EPI International Journal of Engineering
Volume 4, Number 1, February 2021, pp. 45-50
DOI: 10.25042/epi-ije.022021.07

*p*ISSN 2615-5109
*e*ISSN 2621-0541

Fluid Flow Through Serial Parallel Circular Cylinder Arranged in Tandem

Nur Ikhsani[a,*], Nasaruddin Salam[b], Luther Sule[c]

[a]Departmen of Mechanical Engineering, Hasanuddin University, Makassar, Indonesia. Email : mr.nikhsani24@gmail.com
[b]Departmen of Mechanical Engineering, Hasanuddin University, Makassar, Indonesia. Email : nassalam.unhas@yahoo.com
[c]Departmen of Mechanical Engineering, Hasanuddin University, Makassar, Indonesia. Email : luther.sule@yahoo.co.id

Abstract

The Fluid flow through circular cylinders in serial parallel positions arranged in tandem were analyzed computationally and experimentally at nine levels of Reynolds number, Re_D 34,229; 47,921; 61,612; 75,304; 88,996; 102,688; 116,379; 130.071 and 143,763 The variation in the ratio of the distance between the front and rear cylinders is determined as M/D = 0.3, M/D = 0.5, M/D = 0.7, M/D = 0.9, and M/D = 1.1. While the distance between cylinder number 2 and 3 we set constantly and determined as N/D = 5 cm. The results displayed are flow velocity with computational approach validated by flow visualization, computational pressure contour, and drag coefficient through experimental testing. The results showed that the smallest boundary layer thickness was obtained in the model with a distance ratio of M/D = 2.5, using both computational and experimental approaches. The characteristics of the minimum pressure contour and the lowest drag coefficient (C_D) = 0.7572 were also obtained at the ratio of the distance M/D = 0.25 and at upstream speed of 21 m/s.

Keywords: Drag coefficient; flow visualization; FLUENT 6.3.26; tandem serial-parallel circular cylinders; pressure coefficient

1. Introduction

Flow through vertical circular cylinders arranged in tandem is a form often used in structural and transportation engineering. Sharing of round cylindrical applications such as offshore structures, lighthouses, platform and port support structures as well as heat exchangers such as Shell and Tube heat exchangers [1]–[3]. Wind and water loads on a building are some of the main factors that must be considered in the design.

Previous research was conducted to determine the flow phenomenon across a tandem object and its effect on drag force [4]–[6] commonly used geometric bodies are cylindrical, often found inland transport vehicles such as trains and trailers as well as in marine transportation such as barges. The reduction of drag can be obtained by engineering the flow field by adding an object called the inlet fault body, which is placed in front of the main object.

It is known that the wind and water loads on a building in groups are different from single-building structures with the same shape, because the combined disturbance from the flow around the structure in groups shows interesting and unpredictable variations of phenomena [7]. Many researchers have made efforts to reduce friction forces. Some show how to reduce the drag force on a single cylinder or are arranged in tandem by a variety of methods. Research on the reduction of friction using a drag body has been widely applied. Based on Etminan et al. [8]. when the circular cylinder is arranged in tandem in front of the two cylinders are given a T-shaped resistance plate, with a head width of 5 mm and the distance to the cylinder is varied to get the optimal position results, the optimal results are obtained when the variation of cylinder distance with cylinder diameter is in L/D = 1.0 to 1.5.

Salam, et al [9]–[11], conducted a similar study regarding the effect of adding an Inlet Disturbance Body (IDB) in the form of a circular cylinder on the drag acting on a square cylinder in a tandem arrangement through a numerical simulation approach using computational fluid dynamics methods and experimental testing utilizing the subsonic wind tunnel facility. The Reynolds number used is in the range of 30.625 to 96.250. The ratio of IDB diameter and square cylinder diameter (d/D) is 0.08, 0.14, and 0.20, respectively, with the ratio of the distance between the IDB and the square cylinder (L/D) varying from 0.0 to 1.0. The results showed a decrease in the amount of drag coefficient (C_D) and pressure coefficient (C_P) along with the increase in L/D and d/D ratios. The decrease in the highest drag coefficient was 21.5962% and the decrease in pressure coefficient was 14.7059 at L/D = 0.43 and d/D = 0.14. Regards Salam, et al. Also investigated the flow separation that occurs in rectangular cylinders arranged in serial and parallel and found that the smallest flow separation for serial and parallel configurations is at M/D = 0.6.

