BCSIR

Available online at www.banglajol.info

Bangladesh J. Sci. Ind. Res. **57(4)**, 215-220, 2022

BANGLADESH JOURNAL OF SCIENTIFIC AND INDUSTRIAL RESEARCH

E-mail: bjsir07@gmail.com

Numerical study of unsteady flow past compound cylinders

M. M. Mosallem[1,2]*, H. M. Hadidi[1], M. Y. Tharwan[1] and Z. A. Ahmed[3]

[1]*Mechanical Engineering Department, Jazan University, Jazan, 45142, Saudi Arabia*
[2]*Department of Mechanical Power Engineering and Energy, Minia University, Minia, 61519, Egypt*
[3]*Architecture Department, Jazan University, Jazan, 45142, Saudi Arabia*

Abstract

The flow over compound cylinders is simulated using computational fluid dynamic (CFD). Three contacting cylinders of different diameters, arranged in tandem with respect to the oncoming flow were examined at Reynolds number $\mathrm{Re}_D = 333$ based on the diameter of the middle cylinder, D. The proposed configuration simulates the wind flow over the satellite launch vehicles during the period of launch. Computations were performed for diameter-to-diameter ratio, d / D, of 0.8 and 0.6. The numerical results revealed the vortex formation and vortex shedding mechanism in the wake downstream of the cylinders. For $d / D = 0.6$, the near wake is wider than that of $d / D = 0.8$. It is also found that the base pressure coefficient values for $d / D = 0.6$ are lower than those for $d / D = 0.8$. These findings imply that the acting drag force on the cylinders for $d / D = 0.6$ is larger than that for $d / D = 0.8$. Concluding that, the base pressure coefficient depends on the diameter ratio d / D whereas, the computed vortex-shedding Strouhal number value was found to be 0.25 and is independent on the d / D ratio.

Received: 11 April 2022
Revised: 30 October 2022
Accepted: 23 November 2022

DOI: https://doi.org/10.3329/bjsir.v57i4.58987

Keywords: Numerical; Unsteady flow; Compound cylinders; Vortex shedding

Introduction

The flow past arrays of bodies is an increasingly popular field of research. For example, flow over a multiple of cylinders is a very important issue in engineering practice, such as flow around satellite launch vehicles, tube banks, cables, chimneys and buildings. In such applications, vortex shedding induces vibrations that can potentially damage the structure under severe conditions. Therefore, it is necessary to investigate the fluid–structure interactions behind the bodies in order to obtain better structural designs.

Literature review

There are extensive studies on characteristics of flow over multiple-bodies. In his review of the literature, Zdravkovich (1977) showed that the wake characteristics behind two cylinders arranged side-by-side is greatly affected by the spacing between the cylinder. Mosallem (2003) studied the wake characteristics downstream a pair of circular cylinders placed side by side in a uniform flow. The wake behind the cylinders exhibited a quasi-stable behavior at spacing ratio less than 1.0. Sharman *et al.* (2005) analyzed laminar flows (Reynolds number of 100) over two cylinders arranged in tandem using unstructured CFD code. It was found that the predicted fluctuating and mean pressures between the cylinders varied with the cylinder spacing.

A numerical study was carried out by Harichandan and Roy (2010) to investigate the flow around two and three cylinders arranged in-line and also side by side. The tested Reynolds numbers are 100 and 200, respectively. Different wake patterns were noticed behind the cylinders. Furthermore, Bao *et al.* (2010) investigated numerically the flow past three cylinders in equilateral-triangular manner with different spacings, and for three incidence angles. The simulated Reynolds number was 100, and the numerical results illustrated that the Strouhal number was affected by the transitions of interference effect.

