Computers & Fluids 55 (2012) 118–131

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

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

Numerical investigation of flow around an inline square cylinder array with different spacing ratios

Yan Bao [a], Qier Wu [a], Dai Zhou [a,b,*]

[a] School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiaotong University, No. 800, Dongchuan Road, Shanghai 200240, China

[b] State Key Laboratory of Ocean Engineering, Shanghai Jiaotong University, No. 800, Dongchuan Road, Shanghai 200240, China

ARTICLE INFO

Article history:
Received 30 April 2011
Received in revised form 2 July 2011
Accepted 19 November 2011
Available online 26 November 2011

Keywords:
Inline square cylinder array
Flow pattern
Wake interference
Aerodynamic characteristics
Spacing ratio
Drag and lift coefficients
Strouhal number

ABSTRACT

Flow around an inline cylinder array consisting of six square cylinders at a Reynolds number of 100 is investigated numerically by using a second-order characteristic-based split finite element algorithm in this paper. The numerical method and the code for the solution of incompressible Navier–Stokes equations are validated for the flow past a single and two tandem square cylinders, and the numerical results show a good agreement with the available literatures. The study then focuses on the effect of spacing ratio (ratio of center-to-center distance s to cylinder width d, widely ranging in s/d = 1.5–15.0) on flow characteristics by identifying flow patterns and extracting pressure distributions, force statistics as well as wake oscillation frequencies. Numerical results showed six different flow patterns, which appeared successively with the increase of gap spacing, namely, steady wake, non-fully developed vortex street in single row and double-row, fully developed vortex street in double-row, fully developed vortex street in partially recovered single-row and fully developed multiple vortex streets. A shielding effect of the first cylinder and reducing Bernoulli effect on the rear cylinder rows work in the pressure distribution even at very large gap spacing. In the vortex shedding regime, beyond the critical spacing of wake mode transition, force statistics show a periodic variation characteristic for the last four cylinders; moreover, multiple frequency components involve in the vortex shedding oscillation behind these cylinders and the dominant frequency jumps down with the increase of the spacing. Finally, the flow fields around the critical spacing range are comprehensively analyzed to reveal the crucial mechanism behind the observed aerodynamic characteristics.

1. Introduction

The unsteady flow around an array of cylinders is of practical engineering importance, such as flow around buildings, wind turbine farms and chimney stacks. The flow over multiple cylinders has much richer fluid dynamic features [1–5]. The aerodynamic forces and vortex shedding frequencies as well as wake patterns are very different from those of an isolated cylinder, even at the same Reynolds number [6,7]. Sumner et al. [8] investigated experimentally two equal circular cylinders in staggered arrangement and identified nine flow patterns depending on the incidence and distance between the cylinders. Bradshaw [9] carried out some experiments on a row of nine circular cylinders in order to find out the stability limit for the merging of vortices at Re = 1500. Cheng and Moretti [10] investigated experimentally the wake of a row of nine tubes at Re = 200 with a gap ratio (the ratio of the separation between the cylinders and the width) of 3. Le Gal et al. [11] presented flow-visualization and velocity measurements of the wake behind a row of cylinders placed side-by-side at Re = 80 with the spacing of 0.5d and 2d (d is the cylinder diameter). Igarashi and Suzuki [12] performed experiments on flow around three cylinders arranged in-line while Lam and Cheung [13] investigated the flow characteristics and interference effect of three cylinders in different equilateral arrangements. Lam et al. [14] measured the force coefficients and Strouhal numbers on four cylinders in a square configuration at subcritical Reynolds numbers in a wind tunnel. Ziada and Oengören [15] carried out experiments to study the vortex shedding characteristics in an inline tube bundle with different spacing. Recently, Liang et al. [3] investigated numerically the vortex shedding characteristics of laminar flow past an inline tube array. The effect of spacing ratio was mainly studied, but with a very limited spacing range. Their research showed that with the increasing of the spacing ratio, the flow becomes more asymmetric and vortex shedding is induced from the last cylinder then propagates toward upstream.

Flow around square cylinders can be expected to has their own characteristics and be different from that of circular cylinders, since the square cylinders have a tendency to fix the separation points at one of the section edges. Investigations of flow past two square cylinders have been reported in the literatures in the past [16–20].

* Corresponding author at: School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiaotong University, No. 800, Dongchuan Road, Shanghai 200240, China.

E-mail address: zhoudai@sjtu.edu.cn (D. Zhou).

0045-7930/$ - see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compfluid.2011.11.011

Among others, Mizushima and Takemoto [21] investigated numerically the stability of flow over a row of square bars with the assumption that the flow was two-dimensional and incompressible. They found that vortex shedding is nearly independent when the spacing ratio is large while the confluence of several jets appears for small spacing ratio. For some combination of Reynolds number and spacing ratio, both flopping and bi-stable flip-flop behaviors were observed in the wake. Mizushima and Akinaga [22] investigated both numerically and experimentally the interaction of wakes of a row of square bars perpendicular to an incoming uniform flow. They observed in-phase vortex shedding for gap ratio 1.0 and anti-phase shedding for 3.0. Recently, Kumar et al. [1] investigated numerically the flow around a row of nine square cylinders placed normal to the oncoming flow for gap ratio from 0.3 to 12.0 at $Re = 80$. The flow regimes such as synchronized, quasi-periodic and chaotic have been identified for gap ratio smaller than 6.0, which are resulted from the interaction between primary and secondary vortex shedding frequencies. Chatterjee et al. [2] performed a numerical study for the flow around five square cylinders placed side-by-side and normal to the oncoming flow at $Re = 150$. They identified the flow patterns as flip-flopping pattern, in-phase and anti-phase synchronized pattern and non-synchronized pattern.

