Numerical Investigation Aerodynamic Characteristic Installation I-65° Cylinder Type Upstream Bluffbody as Airflow Passive Control

G Sakti¹, B G Cahyo¹, A Wulansari¹, A Regia¹, I A Dharma²
¹ Politeknik Penerbangan Surabaya, Jalan Jemur Andayani I/73 Wonocolo, Surabaya 60236, Indonesia
² Graduate students in Department of Mechanical Engineering, Institut Teknologi Sepuluh Nopember, Kampus Keputih-Sukolilo, Jl. Arif Rahman Hakim, Surabaya, 60111, Indonesia

gunawan.sakti@poltekbangsby.ac.id

Abstract. This report is the basic research that focuses on efforts to reduce the drag force of a cylindrical pipe by placing an interfering cylinder in the area of the incoming flow direction. The aerodynamic behavior of the central cylinder and its disturbances were modeled in 2D are discretized in laminar flow by Finite Volume Method using Ansys Fluent®. Efforts to reduce the drag force are carried out with the main cylinder diameter \(D=60\) mm and the interfering cylinder type I-65° with diameter \(d/D = 0.125\). The distance between the center points of the two cylinders being \(s/D=1.4\) and Reynold number \(Re = 5.3 \times 10^4\) at a speed of \(U_z=14\) m/s. Numerical simulation using variations of turbulent models \(k\)-epsilon (2eq), \(k\)-omega (2eq), and \(transition\ k-kl-omega\ (3eq)\). The results of this research can show better aerodynamic performance. Placing the cylinder I-65° in tandem can reduce the average drag force coefficient by 68% at 700-800 timesteps. In contrast, the average lift coefficient decreased by 13% at the same timestep. The results were obtained with \(transition\ k-kl-omega\ (3eq)\) turbulence models that have been validated and able to approach the referenced experimental data.

1. Introduction

The flow of air flowing in a cylinder has been carried out by extensive research efforts in various engineering sciences[1][2][3]. Various literature[4][5][6] indicates that this flow pattern has many technical applications and is still one of the leading research scourges, especially in aerodynamics. For example, one can be applied to an optimized design heat exchanger, VIV harvesting energy[4][8][9] and the arrangement of hydraulic hoses on a non-retractable aircraft's shock strut landing gear. The cross-section of the heat exchanger capillary tube has identical cross-sectional geometry to a tandem circular cylinder. With varying distances and diameters, it can reduce the pressure drag and provide a hydrodynamic advantage.

Experts say a bluff body has a considerable dynamic drag due to the flow disjunction passing over the cylinder[10]. Two methods to decrease the dynamic \(Cd\) on a bluff body are passive and active control. Active control means controlling the flow by supplying external energy through various means such as acoustic excitation or jet blowing. The second is passive control, namely by modifying the shape of the body or by installing additional devices such as control rods or rough body surfaces [11][12]. The best method of reducing drag is active control, but it is challenging and expensive to implement because it requires extra energy[13].

Research with a direct numerical simulation approach was carried out by [13] with investigating the flow through the slotted cylinder at a low Reynolds Number \(Re=100\). Slots on the cylinder affect...
the boundary layer separation, vortex formation position, recirculation region length, and wake width, which are determined by the variation of the slot. The result is a reduction in the drag force of 1.7% and a lift force of 17%. [14]'s scientific report experimentally tested a circular cylinder with speed variations of 4 m/s to 36 m/s at steady flow conditions, at Reynolds number 8.3x10^4<Re<7.5x10^5 and at the turbulence intensity of 0% to 12%. The results show that the turbulence intensity significantly affects the spectral density distribution of the drag coefficient. The author [1] makes an effort to reduce the drag force on a circular cylinder by using a cylinder type I-65° which is placed in the incoming flow direction. The cutting angle of this cylinder starts from θs = 0⁰, 10⁰, 20⁰, 30⁰, 45⁰, 53⁰, dan 65⁰ with a space of S/d=1.375 at Re=5.3x10^4. The results show that the placement of the type I cylinder can reduce the drag force by 52% with an optimal cutting angle of 65°.

Based on this background and previous literature study, an effort is made to reduce the drag force by placing a cylinder type I-65°, tandem to the main cylinder as passive flow control. The distance between the bluff bodies is fixed at s/D= 1.375, with the main cylinder diameter D=60 mm, Reynolds number Re=5.3x10^4 at the freestream airflow velocity $U_∞ = 14$ m/s. The diameter of the bluff body cut I-65° relative to the main cylinder is determined at d/D=0.125. The 2D tandem cylinders model is discretized in laminar flow by Finite Volume Method using Ansys Fluent®. Several turbulence models such as transition k-kl omega, standard k-ε, standard k-ω, RNG k-ε, SST k-ω were tested with the compared output parameters the pressure distribution on the main cylinder wall.