---

*Corresponding author e-mail: mosall2000@yahoo.com

Liang *et al.* (2009) examined numerically the laminar flow over a cylinder array. The results indicated that the flow asymmetry depends on the gap between the cylinders. Furthermore, vortex shedding starts from the last cylinder and proceeds upstream. Sumner (2010) reviewed the flow around two cylinders immersed in a steady flow. The flow patterns behind the cylinders and Strouhal numbers were demonstrated. For Reynolds numbers (Re ≤ 160), Lee and Yang (2009) studied numerically the flow over two cylinders. Various arrangements of the cylinders with different spacing between them and the inclination angle were considered and ten flow patterns were identified. Additionally, Lam *et al.* (2008) performed flow simulation for four cylinders arranged in square configuration with different distances between the cylinders at two Reynolds (100 and 200). The results revealed three different flow patterns, namely vortex shedding, stable shielding, and wiggling shielding. The sensitivity of spacing between cylinders on the flow phenomenon was studied by Alam *et al.* (2017). The study numerically simulated the flow over three circular cylinders at Re = 200 with varying the spacing between the centers of the cylinders and the flow was found to be very sensitive to the spacing between the cylinders.

Fig. 1. Satellite launch vehicle during the designed period of launch

Selvi *et al.* (2009) studied numerically and experimentally the wind flow over a typical satellite launch vehicle model. The model was exposed to two cases of wind conditions. The results of streamlines obtained from simulation and experiments are compared very well. The predicted force coefficients showed a good agreement with those measured in the wind tunnel. Considering investigations on the cross-flows around multiple cylinders, researchers studied the spacing ratio effect on the flow behind multiple cylinders. However, there is a lack of available literature that is focused on flow over compound cylinders of different diameters. This configuration simulates the wind flow over the satellite launch vehicles for the problem of wind in horizontal direction particularly when there is a possibility of storms, cyclones and tornados near the launching station during the period of launch as shown in Fig.1. The objective of the present work, therefore, is to investigate the flow behavior over three compound cylinders of different diameters.

Numerical simulations

Governing differential equations

The flow to be studied is assumed to be two-dimensional, unsteady laminar, and isothermal and the fluid is considered to be incompressible, Newtonian and of constant viscosity. The flow motion is governed by the continuity and Navier-Stokes Eq. (1-3) as follows:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}=-\frac{\partial p}{\partial x}+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right] \tag{2}$$

$$\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}=-\frac{\partial p}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right] \tag{3}$$

Where u and v are the velocity components along the axes x and y, respectively. p is the pressure, ρ and μ are the density and viscosity of the fluid and t is the time.

Computational details

The particular problem for which a solution of the above equations is required, is that of flow around three tandem compound cylinders with different diameters. Fig. 2 shows the present computational domain of the compound cylinders. The cylinders were placed in a constant velocity uniform flow. Flow approaching from the left direction, where the three main cylinders are one behind the other. The upstream and downstream cylinders have an equal diameter, *d*. The diameter of the middle cylinder is *D*. The examined diameter-to-diameter ratios, d / D were 0.8 and 0.6.

The initial and boundary conditions are defined as follows:

- At the inlet boundary of the domain, a uniform velocity profile is considered, $u = U_\infty$ and $v = 0$
- No-slip condition is applied on duct walls and the cylinders surface, $u = v = 0$
- Atmospheric pressure is assumed at the outlet of the domain
- The initial condition is chosen to be a uniform flow at stream conditions everywhere

Numerical simulations were done at Reynolds number, $\mathrm{Re}_{D} =$ 333 based on the diameter of the middle cylinder, *D* and freestream velocity, U_∞. The computations were done with a commercial CFD software package (Ansys/Flotran). To examine the grid dependence, steady flow computations were performed for the two diameter ratios. Three numerical grids were considered. The grids were built using Ansys software. The grid refinement was applied on the cylinder surfaces and at the walls of the domain. Table I gives the grid number, number of nodes and mean base pressure coefficient at $\mathrm{Re}_D = 333$.

Table I. Grid independence test performed at $\mathrm{Re}_D = 333$

	$d / D = 0.8$		$d / D = 0.6$	
Grid	No. of nodes	$\overline{Cp}_b$	No. of nodes	$\overline{Cp}_b$
1	54477	-0.85535	73216	-0.99333
2	91527	-0.86123	89167	-1.0190
3	108688	-0.86365	105023	-1.0198

Based on the computed values of the mean base pressure coefficient, $\overline{Cp}_b$ shown in Table I, grids No. 2 have been selected for the flow calculations as shown in Fig. 3. The base pressure is defined as the exerted pressure on the base or extreme aft end of a body, i.e., the pressure at point 1 marked on Fig. 2.

