International Journal of Multiphase Flow 68 (2015) 80–92

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Modelling of gas–solid turbulent channel flow with non-spherical particles with large Stokes numbers

Berend van Wachem *, Marian Zastawny, Fan Zhao, George Mallouppas

Division of Thermofluids, Department of Mechanical Engineering, Imperial College London, Exhibition Road, London SW7 2AZ, United Kingdom

ARTICLE INFO

Article history:
Received 7 March 2013
Received in revised form 3 October 2014
Accepted 9 October 2014
Available online 22 October 2014

Keywords:
Non-spherical particles
Turbulent gas–solid flow
Large eddy simulation

ABSTRACT

This paper describes a complete framework to predict the behaviour of interacting non-spherical particles with large Stokes numbers in a turbulent flow. A summary of the rigid body dynamics of particles and particle collisions is presented in the framework of Quaternions. A particle-rough wall interaction model to describe the collisions between non-spherical particles and a rough wall is put forward as well. The framework is coupled with a DNS-LES approach to simulate the behaviour of horizontal turbulent channel flow with 5 differently shaped particles: a sphere, two types of ellipsoids, a disc, and a fibre. The drag and lift forces and the torque on the particles are computed from correlations which are derived using true DNS.

The simulation results show that non-spherical particles tend to locally maximise the drag force, by aligning their longest axis perpendicular to the local flow direction. This phenomenon is further explained by performing resolved direct numerical simulations of an ellipsoid in a flow. These simulations show that the high pressure region on the acute sides of a non-spherical particle result in a torque if an axis of the non-spherical particle is not aligned with the flow. This torque is only zero if the axis of the particle is perpendicular to the local direction of the flow. Moreover, the particle is most stable when the longest axis is aligned perpendicular to the flow.

The alignment of the longest axis of a non-spherical particle perpendicular to the local flow leads to non-spherical particles having a larger average velocity compared to spherical particles with the same equivalent diameter. It is also shown that disc-shaped particles flow in a more steady trajectory compared to elongated particles, such as elongated ellipsoids and fibres. This is related to the magnitude of the pressure gradient on the acute side of the non-spherical particles. Finally, it is shown that the effect of wall roughness affects non-spherical particles differently than spherical particles. Particularly, a collision of a non-spherical particle with a rough wall induces a significant amount of rotational energy, whereas a corresponding collision with a spherical particle results in mostly a change in translational motion.

Introduction

Knowledge of the dynamics of turbulent gas–solid flows has a great importance for the successful design and determination of optimum operating conditions of numerous industrial applications, e.g. pneumatic transport, cyclone separators, fluidised beds, dust collectors, and pulverised-coal combustors to name a few. These systems exhibit complex flow dynamics and interactions between flow components. In particular, the complexity of the interaction between particles and gas-phase turbulence (Vreman, 2007) and the effect of particle–particle and particle–wall collisions (Sommerfeld and Kussin, 2003) have stimulated research work in recent years.

Turbulent gas–solid flows have been studied experimentally (e.g. Snyder and Lumley, 1971; Kulick et al., 1994; Kussin and Sommerfeld, 2002) and numerically. Numerical simulations can be done in an ensemble-averaged framework, in which the particle properties are represented by their mean or a PDF (e.g. Simonin et al., 1993; Minier and Peirano, 2001; van Wachem et al., 2001a). Alternatively, the location and other properties of each individual particle can be tracked, the so-called Lagrangian approach (e.g. Tsuji, 1993; Tsuji et al., 1992; van Wachem et al., 2001b; Kuang et al., 2008). With this approach, various frameworks can be used to account for the interaction of the particle with the surrounding fluid. Most common is the so-called “point-particle” approach, in which an empirical expression is used to estimate the interaction

---

* Corresponding author. Tel.: +44 20 759 47030; fax: +44 20 759 45702.
E-mail address: Berend.van.Wachem@gmail.com (B. van Wachem).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2014.10.006
0301-9322/

between the fluid and the particle, which is added as a momentum source to the fluid. A valid empirical relation between the local fluid properties and the interaction forces for the specific particle must exist in order to use this approach. Moreover, a point-source approach is only valid if the particle is sufficiently small with respect to the Kolmogorov scale of the fluid. Otherwise, a more detailed coupling algorithm must be used, which takes into account the no-slip condition on the surface of each particle (e.g. Patankar et al., 2000; Mittal and Iaccarino, 2005; Mark and van Wachem, 2008). Although this type of coupling is more accurate, it is also computationally very expensive and currently very restrictive in the number of particles it can deal with.

