Computers & Fluids 38 (2009) 778–788

Contents lists available at ScienceDirect

Computers & Fluids

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

Flow patterns past two circular cylinders in proximity

Kyongjun Lee, Kyung-Soo Yang *

Department of Mechanical Engineering, Inha University, 253 Yonghyun-Dong, Nam-Gu, Incheon 402-751, Republic of Korea

A R T I C L E I N F O

Article history:
Received 18 February 2008
Received in revised form 6 June 2008
Accepted 24 July 2008
Available online 5 August 2008

Keywords:
Circular cylinder
Immersed boundary method
Flow pattern
Vortex shedding
Wake interaction

A B S T R A C T

Flow patterns past two nearby circular cylinders of equal diameter immersed in the cross-flow at low Reynolds numbers ($Re \leqslant 160$), were numerically studied using an immersed boundary method. We considered all possible arrangements of the two cylinders in terms of the distance between the two cylinders and the inclination angle of the line connecting the cylinder centers with respect to the direction of the main flow. Ten distinct flow patterns were identified in total based on vorticity contours and streamlines, which are Steady, Near-Steady, Base-Bleed, Biased-Base-Bleed, Shear-Layer-Reattachment, Induced-Separation, Vortex-Impingement, Flip-Flopping, Modulated Periodic, and Synchronized-Vortex-Shedding. Collecting all the numerical results obtained, we propose a general flow-pattern diagram for each Re, and a contour diagram on vortex-shedding frequency for each cylinder at $Re = 100$. The perfect symmetry implied in the geometrical configuration allows one to use these diagrams to identify flow pattern and vortex-shedding frequencies in the presence of two circular cylinders of equal diameter arbitrarily positioned in physical space with respect to the main-flow direction.

1. Introduction

Cross-flow past a group of cylinders is often found in practical engineering applications. For instance, air flow past a bundle of pipes in a chemical plant, coolant flow past tubes in a heat exchanger, sea-water flow past columns of a marine structure, and wind past an array of chimneys of a power plant, to name a few. Flow characteristics past each cylinder are affected by its neighbors via wake interaction, resulting in alteration of the overall flow pattern. Therefore, flow pattern past multiple cylinders heavily depends on their relative positions with respect to the main-flow direction. Consequently, vortex-shedding frequency of the individual cylinder is accordingly determined, and serves as an important factor in generation of flow noise [1]. Being motivated by this, many researchers have been involved in studying wake interaction between two circular cylinders of equal diameter immersed in a cross-freestream as a basic wake-interaction model. Spatial arrangement of two cylinders can be classified into three categories, namely, aligned with the direction of the main flow (in tandem), placed side-by-side, and placed in a staggered arrangement.

When two circular cylinders of equal diameter are placed side-by-side, flow pattern varies with Reynolds number (Re) and the distance between the cylinders. Kang [2] identified six distinct flow patterns depending on Re and the distance between the cylinders. It was also found that when the surface-to-surface distance is longer than $5D$ (here, D is the cylinder diameter), wake interaction vanishes and flow past each cylinder behaves just like the flow past the single cylinder placed in the freestream [2]. Williamson [3] classified the types of vortex shedding by using flow visualization. When $Re = 100$ and the surface-to-surface distance is longer than D, the flow pattern past the two cylinders is periodic, and vortex shedding from each cylinder is either in phase or out of phase by 180° with the other one. However, when the surface-to-surface distance is shorter than D, the flow pattern past the two cylinders is non-periodic and completely irregular [2,3].

When the two cylinders are placed in tandem, there exists the critical distance between the two cylinders, below which vortex shedding of the upstream cylinder (hereafter, called "main cylinder", MC) does not occur; the value of the critical distance varies depending on Re [4–7]. Mizushima and Suehiro [4] revealed that the flow pattern with $L/D = 2.0$ is different from that with $L/D = 4.0$ at $Re = 100$ mainly due to suppression of vortex shedding from MC. Here, L denotes the streamwise distance between the two cylinder centers (Fig. 1(a)). They also predicted existence of bistable solutions in the range of $3.1 \leqslant L/D \leqslant 3.5$ for $Re = 100$. Subsequently, Tasaka et al. [5] performed experimental investigation to verify the prediction of Mizushima and Suehiro [4], and reported $4.6 \leqslant L/D \leqslant 5.0$ as the range of L/D for existence of bistable solutions, being shifted from the numerical prediction of Mizushima and Suehiro [4]. Tasaka et al. [5] attributed the discrepancy to the finite length of the cylinders in their experiments. Carmo and Meneghini [6] investigated the three-dimensional structures behind two circular cylinders when they are in tandem for the range of $160 < Re < 320$. Sharman et al. [7] performed numerical predictions of flows over two tandem circular cylinders at $Re = 100$,

* Corresponding author. Tel.: +82 32 860 7322; fax: +82 32 868 1716.
E-mail address: ksyang@inha.ac.kr (K.-S. Yang).

