Computers & Fluids 109 (2015) 155–167

Contents lists available at ScienceDirect

Computers & Fluids

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

# Control of fluid flow and heat transfer around a square cylinder by uniform suction and blowing at low Reynolds numbers

A. Sohankar [a,*], M. Khodadadi [b], E. Rangraz [b]

[a] Department of Mechanical Engineering, Isfahan University of Technology, Isfahan, Iran
[b] Department of Mechanical Engineering, Yazd University, Yazd, Iran

ARTICLE INFO

*Article history:*
Received 31 March 2014
Received in revised form 16 November 2014
Accepted 23 December 2014
Available online 30 December 2014

*Keywords:*
Active control
Suction/blowing
Heat transfer
Square cylinder
Vortex shedding
Force reduction

ABSTRACT

The effects of uniform suction and blowing through the surfaces of a square cylinder on the vortex shedding, wake flow and heat transfer are investigated, $Re = 70–150$, $Pr = 0.7$. All numerical simulations are performed with a finite-volume code based on a collocated grid arrangement. To find the optimum condition, where vortex shedding suppression occurs and the maximum reductions on the forces and their fluctuations provide, three simple cases are examined. In these cases, the influence of the uniform blowing and suction only through the front surface (case I), rear surface (case II) and top/bottom surface (case III) is studied at $Re = 150$. Based on the obtained advantages of these simple cases, different combinations of the suction and blowing on the cylinder sides are considered and an optimum case is introduced. In this case, suction is applied on the top and bottom surfaces and blowing is employed on the front and rear faces. The effect of the Reynolds number on the results for the optimum configuration is also investigated ($Re = 70, 100, 150$). This study shows that the lift and drag fluctuations for the optimum configuration decay and the maximum reduction on the drag force are 61%, 67% and 72% for $Re = 70, 100, 150$, respectively. An optimum case with respect to heat transfer is also introduced where suction is applied through all surfaces.



## 1. Introduction

Flow-induced vibrations and generation of the fluctuating forces are major problems in most of the flow systems in engineering and industrial applications. If this phenomena is not considered in design of an engineering structure, it can demolish the structure due to oscillations. The structure of the wake region and the shape of vortex formation are main factors for the value of the fluctuating forces exerting on the structures. Thus, the elimination or reduction of the fluctuating forces and vortex shedding from such bodies by the application of the various passive or active flow-control methods is necessary. Passive flow control methods require no external energy input, typically changing the geometrical configurations such as splitter plate, small control cylinder, trip wire or a grooved wall, change of surface roughness, adding fixed mechanical vortex disturbers, ribs or large eddy breakup devices on to the main body to influence vortex shedding [1–7]. Active flow control methods require an energy input. Solid wall motion, blowing or suction from a surface, the injection of micro bubbles or particles, acoustic excitation, periodic rotation or oscillation of a body, and electromagnetic forces can be given as examples to active control methods. These methods due to kinds of energy that is used divided to three kinds: (1) Actuators (electromagnetic, acoustic forcing, etc.), (2) motion of the solid wall (forced cylinder vibrations) and (3) generate secondary flow (suction, blowing, bleed) [8–11].

A limited number of literatures have been published reporting the effects of injection or suction through a bluff body on the aerodynamics parameters and heat transfer characteristics. Mathelin et al. [12] numerically studied the flow around a porous circular cylinder in a cross-flow when complete blowing was applied through the cylinder at Reynolds number range of $3900 < Re < 31{,}000$. They revealed that the pressure defect at the rear of the circular cylinder tended to “fill up” with blowing, leading to lower transverse static pressure gradients in the near wake. Fransson et al. [10] conducted experiments on the flow around a porous circular cylinder subject to continuous suction or blowing at the range of Reynolds numbers of $8.5–25 \times 10^3$. Their results show that the drag coefficient of the cylinder increases linearly with the blowing rate, whereas for suction there is a drastic decrease at a specific rate. Also, it was observed that the Strouhal number decreases with blowing and increases with suction.

* Corresponding author.
*E-mail address:* asohankar@cc.iut.ac.ir (A. Sohankar).

http://dx.doi.org/10.1016/j.compfluid.2014.12.020

Dong et al. [13] present an effective technique for suppressing the vortex-induced vibrations of a circular cylinder for $Re = 500$, 1000 numerically. One result of this study revealed that small amounts of combined windward suction and leeward blowing around the body modify the wake instability and lead to suppression of the fluctuating lift force.

Ling and Fang [14] numerically investigated the effects of surface injection's position and strength on the vortex structure in flow around a circular cylinder, as well as the drag and lift forces at $Re = 100$. The results of this study show that either the suction on the shoulder of the cylinder or the blowing on the rear of the cylinder causes the lift force to reduce greatly. They also revealed that applying suction through the shoulder of the cylinder, when its strength is properly chosen, can reduce the drag force significantly, too.

Lisa et al. [15] numerically analyzed the effects of normal surface suction and blowing through all surfaces of a square cylinder on the vortex shedding frequencies for $Re = 250$. They showed that an initial increase followed by a decreasing behavior in the Strouhal number with increasing the suction velocity occurs. Also a decrease in the Strouhal number with increasing the blowing velocity was observed.

Cuhadaroglu et al. [16] experimentally presented the pressure coefficient distributions around a horizontal square cylinder at Reynolds number $Re = 10{,}000$, 16,000 and 24,000, where the injection through front, top and rear surfaces of the cylinder were applied. They found that applying injection through front face increases the drag coefficient while blowing fluid through the rear one decreases it. They also claimed that injection through the bottom and top faces has no considerable effect on drag coefficient.