The majority of studies involving gas-particle flows assume that particles are perfect spheres. This assumption is very convenient because of several factors: perfect spheres are simple to model, their behaviour is well known, and lastly there is a large availability of models in the literature which describe the particle–fluid interactions (e.g. Fan and Zhu, 1998). However, assuming the particles are perfect spheres may be unrealistic, because most applications deal with non-spherical particles. Analysis of flows with non-spherical particles is considerably more complicated than flows with spherical particles. While a sphere is characterised by its diameter only, even a very simple non-spherical particle like a disc or a fibre needs at least two parameters to be uniquely defined. This makes the rigid body dynamics of non-spherical particles more complex than the corresponding dynamics of spherical particles. Moreover, additional complexities arise in describing the interaction of a non-spherical particle with a fluid. In a uniform flow a sphere experiences only a drag force, whereas a non-spherical body is also affected by a transverse lift force, a pitching torque and a counter-rotational torque. Moreover, all of these forces acting on a non-spherical body depend not only on the Reynolds number, but also on the angle between the axes of the particle and the direction of the incoming flow. Additionally, the framework for describing collisions requires a different approach compared to the one used for perfect spheres; for instance, the orientation of the particle must be taken into account. All of the factors above contribute to the complexity of the investigated problem and are addressed throughout this article.

A comprehensive overview of the available methods to describe the shape, the resulting drag force based on correlations and their associated behaviour of non-spherical particles is presented in Chhabra et al. (1999), Mandø and Rosendahl (2010). A common approach to describe the particle shape is by using a so-called "sphericity factor", Φ (Wadell, 1934). Sphericity is defined as the ratio of the surface area of a sphere over the surface area of a non-spherical particle with the equivalent volume. By definition, the sphericity is less than or equal to one. In most engineering handbooks (e.g. Crowe, 2005) and papers (e.g. Hölzer and Sommerfeld, 2008) the drag of a non-spherical particle is estimated from correlations for spherical particles which are modified to take into account the sphericity factor.

The majority of papers concerning the simulation of the behaviour of non-spherical particles use the framework of Brenner (1964) to determine the hydrodynamic drag interaction and Jeffery (1922) to describe the hydrodynamic torque acting on a particle from a flow (e.g. Marchioli et al., 2010; Marchioli and Soldati, 2013; Njobuenwu and Fairweather, 2013; Zhao and van Wachem, 2013a). However, both models assume creeping flow and Stokes flow conditions, and are in principle not valid to describe gas-particle flows where there is a slip between the particle and fluid velocity. Hence, simulations carried out using these models cannot resolve gas-particle flows with non-spherical particles, where there is a slip between the particle and the fluid flow, *i.e.* particles with finite Stokes numbers.

In Zastawny et al. (2012), the development of models for the drag, lift and torques acting on non-spherical particles with a significant slip has been researched by means of true direct numerical simulation. The term "true" emphasises that not only all the flow scales are resolved but also a no-slip boundary condition is applied at the surface of particle. As all the existing flow scales are resolved, there are no assumptions required at this scale to capture the interaction of the particles with the fluid flow. The true direct numerical simulations in this paper are shown to be grid independent, and a large number of simulations have been performed for each particle shape. Although there is a good agreement from the new drag, lift and torque model with the analytical models of Brenner (1964) and Jeffery (1922), the models show that the behaviour of non-spherical particles at larger slip velocities is quite different from the models put forward by Brenner (1964) and Jeffery (1922).

The most notable difference of the forces on a non-spherical particle in a flow with a significant slip velocity, is the detachment of the flow at the acute edges on the particles. This is illustrated by a result of the resolved direct numerical simulation shown in Fig. 1. This figure shows an ellipsoid in a flow with a slip velocity between the particle and the fluid, the Reynolds number based on the slip velocity is 200. It can be clearly seen that the acute edges of the particle cause the flow to separate. This leads to high pressure regions near these points of detachment, as is indicated by the colours of Fig. 1. This was also confirmed in Hölzer and Sommerfeld (2008).

These high pressure regions cause a net fluid torque to act on the particle, and as a consequence the particle will rotate until the pressure gradients are of equal magnitude on both sides of the particle. Thus, the configuration as shown in Fig. 1 is unstable, and the resulting net torque on the particle originates from the difference in pressure gradients on either side of the particle. This will result in a rotation of the particle in the flow, until the pressure gradients are maximum and of equal magnitude on both sides of the particle. Hence, a non-spherical particle will tend to maximise its drag once there is a slip velocity between the particle and the fluid. This is also commonly observed in nature, as described for instance in Hoerner (1965): leaves that fall from a tree do not fall as fast as possible, but maximise their drag and their falling time. There are numerous other examples of this in nature.

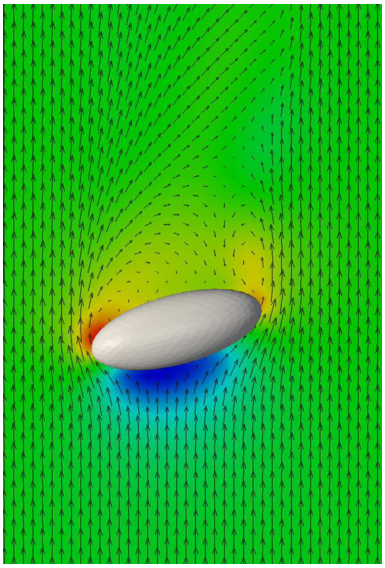

Fig. 1. A snapshot of an ellipsoid in a flow with significant slip ($Re_p = 200$). The vectors indicate the flow velocity and the colours indicate the relative pressure.