Advanced Powder Technology xxx (2018) xxx–xxx

Contents lists available at ScienceDirect

Advanced Powder Technology

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

Original Research Paper

Drag, lift and torque acting on a two-dimensional non-spherical particle near a wall

Ahad Zarghami *, Johan T. Padding

Department of Process and Energy, Delft University of Technology, Delft, The Netherlands

ARTICLE INFO

Article history:
Received 23 January 2018
Received in revised form 8 March 2018
Accepted 15 March 2018
Available online xxxx

Keywords:
Lift
Drag
Torque
Non-spherical particle
Wall-effect

ABSTRACT

Gas-solid granular flows with non-spherical particles occur in many engineering applications such as fluidized beds. Such flows are usually contained by solid walls and always some particles move close to a wall. The proximity of a wall considerably affects the flow fields and changes the hydrodynamic forces and torque acting on particles moving near the wall. In this paper, we numerically investigate the drag, lift and torque acting on a non-spherical particle in the vicinity of a planar wall by means of lattice Boltzmann simulations. To gain an exhaustive understanding of the complex hydrodynamics and study the influence of various geometrical and flow parameters, a single 2D elliptical particle is selected as our case study. In the simulations, the effect of particle Reynolds number, distance to the wall, orientation angle and aspect ratio on drag, lift and torque is studied. Our study shows that the presence of a wall causes significant changes in hydrodynamic forces, with increasing or decreasing drag and lift forces, depending on the distance from the wall. Even the direction of lift and torque may change, depending on both the distance from the wall and particle orientation angle. Also, an ellipse with higher AR experiences larger hydrodynamic forces and torque whatever the gap size and orientation angle.

1. Introduction

Granular systems, i.e. assemblies of solid, macroscopic particles, are worth studying because they are commonly encountered in various engineering applications in the energy, chemical and agricultural industries [1–5]. Hence, understanding the characteristics and hydrodynamics of granular materials in a fluid flow is of great importance to increase the efficiency of these applications.

It has been shown that the shape of a particle is one of the most important parameters influencing the hydrodynamics of a granular system [5,6]. It is believed that a significant fraction (estimated around 70%) of processed granular materials have non-spherical shape [5,7]. However, in many studies that deal with granular systems, particles are assumed to be perfect spheres [8–10] due to the fact that the shape characterization of non-spherical particles is a complex process, with various aspects, such as roundness, aspect ratio, irregularity and sphericity, influencing the hydrodynamics [11,12]. Despite much research on hydrodynamics of non-spherical granular particles [5,13–16], significant efforts are still required to accurately predict the hydrodynamic forces acting on non-spherical particles under various flow conditions.

In engineering equipment with granular particles, such as fluidized beds, some of the particles move in the vicinity of a wall. The presence of the wall will change the drag force acting on those particles and, more importantly, the particles experience a transverse lift force known as wall-induced lift [17]. The wall-induced lift is caused by two different competing mechanisms [18,19]. First, the presence of a wall breaks the symmetry of the generated wakes behind the particle which generally results in an effective lift force directed away from the wall. Second, flow relative to the particle will accelerate faster in the gap between the particle and the wall. According to inviscid theory, the pressure in the gap decreases resulting in a lift force directed toward the wall. The wall effect on hydrodynamic forces decays rapidly as the distance between the particles and wall increases, and for distances of the order of 10 (spherical) particle diameters, the wall effect can be reasonably ignored [20].

