European Journal of Mechanics B/Fluids 60 (2016) 48–61

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

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

Numerical investigation on characteristic flow regions for three staggered stationary circular cylinders

Weiwei Yan [a,*], Jie Wu [b], Shuchao Yang [a,b], Yong Wang [c]

[a] College of Metrology and Measurement Engineering, China Jiliang University, Hangzhou, 310018, China
[b] Department of Aerodynamics, Nanjing University of Aeronautics and Astronautics, Nanjing, 210016, China
[c] Max Planck Institute for Dynamics and Self-Organization, Göttingen, 37077, Germany

ARTICLE INFO

Article history:
Received 21 December 2015
Received in revised form
11 July 2016
Accepted 16 July 2016
Available online 28 July 2016

Keywords:
Lattice Boltzmann method
Immersed boundary method
Steady and unsteady regions
Vortex shedding

ABSTRACT

In this work, the characteristic flow regions for three stationary circular cylinders were numerically investigated by the multiple-relaxation-time (MRT) based lattice Boltzmann method (LBM). The immersed boundary method (IBM) was employed to handle the solid boundary of cylinders to account for the fluid–solid interaction. The cylinders were arranged in a staggered configuration, which means that one cylinder was placed in front of the others with side-by-side arrangement. The calculations were carried out at different spacing ratios T/D (varying from 1 to 10) and fixed spacing ratio $S/D = 3$ with a constant Reynolds number $Re = 200$, which represents a typical unsteady laminar flow. Here, D is the diameter of the cylinders, T is the spacing between the centers of two downstream cylinders, and S is the distance between the centers of the upstream cylinder and downstream cylinders. The experiments based on the laser-induced fluorescence (LIF) flow visualization were performed to verify the reliability of simulation results. The results indicated that the spacing ratio T/D has a significant influence on the wake structures. Two different characteristic steady and unsteady flow regions behind the upstream cylinder were observed. The characteristic steady flow occurs at the regions of $1 \leq T/D \leq 1.2$ and $2.5 \leq T/D \leq 3.1$, and the characteristic unsteady flow happens at the regions of $1.3 \leq T/D \leq 2.4$ and $3.2 \leq T/D \leq 10$. The present results would be helpful for designing multiple piers in the practical application.

1. Introduction

The flow past multiple circular cylinders has received great attentions due to its wide practical applications in industry and engineering, such as tall buildings, offshore platforms, electrical transmission lines, bridge constructions and heat exchangers, to name a few. When the Reynolds number exceeds a critical value, the flow becomes unsteady and the famous Kármán vortex street is formed. As a consequence, the structures are unavoidably damaged due to the oscillatory force and vibration. It is well known that the arrangement of multiple cylinders may influence the flow patterns around them. For instance, since the water over pipes in the sea or the wind past a windmill may damage the structures, the downstream cylinders are generally employed to protect the upstream structure and reduce the impact of vibration. Therefore, it is of great importance to study the effect of cylinders arrangement on the cylinder wake. A particular arrangement can be identified to suppress the vortex shedding and reduce the impact of vibration on the cylinders.

Both numerical and experimental investigations are efficient ways to solve complex flow problems. To date, many studies on the flow over circular cylinders as well as the related flow control have been reported. To control the flow past an isolated cylinder, the rotation of the cylinder is a feasible control technique [1–3], and the use of the auxiliary devices is another choice [4–6]. For the flow past two circular cylinders, a large amount of work has been achieved in the laminar flows [7]. Six different wake patterns were observed within the low Reynolds number regime for two side-by-side cylinders. It is suggested that the appearance of these patterns is sensitive to both the Reynolds number and the gap spacing between the centers of two cylinders (L) [8]. The flow characteristics over two tandem cylinders in the laminar flow have also been studied. Two distinct flow patterns were found, with which single and double vortex streets are generated. It is found that the flow patterns strongly depend on the gap spacing between

* Corresponding author.
E-mail address: yanww@cjlu.edu.cn (W. Yan).

http://dx.doi.org/10.1016/j.euromechflu.2016.07.006
0997-7546/

the centers of tandem cylinders (S). The critical gap spacing (S_c) of the flow patterns alternation occurs at $3.25D < S < 3.5D$, where the Strouhal number experiences a sharp transition [9]. Besides the flow patterns, much attention has also been paid to the time-averaged forces on the two cylinders. For the side-by-side arrangement, the summation of the mean drag coefficient C_d on two cylinders is always less than two times of that of the isolated cylinder at $T/D = 3$ [10], and the mean lift coefficients (C_l) on the cylinders are always repulsive and decreased with the increase of T/D [11]. For the root-mean-square lift and drag coefficients, their maximal values at $T/D = 2.2$ are nearly 46% and 50% higher than those of the isolated cylinder, respectively [12]. The time-averaged forces on two tandem cylinders have also been analyzed. The C_d for the upstream cylinder is always larger than that for the downstream one at $2 \leq S/D \leq 10$ [13]. When $S > S_c$, the value of C_d for the upstream cylinder increases gradually to $C_d \approx 1.33$ for an isolated cylinder. When $S < S_c$, the C_d for the upstream cylinder is much smaller than that of the single cylinder, and the C_d for the downstream cylinder is even negative, owing to the strong suction force induced by the upstream one [9]. The variation trends of the root-mean-square lift and drag coefficients for two tandem cylinders are very similar to each other. Both of them encounter a drastic increase at $S = S_c$, and reach their maximal values at $S \approx 4$ [14].

