European Journal of Mechanics B/Fluids 61 (2017) 187–199

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

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

Effects of fluid injection on dynamics of flow past a circular cylinder

Uddalok Sen [1], Achintya Mukhopadhyay *, Swarnendu Sen

Mechanical Engineering Department, Jadavpur University, Kolkata 700032, India

HIGHLIGHTS

- Numerical simulation of unsteady, 2D laminar fluid flow past a circular cylinder.
- Effect of injection of a second fluid from circumferential slots on the cylinder studied.
- Two injection configurations studied, resulting in distinctly different mixing patterns.
- POD performed to explain dynamics of such different mixing characteristics.

ARTICLE INFO

Article history:
Available online 9 November 2016

Keywords:
Flow past a cylinder
Proper orthogonal decomposition
Numerical simulation
Mass transfer
Bluff body combustor

ABSTRACT

The present paper deals with the numerical simulation of unsteady, two-dimensional laminar fluid flow past a circular cylinder. An unstructured grid finite volume method has been used, and the commercial CFD code ANSYS Fluent was employed to perform the simulations. Effects of injection of a second dissimilar fluid from two diametrically opposite peripheral slots on the cylinder are of prime interest in the study. Two distinct injection arrangements have been investigated – a co-counter flow arrangement, in which the injected fluid is aligned with the free stream flow, and a cross flow arrangement, in which the second fluid is injected perpendicular to the free stream flow direction. A parametric variation of the velocity of the injected stream in the downstream wake region of the cylinder have been studied qualitatively by observing the velocity, vorticity, and mass fraction contours. Effects of injection on the vortex shedding have also been studied by observing the variations of the Strouhal number. The two injection arrangements affected the vortex shedding in distinctly different manners. Effect of lateral walls on the flow dynamics were also investigated. The presence of lateral walls was found to delay the onset of aperiodic wake structures and to promote better mixing in the downstream wake of the cylinder. Proper orthogonal decomposition of the vorticity field was also performed in order to identify the dominant modes and their respective enstrophy contents. Significant differences in the modal structures were observed with increase in the value of the control parameter. At lower values, large scale POD modes dominate, while at higher values, the wake comprised of small scale structures, for both the flow arrangements. A gradual dominance of the injected stream on the free stream was also observed.

1. Introduction

The unsteady fluid flow past a circular cylinder is a widely studied phenomenon in fluid mechanics, primarily because of the different types of flow characteristics observed. The most popular feature of it is the asymmetrical von Kármán vortex shedding pattern observed at a certain range of Reynolds number. Such vortex shedding pattern is particularly of importance in bluff body stabilized combustors, where the recirculation region in the wake of the cylinder is utilized for flame anchoring. In such combustors, if the design of the fuel injection system is such that the fuel is injected from the bluff body itself [1–3], then the interaction between the vortices of the free stream and the injected stream comes into play for governing the dynamics of the entire system.

Flow past an unconfined cylinder has been the topic of active research around the world for a number of decades now. Fornberg [4] presented numerical results of the unsteady flow past a circular cylinder up to Reynolds number of 600 by solving the stream function–vorticity formulation of the Navier–Stokes equation. A comprehensive overview of the flow past smooth and rough circular cylinders for a large range of Reynolds number was presented by Zdravkovich [5], highlighting three transitions—in-the-near-wake,

* Corresponding author.
E-mail address: achintya.mukho@gmail.com (A. Mukhopadhyay).
[1] Present address: Department of Mechanical and Industrial Engineering, University of Illinois at Chicago, Chicago, IL 60607, USA.

http://dx.doi.org/10.1016/j.euromechflu.2016.11.006

along the free shear layers, and along the boundary layers. Several disturbance-sensitive flow regimes associated with each transition were also identified, with the variations of the mean and fluctuating aerodynamic coefficients also highlighted. A review of the dynamics of the shed vortices in a cylinder wake was presented by Williamson [6]. Results of the temporal variation of the lift coefficient for the unconfined flow past a circular cylinder for a large range of Reynolds number was reviewed by Norberg [7].

The suitability of suction and blowing on the cylinder surface for controlling the vortex shedding past a circular cylinder was shown by Li et al. [8]. Muralidharan et al. [9] presented an active control strategy to suppress vortex shedding behind a flexibly mounted circular cylinder by having a suction slot and two blowing slots placed on the periphery. Numerical simulations were performed for Reynolds number of 100 and it was observed that complete suppression of vortex induced vibrations occurred when the ratio of the actuation velocity to the free stream velocity is 3. Recently, Liu and Feng [10] demonstrated a control strategy for the suppression of the lift fluctuations on a circular cylinder by inducing the symmetric vortex shedding mode. This was achieved by two synthetic jets having a top hat sinusoidal velocity profile placed at the two mean separation points.