Investigations of the effect of wall proximity on hydrodynamic forces, i.e. drag and lift forces and torque, acting on non-spherical particles are scarce [21,22] and most of the research work available in the literature is limited to spherical particles [9,10,17,19,23–25]. The purpose of this paper is to investigate the effect of wall

* Corresponding author at: Department of Process and Energy, TU Delft, Leeghwaterstraat 39, 2628CD Delft, The Netherlands.
E-mail address: a.zarghami-1@tudelft.nl (A. Zarghami).

https://doi.org/10.1016/j.apt.2018.03.019
0921-8831/

proximity on hydrodynamic forces and torque acting on non-spherical particles in a shear flow up to moderate Reynolds number. In order to gain a better understanding of the complex hydrodynamics, a single two-dimensional elliptic particle is selected as a case study. The effect of flow Reynolds number, gap size (or minimum separation distance from the wall), and orientation on drag and lift forces are studied. For this purpose, the lattice Boltzmann method (LBM) is applied as a capable and accurate technique for simulating the fluid flow.

The rest of this paper in organized as following. In Section 2, the numerical method is presented briefly. Results and discussions are presented in Section 3, follow by concluding remarks in Section 4.

2. Numerical method

2.1. LBM for fluid flow

In In the past two decades the LBM, a mesoscopic numerical method based on kinetic theory [26], has become an alternative, capable and computationally efficient technique for simulating various complex fluid flows [27–29]. The simple form of the governing equations, space–time locality, straightforward parallelism, easy grid generation and capability of incorporating complex microscopic interactions are the main advantages of LBM compared with conventional CFD methods based on Navier–Stokes equations. The general lattice Boltzmann equation with the BGK collision operator is expressed as [26]:

$$\frac{f_i(\vec{x}+\vec{e}_i\Delta t, t+\Delta t) - f_i(\vec{x},t)}{\Delta t} = -\frac{1}{\tau}(f_i(\vec{x},t) - f_i^{eq}(\vec{x},t)) \quad (1)$$

where $\vec{x}$ is a spatial coordinate, t is time, $f_i(\vec{x},t)$ is the density distribution function associated with discrete velocity direction i, and τ is the relaxation time of the fluid. The discrete velocities $\vec{e}_i$ in the i$^{\text{th}}$-direction, for the D2Q9 lattice are given by $\vec{e}_0 = 0$ and $\vec{e}_i = \lambda_i(\cos\theta_i, \sin\theta_i)$ with $\lambda_i = 1, \theta_i = (i-1)\pi/2$ for $i = 1$—4 and $\lambda_i = \sqrt{2}, \theta_i = (i-5)\pi/2 + \pi/4$ for $i = 5$—8. The order number $i = 1$—4 and $i = 5$—8 represent the rectangular and the diagonal directions of the lattice, respectively. Also, f_i^{eq} is the equilibrium distribution function, defined as:

$$f_i^{eq} = w_i\rho\left[1 + \frac{(\vec{e}_i\cdot\vec{u})}{c_s^2} + \frac{(\vec{e}_i\cdot\vec{u})^2}{2c_s^4} - \frac{(\vec{u}\cdot\vec{u})}{2c_s^2}\right] \quad (2)$$

where $c_s = 1/\sqrt{3}$ is the lattice speed of sound, $\vec{u}$ is the velocity, and w_i are the weighting factors, equal to 4/9 for $i = 0$, 1/9 for $i = 1$—4 and 1/36 for $i = 5$—8. The local mass density, the viscosity, velocity and the pressure in lattice units are calculated as $\rho = \sum_i f_i$, $\nu = (\tau - 0.5)/3$, $\vec{u} = (\sum_i \vec{e}_i f_i)/\rho$ and $p = \rho c_s^2$, respectively.

Eq. (1) is usually solved through standard collision and streaming steps as:

$$\text{Collision}: \tilde{f}_i(\vec{x}, t+\Delta t) = f_i(\vec{x},t) - \frac{\Delta t}{\tau}(f_i(\vec{x},t) - f_i^{eq}(\vec{x},t)) \quad (3)$$

$$\text{Streaming}: f_i(\vec{x}+\vec{e}_i\Delta t, t+\Delta t) = \tilde{f}_i(\vec{x}, t+\Delta t) \quad (4)$$

where $\tilde{f}_i$ represents the post-collision state. The simulation is performed on a square Cartesian grid where for convenience dimensionless lattice units are utilized, i.e. $\Delta\vec{x} = \Delta t = 1$. During the streaming step the particles move from a node to its neighbor node, according to the set of discrete velocities. Computationally speaking, during the streaming step all distribution functions are copied to the adjacent node in the direction of the lattice vector. Therefore, the streaming step involves very little computational effort. During the collision step the particles relax towards local equilibrium according to the collision operator.

