Asia-Pacific Journal of Science and Technology :Volume :26 .Issue :01 .Article ID :.APST-26-01-08. Research Article

Asia-Pacific Journal of Science and Technology

https//:www.tci-thaijo.org/index.php/APST/index

Published by the Research and Graduate Studies,
Khon Kaen University, Thailand

Numerical study of vortex shedding suppression using confined cylinder

Zahrapanah Razaviyan[1], Milad Heidari[2,*], Fatemeh A. Talebbeigi[1] and Pooyan Rahmanivahid[2]

[1]Faculty of Engineering, Persian Gulf University, Boushehr, Iran
[2]Mechanical Engineering and Vehicle Technology Department, Global College of Engineering and Technology (GCET), Muscat, Sultanate of Oman
[*]Corresponding author: milad@gcet.edu.om

Received 21 August 2020
Revised 24 December 2020
Accepted 25 December 2020

Abstract

The application of cylindrical objects, such as pipelines and risers, in the marine industry is irresistible. Vortex shedding in wake flow downstream blunt bodies may cause vibrations, which result in structural failures in case of closeness or matching natural frequencies. This study aims to explore the effects of geometrical arrangements on the suppression of vortex shedding phenomena. Cylinders of different diameters and pitch distances are used to change the flow dynamics. The flow is considered to be incompressible and steady. A numerical study of flow passing a single cylinder with various diameters and two unequal cylinders with varying pitch distances is conducted. Results show a complete vortex suppression at $\delta/D=4$. These results are expected to be useful for the verification of computational models of unsteady Newtonian flows.

Keywords: Confined cylinder, Fluid-induced instability, Numerical study, Vortex shedding

1. Introduction

Vortex-induced vibration (VIV) in elastic blunt bodies is of great importance due to potential destructive effects on offshore structures, heat exchangers, towers and bridges. In the case of cross-flow passing a confined circular cylinder, flow separation starts at Reynolds numbers as low as Re ~2 [1]. The separated boundary layer leads to accumulated vorticities [2]. The circulation causes (Re) the growing vortex. As soon as the vortex reaches a degree of strength, it starts to attract the circulation of the opposing shear layer to the wake. The asymmetry of instable fluid wake induces vortex shedding.

A wide range of studies have predicted that vortex shedding leads to a reduction in fluctuating forces and drag [2]. These efforts have been done to provide better understanding and cure of vortex shedding phenomena, as well as provide efficient solutions to reduce the unwanted disturbances in the past few decades. Its importance is due to the creation of imposing vibrations leading to fatigue failure of structures as a result of a significant increase in horizontal and vertical fluctuations, which sometimes are the source of acoustic noise [2]. The von Karman vortex street appears when the periodical detachment of vortices happens and flow becomes unsteady [3].

The procedure of modeling turbulent flow around a blunt body is challenging. The retarded stagnation is strong, and flow separation and vortex shedding occur. Therefore, some simplifications can be made due to the symmetry and ignoring periodic vortex shedding. Engineering modeling seeks less accuracy than domain name service (DNS) or large eddy simulation (LES) methods, which simplifies modeling a 2D asymmetric system using available standard commercial solver (ANSYS Fluent).

The wake regime of blunt bodies has been vastly explored. Although flow passing a circular cylinder is a classical study, its primary and functional significance in fluid dynamics purposes cannot be ignored. Studies have continued to find better solutions to omit vortex shedding as much as possible. Various techniques have been introduced to control the wake regime, such as geometrical modifications, use of external small elements, and surface roughness modifications [4]. The method of using external components for vortex ignorance is notable from two points of view. First, the velocity profiles in the wake section are remodified, and their stability characteristics are changed. Second, shear stress distributions in the wake zone are altered. This technique may possibly create unwanted dynamisms. Furthermore, this approach is extremely sensitive to the arrangements of the components. Several studies have investigated flow control around a cylinder. Tsutsui explored the location and diameter of the rod to force flow reattachments to occur [5]. Lu et al investigated VIV suppression efficiency using control rods at different attack angles. They concluded that using four control rods is more efficient than using three [6]. Gozmen et al. studied vortex shedding behind a circular obstacle in shallow water using a splitter plate located in the downstream region at a Reynolds number of 6.25×103. The turbulent quantities and average flow change considerably with the dimensional ratios of the splitter plates in shallow flows [7]. Zhu et al. studied two-degree-of-freedom VIVs of a circular cylinder with and without two smaller control cylinders using computational fluid dynamics (CFD) and fluid-structure interaction (FSI) computational methods. The placement of controllers at 45° to the downstream flow vector can result in an appropriate eliminating outcome [8]. Numerical studies on VIV of a circular cylinder in the viscoelastic fluid have shown that viscoelasticity can increase the critical [9]. The latest studies in this field use numbers of rotating control rods (2-8) as an active method of suppression [10,11]. The problem with these methods is the supply of energy requirements to activate the rods.

