Journal of Solid and Fluid Mechanics (JSFM), 15(5): 13-17, 2025

Shahrood University of Technology

Journal of Solid and Fluid Mechanics (JSFM)

DOI: 10.22044/jsfm.2025.16124.3966

Manufacturing Engineering of Iran

Numerical simulation of flow around an immersed cylinder in turbulence generated by networks of upstream cylinders

Sarallah Abbasi[1*], Mohammad Matin Aghadavodi[2]

[1] Assoc. Prof., Mech. Eng., Arak University of Technology, Arak, Iran
[2] BSc. Student, Mech. Eng., Arak University of Technology, Arak, Iran

*Corresponding author: s_abbasi@arakut.ac.ir

Received:04/28/2025 Revised: 06/15/2025 Accepted: 07/15/2025

Abstract

This study aims to investigate the effect of incoming flow turbulence around a circular cylinder. To generate turbulence, an array of upstream circular cylinders was modeled, and its influence on the flow behavior around the main cylinder was analyzed. The flow was simulated using a two-dimensional, transient approach with the commercial software ANSYS Fluent, employing the k-ω SST turbulence model at a Reynolds number of 22,000. For validation, the results of the present study were compared with those of previous research, showing good agreement. The findings indicate that when the distance between the turbulence-generating cylinders and the main cylinder is 6D, the highest level of vortex interaction occurs, increasing the turbulence intensity around the main cylinder up to 0.9. Evaluation of the drag and lift coefficients shows that the drag coefficient decreases by approximately 76%, and the lift coefficient increases by nearly 13 times compared to the base case. These validated results highlight the potential of passive turbulence-generating arrays for flow control in bluff body applications and offer insight into drag reduction and mitigation of vortex-induced vibrations.

Keywords: Circular cylinder; numerical simulation; inlet flow turbulence; vortex; turbulence intensity.

1. Introduction

Turbulent flow has been extensively studied, both experimentally and numerically, over the past several decades. However, due to the complexity of inlet flow disturbances, existing investigations remain insufficient. Turbulent flow around a cylinder exhibits highly variable behaviors. Despite its inherent unpredictability, turbulence often follows certain discernible patterns [1-4]. The following is a review of studies related to this subject. Bearman and colleagues investigated the effects of free-stream turbulence on the dynamic flow around solid bodies. Their findings revealed three fundamental mechanisms governing such flows: rapid transition to turbulence, enhanced mixing or vortex formation, and spontaneous distortion of the free shear layer in turbulent streams [5]. Norberg conducted studies on the effect of increased turbulence and observed that higher turbulence levels lead to vortex formation around circular cylinders. This phenomenon results in pressure reduction and increased flow fluctuations, provided that the Reynolds stresses in the shear layers are sufficiently high [6]. In further experimental research, Norberg examined the effects of Reynolds number and turbulence intensity on the flow around circular cylinders. He concluded that low-intensity disturbances increase the pressure forces in the pre-turbulent regime, while in turbulent flow conditions, the opposite behavior is observed [7]. Williamson's results indicate that if a cylinder is elastically mounted or flexible, vortex-induced vibrations (VIV) may cause structural fatigue damage. This is especially relevant in marine environments where incoming flow is significantly more turbulent compared to uniform laboratory conditions. The hydrodynamic characteristics of turbulent inflow differ substantially from those in uniform flows [8]. In a comprehensive review, Williamson and co-authors emphasized the critical role of vortex shedding in triggering various vibration modes. They highlighted the importance of mass-damping interactions and presented the relationship between hydrodynamic forces and vortex dynamics. Their compilation of studies on forced oscillations also indicated that certain vortex modes can trigger free vibrations [9]. Zhou et al. conducted two-dimensional numerical simulations using the k–ω turbulence model to study the flow around circular cylinders. Their analysis of flow structures and velocity fields revealed that flow regime characteristics are strongly influenced by vortex shedding behavior [10]. Olive and colleagues investigated the effect of turbulence intensity on flow dynamics. They found that at low turbulence levels, the influence of turbulence on flow behavior is poorly understood, and emphasized the necessity of further studies under high turbulence intensity conditions [11]. Sadok et al. experimentally studied flow around a circular cylinder, with a focus on the effect of free-stream turbulence intensity (Ti) on the separating shear layer. Their results indicated that turbulence reinforces the shear layer's instability, accelerates vortex breakdown, and reduces the formation length of large vortices. Ultimately, they developed a model demonstrating that, qualitatively, increasing turbulence intensity produces effects on shear layers similar to those caused by increasing Reynolds number [12]. Tamura and Kua examined the effect of shear flow on a rounded-corner square cylinder at critical Reynolds numbers. Their findings showed that shear flows affect aerodynamic characteristics differently across subcritical and supercritical regimes. Generally, shear effects are less pronounced in the supercritical regime. They also found that the mean and fluctuating pressures were similar across different inflow profiles, while the mean lift direction reversed between subcritical and supercritical conditions [13]. Ji et al. experimentally investigated the hydrodynamic behavior of steel tube bundles subjected to oscillatory flows induced by vessel motion. They observed a strong correlation between local and convective accelerations in the inflow and the resulting hydrodynamic forces. Vortex-induced chaotic behavior was clearly evident in their results [14]. Zhou and colleagues used Direct Numerical Simulation (DNS) to analyze turbulent wake flows behind two side-by-side square cylinders with varying gap ratios. Their work identified two primary turbulent regimes: "weak-gap" and "strong-gap" flows. Only in the strong-gap regime were they able to observe well-defined velocity fluctuations [15]. In another DNS study, Zhou and Tao compared two layouts of multi-cylinder arrays: a conventional uniform layout and a multiscale configuration with varying inter-cylinder spacing. They found that although the pressure drop was similar in both cases, the downstream flow was significantly more turbulent in the multiscale layout, while the conventional layout produced a more uniform wake [16]. Zhang et al. numerically studied vortex-induced forces and drag/lift coefficients in turbulent flows. Their results showed that these coefficients are more sensitive to increased flow velocity under turbulent conditions. The randomness of velocity and its coupling with out-of-plane oscillations were cited as the primary contributors to this behavior [17]. Chandran and collaborators performed numerical analyses of drag coefficients on thick three-dimensional bluff bodies, including flat plates, cylinders, triangular prisms, and semi-circular profiles in pipe flow. Their findings suggested that drag decreases with increasing body thickness. Moreover, their results indicated that 2D and 3D simulations yield significantly different drag predictions, and that body length in confined domains strongly influences the drag coefficient [18]. Song et al. carried out 3D DNS simulations to investigate the influence of inlet turbulence on the wake dynamics of a primary cylinder at Reynolds number 3900. Turbulence was generated by a grid of upstream cylinders. Their findings showed that compared to uniform inflow, turbulence significantly amplified hydrodynamic forces: time-averaged drag increased by 4.29%, and the lift coefficient increased by as much as 500% [19]. This study, by focusing on the quantification of drag and lift coefficients, separation angle, vortex intensity, and pressure fluctuations under realistic turbulent inflow conditions, provides valuable insights into flow control strategies and vortex-related phenomena in bluff-body aerodynamics.

2. Numerical solution method

In this study, an unsteady two-dimensional simulation of flow around a circular cylinder was conducted using the control volume method and the pressure-based solver in ANSYS Fluent. The k-ω SST turbulence model was employed to accurately estimate eddy viscosity. This model is widely favored in the simulation of turbulent heat transfer and flow in industrial applications due to its robustness, computational efficiency, and reliable accuracy over a broad range of turbulent regimes [20]. At the inlet boundary, a uniform velocity of 22 m/s corresponding to a Reynolds number of 22,000 was specified. No-slip wall boundary conditions were applied at the upper and lower domain boundaries, while a zero-gauge static pressure was defined at the outlet boundary. For coupling pressure and velocity fields, the SIMPLE algorithm was utilized. The spatial discretization of the momentum, turbulent kinetic energy (k), and specific dissipation rate (ω) equations was performed using the Second Order Upwind scheme. Time discretization was carried out with the Second Order Implicit method, while the PRESTO! scheme was used for pressure interpolation. The transient simulation was run with a time step of 0.0001 seconds, and 30 inner iterations per time step were carried out to ensure convergence. The total simulation time was set to 0.5 seconds. Convergence was achieved when the residuals of all governing equations dropped below 10^{-5}. To verify mesh independence, simulations were conducted on three different grid densities. The simulation proceeded until a stable and periodic vortex shedding pattern was established. Aerodynamic coefficients were quantitatively extracted and subsequently validated against experimental benchmark data.

3. Boundary conditions

The simulated geometry consists of 20 rigid cylinders upstream and one rigid cylinder downstream. According to Figure 1, in the reference case, the distance from the upstream cylinders to the single cylinder is 9D, and other models have been investigated with respect to the reference case [19].

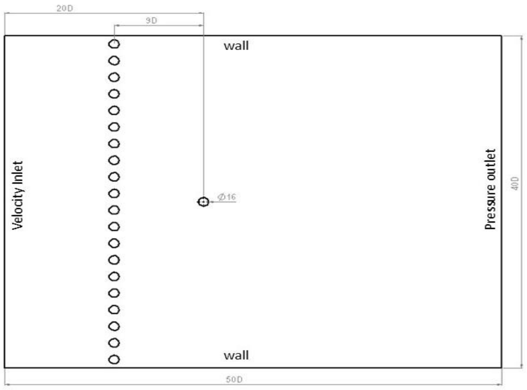

Figure 1. Geometry specifications and boundary conditions of the problem

Results and Discussion

- **Aerodynamic coefficients**

The mean values of the lift and drag coefficients are presented in Table 1. Examination of the results shows that, when the spacing between the turbulence-generating cylinders and the main downstream cylinder is 6D, the mean drag coefficient decreases by approximately 76%, while the mean lift coefficient increases by more than 13 times compared to the baseline case. These variations in drag and lift coefficients are expressed relative to the 0D baseline, in which no upstream cylinders are present. In this 0D case, the drag coefficient is considerably high, whereas the lift coefficient remains nearly zero due to the symmetry of the incoming flow. At a 6D spacing, the introduction o turbulence and flow asymmetry leads to significant instabilities in the wake, resulting in a substantial increase in the lift force, with the lift coefficient exceeding 13 times its baseline value.

Table 1. It shows the average values of lift and drag coefficients in the case without network cylinders and in the three cases 6D, 9D, and 12D.

	No Grid	6D Grid	9D Grid	12D Grid
Drag coefficient	4.01778 3	0.939977	0.96682	0.702334
Lift coefficient	-0.00502	-0.07785	0.02021	-0.03289

- **Flow separation**

In the present study, the zero-degree angle corresponds to the front stagnation point of the main cylinder, while 180 degrees represents the rear side. The flow separation angle was determined using the wall shear stress distribution diagrams shown in Figure 2, for four different configurations: without upstream cylinders (0D spacing), and with spacings of 6D, 9D, and 12D between the upstream array and the main cylinder. In this analysis, the separation point is identified where the wall shear stress approaches zero and changes sign, indicating flow reversal. The results show that in the baseline case (0D spacing), the separation angle is approximately 82 degrees. When upstream cylinders are introduced, the separation angle increases to approximately 93° (12D), 98° (9D), and 110° (6D), respectively. This trend clearly demonstrates the direct effect of incoming turbulence on delaying flow separation from the cylinder surface. In essence, increased turbulence enhances the energy within the boundary layer, enabling it to resist adverse pressure gradients for a longer distance before detaching. Therefore, the presence of appropriately spaced upstream cylinders can serve as an effective passive flow control mechanism to delay separation, offering potential benefits for aerodynamic optimization and drag reduction.

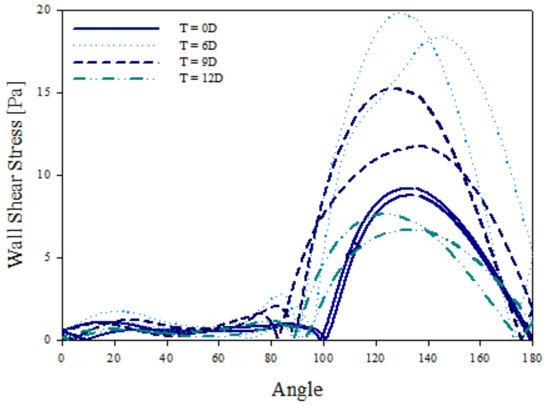

Figure 2. Shear stress on the cylinder wall in different states

In Figure 3, the velocity profiles of the boundary layer at the 90-degree location on the cylinder surface are presented. This angular position is selected as a key region for analyzing flow separation, as it is where the most significant boundary layer variations occur. The plot indicates that, with decreasing spacing between the upstream turbulence-generating cylinders—particularly at 6D—the boundary layer becomes thicker, and the velocity gradient near the wall decreases. This behavior is a clear indicator of weakened momentum transfer and proximity to flow separation. In contrast, at a spacing of 12D, the boundary layer remains thinner, and the velocity increases more rapidly across the wall-normal direction. This suggests a delayed onset of flow separation, attributable to improved boundary layer stability.

These results highlight the local stabilizing or destabilizing effects of incoming turbulence on the boundary layer behavior near the mid-span of the cylinder. The observed trends align well with the earlier analysis of vortex structures and aerodynamic force coefficients, reinforcing the link between turbulence intensity and flow separation dynamics.faster, indicating a delay in the onset of flow separation. These results well demonstrate the effect of the inlet turbulence on the stability or local instability of the flow in the middle region of the cylinder and are in agreement with the analysis of vortices and aerodynamic coefficients.

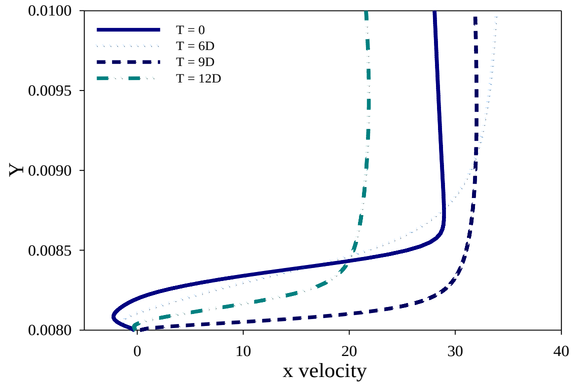

Figure 3. Velocity profile in different states of network cylinders

4. Conclusions

In this study, the effect of upstream-induced turbulence generated by an array of cylinders on the flow behavior around a single downstream cylinder was numerically investigated at a Reynolds number of 22,000. The upstream cylinders were positioned at varying streamwise distances from the main cylinder. Based on the simulation results, the following key conclusions were drawn:

- As the spacing between the upstream and downstream cylinders decreased from 12D to 6D, the intensity of incoming turbulence increased significantly, leading to major modifications in the wake flow structure around the main cylinder.
- The flow separation angle increased from approximately 82° in the reference case (0D spacing) to about 110° at a spacing of 6D. This demonstrates a delay in separation due to increased energy transfer into the boundary layer under the influence of upstream turbulence.
- At 6D spacing, a notable reduction in the drag coefficient (Cd) and a significant increase in the lift coefficient (Cl) were observed. This reflects a less stable flow and intensified vortex shedding oscillations.
- Vorticity contours and streamline visualizations revealed that incoming turbulence enhances the size and intensity of the wake vortices, especially at smaller spacing distances.
- The numerical results showed good agreement with available experimental data, validating the simulation methodology. Additionally, the mesh and time-step independence studies confirmed the stability and accuracy of the numerical solution.

5. References

[1] Yadegari M, Bak Khoshnevis A, Boloki M (2023) An experimental investigation of the effects of helical strakes on the characteristics of the wake around the circular cylinder. *Iran J Sci Technol Trans Mech Eng* 47(1): 67–80.

[2] Hatami, S., et al. "Experimental Investigation of Drag Coefficient in Staggered Circular Cylinders at Different Angles." *J of Mech Eng* (Biquarterly), vol. 31, no. 6, 2023, pp. 3-12.

[3] Bak Khoshnevis A, Boloki M, Yadegari M (2020) The investigation of the effect of the helical strakes' height on the cylindrical wake. *J Solid Fluid Mech* 10(1): 223–236.

[4] Yadegari M, Bak Khoshnevis A (2021) Numerical and experimental study of characteristics of the wake produced behind an elliptic cylinder with trip wires. *Iran J Sci Technol Trans Mech Eng* 45: 265–285.

[5] Bearman P, Morel T (1983) Effect of free stream turbulence on the flow around bluff bodies. *Prog Aerosp Sci* 20(2–3): 97–123.

[6] Norberg C (1986) Interaction between freestream turbulence and vortex shedding for a single tube in cross-flow. *J Wind Eng Ind Aerodyn* 23: 501–514.

[7] Norberg C, Sundén B (1987) Turbulence and Reynolds number effects on the flow and fluid forces on a single cylinder in cross flow. *J Fluids Struct* 1(3): 337–357.

[8] Williamson CHK (1996) Vortex dynamics in the cylinder wake. *Annu Rev Fluid Mech* 28: 477–539.

[9] Williamson CHK, Govardhan R (2004) Vortex-induced vibrations. *Annu Rev Fluid Mech* 36: 413–455.

[10] n H, Cheng L, Zhao M (2009) Steady streaming around a circular cylinder in an oscillatory flow. *Ocean Eng* 36(14): 1089–1097.

[11] McAuliffe BR, Yaras MI (2010) Transition mechanisms in separation bubbles under low-and elevated-freestream turbulence. *In: Proc. ASME Turbo Expo*.

[12] Khabbouchi I, Fellouah H, Ferchichi M, Guellouz MS (2014) Effects of free-stream turbulence and Reynolds number on the separated shear layer from a circular cylinder. *J Wind Eng Ind Aerodyn* 135: 46–56.

[13] Cao Y, Tamura T (2018) Aerodynamic characteristics of a rounded-corner square cylinder in shear flow at subcritical and supercritical Reynolds numbers. *J Fluids Struct* 82: 473–491.

[14] Ji C, Cui Y, Xu D, Yang X, Srinil N (2019) Vortex-induced vibrations of dual-step cylinders with different diameter ratios in laminar flows. *Phys Fluids* 31(7): 073602.

[15]Zhou Y, Nagata K, Sakai Y, Watanabe T (2019) Extreme events and non-Kolmogorov-5/3 spectra in turbulent flows behind two side-by-side square cylinders. *Phys Rev Fluids* 4(12): 124602.

[16] Tao S, Zhou Y (2019) Turbulent flows around side-by-side cylinders with regular and multiscale arrangements. *Phys Rev Fluids* 4(12): 124602.

[17] Zhang M, Fu S, Liu C, Ren H, Xu Y (2021) Experimental investigation on vortex-induced force of a steel catenary riser under in-plane vessel motion. *Mar Struct* 78: 102882.

[18] Chandran J, KR B (2022) Numerical prediction of the drag coefficient of bluff bodies in three-dimensional pipe flow. *J Comput Appl Res Mech Eng* 12(1): 13–29.

[19] Song B, et al. (2022) Direct numerical simulation of flow over a cylinder immersed in the grid-generated turbulence. *Phys Fluids* 34(1): 015109.

[20] Mansouri Z, Yadegari M, Bak Khoshnevis A (2023) Numerical investigation of the effects of installing four trip wires with different diameters on the mean and fluctuation

velocities and characteristics of the wake around the circular cylinder. *J Braz Soc Mech Sci Eng* 45: 459.
[21] Mansouri Z, Yadegari M, Bak Khoshnevis A (2023) Effect of trip wire on Reynolds shear stress and fluctuating velocities in circular cylinder. *J Mech Eng* 32: 31–45.
[22] Sadeghi M, Yadegari M, Khoshnevis AB (2024) Numerical investigation of the flow characteristics around two sequential cylinders with circular and square cross-sections. *J Mar Sci Technol* 29(2): 315–332.