Journal of Fluids and Structures 59 (2015) 22–36

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Numerical investigation of steady suction control of flow around a circular cylinder

Wen-Li Chen [a,b,c,*], Yong Cao [b], Hui Li [a,b], Hui Hu [c]

[a] Key Lab of Structures Dynamic Behavior and Control (Harbin Institute of Technology), Ministry of Education, Harbin, Heilongjiang 150090, China
[b] School of Civil Engineering, Harbin Institute of Technology, Harbin, Heilongjiang 150090, China
[c] Department of Aerospace Engineering, Iowa State University, Ames, Iowa 50011, USA

ARTICLE INFO

Article history:
Received 24 December 2014
Accepted 1 September 2015
Available online 29 September 2015

Keywords:
Circular cylinder
Suction flow control
Aerodynamic force
Vortex shedding
Three-dimensional effect
Numerical investigation

ABSTRACT

This paper numerically investigates the effectiveness of the control of steady suction on a stationary circular cylinder with several isolated suction holes on the surface at a sub-critical Reynolds number. The control effectiveness as a function of the azimuthal position, spanwise spacing and suction flow rate of the suction holes on the control of the aerodynamic forces on the cylinder and the suppression of alternate vortex shedding are taken into account. The study of the azimuthal location of the suction holes indicates that azimuthal angles of $\theta=90^\circ$ and 270°, which are close to the separation point, provide the most substantial decreases in the aerodynamic forces. When restricted to the most effective azimuthal angle, a remarkable control effectiveness can be achieved when the axial spacing between two neighboring suction holes is less than a minimum value even under a small suction momentum coefficient. However, if the axial spacing exceeds the minimum spacing, the control effectiveness will not be saturated even under a very large suction momentum coefficient. Thus, the cause of the effective aerodynamic force control is suggested to be a result of obvious three-dimensional phenomenon in the near wake, which is characterized by the generation of a convergent flow between two neighboring suction hole sections and a stronger, larger three-dimensional vortex pair adjacent to the convergent flow. It has been suggested that this strongly three-dimensional flow pattern is induced by the strong interaction between two neighboring but counter-rotating three-dimensional vortices separately produced by two neighboring suction holes. Moreover, the effects of such three-dimensional flow patterns are investigated in detail based on variations in the flow field and sectional aerodynamic forces in different cross sections. Finally, the upper limit of the axial spacing between two neighboring suction holes to form such a three-dimensional flow pattern is suggested to be between 0.75 D and 1.5 D when the suction flow rate exceeds a certain value.

1. Introduction

The flow around a circular cylinder has always been important for researchers and engineers because of its widespread practical applications (e.g., offshore risers, bridge piers, towers and cables). In the civil engineering field, inclined cables are

* Corresponding author.

http://dx.doi.org/10.1016/j.jfluidstructs.2015.09.002
0889-9746/

key components of long-span bridges that are prone to wind-induced vibration or aerodynamic instability due to the low stiffness and damping rate. Thus, flow control around a circular cylinder is of great significance.

Flow control methods can be divided into two general types: passive and active flow control. The introduction of a splitter plate (aligned with the oncoming flow) downstream of a bluff body is one widely used passive control method (Roshko, 1993; Bearman and Branković, 2004). Owen et al. (2001) attached hemispherical humps to circular cylinders to control the fluctuating amplitudes of the vortex-induced vibrations of cable models.