Since most bluff body stabilized combustors have confined flames, the effect of the lateral walls on the vortex shedding past the bluff body becomes important. The influence of the lateral boundaries on the vortex shedding phenomena of the incompressible 2D flow past a circular cylinder at Reynolds number of 100 was examined by Behr et al. [11] using two finite element formulations—space–time–velocity–pressure formulation and velocity–pressure–stress formulation. It was concluded that the lateral boundaries result in an increased Strouhal number of vortex shedding compared to that of the unconfined flow. Numerical simulations performed by Chakraborty et al. [12] showed that for a fixed blockage ratio, the length of the recirculation zone increases and the drag coefficient decreases with increasing Reynolds number. Singha and Sinhamahapatra [13] used an unstructured collocated grid based finite volume method based on the primitive variable formulation to study the flow past a circular cylinder placed between two parallel walls. It was observed that a decreasing channel height results in a shorter wake, and the presence of lateral boundaries also delays the transition from a steady attached wake to an unsteady detached wake flow. Griffith et al. [14] reported the presence of a two-dimensional instability of the periodic vortex shedding past a circular cylinder placed between two parallel walls, which leads to the beating phenomenon of the drag and lift coefficients.

Since its introduction, proper orthogonal decomposition (POD) has been widely used as an important and effective analysis technique for the identification of mathematical POD modes in spatio-temporally evolving flows. The ability to construct a reduced order model is also facilitated by POD [15–18]. Apart from fluid flow problems [19,20], the POD technique has been used to analyze diverse fields such as thermal convection [21], structural dynamics [22], and combustion [23–25]. The POD analysis of the flow past a cylinder has also been performed by various researchers [26–28].

In the present work, a variation of the traditional flow past a circular cylinder problem has been attempted. The bluff body considered in the present problem is not entirely circular, but has two slots on its periphery, 180° apart. Two different orientations of the placement of the slots, similar to the experimental setup of Mondal et al. [1], have been studied, henceforth referred to as co-counter flow and cross flow arrangements. In the co-counter flow arrangement, the direction of injection of the fluid through the slots is aligned with the free-stream flow. Since the injection takes place through two slots 180° apart, one of the injected jets has the same direction as the free stream while the other is in the opposite direction. This motivates us to call this configuration co-counter flow. In the second configuration, the fluid is injected in a direction perpendicular to the direction of free stream flow. Thus this configuration is called cross flow injection. The bluff body is placed centrally between two lateral boundaries in a steady free stream air flow in the confined flow configuration, and methane is injected at a steady uniform velocity from the two peripheral slots. Although substantial amount of literature exists on the blowing of species from the cylinder surface, none of those have considered the cylinder to be placed in a channel. Moreover, the motivation of the present work is not control of the vortex shedding, but to study the mixing phenomena in such a configuration. The ratio of the injection velocity to the free stream velocity is varied, and its effect on the wake structure and mixing of the two fluid stream has been studied. The commercial CFD package ANSYS Fluent 14.5 was used to carry out the simulations at a free stream Reynolds number of 100. The effect of the presence or absence of the lateral walls was investigated. Based on the results of the numerical simulations, proper orthogonal decomposition of the vorticity field was performed so as to identify the dominant modes and their flow characteristics.

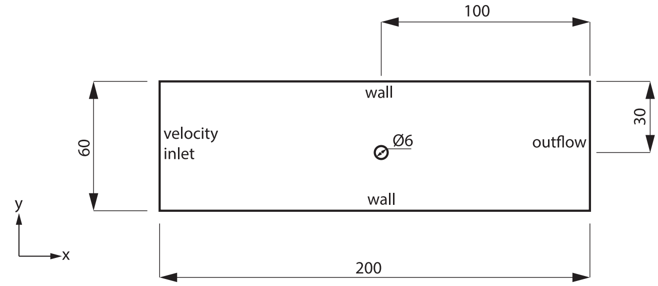

Fig. 1. Geometry and boundary conditions for the confined case (all dimensions in mm).

2. Problem geometry

The geometry used in the present study is shown in Fig. 1. A rectangular flow domain (200 mm × 60 mm) is considered, and the cylinder (6 mm diameter) is placed centrally within the flow domain. The channel width-to-cylinder diameter ratio is 10. On the periphery of the cylinder, two diametrically opposite slots (2 mm width) are placed as shown in Fig. 2. Two different flow configurations considered in the present work are shown in Fig. 2: Fig. 2(a) for the co-counter flow configuration and Fig. 2(b) for the cross flow configuration. In the co-counter flow configuration, injection of methane is along the positive and negative x-directions, whereas in the cross-flow configuration, injection velocity is along the positive and negative y-directions. The directions of injection are shown by red arrows in Fig. 2. The injection of methane is uniform across the entire width of the slots.

The free stream flow comprises of air at $Re = 100$, where Re is the free stream Reynolds number given by

$$Re = \frac{\rho U_\infty D}{\mu}. \tag{1}$$

The ratio of the velocity of methane injection to the velocity of free stream flow (ϵ) is varied, and the effects on the flow and mixing are studied.

3. Governing equations

For incompressible, laminar, two-dimensional flow, the governing equations can be expressed as: