International Journal of Multiphase Flow 79 (2016) 88–106

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Vortex dynamics of a sphere wake in proximity to a wall

Hui Zhao, Xiaofei Liu, Dong Li, Anyang Wei, Kun Luo, Jianren Fan*

State Key Laboratory of Clean Energy Utilization, Zhejiang University, 38 Zheda Road, Hangzhou 310027, People's Republic of China

ARTICLE INFO

Article history:
Received 31 December 2014
Revised 26 September 2015
Accepted 6 October 2015
Available online 10 November 2015

Keywords:
Vortex dynamics
Sphere wake
Direct numerical simulation
Immersed boundary method

ABSTRACT

Toward getting the vortex dynamics characteristics and wake structure of a sphere in proximity to a wall, the effect of a proximal flat plate on the wake of a stationary sphere is investigated via direct numerical simulation. The vortex shedding process and the significant variation of the wake structure are described in detail. The drag coefficient reduces and the wake structure of the sphere becomes complex due to the combined effect of the wake flow and the wall. A jet flow forms between the sphere and the flat plate, which suppresses the vortex separation on the bottom of the sphere. The asymmetric distributions of the coherent structures and the recirculation region behind the sphere are discussed. Besides vortex shedding patterns, the time-averaged velocity distribution, vortex dynamics, distribution regularities of turbulent kinetic energy and enstrophy are investigated.

Introduction

Many studies have been conducted to investigate the flow field around a bluff body in a uniform flow, which has been considered as a classical issue. Among the previous studies of the flow past a bluff body, such as circular cylinder, square cylinder, streamlined body etc., the study of flow field around a sphere is valuable and necessary because of its wide application in the chemical and energy fields. The primary interests of related experimental and numerical studies lie in the visualization of the wake configuration, the vortex shedding mechanisms, and the integral parameters (e.g., the drag coefficient of the sphere) (Ben Salem and Oesterle, 1998; Constantinescu and Squires, 2003; Hassanzadeh et al., 2011; Kim and Durbin, 1988; Sakamoto and Haniu, 1990; Yun et al., 2006).

On the basis of previous research of flow past a sphere, Rodriguez et al. (2011) studied the vortex-shedding dynamics and wake characteristics of the flow behind a sphere at a Reynolds number $Re = 3700$ by means of direct numerical method. They provided the visualization of vortex structures over a long period of time which showed that the wake has a marked helical-like configuration, and the vortices move downstream without circulation in the azimuthal direction. They also found that the shear layer instabilities occur at a random position and the vortices are shed periodically at no particular azimuthal position. However, in contrast to unbounded boundary conditions, a sphere placed over a horizontal wall has received little attention. Influenced by the proximal flat plate, the sphere wake could be significantly changed, and the interactions between the sphere wake and the wall are still unclear. These solutions can also be directly applied to many engineering problems, such as suspended particles in wind tunnels, water channels, chemical containers, and certain architectural structures, such as gas tanks.

In early studies of this problem, Okamoto (1980) performed experimental investigation on the turbulent shear layer behind a sphere placed on a flat plate. According to the research, the length of the recirculation zone was presented as 2.5D and was shortened by the flat plate. Based on the study of the turbulence intensity, the turbulence becomes isotropic at $z/D = 1.2$ in the region where $x/D = 10$, while the streamwise turbulence intensity is larger than the lateral and vertical turbulence intensities for $z/D < 1.0$. The wall wake behind the sphere becomes lower and spreads transversely with increasing the downstream distance, and the drag coefficient is larger than that of a sphere in a uniform flow. Tsutsui (2008) performed an experimental study of the flow around a sphere over a flat plate with a Reynolds number of 8.3×10^4. The gap ratio G/D varied from 0 to 0.526, where G (denoted by S in Tsutsui 2008) was the distance between the bottom point of the sphere and the flat plate surface, and D was the diameter of the sphere. The behaviors of the flow and vortices were observed during flow visualization using the smoke-wire and surface oil-flow patterns methods. For different ranges of G/D, the characteristics of the vortices were different. In the ranges $G/D \leq 0.026$ and ≤ 0.088, horseshoe and arch vortices formed, respectively, and in the range $G/D \leq 0.175$, the vortices were shed from the sphere. The drag coefficient, C_D, corrected by the flow velocity associated with the stagnation streamline approximate the constant value of the drag coefficient of a single sphere in a uniform flow, except for $G/D = 0$. Recently, the flow characteristics around a sphere located above a smooth flat plate in a water channel were experimentally investigated using dye visualization and the particle image velocimetry

* Corresponding author. Tel.: +86 0571 87951764; fax: +86 0571 87951764.
E-mail address: fanjr@zju.edu.cn (J. Fan).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2015.10.005
0301-9322/

(PIV) technique (Ozgoren et al., 2013). The Reynolds number was within the range of $2500 \leq Re \leq 10{,}000$ based on the free-stream velocity and the sphere was fixed in different locations in the range of $0 \leq G/D \leq 1.5$. Instantaneous and time-averaged flow patterns in the wake region of the sphere were examined. The results demonstrated that the wall affects the sphere wake at small gap spacing, constrains the flow passing through the gap, and restricts the vortex shedding from the sphere, especially in the lower section of the sphere.