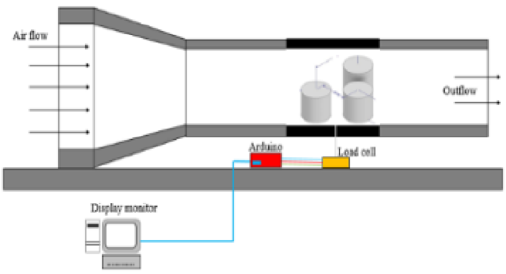
*Corresponding author. Tel.: +62 813-4299-2098
Jalan Poros Malino km. 6 Bontomarannu
Gowa, Indonesia, 92171

Tsutsui and Igarashi [12] investigated the reduction of drag in circular cylinders through the application of a drag rod in front of the object. The results showed that the disturbance diameter and Reynolds number had an effect on the flow pattern. A 63% reduction in resistance was also found. For very blunt objects, such as a flat plate perpendicular to the flow, flow separation occurs at the edges of the plate regardless of the nature of the boundary layer flow [13]–[15].

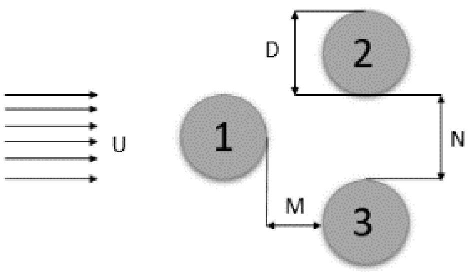
2. Research Methods

The research begins with a numerical simulation using computational fluid dynamic fluent 6.3.26 software before being validated through experimental tests. For numerical simulation, the test model is designed using the Autodesk inventor device and assigned to the computational domain and then through the meshing process using the Gambit device 2.4.6. Numerical simulation is focused on analyzing flow phenomena in the form of velocity trajectories and pressure contours. Details of the model test and computational conditions are shown in Fig. 1 and Table 1.

The experimental test was conducted in a Sub-Sonic Wind Tunnel made by Plint & Partners LTD. Engineers England. The test section has made of transparent acrylic with a thickness of 2.5 cm for flow visualization. The test object was a tandem of circular cylinders as shown in Figure 1. Circular cylinders length, width, and height were equal referred to as square cylinders diameter (D) = 5 cm. The test object was made of acrylic with a thickness of 2 mm. The airflow velocity that entered the wind tunnel (U) was ranged from 5 to 21 m/s. The drag was measured by a *Load Cell* system with a measurement range of 0.00981 Newton to balance the left and right side. The result from the measurement of *Load Cell* toward the object was the drag force (F) of the test object.



(a)



(b)

Figure 1. A parallel series tandem pattern on a circular cylinder body. (a) Position of specimen in the wind tunnel and (b) Computational arrangement condition

Figure 2. Sub-sonic wind tunnel

The circular cylinder size is 5 cm length and 5 cm height. The diameter of the cylinder is constantly 5 cm. Distance between front cylinder and rear cylinder (M) is changed with various distances. Comparison of distances between circular cylinders of the M/D series positions expanded from M/D = 0.3, M/D = 0.5, M/D = 0.7, M/D = 0.9, and M/D = 1.1 in centimeter (cm). Meanwhile, the distance of two vertical circular cylinders 1 and 2 in parallel position (N/D) is set constantly at 5 cm.