2. Numerical method

The aerodynamic characteristics of the cylinder circular and its disturbances were modeled in 2D are discretized in laminar flow by Finite Volume Method using Ansys Fluent®. The solution in this simulation will use standard discretization for pressure, second-order upwind for momentum, Pressure Velocity Coupling is determined to be COUPLE. The convergence criteria used in the iteration process are $10^{-6}$, which means the calculation or running process will continue to iterate to achieve results with an error rate of $10^{-6}$ for residual energy.

2.1. Geometry & Boundary Condition

The geometry is in the form of a conventional circular cylinder and a cylinder with a disturbance I-65° which needs to be made of auxiliary mesh lines to facilitate mesh control, especially in the area near the walls of the main cylinder and cylinder I-65° as shown in Figure 1 and Figure 2. The downstream area is extended 16 times the circular cylinder diameter to prevent backflow and fully developed flow. At the same time, the upstream area is extended ten times the diameter of the circular cylinder. In addition, each wall is built with a distance of 5 times the cylinder diameter, respectively, to reduce the blockage effect.

![Figure 1. Boundary conditions of the single circular cylinder with meshing lines make adjusting the face sizing mesh easier.](image-url)
2.2. Mesh and grid independence study

As a result of the influence of the wall function, a surface meshing with a quadrilateral face meshing type was built with grading in the vertical and horizontal directions with a tighter mesh distribution on the cylinder wall, shown in figure 3. In Figure 3, it is explained that the magnification of A and the magnification of B shows that the mesh used has vertical and horizontal gradations and the higher the density in the area near the circular cylinder wall.

The inflation, first layer thickness $\Delta s$, for the desired $y^+$ is also determined and set on the Savonius turbine and the cylinder circular wall. Finally, it was calculated using the following equation.

$$\Delta s = \frac{y^+ + \nu}{0.013 \times U_f} \sqrt{Re}^n$$

(1)

$U_f$ is expressed as frictional velocity, calculated based on shear stress rate at wall $\tau_w$. Table 1 shows the results of grid independence represent that all the meshes built can reach a value of $y^+ < 1$, with the
lowest value being mesh C with $y^+ = 0.45$ and 180000 nodes. The simulation results show that $y^+ = 0.53$ is the highest value among the built mesh structures for mesh A and B. The maximum skewness value of the constructed mesh as a whole can reach a value of less than 0.9, where the highest maximum skewness is obtained by mesh C with a value of 0.77. In contrast, the lowest maximum skewness value is obtained by mesh B with a value of 0.62.

**Table 1.** Measuring a parameter of mesh quality in single cylinder simulation with $Re= 5.3 \times 10^4$

| Meshes | Nodes   | $y^+$ | Max. Skewness | Avg. Aspect Ratio |
|--------|---------|-------|---------------|-------------------|
| Mesh A | 73,163  | 0.53  | 0.67          | 2.21              |
| Mesh B | 116,439 | 0.53  | 0.62          | 1.75              |
| Mesh C | 180,049 | 0.45  | 0.77          | 1.20              |
| Mesh D | 240,007 | 0.45  | 0.70          | 1.13              |
| Mesh F | 317,100 | 0.46  | 0.75          | 1.12              |

Figure 4 shows the evolution of the average pressure coefficient for each mesh A, mesh B, mesh C, mesh D, and mesh E. It can also be seen that mesh C to mesh E does not significantly change the results with increasing created nodes. So this strengthens the results of the grid analysis to maintain mesh C as the minimum configuration for use in the following research step.

![Figure 4. The average coefficient of pressure value for each mesh configuration](image)

### 3. Result and Discussion

#### 3.1. Turbulence model consideration

The results of this circular cylinder simulation for each variation of the turbulence model are shown in Figure 5. The figure, qualitatively, shows that the turbulence model that can approach the experimental data is the *k-kl omega transition*. The *k-kl omega transition* trend is also near other experimental results [1,11]. So this configuration will be maintained for further analysis in this study.
Figure 5. Turbulence Model analyzed the single-cylinder model at $Re=5.3\times10^4$ with freestream airflow $U_\infty=14$ m/s for distribution of pressure coefficient around cylinder wall.

3.2. Pressure contour consideration
The comparison of the pressure contour between a single-cylinder and an I-65° cylinder installation upstream of the main cylinder can be shown in Figure 6. The results of the qualitative analysis of this pressure contour are the narrowing of the low-pressure and the high-pressure area in the downstream and upstream of the main cylinder, respectively. Therefore, the placement of cylinder I-65° as passive control of the upstream flow of the main cylinder reduces the pressure drag.

