International Journal of Multiphase Flow 59 (2014) 221–243

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

The motion of a single heavy sphere in ambient fluid: A benchmark for interface-resolved particulate flow simulations with significant relative velocities

Markus Uhlmann [a,*], Jan Dušek [b]

[a] Institute for Hydromechanics, Karlsruhe Institute of Technology (KIT), 76131 Karlsruhe, Germany
[b] Institut de Mécanique des Fluides et des Solides, Université de Strasbourg, 67000 Strasbourg, France

ARTICLE INFO

Article history:
Received 31 July 2013
Received in revised form 16 October 2013
Accepted 19 October 2013
Available online 13 November 2013

Keywords:
Sphere wake
Particulate flow
Direct numerical simulation
Interface resolution
Spectral element method
Benchmarking
Immersed boundary method

ABSTRACT

Detailed data describing the motion of a rigid sphere settling in unperturbed fluid is generated by means of highly-accurate spectral/spectral-element simulations with the purpose of serving as a future benchmark case. A single solid-to-fluid density ratio of 1.5 is chosen, while the value of the Galileo number is varied from 144 to 250 such as to cover the four basic regimes of particle motion (steady vertical, steady oblique, oscillating oblique, chaotic). This corresponds to a range of the particle Reynolds number from 185 to 365. In addition to the particle velocity data, extracts of the fluid velocity field are provided, as well as the pressure distribution on the sphere's surface. Furthermore, the same solid–fluid system is simulated with a particular non-boundary-conforming approach, i.e. the immersed boundary method proposed by Uhlmann (2005a), using various spatial resolutions. It is shown that the current benchmark case allows to adjust the resolution requirements for a given error tolerance in each flow regime.

1. Introduction

The gravity-induced settling or rising of a spherical rigid body in a viscous fluid exhibits a rich set of dynamical features, involving a variety of patterns of motion from steady vertical to fully chaotic in different regions of the parameter space. Many aspects of the flow physics have been discussed in a recent review by Ern et al. (2012). When considering spheres settling in a priori quiescent ambient surroundings all deviations from a straight vertical path as well as all unsteadiness originate from the characteristics of the fluid motion in the near-field around the immersed object and in its wake. Therefore, the analysis of the motion of settling/rising objects really implies an investigation of the features of particle wakes.

Beyond their relevance to particle trajectories, wakes generated by moving particles are of significance in the context of particle-induced turbulence generation and modification. One question which is often posed in particulate flow systems pertains to the amount of turbulence enhancement or attenuation due to the addition of particles to a given fluid flow. Elucidating the physics of wakes shed by single (and multiple) mobile particles is expected to contribute to a better understanding of the technologically important problem of turbulence-particle interaction to which a considerable effort has been devoted (Balachandar and Eaton, 2010).

As a complement to modern experimental techniques, it has now become feasible to simulate numerically the flow around a reasonably large amount of moving immersed objects based upon the Navier–Stokes equations (e.g. Ten Cate et al., 2004; Uhlmann, 2008; Lucci et al., 2010, 2011; García-Villalba et al., 2012; Gao et al., 2013). For reasons of computational efficiency, most of the simulations of this kind employ numerical techniques which do not rely on geometry-conforming grids, thereby avoiding the necessity for repeated remeshing and complex data structures. Instead, the general idea of these methods is to allow for the treatment of a single medium throughout the domain occupied by both the fluid and the solid, while imposing locally the constraint of rigid body motion through some kind of manipulation of the Navier–Stokes equations. While the general concept of these non-conforming methods as well as their efficiency has now been widely established, it is felt that rigorous resolution criteria have not yet been determined in all situations.

Typically, finite-size particle flow simulation codes are validated with respect to a sub-set of the following test cases:

* Corresponding author. Tel.: +49 721 60844106; fax: +49 721 60842202.
E-mail addresses: markus.uhlmann@kit.edu (M. Uhlmann), dusek@unistra.fr (J. Dušek).

0301-9322/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijmultiphaseflow.2013.10.010

1. Flow around a fixed sphere with uniform, steady inflow versus standard drag correlations (such as Clift et al., 1978) or high-fidelity numerical data (e.g. Johnson and Patel, 1999; Bouchet et al., 2006).
2. Gravitational settling of a single heavy sphere versus reference data, e.g. by Mordant and Pinton (2000).
3. "Drafting-kissing-tumbling": gravitational settling of a pair of cylinders (in two space dimensions) or spheres initially trailing each other, for which no rigorous reference data exists to our knowledge.
4. Rotation of a single fixed cylinder (in two space-dimensions) or ellipsoid in Couette flow versus reference data from experiments (Zettner and Yoda, 2001), analytical solutions (Jeffery, 1922) or numerical reference data (e.g. Ding and Aidun, 2000).
5. Lateral migration of a single neutrally-buoyant particle in laminar Hagen-Poiseuille flow versus analytical results (Asmolov, 1999) and experimental data (Matas et al., 2004). For this case numerical reference data is available (e.g. Yang et al., 2005). The computationally less demanding case of two-dimensional flow around migrating circular disks in plane channel flow has been studied numerically by Inamuro et al. (2000), Pan and Glowinski (2002), and Joseph and Ocando (2002).