Subsequent research was carried out in the laboratory for experimental testing. For experimental testing, it is focused on collecting flow visualization data to validate the results of numerical and drag calculations using a subsonic wind tunnel which is equipped with a measuring instrument for the actual resistance force of the test object with the principle of force balance as shown in Figure 2. The resistance obtained is then written into dimensionless units through the application of equation:

$$C_D = \frac{FD}{\frac{1}{2}\rho U^2 A} \tag{1}$$

3. Results and Discussion

3.1. Flow field

The effect of changes in distance on the characteristics of flow patterns through numerical simulation approaches and visualization of flow of vertical circular cylinders arranged in tandem with M/D variations respectively at M/D = 0.3, M/D = 0.5, M/D = 0.7, M/D = 0.9, and M/D = 1.1, shown in Figure 3. In all comparisons, the M/D distance shows a significant increase in flow velocity when the fluid reaches the side of the specimen. This is due to a decrease in the intensity of the direct impact against the front side of the cylinder which minimizes the loss of flow momentum to move towards the boundary layer. The thicker the boundary layer that is formed, the greater its influence on the amount of drag that works. It can be seen that the model at M/D = 0.3 has a boundary layer that tends to be larger than the other models. Meanwhile, the smallest boundary layer thickness is obtained at M/D = 1.1.