In this research for computing the fluid force on a body, the momentum exchange approach [30] after the streaming step is applied:

$$\vec{F} = \sum_{all\ \vec{x}_b}\sum_{i=1}^{8}\vec{e}_i[\tilde{f}_i(\vec{x}_b,t) + \tilde{f}_{\bar{i}}(\vec{x}_f,t)] \quad (5)$$

where $\bar{i} = -i$ and the summation is done over all boundary nodes $\vec{x}_b$, which are connected to a fluid node in the i direction according to $\vec{x}_b = \vec{x}_f + \vec{e}_i\Delta t$ (see Fig. 1).

2.2. No-slip boundary condition

One important and outstanding issue in the LBM is that of solid boundary conditions needed to effectively model the interactions between fluid flow and a solid wall. In LBM, the boundary conditions influence the accuracy and stability of the computation. In the conventional CFD methods, the boundary conditions are defined using macroscopic variables (such as velocity), whereas in the LBM, the boundary conditions are enforced by changing the distribution functions, but no physically based boundary constraints for the distribution functions are provided. Therefore compared to traditional CFD methods, the LBM suffers from unknown variables at boundaries. In other words, the difficulty of applying boundary conditions in the LBM is to determine the particle distribution functions leaving into the fluid domain, but which originate from outside the bulk fluid domain (e.g. solid walls).

Fig. 1 depicts the schematic of the wall boundary condition for a one-dimensional arbitrary shaped moving object between the lattice nodes of spacing Δx. Because a no-slip boundary with a non-moving wall can be modelled by mirroring the fluxes at the boundary along each possible velocity directions separately, for one dimension it is sufficient to only consider a one-dimensional projection of lattice direction with its intersection with a (curved) wall boundary surface at x_w. The curved-wall boundary may be located at an arbitrary position between the solid and fluid nodes (i.e. x_b and x_f). The fraction of an intersected link in the fluid region is expressed using a parameter q as $q = (x_w - x_f)/\Delta x$. Obviously, $0 \leqslant q \leqslant 1$ and the distance between x_b and x_w is $q\Delta\vec{x}$.

We model the surface of a non-spherical particle as a no-slip boundary, using the so-called Bouzidi scheme [31]. Bouzidi et al. [31] proposed a method with second-order accuracy that does not require the extrapolations from the ghost nodes in solid wall. The Bouzidi scheme combines the bounce-back concept with quadratic interpolation of the distribution functions from the internal fluid nodes. For $q < 0.5$:

$$\tilde{f}_{\bar{i}}(x_f, t+\Delta t) = q(1+2q)\tilde{f}_i(x_f,t) + (1-4q^2)\tilde{f}_i(x_{ff},t) - q(1-2q)\tilde{f}_i(x_{fff},t) \quad (6)$$

and for $q \geqslant 0.5$:

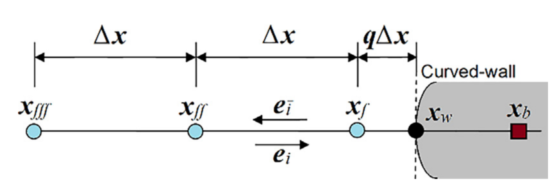

Fig. 1. One-dimensional representation of a regular lattice and a curved-wall boundary.

Please cite this article in press as: A. Zarghami, J.T. Padding, Drag, lift and torque acting on a two-dimensional non-spherical particle near a wall, Advanced Powder Technology (2018), https://doi.org/10.1016/j.apt.2018.03.019