Figure 6. Pressure contour of single cylinder (left) and cylinder with installation of cylinder type I-65° (right) for $Re = 5.3 \times 10^4$, with center to center distance $S/D = 1.375$. (pascal)

Figure 7 shows a streamlined overlay of flow passing through a single-cylinder and a cylinder with a bluff body placement of an I-65° type cylinder. With the placement of the cylinder, I-65° will release the shear layer, which will experience reattachment on the surface wall of the cylinder, so that the flow separation will be slightly delayed backward, causing the wake area in the downstream area of low pressure to narrow. In addition, the area between the main cylinder and cylinder I-65° will experience a decrease in pressure. As a result, the pressure difference between the downstream area and the upstream area of the main cylinder will decrease so that the pressure drag will decrease.
3.3. Velocity contour consideration

Figure 8. shows a streamlined overlay of airflow on the velocity contour for the two configurations tested at \( Re = 5.3 \times 10^4 \). The blue area indicates an area with low velocity, and a streamlined direction and negative velocity indicate backflow. This velocity contour consideration reinforces the delayed separation phenomenon due to the type I-65° cylinder upstream of the main cylinder. In the reattachment area, the air velocity decreases, until the laminar sublayer flow, then the air velocity increases again when separated from the main cylinder wall.

3.4. Pressure coefficient distribution consideration

Figure 9 shows, the reattachment points for the tandem cylinder configuration are at degrees = 35° and = 325°, respectively. A constant pressure coefficient distribution determines the separation point because the laminar boundary layer has been separated from the cylindrical surface to form a turbulence boundary layer. From this graph, the separation point for a single-cylinder occurs at positions = 85° and = 275°, while for tandem cylinder type I-65°, the separation point occurs at positions = 120° and = 240°. So, it can be concluded that the placement interfering cylinder type I-65° with a diameter of \( d/D = 0.125 \) can delay the separation point each as far as 35° towards the downstream. In addition, this graph can also show a decrease in the difference in the distribution value of the pressure coefficient between the upstream cylinder and the downstream cylinder. So that the placement of the cylinder type I-65° significantly reduces the pressure drag of the main cylinder.
3.5. Coefficient of drag consideration

Figure 9, qualitatively, shows a decrease in the coefficient of drag on a tandem cylinder configuration compared to a single cylinder. At a timestep of less than 100, the force values of the two configurations of the cylinder arrangement show fluctuations. The next 100th timestep is slightly stable and starts to be constant after the 200th timestep. From this figure, it can be concluded that the configuration of the I-65° cylinder in front of the main cylinder can decrease the $Cd$ compared to a single-cylinder configuration at Reynolds number $Re = 5.3 \times 10^4$.

3.6. Coefficient of lift consideration

Figure 11 shows the comparison of the coefficient of lift graph for single cylinder and tandem cylinder configurations as qualitative data to analyze the evolution of lift force on cylinders with cylinder
installations of type I-65°. This figure indicates that the placement of the bluff body cut I-65° also reduces the lift coefficient in a single-cylinder configuration. The fluctuations occur at a timestep of less than 400. Meanwhile, there are small fluctuations for the rest, and it becomes more stable as the timestep increases. The value of the coefficient of the lift force itself is minimal because it only shows a value of one-hundredth of a thousand, so with the consideration of uncertainty, it can be said that there is no lift force or a value of 0. This can be agreed upon considering that the shape of the cylinder being tested is symmetrical.

Figure 11. Graph of the evolution of the coefficient of lift as a function of timestep for single cylinders and tandem cylinders at \( Re = 5.3 \times 10^4 \).

Table 2 shows the transient simulation results for each drag coefficient and lift coefficient at \( Re = 5.3 \times 10^4 \). The average value of the \( Cd \) of the cylinder tandem with the bluff body cut I-65° in the timestep range 0-600 shows a decrease of 67.7% compared to a single cylinder. It can also be observed that the single-cylinder drag coefficient has a more stable value at the 400th timestep. In the simulation tandem cylinder, I-65° in front of the main cylinder, the drag coefficient shows a stable value since the 200th timestep. The decrease in the value of \( Cd \) is determined to be 68.8% if it is calculated when the average value of each 100th timestep interval shows a stable number. So, it is concluded that with the installation of a cylinder type I-65° upstream, the central cylinder successfully reduces the coefficient of drag lower than the single-cylinder configuration.

Table 2. The value of the coefficient of drag and coefficient of lift for simulation of single-cylinder and tandem cylinder type I-65° at \( Re= 5.3 \times 10^4 \).

| time step | \( Cd \) Avg. | \( Cl \) Avg. | \( \Delta \) | \( \Delta \) |
|-----------|---------------|---------------|--------------|--------------|
| Single Cylinder | Tandem Cylinder I-65 | Single Cylinder | Tandem Cylinder I-65 | Single Cylinder | Tandem Cylinder I-65 |
| 0-600 | 0.106 | 0.035 | 67.075 | 0.00018744 | -0.00004830 | -25.77 |
| 0-100 | 0.103 | 0.046 | 55.706 | 0.00057720 | -0.00038207 | -66.19 |
| 100-200 | 0.107 | 0.033 | 68.832 | 0.00032285 | -0.00001867 | -5.78 |
| 200-300 | 0.106 | 0.034 | 68.446 | 0.00027434 | 0.0000273 | 1.00 |
| 400-500 | 0.107 | 0.034 | 68.395 | 0.00010829 | -0.00000091 | -0.84 |
| 500-600 | 0.107 | 0.034 | 68.36 | 0.00010970 | 0.00000027 | 0.24 |
| 600-700 | 0.107 | 0.034 | 68.352 | 0.00010570 | 0.00000025 | 0.24 |
| 700-800 | 0.107 | 0.034 | 68.37 | 0.00009550 | 0.00000012 | 0.13 |


4. Conclusion
The simulation and analysis on the placement of the bluff-body cut I-65° upstream of the main cylinder can be judged that the numerical method used to measure and display the pressure and velocity contours as a comparison between the main cylinder and the I-65° cylinder is valid and verified. Furthermore, qualitative and quantitative data analysis of the drag and lift coefficient can prove a decrease in the drag coefficient. For evidence, the value of $Cd$ decreased by 68.8% due to the installation of cylinder type I-65° at a distance of $S/D = 1.375$. Furthermore, the distribution of the coefficient of pressure on the surface of the main cylinder wall can show the location of the flow separation in single cylinder and tandem cylinder configurations. Thus, the cylinder placement increased pressure in the downstream area and visually decreased the upstream pressure area. Therefore, the difference between the two is more negligible, the pressure drags decreases.

5. References
[1] Triyogi Y, Suprayogi D and Spirda E 2009 Reducing the drag on a circular cylinder by upstream installation of an I-type bluff body as passive control Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science 223 2291–6
[2] Rashad A M, Khan W A, EL-Kabeir S M M and EL-Hakiem A M A 2019 Mixed convective flow of micropolar nanofluid across a horizontal cylinder in saturated porous medium Applied Sciences (Switzerland) 9
[3] Lai J, Sun L, Yang J, Li P, Xi Z, Gao L, and He C 2018 Experimental and numerical analyses on flow-induced vibration of a nuclear engineering test reactor internal Nuclear Engineering and Design 340
[4] Zhang X 2019 Flow control over a circular cylinder using virtual moving surface boundary-layer control Experiments in Fluids 60
[5] Ren Q 2018 Flow past a rotating circular cylinder with superhydrophobic surfaces Acta Mechanica 229 3613–27
[6] Saisanthosh I, Arunkumar K, Ajithkumar R and Srikrishnan A R 2017 Combined Effect of Surface Roughness and Wake Splitter Plate on the Aerodynamic Characteristics of a Circular Cylinder IOP Conference Series: Materials Science and Engineering vol 234
[7] Zhang L B, Dai H L, Abdelkefi A and Wang L 2019 Experimental investigation of aerodynamic energy harvester with different interference cylinder cross-sections Energy 167 970–81
[8] Zou Q, Ding L, Wang H, Wang J and Zhang L 2019 Two-degree-of-freedom flow-induced vibration of a rotating cylinder Ocean Engineering 191
[9] Miao Y, Jiang Y, Zhang X and Li K 2021 Transfer learning for isolated cylinder vibration induced by vortex shedding IOP Conference Series: Earth and Environmental Science vol 774
[10] Achenbach E 1971 Influence of surface roughness on the cross-flow around a circular cylinder Journal of Fluid Mechanics 46
[11] Lee S-J, Lee S-I and Park C-W 2004 Reducing the drag on a circular cylinder by upstream installation of a small control rod Fluid Dynamics Research 34
[12] Sakamoto H, Tan K and Haniu H 1991 An optimum suppression of fluid forces by controlling a shear layer separated from a square prism Journal of Fluids Engineering, Transactions of the ASME 113
[13] Zhu H, Zhao H and Zhou T 2019 Direct numerical simulation of flow over a slotted cylinder at low Reynolds number Applied Ocean Research 87
[14] Yao J, Shen G and Lou W 2020 Effect of turbulence on aerodynamic coefficient of cylinder IOP Conference Series: Earth and Environmental Science vol 510