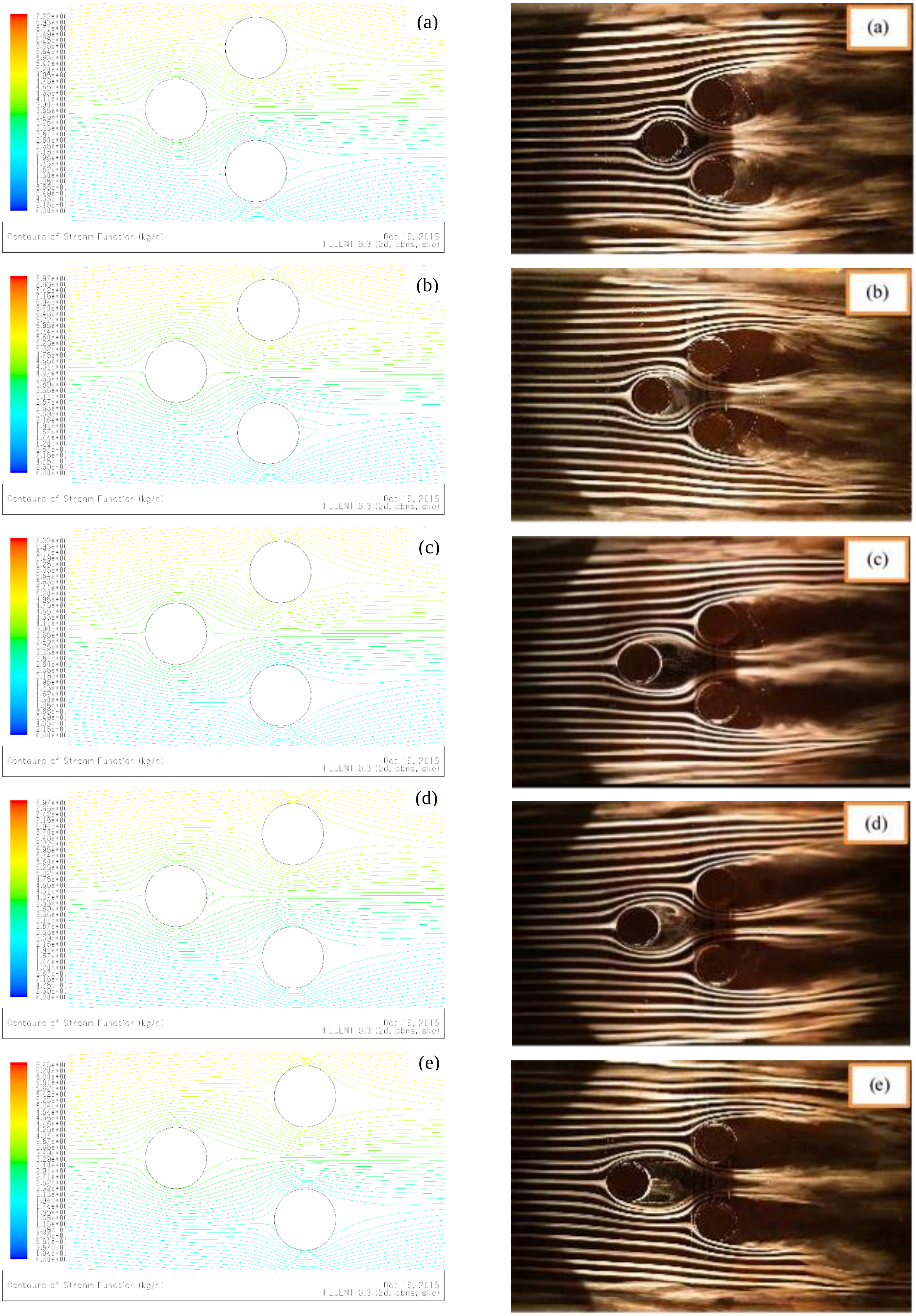


Figure 3. Computational and experimental streamlined comparison of (a) M/D = 0.3, (b) M/D = 0.5, (c) M/D = 0.7, (d) M/D = 0.9, and (e) M/D = 1.1 at U = 11m/s (Re_D = 75.304)

For M/D = 1.1 has the thickest boundary layer than four others. This was caused by the fact that vortex was not damped between serial-parallel circular cylinders and it pushed the flow apart from the cylinder that causes flow separation earlier in the upstream side of parallel cylinders. The almost same phenomenon occurred at M/D = 0.7, namely the vortex was not damped anymore and it rolls the flow apart from the cylinder so that separation of flow occurred earlier

3.2. Drag field

The drag coefficient is obtained through experimental testing at variations of each M/D = 0.3, M/D = 0.5, M/D = 0.7, M/D = 0.9, and M/D = 1.1 with variations in the upstream speed of 5 m/s, 7 m/s, 9 m/s, 11 m/s, 13 m/s, 15 m/s, 17 m/s, 19 m/s and 21 m/s is shown in Table 1. For the upstream speed of 5 m/s, the lowest drag coefficient comes at position M/D = 0,5 is 1.0783 and the highest drag coefficient is obtained at the distance ratio M/D = 0.3 of 1.0923. For the speed of 7 m/s, the lowest drag coefficient is obtained at M/D = 0.7 of 0.9612 and the highest is at M/D = 0.3 of 0.9755. For the speed of 9 m/s, the lowest drag coefficient is obtained at M/D = 0.5 with C_D = 0.8970 and the highest is obtained at M/D = 0.3 with C_D = 0.9083 and almost equal with M/D = 0.9 at C_D = 0.9018. For the highest upstream speed variation, namely 21 m/s, it was also found that the lowest drag coefficient was obtained at M/D = 0.5 at C_D = 0.7572 and the highest at M/D = 0.3 at C_D = 0.7647.

Table 1. Coefficient drag by CFD, M/D

U(m/s)	Re	M/D (cm)				
		0.3	0.5	0.7	0.9	1.1
5	34,229	1.0923	1.0783	1.0790	1.0816	1.0831
7	47,921	0.9755	0.9632	0.9612	0.9643	0.9650
9	61,612	0.9083	0.8970	0.8993	0.9018	0.8991
11	75,304	0.8638	0.8535	0.8523	0.8519	0.8516
13	88,996	0.8322	0.8223	0.8251	0.8261	0.8286
15	102,688	0.8091	0.8022	0.8017	0.8043	0.8061
17	116,379	0.7909	0.7824	0.7845	0.7870	0.7893
19	130,071	0.7768	0.7682	0.7711	0.7732	0.7748
21	143,763	0.7647	0.7572	0.7591	0.7618	07639

