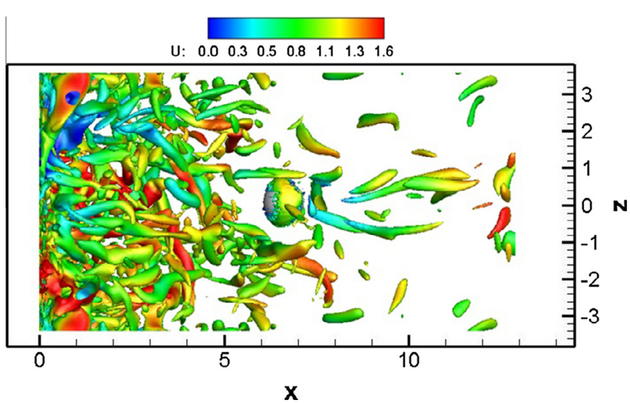

Fig. 2. An instantaneous vortex structure of domain B at the side.

research to the extent that point-particle model cannot ever achieve.

In the present work, we report direct numerical simulation results of turbulent flow with a mean velocity U flowing past two sphere particles aligned in stream-wise direction at two particle Reynolds numbers $Re_p = 2aU/v$ = 80 and 200, with a the particle radius and v the fluid kinematic viscosity. The turbulent field is generated in a computational cubic domain and convected over the particles as it spatially decays. As a validation, we compare the obtained results with former literature. Then, the turbulent flow past two fixed sphere particles with different distances between the centers of each other of 1.5D, 2.5D and 3.5D, in which D is the particle diameter, is simulated. By studying this set of particle configuration, wake effect behind a bluff body is discussed.

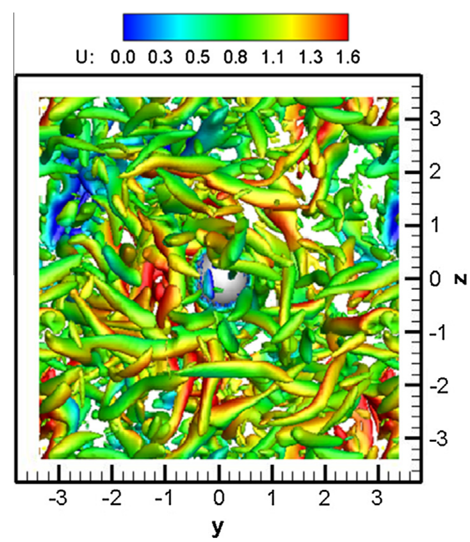

Fig. 3. An instantaneous vortex structure of domain B at the rear.

2. Numerical description

2.1. Governing equations for gas-phase flow

The dimensionless governing equations for the incompressible viscous flow in the entire computational domain are:

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

$$\frac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u} \cdot \nabla \boldsymbol{u} = -\nabla P + \frac{1}{Re}\nabla^2 \boldsymbol{u} + \boldsymbol{f} \tag{2}$$

where $\boldsymbol{u}$ is the fluid velocity, P is the pressure and Re is the Reynolds number defined as $Re = \frac{\rho_f \cdot U \cdot L}{\mu}$, where ρ_f is the density of fluid, U is the mean convecting velocity, L is the cubic box length of the turbulent inflow field ($L = 16a$) and μ is the fluid viscosity. $\boldsymbol{f}$ is an external force exerted on the flow field to represent the mutual interaction between fluid and the immersed bodies. Following the spirit of immersed boundary method [12,13], it can be expressed as:

$$\boldsymbol{f}(\boldsymbol{x}) = \int_\Omega \boldsymbol{F}_k(\boldsymbol{x}_k) \cdot \delta(\boldsymbol{x} - \boldsymbol{x}_k) d\boldsymbol{x}_k \tag{3}$$