J. Wind Eng. Ind. Aerodyn. 146 (2015) 172–184

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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

Large eddy simulation of flow around an inclined finite square cylinder

Gang Hu [a], K.T. Tse [a,*], K.C.S. Kwok [a,b], Yu Zhang [b]

[a] Department of Civil and Environmental Engineering, The Hong Kong University of Science and Technology, Clear Water Bay, Kowloon, Hong Kong, China
[b] Institute for Infrastructure Engineering, University of Western Sydney, Penrith, NSW 2751, Australia

ARTICLE INFO

Article history:
Received 10 February 2015
Received in revised form
31 August 2015
Accepted 31 August 2015
Available online 29 September 2015

Keywords:
Large eddy simulation
Inclined cylinder
Aerodynamic characteristic
Axial flow
Wake

ABSTRACT

The flow field around a rigid finite square cylinder inclined away from the vertical direction was numerically investigated via large eddy simulations. The cylinder under investigation had a height-to-width ratio of 18 and the inclination angle, defined as the angle between the cylinder orientation and the vertical direction, varied from forward inclination to backward inclination. Pressure measurements and flow visualizations were taken in a wind tunnel to validate the numerical simulations. The forward inclination is found to enhance the downwash, which is initially observed behind the free end of the vertical cylinder, and amplify it to a downward axial flow. Conversely, the backward inclination promotes upwash, which originates behind the base of the vertical cylinder, to be an upward axial flow. On the other hand, the vertical cylinder produces two pairs of counter-rotating streamwise vortices (quadrupole wake) in its wake, but only one pair of vortices (dipole wake) is observed to form behind both the forward and backward inclined cylinders. Moreover, only a free-end vortex pair is exhibited behind the forward inclined cylinder whereas a base vortex pair exists behind the backward inclined cylinder. They are considered to generate the downward and upward axial flow respectively. It is anticipated that the presence of the axial flow significantly influence both the pedestrian-level wind conditions and the aerodynamic characteristics of the cylinder.

1. Introduction

A considerable number of studies have been conducted focusing on the flow field around or the aerodynamic characteristics of three-dimensional finite cylinders. The investigated cylinders are of different cross-section shapes, including circular, square, and rectangular sections. These cylinders are the prototype of numerous engineering structures such as cooling towers, chimney stacks, tall buildings and bridge pylons.

The flow field around a vertical finite circular cylinder is much more complex than that around a two-dimensional cylinder. Specifically, the flow field is strongly three-dimensional because of the free end and the cylinder–wall junction. A downward-directed velocity field (downwash) and an upward-directed velocity field (upwash) have been widely observed behind the free end and the base of rigid finite cylinders respectively (Donnert et al., 2007; Etzold and Fiedler, 1976; Kawamura et al., 1984; Park and Lee, 2000; Rostamy et al., 2012; Sumner and Heseltine, 2008; Sumner et al., 2004). Meanwhile, the structure of the vortices shedding from the finite cylinder is more diverse than the well-known Karman vortex shedding from a two-dimensional cylinder. The vortices shedding from a rigid finite cylinder generally consist of the Karman vortex shedding from both sides of the cylinder, two pairs of streamwise counter-rotating vortices exhibited behind the free end and the base of the cylinder, and a horseshoe vortex forming upstream at the cylinder–wall junction (Sumner, 2013).

The streamwise counter-rotating vortex pairs behind the free end and the base of the rigid cylinder are conventionally termed the "tip" vortex pair and the "base" vortex pair respectively. The tip pair is thought to be responsible for the downwash flow while the base pair is believed to generate the upwash flow (Adaramola et al., 2006; Johnston and Wilson, 1997; Sumner, 2013; Sumner et al., 2004). Furthermore, both the tip and base pair have been found to significantly affect the aerodynamic characteristics of the cylinder. For example, the tip and base vortices interact with the Karman vortex shedding and accordingly alter the vortex shedding frequency (Okamoto and Sunabashiri, 1992; Park and Lee, 2000; Sumner et al., 2004). The base vortex also causes a jump in drag coefficient (Sumner et al., 2004), and noticeably influences the momentum transportation and Reynolds stresses in the wake (Wang et al., 2006). These changes of aerodynamic properties have practical implications on predicting the flow-induced vibrations of high-rise structures, pollutant dispersions from chimney stacks, and the pedestrian-level wind field. It is worth mentioning that the presence of the base vortex pair highly depends on the aspect

* Corresponding author. Tel.: +852 23588763; fax: +852 23581534.
E-mail addresses: ghuaa@ust.hk (G. Hu), timkttse@ust.hk (K.T. Tse), k.kwok@uws.edu.au (K.C.S. Kwok), yu.zhang@uws.edu.au (Y. Zhang).

http://dx.doi.org/10.1016/j.jweia.2015.08.008
0167-6105/

ratio (AR) of the cylinder (Sumner et al., 2004). Only when the AR is greater than a specific critical value, the base vortex pair can appear. Based on the findings made in previous studies (Okamoto and Sunabashiri, 1992; Sakamoto and Arie, 1983; Sumner et al., 2004), the value of the critical AR varies with experimental conditions.