Table 2. Coefficient drag by experiment, M/D

U(m/s)	Re	M/D (cm)				
		0.3	0.5	0.7	0.9	1.1
5	34,229	1.0443	1.0498	1.0326	1.0306	1.0374
7	47,921	0.9360	0.9135	0.9237	0.9152	0.9215
9	61,612	0.8643	0.8528	0.8807	0.8673	0.8585
11	75,304	0.8423	0.8034	0.8259	0.8309	0.8288
13	88,996	0.8123	0.7888	0.8063	0.8016	0.7972
15	102,688	0.7870	0.7663	0.7808	0.7847	0.7790
17	116,379	0.7739	0.7435	0.7383	0.7662	0.7512
19	130,071	0.7574	0.7430	0.7509	0.7549	0.7519
21	143,763	0.7465	0.7318	0.7405	0.7408	0.7333

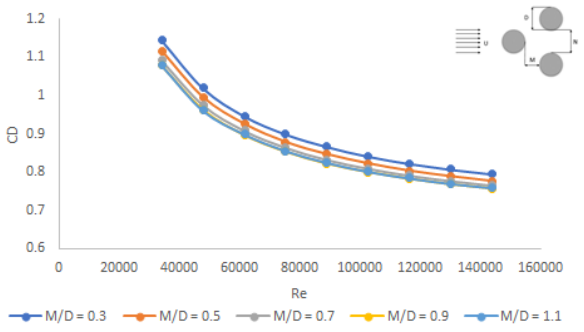


Figure 4. Comparison of the CFD drag coefficient against Reynolds number

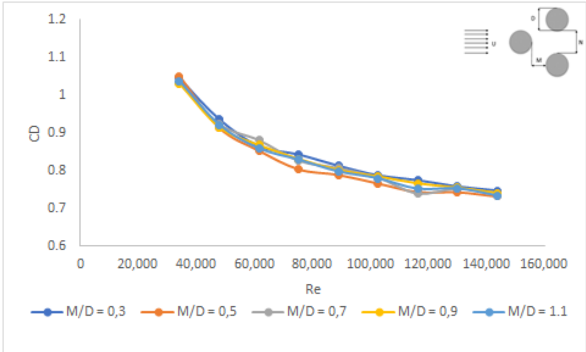


Figure 5. Comparison of the experimental drag coefficient against Reynolds number

Based on the graph in Fig. 5 we can summarize that the value of drag coefficient (C_D) is getting lower when the Reynolds number (Re) is getting more. It changed identically for every various distance (M/D) shows that the value is relevant each other. But although the trend for every M/D is almost the same, the value of drag coefficient at every distance (M/D) does not always change harmonically, as we can see on the table the lowest and the highest value did not settle at any specific distance (M/D) for every Reynolds number (Re). Based on this unique phenomenon explain how important to do and improve this tandem research with larger and more variables.

3.3. Pressure field

The effect of adding variation in distance with a ratio of M/D = 0.3, M/D = 0.5, M/D = 0.7, M/D = 0.9 and M/D = 1.1 for the pressure profile shown in Figure 4. The arrangement of serial parallel circular cylinder, the first specimen already reduces the direct collision between the fluid flow and rear side (second and third specimen) of the circular cylinder. It can be seen that the model with the distance ratio M/D = 1.1 has the lowest pressure contour when compared to other models.

Figure 4 also shows that the region of pressure different around the circular cylinders is the narrowest at M/D = 1.1. Consequently, the pressure difference does not strong enough to push the flow apart from the cylinder while the region of others M/D has pressure different around circular cylinders which getting larger and strong enough to push the flow apart from the cylinder so that the pressure drop is getting larger. This indicates that the smallest pressure coefficient at M/D = 1.1.