Active flow control approaches include forced rotary oscillation, moving-wall, blowing and suction flow control. Tokumaru and Dimotakis (1991) examined the control efficacy of the sinusoidal oscillatory cylinder rotation for actively controlling the cylinder wake in their experimental study. A significant drag reduction of 80% was achieved at the Reynolds number (Re) of 15 000 for the optimal frequency and amplitude parameters of the rotary oscillation. Shiels and Leonard (2001) conducted a numerical simulation to explore the same flow control method, but with a larger range of Reynolds numbers (i.e., from 150 to 15 000). They confirmed the experimental observations at Re=15 000 by Tokumara and Dimotakis (1991) and further suggested that higher control effectiveness of drag could be obtained at higher Reynolds numbers. Wu et al. (2007) employed a flexible surface on the downstream half of the circular cylinder to generate a traveling wave when Re=500. This moving-wall control method successfully impeded the flow separation near the wall even under a strong adverse pressure gradient and eliminated the vortex shedding in the wake. Amitay et al. (1998) and Glezer and Amitay (2002) investigated the interactions of synthetic jets with cross-flows. The overall flow configurations around bluff bodies were modified dramatically by the formation of synthetic flow recirculation regions near the locations of the synthetic jets. While synthetic jets were usually placed near the separation points for flow control, Feng et al. (2010) and Feng and Wang (2012) applied synthetic jets in the rear stagnation points of circular cylinders and temporally alternated suction and blowing of synthetic jests. By increasing the suction duty cycle factor (defined as the ratio of the time duration of the suction cycle to the blowing cycle), a flow separation delay and drag reduction by up to 29% were realized.

Suction flow control methods have also been applied to various flow configurations. The surface suction slot seems to be used most widely among them, e.g., to a turbulent junction flow by Seal and Smith (1999), a wall-mounted hump by Greenblatt et al. (2006), an airfoil at transonic speeds by Qin et al. (1998) and Chng et al. (2009). Fransson et al. (2004) studied the flow field around a smooth, porous cylinder with flow suction pores uniformly distributed over its entire surface. Distinctly different from these suction methods, Chen et al. (2013a),(2013b) and Chen et al. (2014) introduced a limited number of small, isolated suction holes to manipulate vortex shedding in the wake behind a circular cylinder. Because of its steady suction, this method is called the steady suction flow control method. The method has several attractive advantages in comparison with other flow control techniques. For example, the control effectiveness of the model is considerably high (e.g., reduction of lift fluctuation by 80%). Experimental results showed that the azimuthal locations of the suction holes with respect to the oncoming airflow, the spanwise spacing between the suction holes and the suction flow rate through the suction holes have a significant effect on the wake flow characteristics, surface pressure distribution and the resultant dynamic wind loads. However, a deeper and systematic understanding of the underlying mechanism of the suction flow control is still needed, which is very useful for various practical engineering applications and further developments.

Following the experimental work of Chen et al. (2013b), (2014), the authors take advantage of more detailed information obtained by numerical simulation under the same physical conditions to further investigate the steady suction control method. Another spanwise arrangement of suction holes is also simulated. We attempt to (1) verify the applicability of the CFD technique to the suction flow control method, (2) discover the causes of the effective suction control of aerodynamic forces and vortex shedding of a circular cylinder and (3) provide further discussion on how to obtain effective control by employing this method.

2. Numerical method and validation

2.1. Governing equations

An incompressible and viscous fluid with a uniform velocity U_∞ at infinity is assumed to flow past a stationary circular cylinder. For transient flows, the incompressible ensemble-averaged governing equations (or called unsteady Reynolds averaged Navier–Stokes equations) are

$$\frac{\partial\langle u_i\rangle}{\partial x_i}=0, \tag{1}$$

$$\frac{\partial\langle u_i\rangle}{\partial t}+\langle u_j\rangle\frac{\partial\langle u_i\rangle}{\partial x_j}=-\frac{1}{\rho}\frac{\partial\langle p\rangle}{\partial x_i}+\nu\frac{\partial^2\langle u_i\rangle}{\partial x_k^2}-\frac{\partial\langle u_i^{'}u_j^{'}\rangle}{\partial x_j}, \tag{2}$$

The operation $\langle\ \rangle$ is ensemble averaging. The Reynolds stresses $\left\langle u_i^{'}u_j^{'}\right\rangle$ in Eq. (2) are modeled via the turbulent viscosity hypothesis to relate the Reynolds stress with the mean velocity gradients and eddy viscosity ν_T (see Eq. (3).

$$\langle u_i^{'}u_j^{'}\rangle-\frac{2}{3}k\delta_{ij}=-\nu_T\left(\frac{\partial\langle u_i\rangle}{\partial x_j}+\frac{\partial\langle u_j\rangle}{\partial x_i}\right), \tag{3}$$