This study presents the numerical study of the geometrical effects of the 2D-confined cylinder(s) on suppressing vortex shedding at various ranges of Reynolds numbers ($3<Re<6\times10^{6}$). This study aims to examine whether disturbing the wake regime can reduce the vortex flow. Although different methods are provided, they are still expensive. In this regard, cylinder(s) with zero aspect ratio, lift-to-drag (AR=L/d=0), which represents flow perpendicular to the disc, is simulated. Detailed results, including the lift coefficient, drag coefficient, and velocity contours of vortex shedding are obtained. Moreover, the effect of Reynolds number is investigated. The study mainly investigates if a wake control cylinder of diameter d, located in the downstream region of the primary circular cylinder of diameter D, can control suppress vortex shedding and the wake.

2. Materials and methods

The generalized geometrical arrangement and corresponding boundary conditions are illustrated in Figure 1. The diameter of the main cylinder (D) and the gap ratio between the control cylinder and delta/the main cylinder (δ/D) were the main parameters of the present investigation. For this purpose, two problem sets were simulated using Ansys Fluent. To observe the vortex shedding scheme downstream a single cylinder, a 2D circle (as the cross-section of a tube) was located at distance 10D from the inlet boundary. Different diameters were applied to the main cylinder, and its effects on lift and drag forces in terms of lift coefficient and drag coefficient were explored. In this phase, the cylinder was fixed at its specified position. Four diameters were assigned (1, 1.5, 2, and 2.5 m). For each diameter, 10 inlet velocities in the case of each Re number (Re=0.3, 25, 100, 6000000) were applied to include all types of flow regime.

In the latter step, a control cylinder with a smaller diameter was added to the system to investigate the possibilities to dampen the vortex formation. The goal of this system setup is to investigate the effect of the distance between the cylinders on the reduction of flow disturbances. In this case, the cylinder's diameter ratio (d/D) was kept constant, whereas variable δ/D was investigated. Although a wide spectrum of Reynolds numbers ranging $3<Re<6\times 10^{6}$ had been used to meet all fluid flow regimes (i.e., subcritical, critical, and supercritical), the inlet velocity was kept constant at v=0.2 m/s. Subsequently, the main cylinder (D=2 m) and the control rod (d=1 m) were exposed to an inlet velocity of 6 m/s. The von Karman vortex street formation at different pitch ratios was explored and reported in the following section. Convergence study was conducted, such that the flow around the cylinder was not impressed by the boundaries (Figure 2).

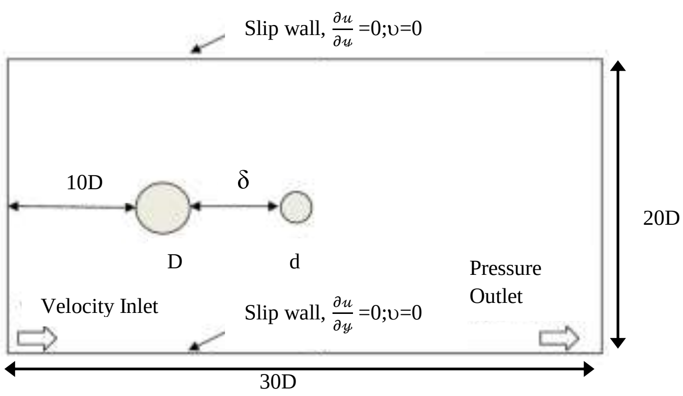


Figure 1 Geometrical parameters for the specified boundary.

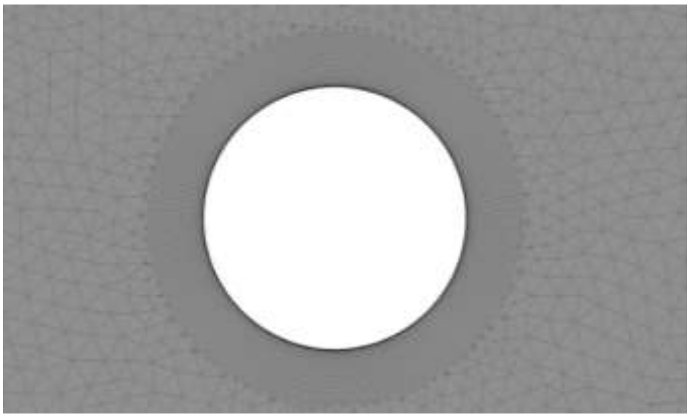


Figure 2 Computational domain.