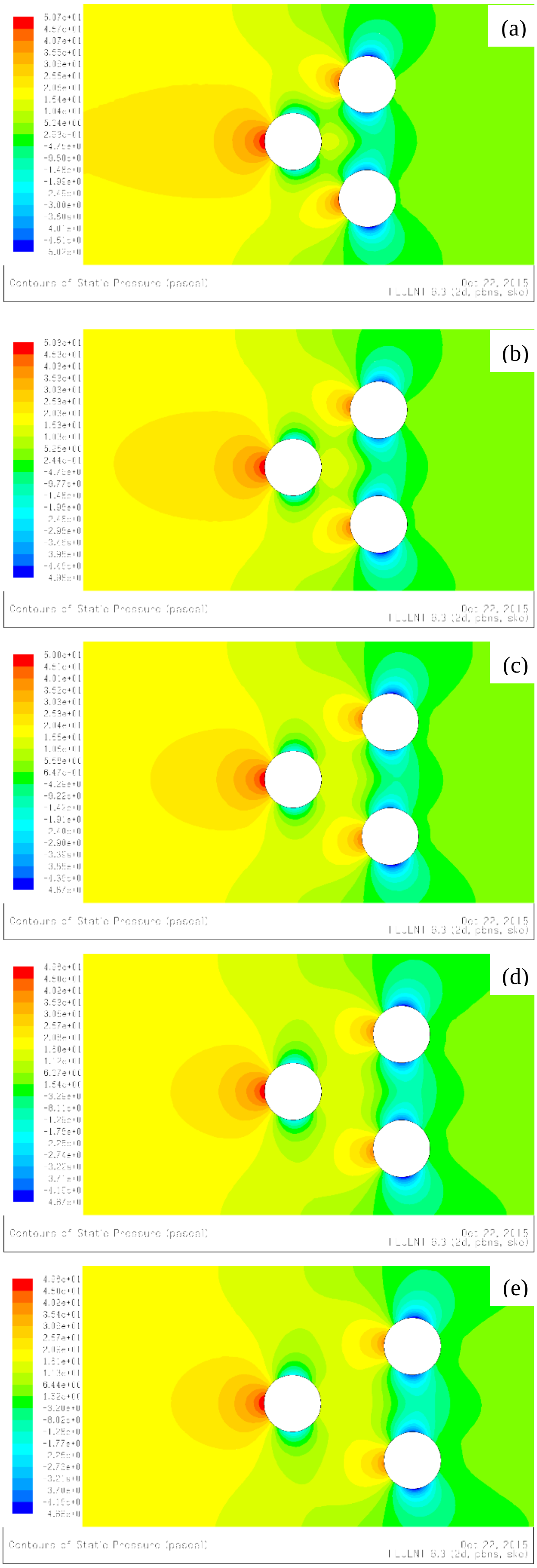


Figure 6. Contour profile on coefficient of flow pressure at U= 11 m/s (Re_D = 75.304). (a) M/D = 0.3; (b) M/D = 0.5; (c) M/D = 0.7; (d) M/D = 0.9 and (e) M/D = 1.1.

Numerical simulation result of pressure contour at various M/D with U = 11 m/s is shown in Figure 6. Figure 4 (a) shows that M/D = 0.3 the pressure changes over a wide region around the circular cylinder. The pressure difference between the upstream and the downstream is very large compared with other distances. Meanwhile, the smallest region of pressure is shown at M/D = 1.1 either in CFD method or experimental method. That is why we can see more blue color between the back tandem which shows a lower value of drag.

4. Conclusion

Experimental analysis and simulation of flow through a tandem of serial parallel circular cylinders has been done at 5 various M/D. The experimental data were validated with the flow visualization and flow simulation.

The smaller the distance between circular cylinders the stronger the vortex between the cylinders which tends to move toward the cylinders. This produces a thicker boundary layer that possibly induces a greater drag coefficient. Meanwhile, the smallest coefficient drag with U = 11 m/s is 0.8516 which occur at M/D = 1.1 and the biggest is 0.8638 at M/D = 0.5. The largest drag decreasing for all levels of Reynolds numbers, occurred when the tandem of serial parallel circular cylinders at M/D = 0.5 which is 0.7572 by CFD method and 0.7318 by experimental method with the comparison of percentage is 1.96% of difference.

Acknowledgements

We gratefully thanks the Head of the Fluid Mechanics Laboratory of Hasanuddin University who has facilitated the data collection process.

References

[1] A. Daloglu, “Pressure Drop in a Channel with Cylinder in Tandem Arrangement,” *Int. Commun. Heat Mass Transf.*, vol. 35, pp. 76–83, 2008.

[2] H. Hiroshi, W. Kenji, W. Yuki, and T. N. Takashi, “Experimental Study On The Flow Field Between Two Square Cylinders In Tandem Arrangement,” in *The Seventh Asia-Pacific Conference on Wind Engineering (APCWE-VII)*, 2009.

[3] A. Lankadasu and S. Vengadesan, “Interference Effect of Two Equal-Sized Square Cylinders in Tandem Arrangement: with Planar Shear Flow,” *Int. J. Numer. Methods Fluids*, 2007.

[4] K. S. Kumar and L. Kumaraswamidhas, “Numerical Study on Fluid Flow Characteristics Over the Side-By-Side Square Cylinders at Different Spacing Ratios,” *Int. Rev. Mech. Eng.*, vol. 8, no. 5, pp. 962–969, 2014.

[5] D. Sumner, S. J. Price, and M. P. Paidoussis, “Flow-pattern Identification for Two Staggered Circular Cylinders in Cross-Flow,” *J. Fluid Mech.*, vol. 411, pp. 263–303, 2000.

[6] M. B. S. Kumar and S. Vengadesan, “A Study On The Influence Of Gap Ratio On Turbulent Flow Past Two Equal Sized Square Cylinders Placed Side-By-Side,” in *The 37th National & 4th International Conference on Fluid Mechanics and Fluid Power December*, 2010, pp. 16–18.

[7] D. V. Patil and K. N. Lakshmisha, “Two-dimensional flow past circular cylinders using finite volume lattice Boltzmann formulation,” *Int. J. Numer. Methods Fluids Int. J. Numer. Meth. Fluids*, vol. 69, pp. 1149–1164, 2012.

[8] A. Etminan, M. Moosavi, and N. Ghaedsharafi, “Characteristics of Aerodynamics Forces Acting on Two Square Cylinders in the Streamwise Direction and its Wake Patterns,” *Adv. Control. Chem. Eng. Civ. Eng. Mech. Eng.*, pp. 209–217, 2010.

[9] N. Salam, R. Tarakka, Jalaluddin, and R. Bachmid, “The Effect of the Addition of Inlet Disturbance Body (IDB) of Flow Resistance Through the Square Cylinder Arranged in Tandem,” *Int. Rev.*

Mech. Eng., vol. 11, no. 3, 2017.

[10] N. Salam, R. Tarakka, Jalaluddin, and M. Ihsan, “Flow Separation Across Three Square Cylinders Arranged in Serial and Parallel Tandem Configuration,” *Int. J. Eng. Appl.*, vol. 3, pp. 96–106, 2020.

[11] N. Salam, R. Tarakka, Jalaluddin, and R. Bachmid, “The Effect of the Addition of Inlet Disturbance Body (IDB) to flow Resistance Through the Square Cylinders Arranged in Tandem,” *IREME*, vol. 11, no. 3, pp. 1970–8734, 2017.

[12] T. Tsutsui and T. Igarashi, “Drag Reduction of a Circular Cylinder in an Air-Stream,” *J. Wind Eng. Ind. Aerodyn.*, vol. 90, pp. 527–541, 2002.

[13] Subagyo and Rachmat, “Experimental Study of External Flow Characteristics on Obtuse Quadrilaterals with Elliptical Edges,” *J. Teknol. Technoscientia*, vol. 4, no. 2, p. 2012, 2012.

[14] N. