



The Influence of Bluff Body Shape Variations on Aerodynamic Characteristics in Flow Around a Cylinder

Christian Jovie Yudhananta¹, Fitri Wahyuni^{1*}, James Julian¹, Riki Hendra Purba¹, Fathin Muhammad Mahdhudhu², Elvi Armadani³, Nely Toding Bunga⁴

¹Departement of Mechanical Engineering, Universitas Pembangunan Nasional Veteran Jakarta, Indonesia

²Departement of Naval Architecture, Universitas Pembangunan Nasional Veteran Jakarta, Indonesia

³Departement of Industrial Engineering, Universitas Pembangunan Nasional Veteran Jakarta, Indonesia

⁴Departement of Mechanical Engineering, Universitas Pancasila, Indonesia

*Correspondence email: fitriwahyuni@upnvj.ac.id

Received: 22/11/2025

Revised: 15/01/2026

Accepted: 17/01/2026

Abstract: Fluid flow around an object plays an important role in various fields, including aerodynamics, building design, and transportation. This study examines the effect of three two-dimensional bluff body shapes, including Circular, Square, and Diamond, on the flow pattern, drag and lift coefficients, and vortex shedding characteristics. The methodology used is Computational Fluid Dynamics (CFD) simulation for incompressible flow with a Reynolds number of 100. The results show that variations in geometric shapes significantly affect the shape formation, flow stability, and aerodynamic force response. The three bluff body variations were found to produce fluctuating aerodynamic characteristics due to the influence of the von Kármán vortex phenomenon. In the diamond cylinder variation, it was found to be the configuration with the widest shape, having the highest average drag coefficient value of 1.84, and exhibiting the most significant force amplification, with an average lift coefficient value of 0.36 and a Strouhal number (St) of 0.18. Overall, variations in bluff body shapes significantly affect the flow pattern, Vortex Shedding frequency, and the total aerodynamic force acting on the object.

Keywords: bluff; CFD; drag; vortex; aerodynamic



SDGs Problem Solving

The research and problem-solving in this study are in line with the SDGs Chapter #9. This study employs CFD simulations to assess the aerodynamic performance of bluff body shapes, thereby enhancing engineering design efficiency, reducing energy losses, and promoting innovation in transportation and infrastructure systems in alignment with sustainable industrial development.

Conflict of Interest Statement: The author(s) declares that the research was conducted in the absence of any commercial or financial relationship that could be construed as a potential conflict of interest.

Copyright: © ASIIMETRIK. This is an open access article distributed under the terms of the Creative Commons Attribution 4.0 International License (CC-BY 4.0), which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.

Jurnal Asimetrik: Jurnal Ilmiah Rekayasa dan Inovasi - ASIIMETRIK is an open access and peer-reviewed journal published by Faculty of Engineering, Universitas Pancasila, Indonesia.



1. INTRODUCTION

Fluid flow past an object is a classic case in fluid mechanics and serves as a fundamental basis for various engineering fields, including submarine design, aircraft development, wind turbine design, piping systems, and bridge infrastructure. When a fluid interacts with the surface of an object, it causes flow separation and vortex shedding behind the body (Wang, Fan and Lin, 2020; Alam, 2021). This phenomenon generates aerodynamic forces in the form of drag and lift, which are expressed through the drag and lift coefficients. These two parameters are strongly influenced by the Reynolds number, which represents the ratio between inertial and viscous forces in the fluid (Anderson, 2017). At low Reynolds numbers, flow is laminar, causing alternating separation from both surfaces of a body, which forms a pattern known as the Von Kármán vortex street (Norberg, 2003). Each bluff body shape exhibits unique characteristics in its vortex shedding pattern, which significantly affect the drag and lift coefficients. Therefore, the geometry of the body is one of the key factors determining flow characteristics and forms the fundamental basis of aerodynamic and hydrodynamic analysis.

Numerous studies have been conducted on fluid flow past objects with various geometries. Soyler et al., investigated flow characteristics around smooth and grooved cylinders using Particle Image Velocimetry (PIV) and Computational Fluid Dynamics (CFD) (Soyler et al., 2022). The results showed that adding square grooves could reduce the drag coefficient by up to 30% at $Re = 20,000$ and narrow the wake width, indicating a passive control effect on vortex formation (Soyler et al., 2022). Al-Haboobi et al., applied the Taylor–Galerkin/Pressure Correction algorithm to simulate flow around differing geometries, demonstrating that square cylinders generate a significantly higher drag coefficient than their circular counterparts (Al-Haboobi, Al-Juaifri and Al-Muslimawi, 2024). Singh et al., conducted CFD simulations on a circular cylinder and concluded that transient simulations are the most suitable for accurately capturing the vortex shedding phenomenon around the cylinder (R. Singh et al., 2025). Lastly, Onel et al., analyzed the wake characteristics of a diamond-shaped cylinder equipped with a splitter plate in laminar flow (Onel et al., 2025). Their findings indicated that the addition of a splitter plate could effectively reduce drag and suppress the wake.

Each bluff body geometry shows distinct patterns of flow separation and vortex shedding. The circular cylinder is a key reference in fluid flow studies because it produces a clear, periodic vortex shedding pattern. Based on previous studies that primarily focused on higher Reynolds numbers (turbulent regime), three-dimensional configurations, or the application of passive flow control techniques (Soyler et al., 2022; Al-Haboobi, Al-Juaifri and Al-Muslimawi, 2024; I. S. Singh et al., 2025; Onel et al., 2025), systematic comparisons of bluff body shape variations in two-dimensional flow past cylinders at a Reynolds number of 100 using CFD simulations remain limited. Moreover, this flow regime involves complex behaviors with significant engineering implications, particularly Kármán vortex shedding. Therefore, the present study conducts a numerical investigation to examine flow structures around cylinders with various bluff body shapes, namely circular, square, and diamond. It also examines the aerodynamic parameters of these bodies to better understand the interactions between the solid surface and the fluid, as well as the resulting wake effects.

2. METHODOLOGY

2.1. Geometry

The geometry used in this study is two-dimensional (2D) and consists of three shapes: circular, square, and diamond. Each geometry has a characteristic length or diameter of 2 meters. The model is placed at the center of the fluid domain to ensure optimal flow contour results. Figure 1 illustrates the geometries.

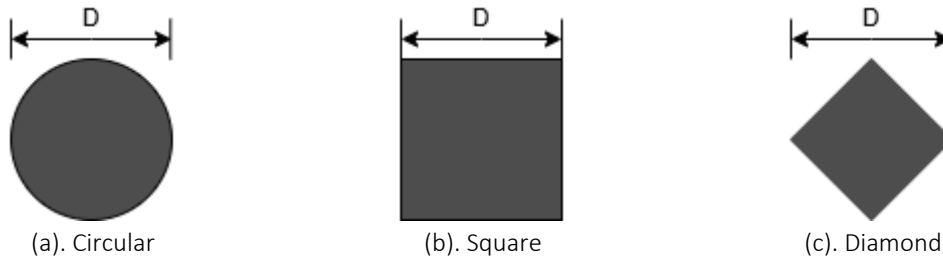


Figure 1. Geometries

2.2. Governing Equation

The equations used in this study are the Navier–Stokes equations. These equations are essential for understanding incompressible and laminar fluid flow, illustrating the relationship between pressure, flow velocity, and viscous forces acting on a fluid element (Versteeg and Malalasekera, 2011). Mathematically, the Navier–Stokes equations consist of the law of mass conservation and the law of momentum conservation. Equation (1) represents the conservation of mass, while Equations (2) and (3) describe the conservation of momentum in the x and y directions, respectively (Liu and Kopp, 2011; Mehdipour *et al.*, 2022; I. S. Singh *et al.*, 2025).

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{Re} \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (2)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{Re} \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (3)$$

2.3. Aerodynamic Parameters

In analyzing aerodynamic forces, the main parameters considered are the drag coefficient and lift coefficient. The drag coefficient represents the resistive force acting parallel to the direction of the fluid flow, while the lift coefficient represents the lifting force acting perpendicular to the flow direction. The values of these coefficients depend on the geometric shape, Reynolds number, and surface flow conditions (Hishikar *et al.*, 2022). In general, higher pressure around the surface of an object results in a greater drag coefficient, whereas a symmetrical pressure distribution produces a more stable lift coefficient (Yadav *et al.*, 2021). The drag and lift coefficients are expressed by Equations (4) and (5).

$$C_D = \frac{F_D}{\frac{1}{2} \rho V^2 A} \quad (4)$$

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A} \quad (5)$$

2.4. Strouhal Number

The Strouhal number is a dimensionless quantity that represents the frequency pattern of vortex shedding in a fluid flow (Lekkala et al., 2022). The Strouhal number is described, which connects the frequency of oscillation in the flow to the velocity of the flow and the length of the object's surface. It is linked to the lift coefficient, as higher shedding frequencies result in periodic oscillations of the lift force (Mathupriya et al., 2018). Mathematically, the relationship between the frequency (f), characteristic length (L), and flow velocity (U) can be expressed by Equation (6).

$$St = \frac{f \times L}{U} \quad (6)$$

2.5. Mesh and Boundary Conditions

The mesh used in this study is a structured mesh composed of quadrilateral elements, as shown in Figure 2a. A structured mesh was selected due to its high numerical stability and its ability to produce smoother velocity distribution compared to an unstructured mesh (Cao and Tamura, 2016). The structured mesh employed in this study is graded, meaning that the mesh becomes finer near the cylinder wall, approaching $Y \approx 1$. The simulation domain used in this study is illustrated in Figure 2b. The boundary conditions were defined for a Reynolds number of 100, with a time step of 0.001. The computational domain was divided into five regions: inlet, outlet, wall, symmetry 1, and symmetry 2. The inlet represents the region where the fluid first enters the domain, with a specified inlet velocity. The cylinder wall applies to a no-slip condition, while the outlet is defined by a pressure outlet with a zero value, representing the exit of the fluid flow. The remaining boundaries are defined as symmetry conditions. A summary of the boundary conditions used in this study is presented in Table 1.

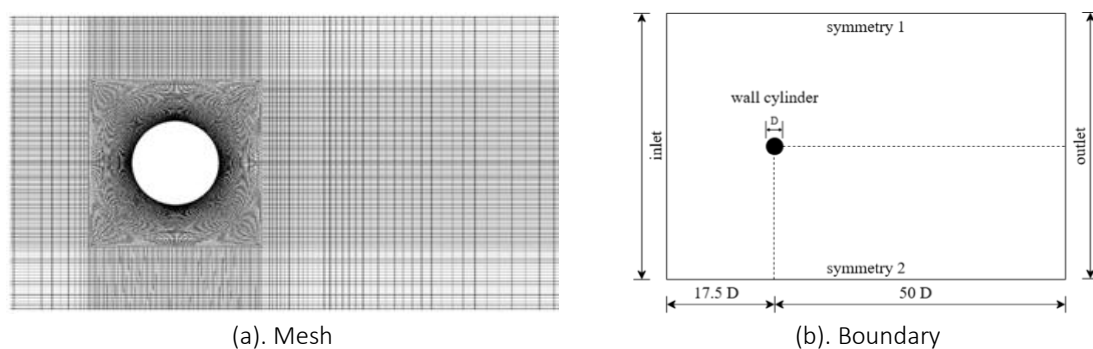


Figure 2. Mesh and Boundary Conditions

Table 1. Detail of boundary conditions

Boundary Conditions	
Inlet	2.5 m/s
Outlet	0
Bluff body	Wall (non-slip)
Symmetry 1 dan 2	Symmetry

2.6. Mesh Independence Test

The mesh independence test is a process used to confirm that the simulation outcomes are precise and not affected by the mesh resolution. In this study, the mesh independence test was performed using Richardson's Extrapolation method (Roache, 1994). First, Equation (7) was used to calculate the mesh variation ratio. The next step involved determining the order of accuracy using Equation (8). Subsequently, the Grid Convergence Index (GCI) was calculated to quantify the numerical error associated with the mesh variations. After completing these calculations, the results were verified using Equations (13) to ensure that the selected mesh variations were within an acceptable convergence range, thereby confirming their suitability for use in this study.

$$r = \frac{h_2}{h_1} \quad (7)$$

$$\bar{p} = \frac{\ln\left(\frac{f_3 - f_2}{f_2 - f_1}\right)}{\ln(r)} \quad (8)$$

$$GCI_{fine} = \frac{F_s |\epsilon|}{(r^{\bar{p}} - 1)} \quad (9)$$

$$GCI_{coarse} = \frac{F_s |\epsilon| r^{\bar{p}}}{(r^{\bar{p}} - 1)} \quad (10)$$

$$\epsilon = \frac{f_{n+1} - f_n}{f_n} \quad (11)$$

$$\frac{GCI_{coarse}}{GCI_{fine} r^{\bar{p}}} \approx 1 \quad (12)$$

$$f_{r_{h=0}} = f_1 + \frac{(f_1 - f_2)}{(r^{\bar{p}} - 1)} \quad (13)$$

The sample data used in the mesh independence test were taken at the coordinates $x = 0.4$ and $y = 1.7$. Three mesh variations were employed: fine mesh (100,000 elements), medium mesh (50,000 elements), and coarse mesh (25,000 elements). Figure 3 shows the visualization of the mesh variations, and Table 2 summarizes the results of the mesh independence test. The fine mesh had the smallest error value, so it was chosen for data extraction and further analysis.

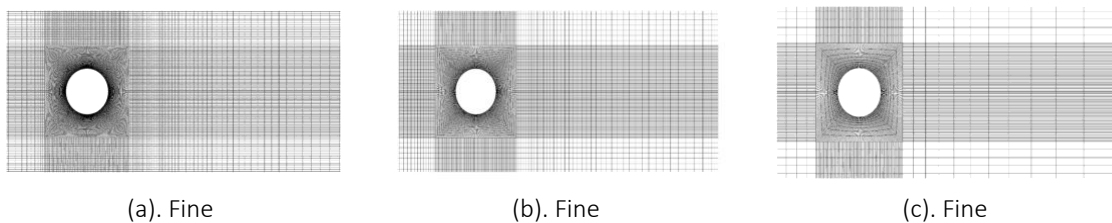


Figure 3. Mesh variations

3. RESULTS AND DISCUSSION

3.1. Validation

This section describes the validation process carried out to ensure that the results of the study meet the required accuracy standards. The validation was performed at a Reynolds number of 100 by comparing the drag coefficient, lift coefficient, and Strouhal number with previously reported

experimental and numerical data in the literature. Several fundamental studies were selected as reference cases due to their enduring relevance in investigating fluid flow past a circular cylinder at low Reynolds numbers (Braza, Chassaing and Minh, 1986; Ding et al., 2007; Harichandan and Roy, 2010). The comparison of the results is presented in Table 3. The comparison indicates that the present study shows a reasonable level of accuracy with respect to previous studies, and therefore the results can be considered valid.

Table 2. Results of the mesh independence study

Mesh	Fine	Medium	Coarse
Difference of pressure	3.28066	3.23228	2.99093
$\bar{p}$		2.318644152	
r		2	
GCI_{fine}		0.469%	
GCI_{coarse}		2.3400%	
$f_{rh} = 0$		3.292789473	
$\frac{GCI_{coarse}}{GCI_{fine} r^{\bar{p}}}$		1	
Error	0.368365%	1.837636%	9.167287%

Table 3. Validation result

Source	C_D	C_L	St
(Braza, Chassaing and Minh, 1986)	1.386	0.25	0.160
(Ding et al., 2007)	1.356	0.287	0.166
(Harichandan and Roy, 2010)	1.352	0.278	0.161
Present Study	1.384	0.241	0.168

3.2. Analysis

The simulation results analyzed the impact of bluff body geometry on flow structure at a Reynolds number of 100 using CFD. Figure 4 presents the drag coefficient as a function of time, initially decreasing, then increasing, and ultimately stabilizing. The sharp decrease in C_D at approximately 30 seconds marks a transient phase during which the boundary layer forms and adapts to the surface. The value subsequently rises and shows periodic oscillations, signalling the beginning of stable vortex shedding. At the beginning of the test ($t \approx 0$ s), all three shapes exhibit a relatively high drag coefficient, close to 2.0. This high initial value is associated with the sudden interaction between the fluid flow and the body surface, which generates strong flow separation and unstable wake formation. As time progresses during the first 30–50 seconds, the drag coefficient for all geometries decreases significantly. This reduction indicates the development of a more organized flow structure and partial stabilization of vortex shedding behind the body.

Among the three geometries, the circular shape shows the lowest minimum drag coefficient during the transient phase (approximately 1.13–1.15), suggesting smoother flow separation and reduced pressure drag compared to the square and diamond shapes. The square configuration stabilizes at a moderate C_D value (around 1.53), while the diamond shape eventually exhibits the highest steady drag coefficient (around 1.85).

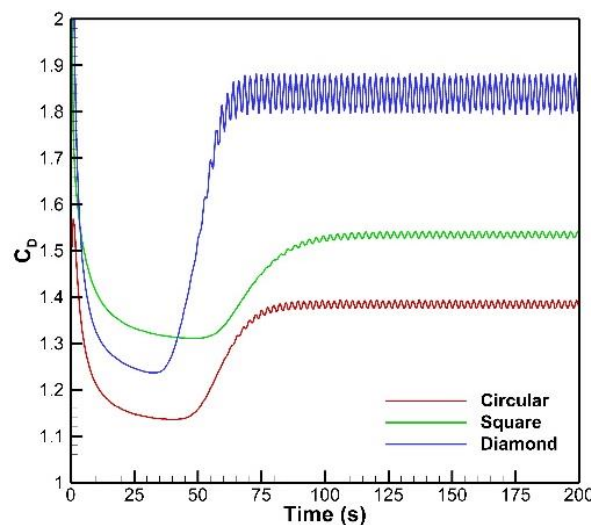


Figure 4. Graph of drag coefficient versus time

After approximately 60 seconds, the drag coefficients begin to rise and then fluctuate around relatively constant mean values. These oscillations are caused by periodic vortex shedding in the wake region. The diamond shape displays the largest amplitude oscillations, indicating stronger unsteady wake dynamics and higher aerodynamic instability. In contrast, the circular shape shows smaller oscillations, implying more stable flow behaviour.

The flow stabilization for the circular cylinder occurs at around 100 seconds, which is the longest duration observed among the various geometries. This shape exhibits a smooth drag coefficient curve with minimal fluctuation amplitude and an average C_D of 1.384. These outcomes are attributed to the aerodynamic properties of the circular cylinder, which facilitate the gradual formation of a uniform wake and reduced drag. The square cylinder stabilizes at around 80 seconds, with moderate fluctuation amplitude and an average C_D of 1.534. Its sharp edges promote earlier flow separation, resulting in a wider wake and increased drag relative to the circular cylinder. The diamond cylinder stabilizes at approximately 75 seconds, exhibiting the largest fluctuations and an average C_D of 1.84, which reflects pronounced and less stable flow patterns, as well as the highest drag force among the analyzed geometries.

Figure 5 presents the lift coefficient (C_L) as a function of time. Initially, the graph reflects a transient state characterized by continuously changing amplitude and a non-periodic oscillation pattern. For each geometry, the curves transition to oscillations with nearly constant amplitude at different times, signifying the establishment of a stable periodic vortex pattern. The diamond cylinder exhibits the earliest onset of fluctuations at approximately 30 seconds, produces the largest amplitude, and stabilizes at around 65 seconds.

As time progresses beyond approximately 70 seconds, the oscillations become clearly periodic and reach a nearly constant amplitude, indicating that the flow has reached a limit cycle state. The diamond geometry exhibits the largest oscillation amplitude, with peak lift coefficients approaching approximately ± 0.7 . This suggests stronger vortex shedding intensity and larger alternating aerodynamic forces acting on the body. In contrast, the circular geometry shows moderate oscillation amplitudes, while the square geometry produces slightly smaller fluctuations compared

to the diamond case. The higher amplitude in the diamond configuration can be attributed to its sharper edges, which promote earlier flow separation and stronger vortex formation.

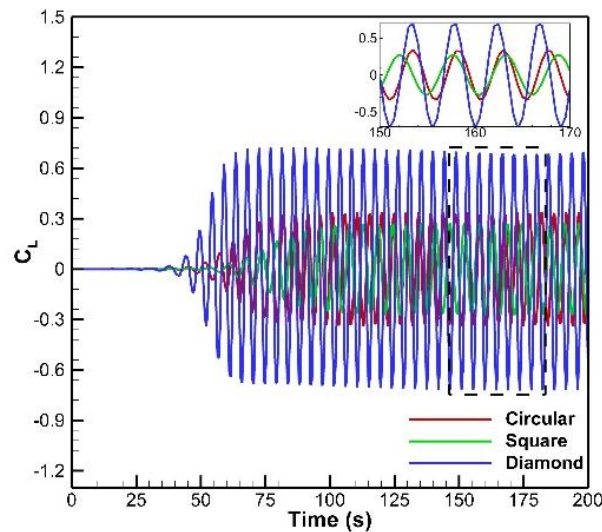


Figure 5. Graph of lift coefficient versus time

The circular cylinder begins to fluctuate at approximately 40 seconds and stabilizes at 75 seconds, while the square cylinder starts fluctuating at around 60 seconds and stabilizes at 100 seconds. For the square cylinder, the average C_L value is 0.17. The square cylinder has an average C_L value of 0.17, attributed to its flat front surface, which generates a larger wake region and irregular vortex shedding. This results in lower and less consistent lift forces, with a maximum value of 0.276. The circular cylinder exhibits an average C_L of 0.241, intermediate between the diamond and square cases. Its smooth surface promotes a more regular and periodic Von Kármán vortex street, resulting in moderate and stable lift fluctuations with a maximum of approximately 0.342. The diamond cylinder exhibits the highest average C_L at 0.362, resulting from vortex shedding along inclined surfaces that divide the flow into upper and lower streams. This configuration produces strong lift fluctuations with a peak amplitude of 0.476, the highest among the geometries. These variations in C_L oscillation characteristics underscore the significant influence of bluff body geometry on the fluctuation pattern and intensity of the lift force.

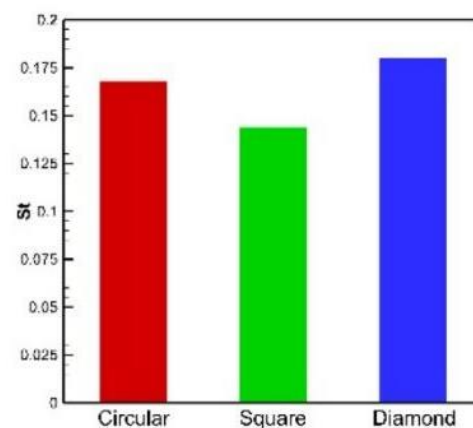


Figure 6. Comparison of Strouhal number (St) for circular, square, and diamond

Figure 6 presents the comparison of the Strouhal numbers for each bluff body shape, obtained from the dominant oscillation frequency of the lift coefficient. From the Figure 6, the diamond shape exhibits the highest Strouhal number, approximately 0.18. The circular shape follows with a value around 0.17, while the square shape shows the lowest value, approximately 0.145. These differences indicate that the vortex shedding frequency varies depending on the geometry of the object exposed to the flow. The higher Strouhal number for the diamond geometry suggests that it produces vortices at a higher frequency relative to the incoming flow velocity. This can be attributed to its sharp leading edges and inclined surfaces, which promote earlier flow separation and more rapid formation of alternating vortices in the wake region. The circular geometry, having a smooth surface, generates a relatively stable and well-known vortex shedding pattern (the classical Kármán vortex street), resulting in a moderate Strouhal number. In contrast, the square geometry produces the lowest Strouhal number. The flat frontal surface and sharp corners cause strong flow separation at the leading edges, but the wake structure tends to be broader and more stable, leading to a slightly lower shedding frequency compared to the other shapes.

The results indicate that the diamond cylinder has the highest Strouhal number at 0.18, followed by the circular cylinder at 0.168 and the square cylinder at 0.144, highlighting the impact of cylinder shape on flow characteristics. The diamond cylinder exhibits the highest oscillation frequency because its flow system reaches stability more quickly with large CL amplitudes, while the square cylinder shows the lowest frequency due to slower stabilization and moderate CL amplitudes. The circular cylinder lies between the two, with moderate frequency and small amplitude. These results show a noticeable difference compared to the study conducted by Singh et al., on a circular cylinder, which reported a Strouhal number of 0.076 (R. Singh et al., 2025). The low Strouhal number indicates a quasi-steady flow condition, in which vortex shedding is relatively weak and results in small lift force fluctuations. Therefore, it can be concluded that the Strouhal number, which is influenced by the geometric shape, plays an important role in determining the oscillation frequency of the flow.

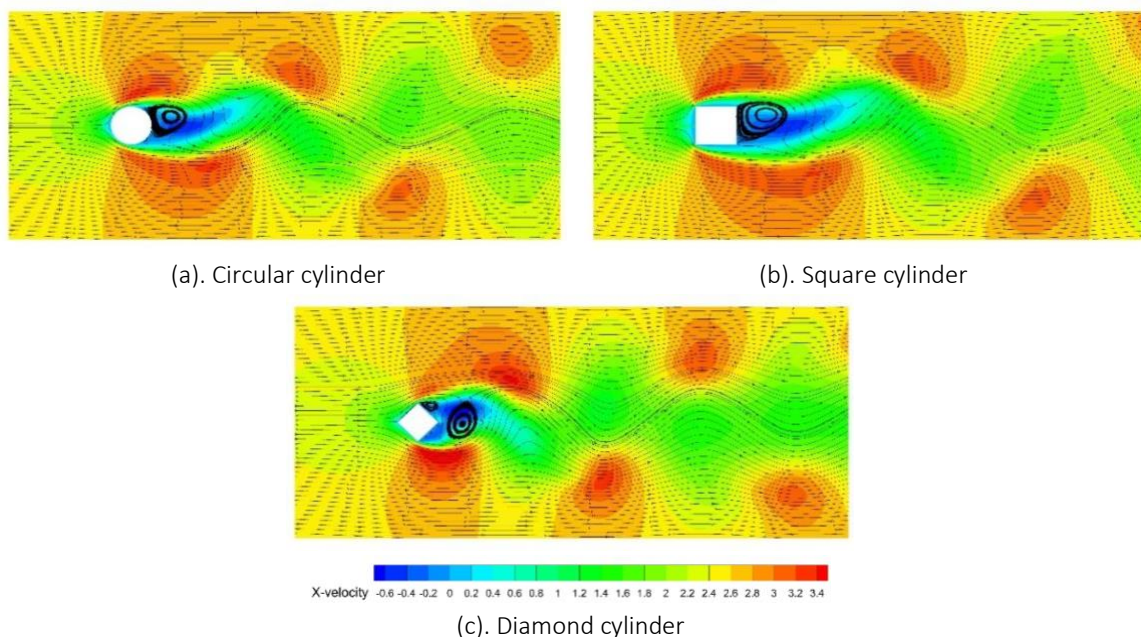


Figure 7. Velocity contours and streamlines at $Re = 100$

Figure 7 shown the velocity contours and streamlines for the three bluff body shapes. The results reveal distinct flow patterns and wake regions, which correspond to the stability of the aerodynamic forces described previously. For the circular cylinder, velocity decreases upon interaction with the frontal surface, followed by acceleration along the sides and flow separation at the rear. This sequence produces a short recirculation zone and a relatively narrow wake, resulting in periodic vortices that signify stable flow with a regular vortex shedding pattern. This observation is consistent with the lift force results, which show moderate amplitude and stable oscillation frequency.

The square cylinder exhibits early flow separation caused by sharp corners, resulting in a wider and longer wake, a larger recirculation region, and more irregular vortices compared to the circular cylinder. In the diamond cylinder, stagnation points at the upstream corners cause the flow to split along the inclined surfaces and rejoin at the downstream corner, forming two recirculation zones close to the body. This configuration results in a shorter and narrower wake but with more intense vortex activity than the square cylinder. These features account for the diamond cylinder's highest oscillation frequency and amplitude among the three geometries. Previous studies have reported that the diamond cylinder exhibits a complex wake topology, characterized by the formation of a sub-wake that is not observed in circular cylinders, as well as the development of secondary separation patterns on the diamond cylinder (Jiang, 2021; Yadav et al., 2021).

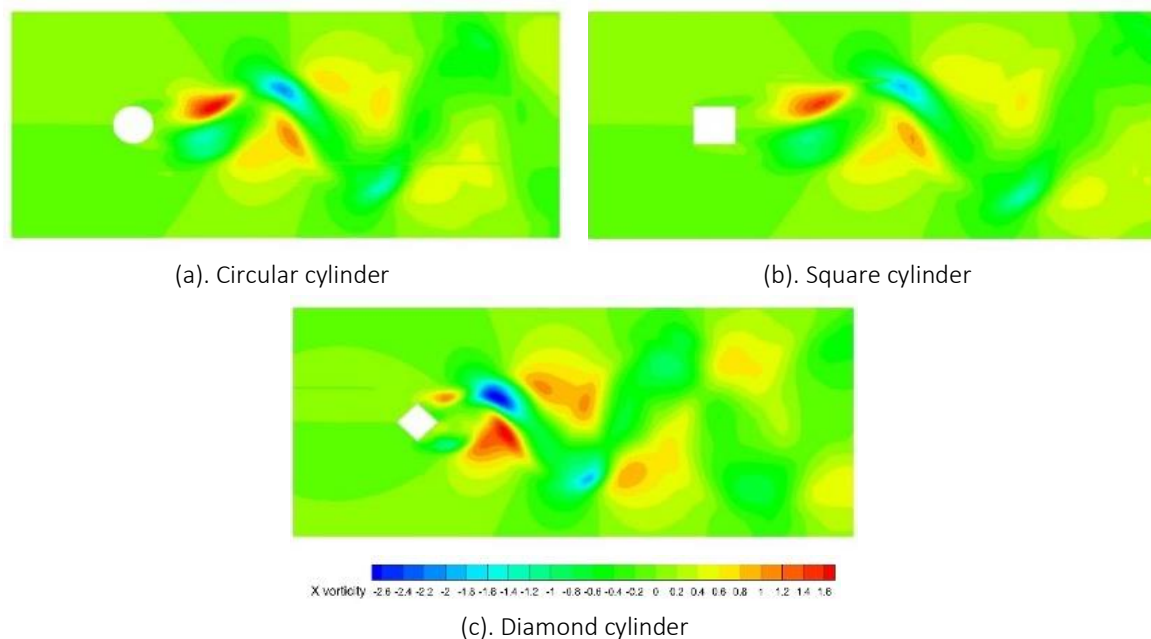


Figure 8. Vorticity contours at $Re = 100$

Figure 8 presents the vorticity contours, which further clarify the vortex structures illustrated in Figure 7. For the circular cylinder, an organized vorticity region can be observed behind the body, aligned along the flow direction, representing the characteristic von Kármán vortex shedding pattern. This behaviour is consistent with the lowest drag coefficient and the smallest fluctuations in lift. In the square cylinder, the vorticity region exhibits a wider magnitude, resulting in a larger wake compared to the circular cylinder. This corresponds to the higher drag and greater fluctuations in lift observed for the square geometry. In the diamond cylinder, regions of high vorticity magnitude are concentrated near the body, and the wake distribution appears inclined with an irregular and

asymmetric flow pattern. This explains the highest drag values and the strongest lift fluctuations among the three geometries. Therefore, the vorticity contours clearly illustrate the relationship between wake structure and the aerodynamic force behaviour for each bluff body geometry.

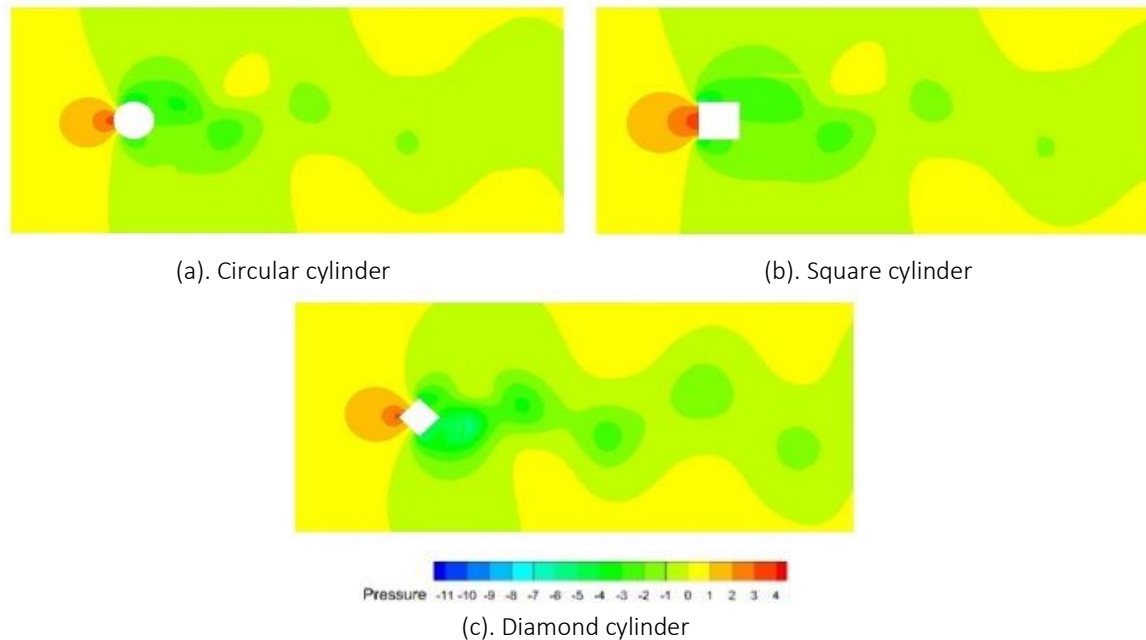


Figure 9. Pressure contours at $Re = 100$

Figure 9 presents the pressure contours surrounding the bluff body geometries. In the circular cylinder, maximum pressure is observed at the stagnation point and decreases toward the rear, forming a relatively narrow low-pressure region. This distribution leads to a smaller longitudinal pressure difference and, consequently, a lower drag force. For the square cylinder, the flat front surface increases flow resistance and induces moderate fluctuations in force. This effect enlarges the upstream stagnation zone and expands the downstream low-pressure region, resulting in a wider wake and less stable flow behind the body. In the diamond cylinder, stagnation points at the upstream corners generate a steeper pressure gradient along the inclined surfaces. A stronger and more extensive low-pressure region forms behind the body, resulting in the largest pressure difference between the upstream and downstream regions among the three shapes. This results in the highest drag force, despite the smallest stagnation area. Consequently, the pressure distribution is closely linked to the aerodynamic forces experienced by each bluff body shape.

4. CONCLUSION

This study investigates the effect of 2D bluff-body shape variations (circular, square, and diamond) on flow structure and aerodynamic forces in incompressible laminar flow at a Reynolds number of 100. The simulation results show that the circular cylinder produces the most stable force response, characterized by the lowest mean drag coefficient of 1.384, a mean lift coefficient of 0.241, and a Strouhal number of 0.168. The square cylinder promotes earlier flow separation, resulting in a wider wake region that is associated with an increased mean drag coefficient of 1.534, a reduced mean lift coefficient of 0.170, and a lower vortex shedding frequency ($St = 0.144$). In contrast, the diamond cylinder produces the widest wake and the strongest force fluctuations, yielding the highest mean

drag coefficient of 1.84, the largest mean lift coefficient of 0.362, and the highest Strouhal number of 0.18. These results demonstrate that bluff body geometry strongly influences flow separation patterns, wake development, and vortex shedding behavior, which directly affect the values of C_D , C_L , and St . Therefore, in flow past bluff bodies, the selection of bluff-body geometry should be considered a primary design parameter for controlling vortex shedding behavior and the resulting aerodynamic forces.

REFERENCES

- Alam, M.M. (2021) 'Effects of Mass and Damping on Flow-Induced Vibration of a Cylinder Interacting with the Wake of Another Cylinder at High Reduced Velocities', *Energies*, 14(16), p. 5148. Available at: <https://doi.org/10.3390/en14165148>.
- Al-Haboobi, A., Al-Juaifri, G.A. and Al-Muslimawi, A.H. (2024) 'Numerical study of Newtonian laminar Flow around circular and square cylinders', *Results in Control and Optimization*, 14, p. 100328. Available at: <https://doi.org/10.1016/j.rico.2023.100328>.
- Anderson, J.D. (2017) *Fundamentals of Aerodynamics*. United Kingdom: McGraw-Hill Education. [Print].
- Braza, M., Chassaing, P. and Minh, H.H. (1986) 'Numerical study and physical analysis of the pressure and velocity fields in the near wake of a circular cylinder', *Journal of Fluid Mechanics*, 165, pp. 79–130. Available at: <https://doi.org/10.1017/S0022112086003014>.
- Cao, Y. and Tamura, T. (2016) 'Large-eddy simulations of flow past a square cylinder using structured and unstructured grids', *Computers & Fluids*, 137, pp. 36–54. Available at: <https://doi.org/10.1016/j.compfluid.2016.07.013>.
- Ding, H. et al. (2007) 'Numerical simulation of flows around two circular cylinders by mesh-free least square-based finite difference methods', *International Journal for Numerical Methods in Fluids*, 53(2), pp. 305–332. Available at: <https://doi.org/10.1002/fld.1281>.
- Harichandan, A.B. 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(2), pp. 154–171. Available at: <https://doi.org/10.1016/j.ijheatfluidflow.2010.01.007>.
- Hishikar, P. et al. (2022) 'Analysis of flow characteristics of two circular cylinders in cross-flow with varying Reynolds number: a review', *Journal of Thermal Analysis and Calorimetry*, 147(10), pp. 5549–5574. Available at: <https://doi.org/10.1007/s10973-021-10933-w>.
- Jiang, H. (2021) 'Three-dimensional wake transition of a diamond-shaped cylinder', *Journal of Fluid Mechanics*, 918, p. A35. Available at: <https://doi.org/10.1017/jfm.2021.354>.
- Lekkala, M.R. et al. (2022) 'Recent advances in understanding the flow over bluff bodies with different geometries at moderate Reynolds numbers', *Ocean Engineering*, 261, p. 111611. Available at: <https://doi.org/10.1016/j.oceaneng.2022.111611>.
- Liu, Z. and Kopp, G.A. (2011) 'High-resolution vortex particle simulations of flows around rectangular cylinders', *Computers & Fluids*, 40(1), pp. 2–11. Available at: <https://doi.org/10.1016/j.compfluid.2010.07.011>.
- Mathupriya, P. et al. (2018) 'Numerical Study of Flow Characteristics Around Confined Cylinder using OpenFOAM', *International Journal of Engineering and Technology*, 7(4.35), pp. 617–623. Available at: <https://doi.org/10.14419/ijet.v7i4.35.22925>.
- Mehdipour, I. et al. (2022) 'Comprehensive experimental study on bluff body shapes for vortex-induced vibration piezoelectric energy harvesting mechanisms', *Energy Conversion and Management: X*, 13, p. 100174. Available at: <https://doi.org/10.1016/j.ecmx.2021.100174>.
- Norberg, C. (2003) 'Fluctuating lift on a circular cylinder: review and new measurements', *Journal of Fluids and Structures*, 17(1), pp. 57–96. Available at: [https://doi.org/10.1016/S0889-9746\(02\)00099-3](https://doi.org/10.1016/S0889-9746(02)00099-3).
-

- Onel, H.C. *et al.* (2025) 'Wake flow characteristics of diamond-shaped cylinder with splitter plate', *Ocean Engineering*, 326, p. 120933. Available at: <https://doi.org/10.1016/j.oceaneng.2025.120933>.
- Roache, P.J. (1994) 'Perspective: A Method for Uniform Reporting of Grid Refinement Studies', *Journal of Fluids Engineering*, 116(3), pp. 405–413. Available at: <https://doi.org/10.1115/1.2910291>.
- Singh, I.S. *et al.* (2025) 'Assessment of Existing OpenFOAM Solvers for Incompressible Flow over a 2D Prism', *CFD Letters*, 17(10), pp. 184–197. Available at: <https://doi.org/10.37934/cfdl.17.10.184197>.
- Singh, R. *et al.* (2025) 'Computational fluid dynamics of flow over a cylindrical tube for steady and transient flow conditions', *International Journal on Interactive Design and Manufacturing (IJIDeM)*, 19(4), pp. 2925–2935. Available at: <https://doi.org/10.1007/s12008-024-01970-4>.
- Soyler, M. *et al.* (2022) 'Experimental and numerical investigation of flow structure of grooved cylinder', *Ocean Engineering*, 264, p. 112478. Available at: <https://doi.org/10.1016/j.oceaneng.2022.112478>.
- Versteeg, H.K. and Malalasekera, W. (2011) *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. United Kingdom: Pearson Education, Limited. [Print].
- Wang, J., Fan, D. and Lin, K. (2020) 'A review on flow-induced vibration of offshore circular cylinders', *Journal of Hydrodynamics*, 32(3), pp. 415–440. Available at: <https://doi.org/10.1007/s42241-020-0032-2>.
- Yadav, P.K. *et al.* (2021) 'Flow around a diamond-section cylinder at low Reynolds numbers', *Physics of Fluids*, 33(5), p. 053611. Available at: <https://doi.org/10.1063/5.0049811>.

This page is blank on purpose