The 2D boundary layer mesh was applied to the computational domain. Fifty layers of prism-type structural mesh were utilized close to the cylinders to improve accuracy. The triangular mesh was applied to other parts for the sake of comfortability. Mesh convergence test studies were performed until the suitable final mesh was found (Figure 3).

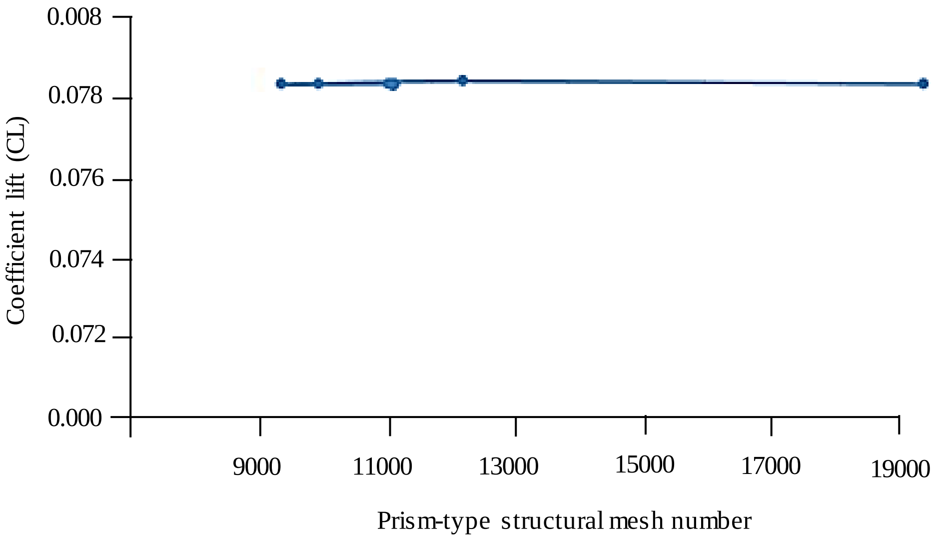


Figure 3 Mesh convergence.

For all cases, the problem was assumed to be in a steady-state. The energy equations were neglected to not consider the energy transfer. The Re was then calculated based on the inlet flow velocity and the tube's diameter. In the case of high Re, which represents turbulent flow, the standard k-epsilon method was used in the simulations, otherwise, the laminar scheme was used.

3. Results and discussion

3.1 Validation

The flow around a cylinder of diameter D=1m at Re = 100 was numerically modeled to establish a source for validation. The cylinder was located at an 8D distance from the inlet. From the hydrodynamic constants reported in Figure 4, the vortex shedding reached a steady-state after 100 iterations from the beginning of the simulation. The average drag (CD) and the lift (CL) coefficients were compared with other two-dimensional (2D) numerical investigations (Table 1).

Table 1 Validation of hydrodynamic coefficients.

Studies	CD	CL
Present Study	1.409	0.284
G.R.S. Assi et al. [10]	1.401	0.227
Blanchard et al. [12]	1.392	0.242