Concerning flows with significant relative velocities between the solid and the fluid phase (e.g. due to buoyancy effects as in sedimentation systems), our personal experience has shown that the above array of validation tests might not be representative of all relevant flow features. In particular, the subtle dynamics of particle motion due to differences in wake characteristics in the various regions of the parameter space may not be sufficiently captured by a numerical code at a given resolution although it might perform reasonably well in the above cases. Therefore, the purpose of the present work is to provide a further benchmark configuration serving as a test of simulation tools for fully-resolved fluid-particle motion.

The case of a single settling unconfined sphere in the absence of solid boundaries appears an attractive configuration in this context. On one hand, high-fidelity data can be generated by means of relatively efficient reference simulations with spectral accuracy (Jenny and Dušek, 2004). In the reference method, the mesh is translated with the immersed object which avoids remeshing (Mougin and Magnaudet, 2002). On the other hand, as mentioned above, the settling process of a single sphere covers all the essential dynamics involved in general sedimentation problems, including very subtle effects of wake-induced non-trivial trajectories, while excluding additional complexity due to inter-particle collisions. It is as such a challenging and rigorous test case for any non-geometry-conforming numerical simulation method. At the same time the benchmark simulations need not be excessively demanding, since the size of the computational domain can be kept relatively small. Furthermore, the initial state and the boundary conditions of the problem are simple and well-defined.

For this purpose we have generated detailed data for the flow field and the rigid body motion in the case of a single heavy sphere settling in quiescent surroundings, using a highly accurate spectral/spectral-element method. The simulations are similar to those performed by and described in Jenny et al. (2004). However, in the present work the computational domain was purposefully kept small, thereby requiring new simulations. Furthermore, in the present paper we aim at reporting a complete set of data (contrary to the previous publication of Jenny et al., 2004) for the purpose of validating alternative numerical methods.

In parallel, we report results from computations of the same flow configuration obtained by means of a non-geometry-conforming code based upon an immersed boundary method (IBM, Uhlmann, 2005a). We have performed refinement tests from which the required small-scale resolution can be deduced in each flow regime.

The outline of the paper is the following. In Section 2 the flow geometry, boundary conditions and the numerical method used to generate the reference data is described, before we proceed to present the benchmark data. In Section 3 we further illustrate the validation procedure by describing simulations performed with an immersed boundary method; the numerical approach is first summarized (Section 3.1) and then the results are compared to the reference data (Section 3.2). The paper closes with a summary and discussion in Section 4.

2. Reference case

2.1. Flow configuration and governing equations

We are considering the motion of a spherical solid body with diameter d immersed in a fluid under the action of a gravitational field. Fig. 1 illustrates the geometry of the problem as well as the definition of the different coordinate systems which will be used in the following. The first set of Cartesian coordinates (x, y, z) describes a position with respect to the center of the sphere. Secondly, the Cartesian coordinates with respect to a fixed origin are denoted as $(x_{fix}, y_{fix}, z_{fix})$. The directions of the axes in both of these Cartesian coordinate systems are the same, with the z and z_{fix} axes pointing into the direction opposite to gravity. The position of the sphere in the fixed coordinate system is henceforth denoted as $\mathbf{x}_{sphere}$. Alternatively, we use a cylindrical coordinate system (the origin of which is attached to the center of the particle), with the coordinates denoted as (z, r, θ), r being the radial coordinate and θ the azimuthal angle in the horizontal plane.

The equations for the flow of a viscous incompressible fluid can be written as

$$\partial_t \mathbf{u} + \left([\mathbf{u} - \mathbf{u}_p] \cdot \nabla \right) \mathbf{u} + \nabla p = \frac{1}{G} \nabla^2 \mathbf{u}, \tag{1a}$$

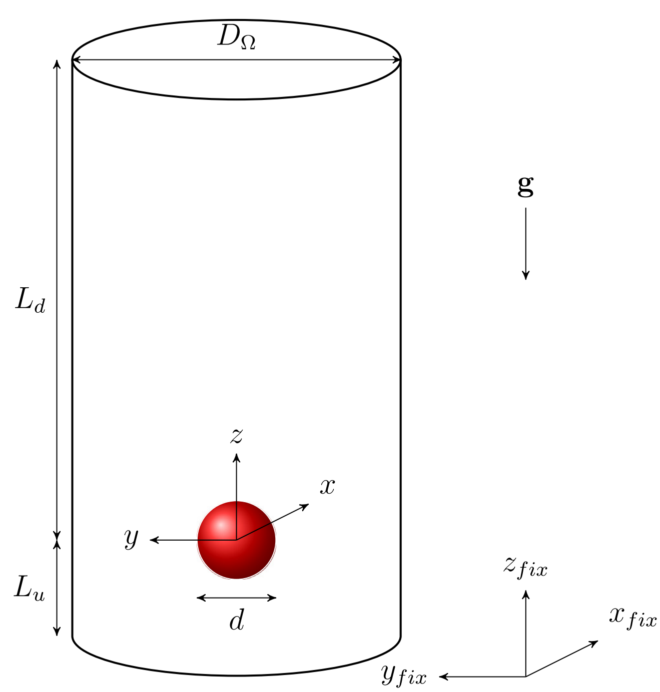

Fig. 1. The geometry of the problem and the computational domain as employed in the reference method described in Section 2.