In addition to circular cylinders, a number of studies have also focused on the flow field around finite square cylinders. Topologically, the flow field around a circular or square section cylinder is expected to share similar features. Different mechanisms for flow separation, however, would induce appreciable differences in the flow fields around cylinders with different cross-sections. For a circular section, an adverse pressure gradient in the longitudinal direction separates the flow. Thus, the separation points are not fixed but are depend on the Reynolds number. However, for a square section, the flow separates at the upstream corners regardless of the Reynolds number. In view of this, flow models have been proposed to explain the flow field around a finite square cylinder. Wang and his coworkers (Wang and Zhou, 2009; Wang et al., 2006) experimentally investigate tip vortices, span-wise vortices and base vortices in the near wake behind a rigid square cylinder. They proposed an innovative model. The model has an arch-type structure that consists of two spanwise vortical legs on each side of the cylinder and a connection of these two legs near the free end. The arch-type structure projects in the streamwise direction to create the tip and base vortices, which are the causes of the downwash and upwash flow in the wake. Recently, a very different model was constructed by Hosseini et al. (2013). Their experimental results exhibited a dipole wake (one streamwise vortex pair behind the free end) behind a cylinder with an aspect ratio of 8 immersed in a thin boundary layer, and a quadrupole wake (two streamwise vortex pairs behind the free end and the base) in a thicker boundary layer. They argued that the connections of the vortices behind the free end are much more complicated than the arch-type structures proposed by Wang and Zhou (2009).

The above studies all concerned the flow field of three-dimensional cylinder with finite length, whose vertical axes are normal to the oncoming flow. Intuitively, inclinations would alter the flow field around the cylinder and hence change the aerodynamic properties of the cylinder. A lot of studies have been conducted focusing on a two-dimensional circular cylinder inclined from the vertical direction (Chen et al., 2013; Vakil and Green, 2009; Yeo and Jones, 2011a, 2011b; Zhao et al., 2009). Most of such studies aimed to investigate the aerodynamic characteristics of cables. Perhaps Shirakashi et al. (1986) is the first to identify an intense secondary flow in the near wake of an inclined circular cylinder in a wind tunnel, and the secondary flow was considered to depress the vortex shedding frequency. Matsumoto and his coworkers (Matsumoto et al., 1990, 2001, 2010) also observed a secondary axial flow propagating along the leeward face of an inclined circular cylinder. They found that the axial flow played a similar role to that of a splitter plate submerged in the wake of the cylinder, and caused an unstable aerodynamic force that excites the cylinder.

Recent development in computational fluid dynamic (CFD) technology facilitates the investigation of flow fields around inclined cylinders. For example, swirling flows around inclined circular cylinders were investigated by using numerical simulation (Yeo and Jones, 2008). Hoftyzer and Dragomirescu (2010) revealed an axial flow attached on the leeward side of an inclined cable via LES. Zhao et al. (2009) employed direct numerical simulation (DNS) to investigate wake structures, vortex shedding frequencies and hydrodynamic forces of an inclined infinite circular cylinder.

Compared with the number of studies focusing on the inclined cylinders with infinite length, which is the prototype of cables, very little attention has been put on inclined finite cylinders especially inclined finite square cylinders, which is the prototype of bridge pylons and architectural features attached to modern buildings. In view of the aforementioned complex vortex structures resulting from the presence of the free end and the cylinder–wall junction of a finite cylinder, it is speculated that the flow field around an inclined finite square cylinder is significantly different from that around a vertical cylinder or an inclined infinite cylinder. To our best knowledge, the flow field and aerodynamic characteristics of an inclined finite square cylinder have not yet been systematically investigated.

The present study investigates the flow field around a rigid inclined finite square cylinder in order to understand its aerodynamic behavior and flow-induced vibration, which have been demonstrated to be different from a vertical finite square cylinder (Hu et al., 2015; Tse et al., 2014). A series of LES and wind-tunnel experiments were conducted. The inclination angle, which was defined as the angle between the cylinder axis and the vertical direction, took the value in the range of $\alpha = 30°$, $15°$, $0°$, $-15°$ and $-30°$. While the positive inclination angle indicated the forward inclination, the negative value represented the backward inclination. The vertical case ($\alpha = 0°$) was included as a benchmark. In wind-tunnel experiments, pressure measurements on the cylinder surfaces were taken and visualizations of the flow field were made. Through comparing to the wind-tunnel test results, the flow fields output from LESs were validated and investigated to reveal the flow field details and aerodynamic properties of the cylinders.

2. Computational method

2.1. Governing equations and numerical methods

LES has been verified to accurately predict both mean and instantaneous flow fields around bluff bodies (Blocken, 2014; Gousseau et al., 2013; Krajnovic, 2011; Tamura, 2008; Tamura et al., 2008). Therefore, the three-dimensional LES turbulence model, which has been incorporated in the ANSYS Fluent software package, was used in the present study. LES simulates large scale eddies by solving the filtered Navier–Stokes equations, in which the effect of small eddies are calculated using a subgrid scale (SGS) model. The finite-volume discretization implicitly used in Fluent provides a filtering operation:

$$\overline{\phi}(x) = \frac{1}{V}\int_V \phi(x')dx', \quad x' \in V, \tag{1}$$

where ϕ is the original (unfiltered) function, x' is the spatial coordinate, x is the spatial coordinate after filtering, and V is the volume of a computational cell. The filter function is

$$G(x, x') = \begin{cases} 1/V, & x' \in V \\ 0, & x' \notin V \end{cases}. \tag{2}$$

The flow is assumed to be Newtonian and incompressible. The filtered Navier–Stokes equations are given as

$$\begin{cases} \frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \overline{u}_i \overline{u}_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + v\frac{\partial^2 \overline{u}_i}{\partial x_j x_j} - \frac{\partial \tau_{ij}}{\partial x_j}, \\ \frac{\partial \overline{u}_i}{\partial x_i} = 0, \end{cases} \tag{3}$$

where $i, j =$ 1, 2 and 3; u_1, u_2 and u_3 are the velocity components along x_1 (i.e. longitudinal direction), x_2 (lateral direction) and x_3 (vertical direction) of the Cartesian coordinate system, respectively. The overbar denotes the filtering operator. ρ and v are air density and kinematic viscosity respectively. $\overline{p}$ is the pressure and