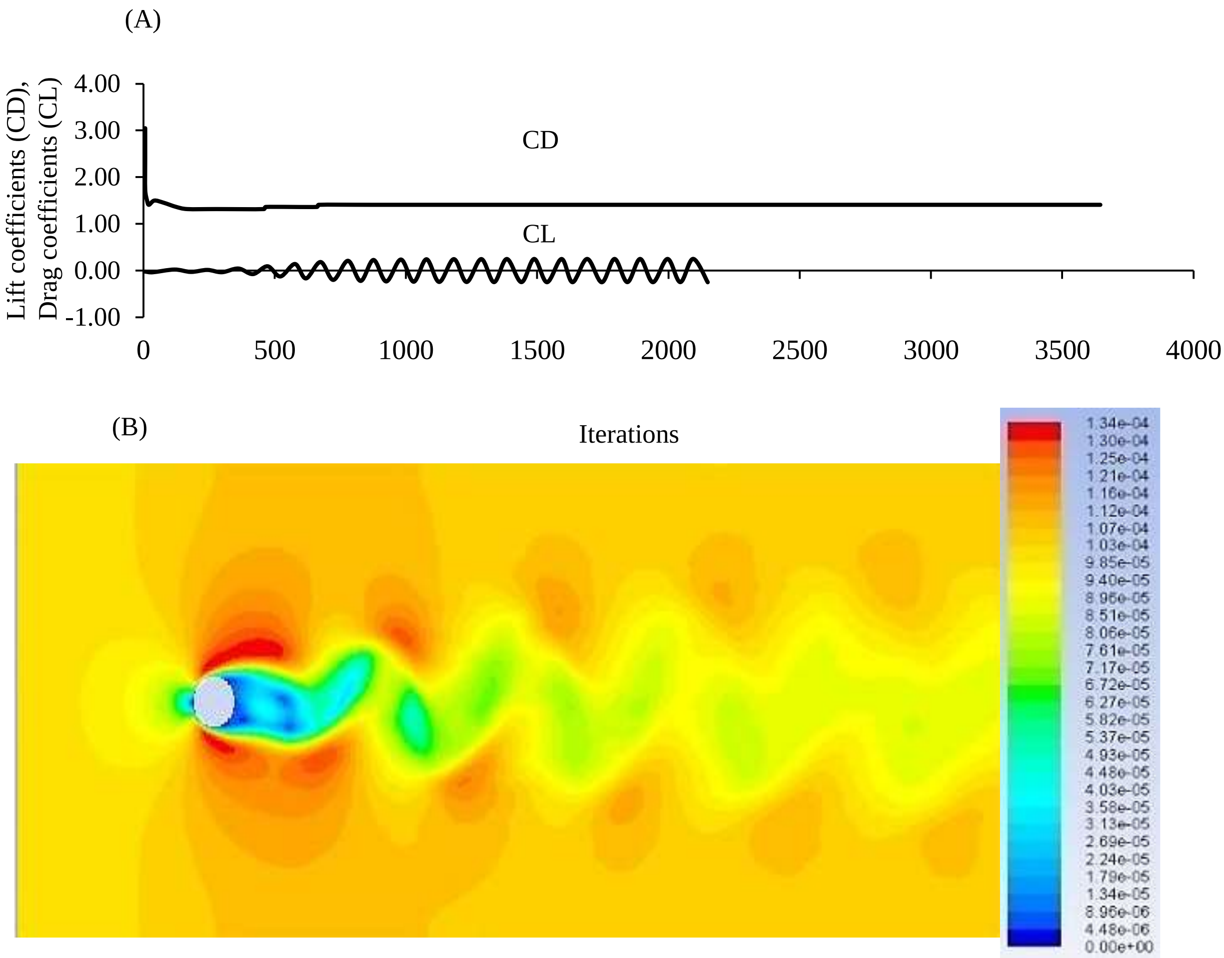


Figure 4 (A) Iterative series of CL, CD and (B) velocity contour at Re= 100.

3.2 Single cylinder

The flow around a bare cylinder (without control cylinder) at different Reynold's numbers was simulated to investigate the effects of diameter on the numerical domain results in terms of CL and CD. A wide range of Reynolds numbers has been studied to explore all possible kinds of fluid regimes including laminar, critical, and turbulent flows. From the data shown in Figure 5(A), a vortex shedding reached a steady-state after 600 iterations from the beginning of the simulation.

Similarly, Figure 5(B) shows the vorticity field downstream of the bare cylinder. The highlighted separated flow at the base of the cylinder by the streamlines was recognized as a typical Karman vortex street. The Reynold's number (defined as UD/υ; where U, D, and υ are the inlet velocity, the diameter, and the fluid's kinematic viscosity, respectively), the CD, and CL are presented in figure 45. As it is shown, in the case of the vortex shedding phenomenon, horizontal forces (represented in form of CD) fluctuate around a non-zero value with a 2f frequency. Provided that f = ω/2π is the shedding frequency. In the case of the studied model in this analysis at D = 2m, the average CD value has been 2.488, while the corresponding average CL has been 0.0048. According to this image, the value of CD increases by the size of the cylinder. It is to be highlighted that the CD reaches its maximum value at Re= 2×10^6. Because of the fluctuating nature of the lift forces, making a simultaneous comparison between all the values is not possible. Hence, it can be seen that the cylinder with D=2m experienced the least lift forces. Also, more lift fluctuations have taken place around the cylinders with D=1.5 and D=2.5.