Numerical studies of the flow around a stationary or rotating circular cylinder next to a wall have increased in recent years, but three-dimensional simulations of a sphere embedded over a flat plate are still limited at present. Lee et al. (2011) conducted direct numerical simulations of a rolling particle for shear Reynolds number up to 100. The far-field ambient flow surrounding the particle is a linear shear flow, and the immersed boundary (IB) method proposed by Uhlmann (2005) was used to enforce the no-slip condition on the particle. The influence of the shear Reynolds number and the translational velocity of the particle on the hydrodynamic forces of the particle were investigated under both transient and the final drag-free and torque-free steady state. It was observed that the presence of the wall has a strong effect on the unsteady added-mass and history forces. Zeng et al. (2007) and Zeng et al. (2010) performed fully resolved direct numerical simulations of a turbulent channel flow over an isolated spherical particle of finite size for Reynolds number of a few hundreds. Two different particle locations and a range of particle sizes were considered in their study. The time-averaged mean wake length decreases when the ambient flow is turbulent and the total kinetic energy is significantly reduced in the wake region. The influence of turbulent structures on particle force and the effect of the particle wake dynamics and vortex shedding on turbulence modulation were analyzed in detail. Homann et al. (2013) investigated the impact of turbulent fluctuations on the forces exerted by a fluid on a towed spherical particle by means of high-resolution direct numerical simulations. Both the laminar upstream flow and the turbulent upstream flow were adopted, and the particle Reynolds number was up to 400. The study of the turbulent wake downstream of the sphere was reported in that work, and it was shown that the average drag force increases as a function of the turbulent intensity and the particle Reynolds number.

Despite the fact that the study of the sphere wake influenced by a wall has begun to attract attention, numerical simulations of the interactions between them are still lacking, especially in the critical Reynolds number regime. Thus, the present investigation describes the wake flow of a stationary sphere located in the proximity of a flat plate using direct numerical simulation. The structure of the paper is as follows: first, the numerical method, including the basic governing equations, is introduced. The next section discusses the simulation results, including time-averaged flow features, vortex dynamics and turbulent kinetic energy. The influence of the gap ratio on the flow field is also discussed. Finally, the conclusions are presented.

Numerical methodology

Governing equations and the immersed boundary method

The immersed boundary method has been widely used in many different simulation areas (Lee et al., 2009; Lu et al., 2012). The present simulation uses the direct numerical simulation with a combined multiple-direct forcing and immersed boundary (MDF/IB) method (Luo et al., 2007; Zhou et al., 2010). The MDF/IB method has a wide applicability due to its many advantages, including the convenience of grid generation, the easy satisfaction of the no-slip condition, the high efficiency of the calculation, and the relaxed stability constraint.

In the following description, the computational domain comprising the actual fluid domain and the space occupied by the solid object is denoted by Ω. The non-dimensional governing equations for the incompressible flow can be expressed as:

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

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

where $\mathbf{u}$ is the velocity of fluid, p is the pressure, Re is the Reynolds number, and $\mathbf{f}$ is an external force that represents the mutual interaction force between the fluid and the immersed boundary. $\mathbf{f}$ is expressed as

$$\boldsymbol{f}(x) = \sum_{k=1}^{N_L} \mathbf{F}_k(X_k)\delta(x - X_k)\Delta V_k, \tag{3}$$

where $\delta(x - X_k)$ is the Dirac-delta function (Peskin, 2002) through which the two-way coupling between the Lagrangian points and the Eulerian grids could be achieved. N_L represents the total Lagrangian points on the immersed boundary, and ΔV_k is the volume of each force point such that the union of all these volumes forms a thin shell with a thickness equal to one mesh width around each immersed boundary (Uhlmann, 2005). To ensure the fact that the no-slip boundary condition of the velocity at the immersed boundary could be satisfied, direct forcing, $\boldsymbol{F}_k(X_k)$, is introduced.

When spreading the effect of the forcing from Lagrangian points to Eulerian grids with the direct forcing scheme, the force acted on the Lagrangian point which contains the desired velocity could be calculated. However, the velocities on the Lagrangian points may not satisfy the no-slip boundary condition very well during the process of interpolation to obtain the simulated velocity on the Lagrangian points and extrapolation to spread the forcing effect to its surrounding Eulerian grids. Therefore, the multi-direct forcing technique is applied. The multi-direct forcing process is composed of many cycles (as many as NF times of exerting multi-direct forcing), and the total force exerted on each Lagrangian point is the sum of the force exerted on the Lagrangian point each time during NF cycles; thus, the velocity at the immersed boundary can approach the desired velocity under the no-slip conditions:

$$\boldsymbol{F}_k(X_k) = \sum_{i=1}^{NF} \boldsymbol{F}_k^i(X_k), \tag{4}$$

and

$$\boldsymbol{f}(x) = \sum_{k=1}^{N_L} \sum_{i=1}^{NF} \mathbf{F}_k(X_k)\delta(x - X_k)\Delta V_k. \tag{5}$$

The detailed formulas are explained by Mohd-Yusof (1997), Luo et al. (2007, 2010).

The parameter NF is crucial to the accuracy of the multi-direct method and the most important parameter to properly implement the method. A better no-slip boundary condition at the immersed boundary can be reached as the number of times that the multi-direct forcing scheme is performed increases, which indicates that, when the value of NF is higher, the accuracy is better; however, to maintain computational efficiency, a low value is preferable (Breugem, 2010). Luo et al. (2007) had examined the response of the velocity on the immersed boundary to the time of exerting multi-direct forcing, and suggested that the no-slip condition was satisfied well with $NF \geq 10$.

In the current work, the governing equations are solved using a finite difference scheme (Harlow and Welch, 1965). To ensure the stability of computational process, a third-order accuracy upwind compact scheme (Fu and Ma, 1997) was used to represent the convective term of the momentum equation, and a sixth-order-accurate centered compact (Lele, 1992) scheme was used to represent other first-order spatial derivatives. A low-storage four-step-fourth-order Runge–Kutta scheme (Jameson and Schmidt, 1985) was used to step