The literature survey shows that not much attention has been paid to an inline cylinder array consisting of multiple (more than two) square cylinders, in particular, the flow characteristic has not been studied yet for a wide range of gap spacing. In Liang et al. [3], the spacing ratio is limited in the range from 2.1 to 4.0. This limitation is however not there in this work. In our investigation, the effect of spacing ratio in the range $1.5 \leqslant s/d \leqslant 15.0$ is documented in terms of aerodynamic characteristics and the ensuing flow patterns, and further examines how the flow mechanism actually works for those phenomena. The motivation of current study is of a fundamental nature, yet it also relevant to some practical engineering applications such as large building blocks, heat exchangers tube bundles, and electronic devices. For example, in the design of high-rise building structures, the spacing distance of adjacent structures is a very important parameter due to the fact that shedding frequency and the aerodynamic forces may differ significantly for different spacing regimes. To get reliable knowledge of design parameters, such as drag and lift coefficients, vortex shedding natural frequency, and wake size, understanding of basic fluid mechanics that occur in multiple bluff bodies is fundamental prerequisite in the design. In the present study, for the representation of physical reality of exceedingly complex multi-cylinder configurations, more than two cylinders are needed; therefore, we follow Liang et al. [3] to use six cylinders in tandem arrangement.

This work is organized as follow: Section 2 presents the governing equations for incompressible viscous flow and provides details on the employed numerical approach for the solutions. For the validation of the computer code and to provide a reference for further study, the results of laminar flow around a single cylinder and two tandem square cylinders with different spacing ratios are presented in Section 3. In Section 4, results for an in-line square cylinder array consisting of six cylinders with different spacing ratios are presented and discussed. Main findings from the present work are summarized in Section 5.

2. Governing equations and numerical details

The non-dimensional governing equations for incompressible viscous fluid flow moving in a domain Ω and in a time interval $[0, T]$ are written in tensorial Cartesian form as:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 u_i}{\partial x_j \partial x_j} \quad (1)$$

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (2)$$

A developed second-order CBS finite element algorithm [23] is employed to solve the above equations. In this algorithm, the form of characteristic-based semi-discretized scheme [24] is obtained by treating the time discretization along the characteristics explicitly, while the spatial discretization for the CBS algorithm is performed with standard Galerkin procedure [25] after the temporal discretization. In the framework of incremental projection method, the second-order CBS scheme requires an additional effort to be made to handle the pressure instability. In Bao et al. [23], two versions of pressure stabilization techniques were integrated into the CBS scheme. One uses T4/C3 MINI element to approximate both the velocity field and pressure field, the other is pressure gradient projection based stabilization method allowing to use equal order interpolation functions for both velocity and pressure. The former is employed in this paper, where the T4/C3 MINI element is obtained by adding a cubic bubble to a piecewise linear approximation of velocity while pressure remains piecewise linear in each triangular element. As compared with the traditional linear triangular element, the T4/C3 is more capable to approach the realistic physics of flow by taking into consideration of the central physical behavior of element. A Bi-conjugate gradient (BiCG) method was used for the solution of pressure Poisson equation, while velocity fields were explicitly obtained from previous time step.

An assumption of laminar flow is valid at this considered low Reynolds number; therefore, the present numerical simulation is performed in two dimensional space, starting from zero velocity and pressure fields. The simulation runs until the flow develops fully with adequate length of data for the statistical analysis. The parallel programming of the simulation code based on the Message-Passing Interface (MPI) standard is unavailable at this time, but it is expected that a substantial improvement of computational efficiency will be realized by using of parallel BiCG iterative method based on the element by element (EBE) technique for the solution of pressure Poisson equation. The simulations were carried out on a PC with Intel Quad-core processor i7-630 (2.80 GHz) and 6 GB of RAM. The simulation takes around 71 h for the case of $s/d = 5.5$ on the grid system containing 133,498 cells, where the total non-dimensionalized time length $T = 500$ with the time step size of 0.005.

Some global flow parameters involved in the discussions, including drag coefficient, C_D, lift coefficient, C_L, Strouhal number, St, and pressure coefficient, C_p, are defined as following,

$$C_D = \frac{2F_D}{\rho U_\infty^2 d}, \quad C_L = \frac{2F_L}{\rho U_\infty^2 d}, \quad St = \frac{f_s d}{U_\infty}, \quad C_p = \frac{2(p - p_\infty)}{\rho U_\infty^2} \quad (3)$$

where F_D and F_L are the force components in the stream-wise and transverse directions, respectively; f_s is vortex shedding frequency determined from the power spectrum analysis of the fluctuating lift force; ρ is the fluid density, U_∞ and p_∞ are characteristic velocity and pressure of the problem, respectively; and d is the characteristic length scale and taken here as the cylinder width. Here, d and U_∞ takes the unit value respectively, so the St in Eq. (3) is equal to the vortex shedding frequency.

3. Validation study

3.1. Flow around a single square cylinder

Here, the unsteady flow over a single square cylinder at Reynolds number of 100 is simulated to serve as a reference for further investigation of multiple cylinders. A computational domain of $\Omega = [-20d, 35d] \times [-25d, 25d]$ is used for simulation and the inlet boundary is located $20d$ upstream from the center of the cylinder