Çuhadaroğlu and Turan [17] reported that the applying the injection or suction through various surfaces of the square cylinder significantly influences on the drag coefficient and vortex shedding frequency as well as heat transfer for $Re = 21{,}400$.

Cuhadaroglu [18] numerically investigated the effects of uniform injection or suction through a porous square cylinder on the flow field and on some aerodynamic parameters at $Re = 21{,}400$. The results show that increasing suction velocity decreases drag coefficient for all suction configurations except that of suction through rear surface. The author also found that applying suction through top and bottom surfaces weakens the vortex shedding motion.

Turhal and Çuhadaroğlu [19] experimentally investigated the flow around perforated horizontal and diagonal square cylinder with surface injection through various surfaces for $Re = 10{,}000$, 16,000 and 24,000. They revealed that surface injection through the top-rear, rear and all surfaces of a diagonal square cylinder reduces the drag coefficient while the injection through all surfaces only reduces the drag of a horizontal square cylinder.

In summary, our review shows that the active flow control over a square cylinder via suction and blowing has not been broadly investigated especially at low Reynolds numbers and very limited results are available in the literatures. Therefore, the main objective of this work is to study the effects of suction and blowing on the flow and heat transfer characteristics of an unconfined square cylinder in cross-flow. At first, influence of uniform blowing and suction through the front, rear and top/bottom surfaces has been studied at a Reynolds number of $Re = 150$ (simple cases). Based on the obtained advantages of these simple cases, different combinations of the suction and blowing on the cylinder sides are considered and an optimum case is introduced, where the elimination or reduction of the fluctuating forces and vortex shedding from body is achieved.

This paper is organized as follows. A presentation on the problem under consideration, numerical method, boundary conditions and governing equations is given in Section 2. Section 3 is devoted to grid study and description of the results including an analysis of the global, time-averaged and instantaneous results. Some conclusions are drawn in the final section.

## 2. Problem under consideration and governing equations

The computational domain is presented schematically in Fig. 1. As seen, a 2D square cylinder of side length $d$ placed in uniform flow with velocity $U_{in}$. To minimize the effect of the outer boundaries on the flow pattern around the cylinder, the lengths of the computational domain for the upstream distance, the downstream distance and the width of computational domain are assigned constant values of 10$d$, 25$d$ and 20$d$, respectively. These values are chosen based on the previous work of the first author [20–23]. The ratio of the square cylinder side length to the width of the computational domain is called the blockage ratio. It was shown that the reduction of the blockage ratio from 5% to 2.5% has no significant effect (less than 1.5%) on Strouhal number, time-averaged drag, and rms lift [20,21]. In the present study, the blockage is set as 5%, thus the width of computational domain is equal to 20$d$ [20–23].

The unsteady governing equations in dimensionless form for two-dimensional incompressible flow and temperature field are given as follows ($U = U_1, V = U_2$):

$$\frac{\partial U_j}{\partial X_j} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial \tau} + \frac{\partial (U_i U_j)}{\partial X_j} = -\frac{\partial P}{\partial X_i} + \frac{1}{Re}\frac{\partial^2 U_i}{\partial X_j \partial X_j} \tag{2}$$

$$\frac{\partial \theta}{\partial \tau} + \frac{\partial (U_j \theta)}{\partial X_j} = \frac{1}{Re\,Pr}\frac{\partial^2 \theta}{\partial X_j \partial X_j} \tag{3}$$

All the equations are scaled with the following relations:

$$U = \frac{u}{U_{in}}, \quad V = \frac{v}{U_{in}}, \quad \theta = \frac{T - T_{in}}{T_W - T_{in}}, \quad \tau = \frac{tU_{in}}{d}, \quad X_i = \frac{x_i}{d}, \quad P = \frac{p}{\rho U_{in}^2}, \quad \Gamma = \frac{V_w}{U_{in}} \tag{4}$$

$U_{in}$ and $T_{in}$ are the velocity and temperature of the free stream at the inlet of the computational domain, respectively. $T_W$ is the temperature of the cylinder, $\rho$ is the density, $x_i$ (or $x$ and $y$) are the streamwise and cross-stream dimension coordinates, respectively. $P$ is pressure, $T$ is temperature and $t$ is the time. $V_w$ is the suction or blowing velocity through the cylinder walls. Furthermore, Reynolds and Prandtl numbers are defined as $Re = U_{in}\, d/\upsilon$ and $Pr = \upsilon/\alpha$, respectively, where $\upsilon$ is the kinematic viscosity and $\alpha$ is the thermal diffusion coefficient. The dimensionless frequency is known as Strouhal number ($St = fd/U_{in}$). $f$ is the frequency of the vortex shedding, which is calculated by taking the FFT (Fast Fourier Transform) of the time series of the lift signal.

It is useful to note that the lift and drag forces are made up of two components, namely, the form (pressure) and the friction component. The pressure component of the forces is evaluated by integrating the pressure acting on the body faces in the $x$ (pressure drag) and $y$ (pressure lift) directions. Similarly, the friction component is obtained by integrating the shear forces acting on the body faces in the $x$ (friction drag) and $y$ (friction lift) directions.

The local Nusselt number on the cylinder surface is defined in this study as $Nu = \lambda d/k$, where $\lambda$ is the local convective heat transfer coefficient and $k$ is the thermal conductivity. To find the surface averaged values of the Nusselt number and/or the overall value of the Nusselt number for the whole cylinder, the local values are integrated over each face or over the whole cylinder to estimate the rate of heat transfer from the cylinder. It should be noted that