Besides, the flow past multiple circular cylinders has also been investigated in recent years owing to its wide applications. In the experimental studies, the flow around three side-by-side equal circular cylinders with transverse gap ratio of $G/D = 1.0$–3.0 in shallow water has been conducted using the particle image velocimetry (PIV). An asymmetrical flow structure at small gap ratio (G/D) and a symmetrical flow structure at intermediate gap ratio ($2 \leq G/D \leq 10$) were observed [15]. The flow over three equal circular cylinders in an equilateral-triangular manner with different angles of incidence and spacing ratios (L/D) has been experimentally studied. The results show that the bistable flow pattern is produced at $L/D < 2.29$. When $L/D < 4.65$, there always exists an angle at which the vortex shedding behind an upstream cylinder is suppressed by a nearest downstream one [16]. The wind tunnel experiments have also been conducted on three equal equilateral-triangular cylinders to measure the time-averaged force coefficients and Strouhal numbers at spacing ratios $L/D = 1.5$–4.0. For $L/D > 2$, the upstream cylinder experiences a lower C_d compared with the downstream ones. The minimal values of C_d for the downstream cylinders occur at $L/D = 1.5$ and 2.0, owing to no vortex shedding from the foregoing cylinders. It is indicated that the Strouhal number for the upstream cylinder increases with the decreasing of L/D [17]. In the numerical simulations, the flow around three staggered cylinders with one cylinder performing in-line vibration has been investigated. The numerical results indicate that the spacing ratio T/D has an obvious influence on the wake patterns and the formation of vortex shedding. In addition, the flow past three tandem and side-by-side cylinders at $Re = 100$ and 200 has been simulated. For the tandem arrangement, it is found that the downstream cylinder experiences very large unsteady forces that can give rise to wake-induced flutter. For the side-by-side arrangement, modulated synchronized and bifurcated parallel flip-flopping wake patterns can be observed depending on the Reynolds number and the gap spacing [18]. From the work mentioned above, one can find that the flow pattern and force coefficients are basically dependent on the spacing ratios. For the cases of three cylinders, the current investigations are focused on the flow patterns and wake interactions. However, the flow control of the upstream cylinder in the staggered arrangement has been limitedly analyzed.

As an alternative to the Navier–Stokes solvers, the LBM has achieved a great success in modeling complex fluid flows. To date, numerous applications have been developed in the LBM, such as multi-phase flow, porous media flow, particulate suspension flow, and even fluid–structure interaction [19]. The simplest LBM is the BGK models based on a single-relaxation time (SRT) approximation [20]. However, the simplicity of SRT-LBM comes at the expense of numerical instability and inaccuracy in implementing boundary conditions [21]. Recently, the MRT–LBM has become increasingly popular since it can overcome the deficiencies in the SRT-LBM [21,22]. On the other hand, the IBM has also received great attentions due to its obvious merit in dealing with complex or moving boundary problems. The IBM was firstly developed for modeling the complex blood flow in the heart [23]. Thereafter, it has undergone various modifications and improvements to become a useful method for fluid–structure interaction problems. Compared to the interpolated bounce-back method (IBBM), the IBM has the virtue of better numerical stability and easier implementation with comparable overall accuracy [24] and analogous convergence rate [25]. Although two methods have comparable computational speed, the IBM is proved to be less computationally efficient [25]. Hence, the multi-block mesh refinement technique [26] is adopted here to improve the computational efficiency. In this work, the characteristic flow regions for three stationary circular cylinders in staggered arrangement are numerically studied. The MRT–LBM is employed to compute the flow field, and the IBM is adopted to treat the solid boundaries of circular cylinders. The main objective of this work is to examine the influence of the spacing ratio T/D on the flow patterns at a fixed spacing ratio $S/D = 3$ at $Re = 200$. Here, D is the diameter of the cylinder, T is the spacing between the centers of two downstream cylinders, and S is the distance between the centers of the upstream and downstream cylinders. In addition, a series of experiments based on the laser-induced fluorescence (LIF) flow visualization technique has been performed to demonstrate the reliability of the numerical results.

2. Problem description and methodology

2.1. Problem description

As shown in Fig. 1, three equal stationary circular cylinders (cylinder 1 was the upstream cylinder, cylinder 2 and 3 were two side-by-side downstream cylinders) were placed in a 2-D cross flow with a constant uniform free-stream velocity $u_\infty = 0.1$. To eliminate the effect of the boundaries, the centers of the cylinders were located far enough from the boundaries of the flow field. The distance from the inlet to the center of cylinder 1 was set as $L_e = 10D$, and the distances from the lateral sides to the centers of cylinder 2 and 3 were also set as $T_b = L_t = 10D$. The spacing between the centers of cylinder 1 and two downstream cylinders was fixed to be $S = 3D$. The total stream-wise length of flow domain was $L_{total} = 35D$, and the total cross-stream width of flow domain was $W_{total} = T + T_b + T_t$, where the spacing T between the centers of cylinder 2 and 3 varied from D to $10D$. A Cartesian coordinate system (x, y) was used to describe the flow field. The stream-wise direction was donated as x, and the cross-stream direction was donated as y. The boundary conditions were enforced as follows: (1) at the inlet, $u_x = u_\infty$ and $u_y = 0$; (2) at the outlet, $\partial u_x/\partial x = \partial u_y/\partial x = 0$. (3) at the top and bottom, $\partial u_x/\partial y = \partial u_y/\partial y = 0$; (4) at the surface of circular cylinders, $u_x = u_y = 0$. The non-equilibrium extrapolation scheme [27] was adopted to treat the inlet, outlet, top and bottom boundaries, and the IBM [23] was used to solve the circular cylinders boundaries. The present simulation problem was characterized by a Mach number (Ma), which is primarily applied to determine the approximation with which a flow can be treated as an incompressible flow. In our simulations, $Ma = u_\infty/c_s \approx 0.17$, where $c_s = 1/\sqrt{3}$ is the speed of sound. Since the Mach number was sufficiently low to annihilate