0045-7930/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compfluid.2008.07.005

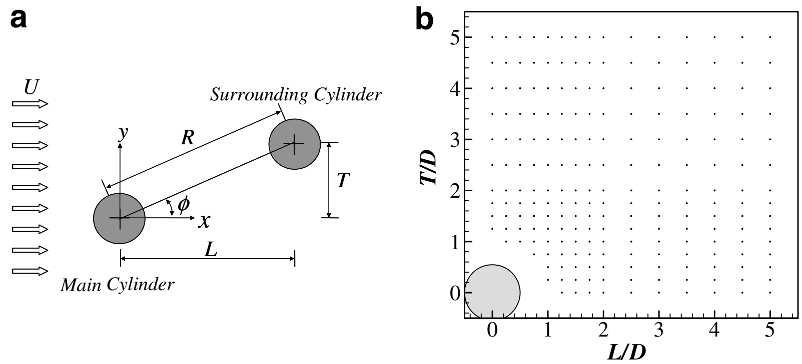

Fig. 1. Physical configuration: (a) staggered position of two circular cylinders, (b) locations of the center of the surrounding circular cylinder, indicated by dots ($Re = 100$).

revealing a sudden change in key flow parameters between $L/D = 3.75$ and 4.0.

Sumner et al. [8,9] experimentally studied flow-induced forces and flow patterns for high-Re ($850 \leqslant Re \leqslant 1900$) flows past two circular cylinders in staggered arrangements; they identified nine distinctive flow patterns. Akbari and Price [10] identified five distinct flow patterns at $Re = 800$. Gu and Sun [11] and Sun et al. [12] also reported flow patterns in turbulent flow regime.

In spite of the numerous studies carried out so far, the case where the two cylinders are placed in a staggered position has not been systematically studied, especially in the laminar flow regime. In this investigation, flow patterns past two nearby circular cylinders of equal diameter immersed in the cross-flow at $Re \leqslant 160$, based on the freestream velocity (U) and D, were numerically studied as a basic model for laminar wake interaction. An immersed boundary method [13] was employed for effective treatment of the cylinders on a Cartesian grid system. We consider all possible arrangements of the two circular cylinders in terms of the distance between them and the inclination angle of the line connecting their centers with respect to the main flow direction. The current authors recently reported their numerical results on the flow-induced forces in the same flow setting [14]. As a follow-up study, the present paper classifies the flow patterns observed, and describes distinctive characteristics of each flow pattern. Collecting all the results obtained, we propose a flow-pattern diagram ("map") for prediction of the flow pattern associated with a particular arrangement of the two cylinders as well as a Strouhal-number diagram for vortex shedding of each cylinder to provide an overall picture on the wake interaction. The perfect symmetry implied in the geometrical configuration allows one to use these diagrams to identify flow pattern and vortex-shedding frequencies in the presence of two circular cylinders of equal diameter arbitrarily positioned in physical space with respect to the main flow direction.

2. Formulation and numerical methodology

The current investigation requires a parametric study where numerous numerical simulations must be performed with various values of L and T. Here, T represents the vertical distance between the two cylinder centers (Fig. 1(a)). This kind of parametric study demands considerable amount of computing resources; the computing efforts can be significantly reduced by employing an immersed boundary method which facilitates implementing the solid surfaces of the arbitrarily-positioned cylinders on a Cartesian grid system.

The governing equations for two-dimensional incompressible flow, modified for the immersed boundary method [13], are as follows:

$$\frac{\partial u_j}{\partial x_j} - q = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re}\frac{\partial^2 u_i}{\partial x_j \partial x_j} + f_i \quad i = 1,2 \tag{2}$$

where u_i (or u, v), p, q and f_i represent velocity component in x_i (or x, y) direction, pressure, mass source/sink, and momentum forcing, respectively. All the physical variables except p are non-dimensionalized by U and D; pressure is non-dimensionalized by far-field pressure (P_∞) and the dynamic pressure. The governing equations were discretized using a finite-volume method in a nonuniform staggered Cartesian grid system. Spatial discretization is second-order accurate. A hybrid scheme is used for time advancement; nonlinear terms are explicitly advanced by a third-order Runge–Kutta scheme, and the other terms are implicitly advanced by the Crank–Nicolson method. A fractional step method [15] was employed to decouple the continuity and momentum equations. The Poisson equation resulted from the second stage of the fractional step method was solved by a multigrid method. For detailed description of the numerical method used in the current investigation, see Yang and Ferziger [16].

3. Boundary conditions and numerical parameters

The main cylinder is fixed at the origin of the coordinate system, and the downstream cylinder (hereafter, called "surrounding cylinder", SC) is placed at various locations relative to MC, which are represented by dots in Fig. 1(b). The total number of cases computed is 208 for $Re = 100$, and 58 for $Re = 40$, 50, and 160. In the case where the center distance (R, Fig. 1(a)) is minimum, $L/D = 0.25$, $T/D = 1.0$, and the inclination angle (ϕ, Fig. 1(a)) is 75.96°. The entire computational domain was defined as $-35D \leqslant x \leqslant 35D$, and $-50D \leqslant y \leqslant 50D$. For each cylinder, 32×32 uniform grid cells in x- and y-directions, respectively, were allocated, and uniform grid cells of the same cell size as in the cylinder region were employed between the cylinders. In the other region of the domain, nonuniform grid cells were used. The numerical resolution was determined by a grid-refinement study to ensure grid-independency. In the case of $L/D = 1.5$, $T/D = 0.5$ (Fig. 2), for instance, the total number of grid cells was 368×208, while 480×352 grid cells were allocated in the most demanding case ($L/D = 5.0$, $T/D = 5.0$).

No-slip condition was imposed on the cylinder surfaces; a Dirichlet boundary condition ($u = U$, $v = 0$) was used on the inlet boundary of the computational domain, while a convective boundary condition [17] was employed at the outlet. A slip boundary condition ($\partial u/\partial y = 0$, $v = 0$) was imposed on the other boundaries.