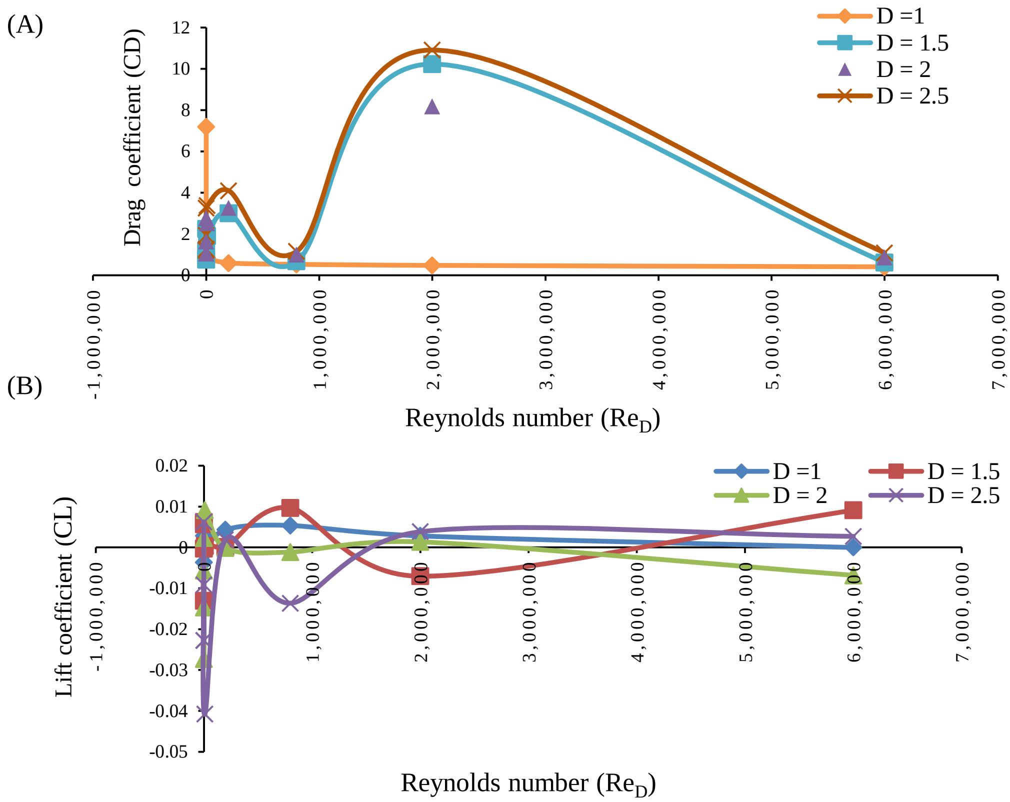


Figure 5 (A) Mean drag coefficient and (B) lift coefficient around bare cylinder.

As it was expected, the topology of the system had a serious effect on the wake regime. More details on flow conditions have been vastly explored before. To investigate the reduction of wake instabilities, a control rod has been used as reported in section 3.3.

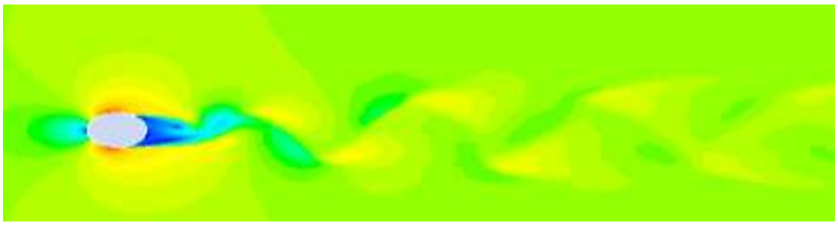
3.3 Double cylinders with control rod

In the case of using a control cylinder in the downstream wake, the control cylinder has been situated in a Pitch distance (δ) from the main cylinder to interfere with the wake regime. Presuming a constant D/d ratio of 2, δ value has been changed to find the optimum δ/d value. Constant inlet velocity of 0.2 m/s has been applied so as the fluid to meet a fixed pair of vortices in the wake and to ensure that a steady-state vortex shedding in the wake regime has occurred. The distance between cylinders (δ/d) has been increased in steps.
The controller induced no separate vortex, hence the vortex was produced by both of the cylinders acting as one body. The case of δ/d = 0 (no control cylinder), in Figure 6 gives the reader a better understanding of the wake regime and the controller's position in the domain. This figure is a visualized fluid flow at different pitch ratios. By variation in the pitch ratio, two diverse patterns were seen in the flow. No cyclic shedding from the major cylinder was detected for a pitch ratio smaller than 2 m.

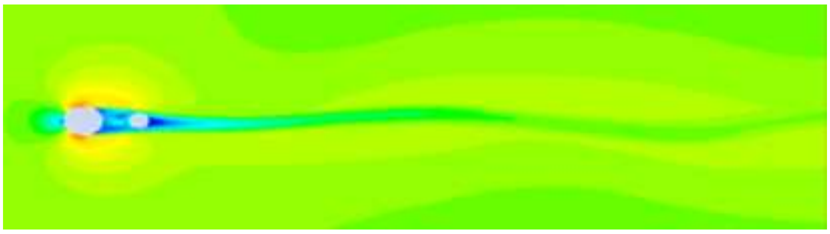
The detached shear layer from the main cylinder smoothly reattaches the control rod at approximate θ = ±40°. This point approves the reattachment location reported in previous studies. In these pitch distances, no vortex shedding is observed upstream of the control rod. It is called the cavity mode. It has been concluded that this cavity plays a vital role in drag reduction on the control cylinder. A pitch takes a distance bigger than the critical pitch distance (δ > 2 m), periodic vortex street starts to form behind the main cylinder. These vortices meet the control rod's face in downstream.

For δ/d=3, quasi-stationary vortices, technically called wake-impingement mode, are produced. It is important to comprehend that in this case, the flow speed touching the control rod is greater than the same quality associated with the cavity mode. As a result, it is expected that the drag reduction in this situation is less than that of the cavity mode. Comparing different pitch ratios in figure 5 shows that by increasing the wake width and shear layer inclination angle, flow separation starts at the front surface. The flow separation point on the control rod moves toward the upstream direction. These changes longer the vortex area in the downstream fluid flow.

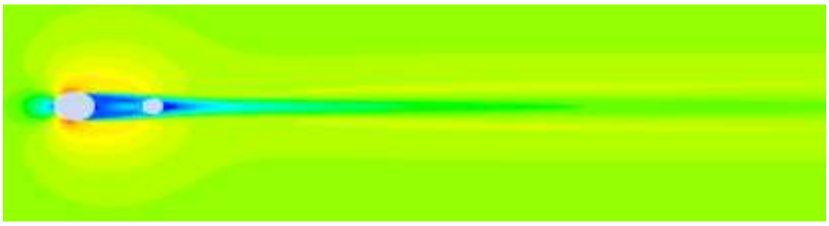
The oncoming flow resulted in turbulent flow separation on the surface of the control rod. This is probably due to a control rod installed in the downstream regime of the main cylinder (Figure 6). This initiates turbulent flow separation on the controller, which ends up a smaller vortex development and wake width compared to the flow around the cylinder. The flow oscillations have been faced with major damp at δ/d=3. But still, a low-frequency shedding can be observed. As shown in this figure, a complete wake suppression has occurred in δ/d=4. By increasing δ/d ratio, the fluid hydrodynamic responses differ. In cases δ/d=5 and δ/d=7, the system shows a high-frequency behavior; while in the case of δ/d = 6, again a low-frequency behavior is observed.



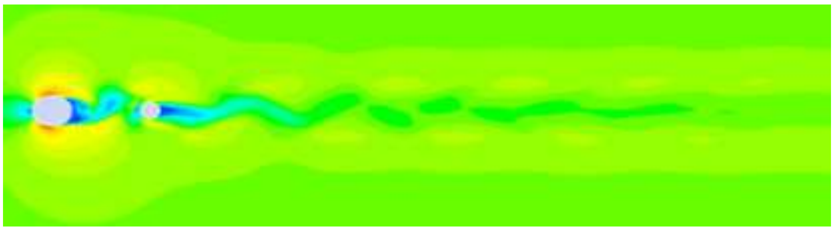
δ/d = 0



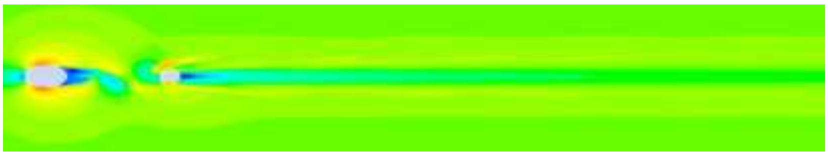
δ/d = 3



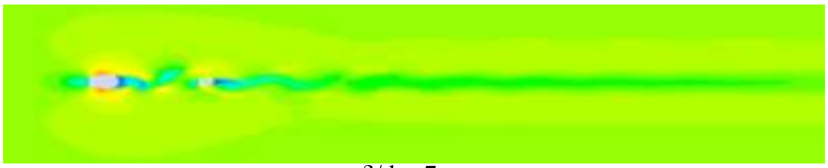
δ/d = 4



δ/d = 5



δ/d = 6



δ/d = 7

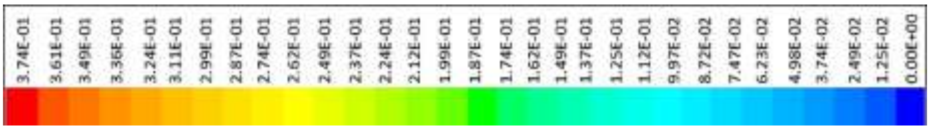


Figure 6 Vorticity contours around a cylinder with one control rod at different distances (δ/d = 3 to 7).

At different δ/d rations provided in figure 6, increasing the pitch distance between cylinders (δ) has little effect on the main cylinder's drag forces. Although it has been widely studied in the available literature, total drag coefficients have been investigated in this study. Considering the drag forces on the whole system (cylinder and control rod) validates the effectiveness of the used method. Total drag force is calculated using the equation offered by Schlichting:

$$D_{tot.} = \rho C \int_{-\infty}^{+\infty} U(U_0 - U) d_y \quad (1)$$

Where $U_0$the inlet velocity and C is is the cylinder's length. Total drag force for all the cases has been extracted to validate the measured drag against this equation. Figure 7(A) illustrates the drag coefficient (CD) of the whole system as well as individual components as a function of pitch ratio (δ/d) between cylinders. As shown in this figure, the drag coefficient of the main cylinder in presence of a control rod differs from that of the bare cylinder of the same setup. In the same way, the drag coefficient of the whole system is different from individual cylinders. For δ/d values higher than 4, the control rod's drag forces get increased. As a result, the whole system's drag trend is ascending. 4<δ/d <5 is a critical δvalue that has to be avoided in designs. The absolute values of lift forces show that the main cylinder had experienced a greater lift in all situations. It is obvious that in δ/d = 4, the lift coefficients related to the main cylinder, controller rod, whole system are perfectly matched to that of the bare cylinder. It can be concluded that vortex shedding suppression can be achieved by controlling the lift forces.

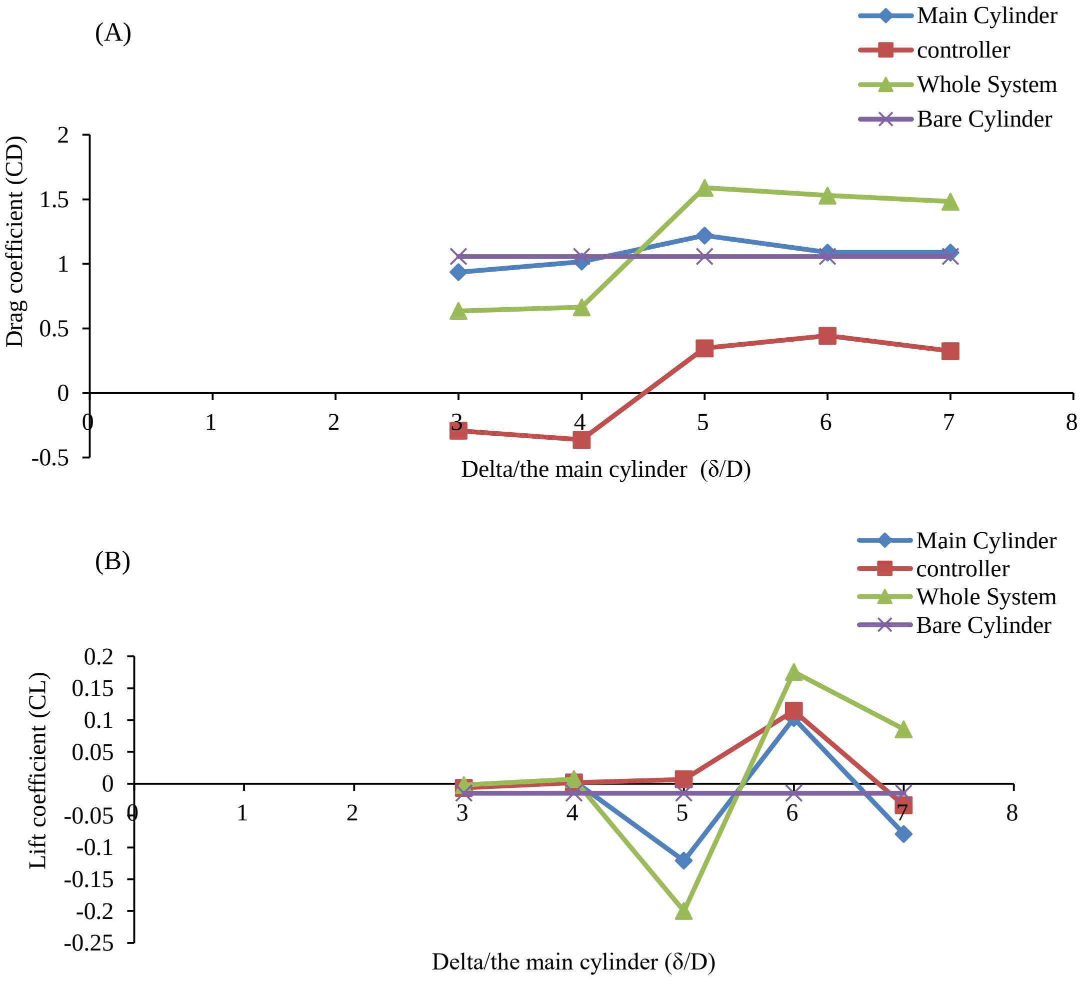


Figure 7 (A) Mean drag and (B) absolute mean lift for the case with control rod.

Taking advantage of simplicity in the study of 2D vortex shedding, which is indeed much more complex in 3D problems, is a feasible tool for understanding the mechanics of the system. Perhaps the most useful outcome of this study is to help better understanding of wake instability related issues. In the case of practical applications (namely offshore and energy fields), it is better to use control rods rather than bare cylinders. In cases that fluid flow direction is not known, it has been offered to use multiple control rods to reduce vortex shedding in any direction [12]. Because many significant parameters play role in this physical phenomenon, any produced procedure has to be investigated to reach an optimal solution. Among the studied parameters in this work, both cylinder size and Re decided the wake regime hydrodynamic. The extent of vortex shedding suppression by using a controller cylinder depends on its geometry and distance to the main cylinder. According to results, vorticity diffusion of separating shear layer is the driving factor for suppression.

In this study triangular mesh was used and the effect of mesh types on results did not investigate. However, the other types including quadrilateral or tetrahedron may present more reliable results. It is hoped that future studies may be able to accommodate this deficiency which may or may not play an important role in the more accurate results.

4. Conclusion

In this study, flow passing a circular cylinder with varying diameters was explored to offer a simplified view of the vortex phenomenon. Offered by a wake control cylindrical rod downstream the main cylinder, vortex shedding was investigated using available numerical methods at different Reynolds numbers. The controller could suppress the vortex mechanism of the major cylinder for a certain pitch distance ($\delta/d=4$). As a consequence of removing the Karman vortex street, the system’s total drag was reduced. The lowest drag coefficient of $CD \approx 0.1$ was observed for $\delta/d=4$. The controller also experienced high lift, which could be useful in the case of controlling the acting lateral forces. The mentioned forces were initiated due to the interaction of controllers with the developed Von-Karman Street. The actual behavior of such control mainly relies on Reynolds number and 3D properties.

5. Acknowledgments

We acknowledge Global College of Engineering and Technology of Oman for supporting the study.

6. References

[1] Toukir I, Rakibul HSM, Mohammad A. Flow separation phenomena for steady flow over a circular cylinder at low reynolds number. Int J Automot Mech Eng. 2013;8:1406-1415.

[2] Abhinav G. Arun S. Suppression of vortex shedding in flow around a square cylinder using control cylinder. Eur J Mech B Fluids. 2019;76:276-291.

[3] Shams UI, Raheela M, Zia UI, Shazia K, Zhou CY. A computational study of drag reduction and vortex shedding suppression of flow past a square cylinder in presence of small control cylinders. AIP Adv. 2017;7(4):19.

[4] Rashihi S, Hayatdavoodi M, Esfahani J. Vortex shedding suppression and wake control: a review. Ocean Eng. 2016;126:57-80.

[5] Tsutsui T. Flow control around a circular cylinder by a small rod. ACC J. 2019;25(1):78-86.

[6] Yan L, Yangyang L, Bin L, Wanhai X. Cross-flow vortex-induced vibration reduction of a long flexible cylinder using 3 and 4 control rods with different configurations. Appl Ocean Res. 2019;91:1-13.

[7] Gozmen B, Akilli H, Sahin B. Passive control of circular cylinder wake in shallow flow. Meas. 2013;46(3):1125-1136.

[8] Hongjun Z, Jie Y, Yue M, Hongnan Z, Youbo T. Simultaneous CFD evaluation of VIV suppression using smaller control cylinders. J Fluids Struct. 2015;57:66-80.

[9] Xiong Y, Peng S, Zhang M, Yang D. Numerical study on the vortex-induced vibration of a circular cylinder in viscoelastic fluids. J Nonnewton Fluid Mech. 2019;272:1-13.

[10] Assi GRS, Orselli R, Silva OM. Control of vortex shedding from a circular cylinder surrounded by eight rotating wake-control cylinders at Re=100. J Fluids Struct. 2019;89;13-24.

[11] Manish KC, Sushanta D, Bhupendra KG. Wake flow modification behind a square cylinder using control rods. J Wind Eng Ind Aerod. 2019;184:342-361.

[12] Blanchard A, Bergman LA, Vakakis AF. Vortex-induced vibration of a linearly sprung cylinder with an internal rotational nonlinear energy sink in turbulent flow. Nonlinear Dyn. 2020;99(1):593-609.