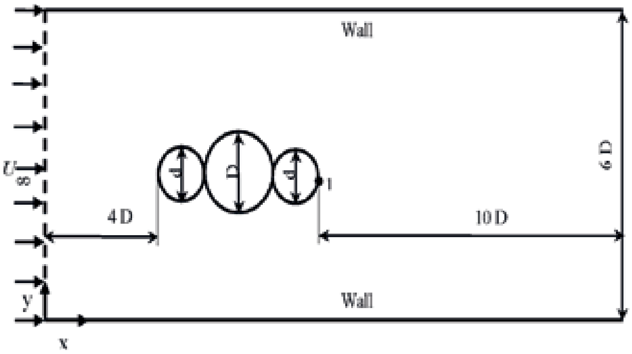
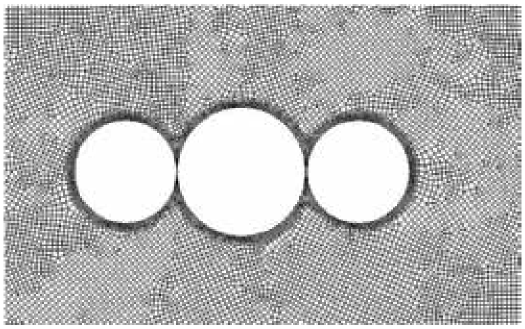


Fig. 2. Computational domain

$d / D = 0.8$



$d / D = 0.6$

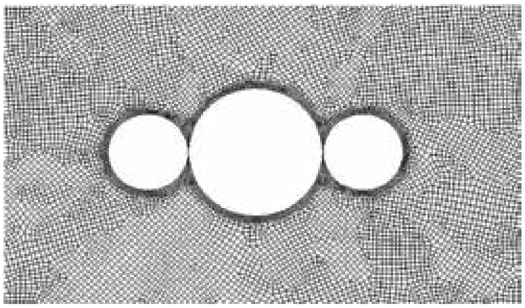


Fig. 3. Grid distributions around the three cylinders

Results and discussion

The flow around the compound cylinders was computed. The first part of the numerical results presents the flow pattern characteristics. In order to describe and understand the vortex shedding behavior, snapshots of instantaneous streamlines for the two examined diameter-to-diameter ratios are shown

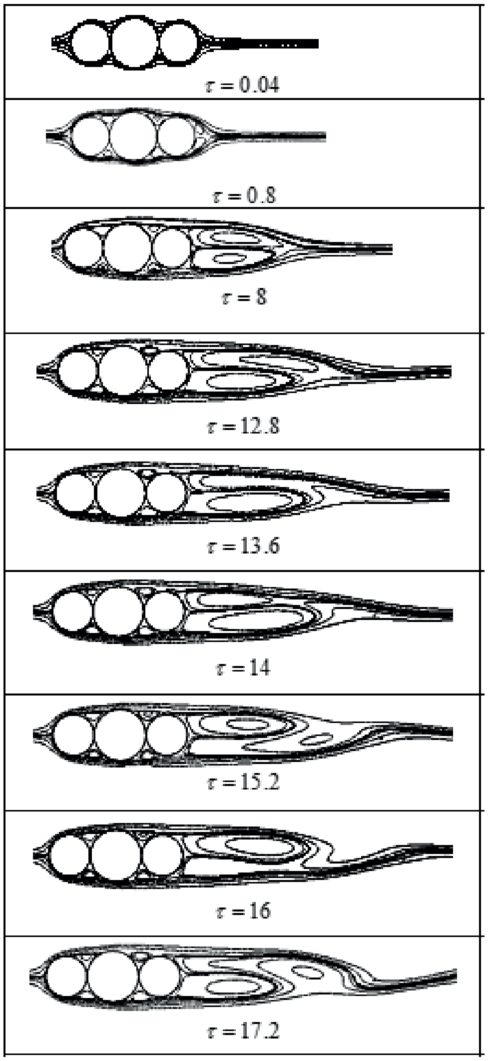


Fig. 4. Instantaneous streamlines map for $d/D = 0.8$

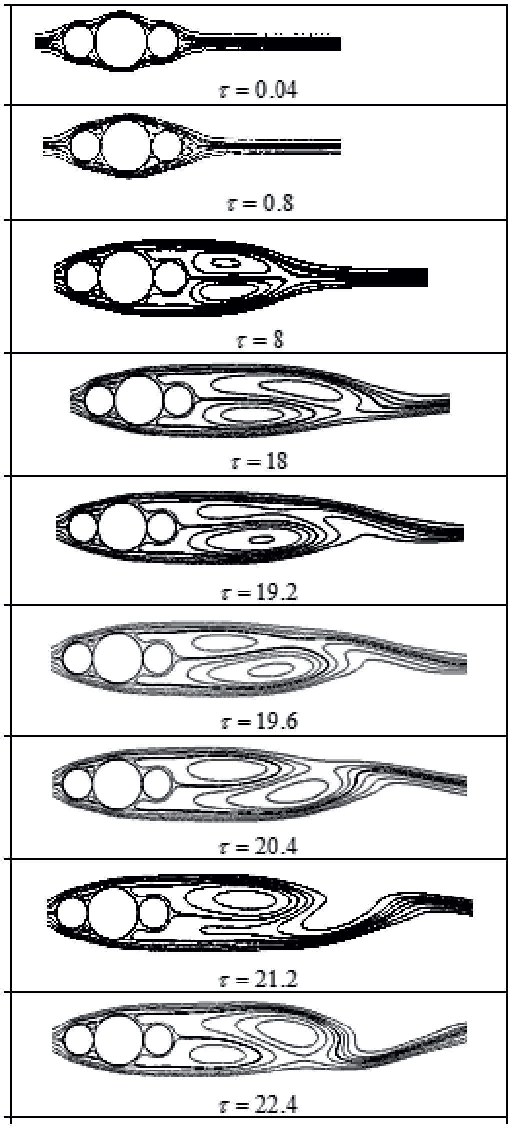


Fig. 5. Instantaneous streamlines map for $d/D = 0.6$

in Figs 3 and 4. The dimensionless time, τ is $\tau = tU_\infty/D$. The computations were begun from an impulsive start. For $d/D = 0.8$, at $\tau = 0.04$, (Fig. 4), the streamlines are uniform. As the time increases to $\tau = 0.8$, the streamlines show the appearance of closed circulation zones in the corners between the cylinders. Furthermore, the formation of a pair of small vortices behind the cylinders can be noticed. However, when time goes by, $\tau = 8$, the small vortices grow larger in size. When the computation time further increases the lower vortex in the near wake grows in size and squashes the upper vortex ($\tau = 12.8$ and $\tau = 13.6$). At time $\tau = 15.2$, the lower vortex starts to shed from the downstream cylinder and the upper vortex grows larger in size. This process is repeated in the reverse sense ($\tau = 17.2$). Therefore, a periodic and alternate vortex shedding develops in the wake of the cylinders.

For $d/D = 0.6$ at $\tau = 8$, (Fig. 5), the one visible change in the flow features is that the separated shear layer from the middle cylinder does not reattach onto the surface of the downstream cylinder and consequently, the circulation zones do not exist between the middle and the downstream cylinders. In addition, the near wake is significantly wider than that of $d / D = 0.8$ $d / D = 0.8$. The second part of the numerical results is concerned with the base pressure coefficient data. Fig. 6 presents the time history of the computed base pressure coefficient, Cp_b for both d / D ratios. After the fully developed state is reached and the initial effect completely disappeared, the base pressure coefficient exhibits a periodic variation with time. The clear periodicity illustrated in the base pressure coefficient denotes the periodic vortex shedding behind the compound cylinders. It is also obvious

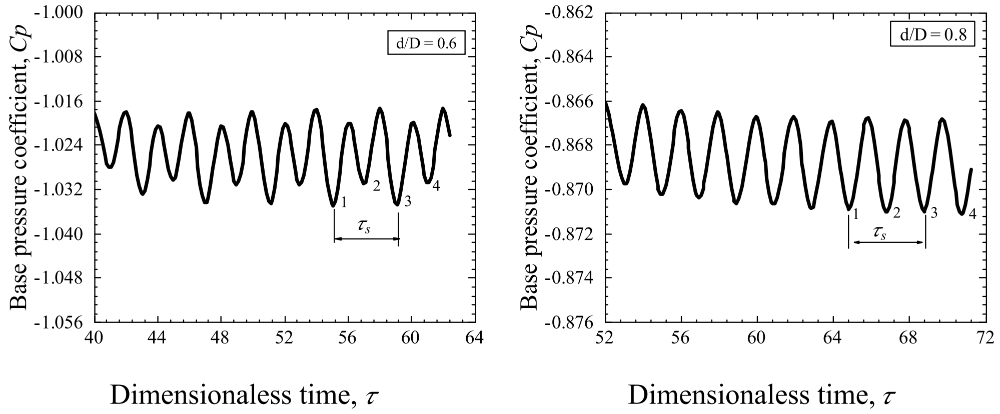


Fig. 6. The time history of the base pressure coefficient

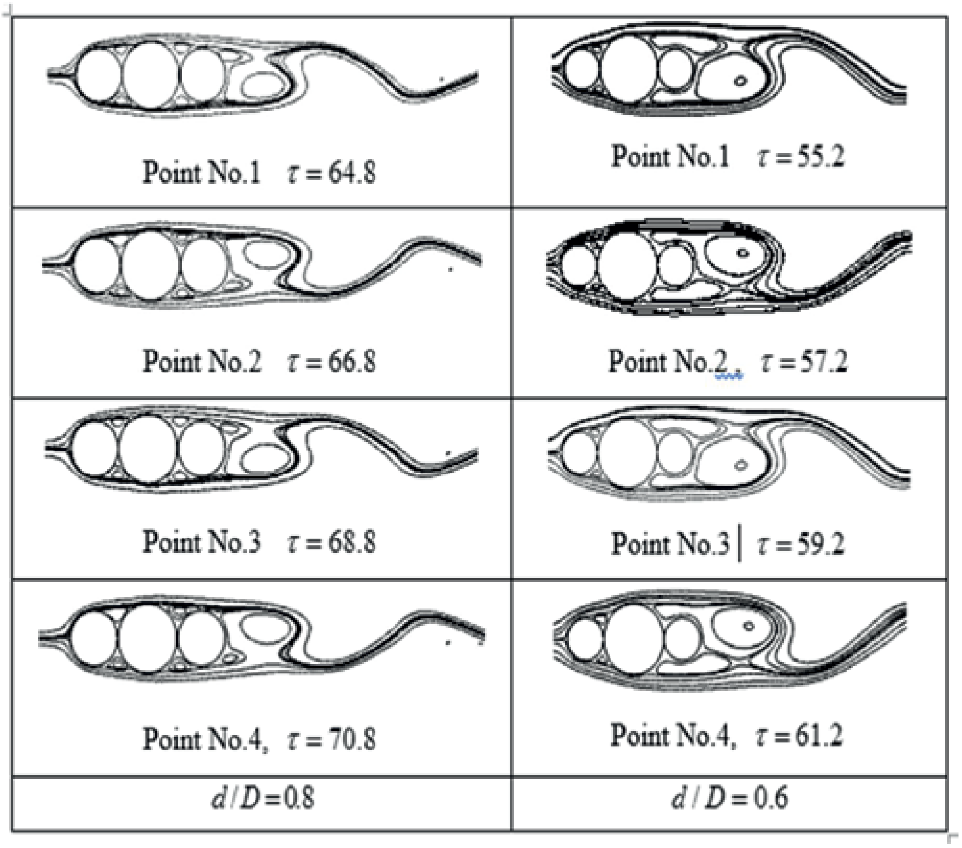


Fig. 7. Periodic steady state

that the base pressure coefficient values for $d / D = 0.6$ are lower than those for $d / D = 0.8$. This implies that the acting drag force on the cylinders for $d / D = 0.6$ is larger than that for $d / D = 0.8$.

The computation is terminated when the solution reaches periodic stability. The shedding Strouhal number, can be determined from the fluctuating base pressure coefficient as follows:

$$St = 1/\tau_s \tag{4}$$

where τ_S is shown in Fig. 6. The value of Strouhal number obtained for both ratios ($d / D = 0.8$ and $d / D = 0.6$) is equal to 0.25. This value is near to that of a single cylinder (0.20). Therefore, the shedding frequency is independent on the d / D ratio for the present flow conditions.

Fig. 7 shows the predicted streamlines map at the periodic steady-state corresponding to the points 1, 2, 3 and 4 marked on Fig. 6. It is evident that at points 1 and 3, the vortex shed away from the lower surface of the downstream cylinder while another vortex begins to form at the upper surface of the lower cylinder. At points 2 and 4 the process is repeated in the reverse sense. This confirms the fully periodic behavior of the vortex shedding behind the compound cylinders. The vortices start immediately behind the downstream cylinder for $d / D = 0.8$. In the case of $d / D = 0.6$, the formation of vortices begins in the corner region between the middle cylinder and downstream cylinder. These vortices stretch in the flow direction and travel downstream to the near wake behind the cylinders and then shed away.

Conclusion

Numerical simulations were performed to investigate the flow over three compound cylinders of different diameters at. $Re_D = 333$. The examined diameter-to-diameter ratios, d / D were 0.8 and 0.6. The results showed the vortex formation and vortex shedding mechanism in the near wake behind the cylinders. The base pressure coefficient exhibits regular variation with the time and the flow is periodic. Furthermore, the base pressure coefficient depends on the diameter ratio d / D. The predicted value of the shedding Strouhal number was found to be 0.25 and it is independent on the d / D ratio within the examined range.

References

Alam MM, Derakhshandeh JF, Zheng Q, Rehman S, Ji C and Zafar F (2017), The flow around three tandem circular cylinders. *Advances in Structural Engineering and Mechanics (ASEM17), LLsan (Seoul) Korea.*

BaoY, Zhou D and Huang C (2010), Numerical simulation of flow over three circular cylinders in equilateral arrangements at low Reynolds number by a second-order characteristic-based split finite element method, *Computers & Fluids* (**39):** 882-899. DOI: org/10.1016/j.compfluid.2010.01.002

Harichandan AB and Roy A (2010), Numerical investigation of low Reynolds number flow past two and three circular cylinders using unstructured grid CFR scheme, *International Journal of Heat and Fluid Flow* **31**: 154-171. DOI: org/10.1016/ j.ijheatfluidflow.2010.01.007

Lam K, Gong WQ and So RMC (2008), Numerical simulation of cross-flow around four cylinders in an in-line square configuration, *Journal of Fluids and Structures* **24**: 34-57. DOI: org/10.1016/ j.jfluidstructs.2007.06.003

Lee K and Yang K (2009), Flow patterns past two circular cylinders in proximity, *Computers & Fluids* **38**: 778-788. DOI: org/10.1016/j.compfluid.2008.07.005

Liang C, Papadakis G and Luo X (2009), Effect of tube spacing on the vortex shedding characteristics of laminar flow past an inline tube array: A numerical study, *Computers & Fluids* **38**: 950-964. DOI: org/10.1016/j.compfluid.2008.10.005

Mosallem MM (2003), Investigation of wake characteristics downstream a pair of circular cylinders in a uniform flow, *Bulletin of the Faculty of Engineering, Minia university* **22**: 112-122. DOI: org/10.1299/ jsmeb.42.178

Selvi RS, Santhoshkumar M, Lakshmanan N, Nadaraja PS and Paramasivam M (2009), CFD analysis and wind tunnel experiment on a typical launch vehicle model, *Tamkang Journal of Science and Engineering* **12**(3): 223-229. DOI: org/10.6180/jase.2009.12.3.01

Sharman B, Lien FS., Davidson L and Norberg C (2005), Numerical predictions of low Reynolds number flows over two tandem circular cylinders, *Int. J. Numer. Meth. Fluids* **47**: 423-447. DOI: org/10.1002/fld.812

Sumner D (2010), Two circular cylinders in cross-flow: A review, *Journal of Fluids and Structures* **26**: 849-899. DOI: org/10.1016/j.jfluidstructs.2010.07.001

Zdravokovich MM (1977), Review of flow interference between two circular cylinders in various arrangements, *Trans. ASME J. Fluids Engng*. **99**: 618-633.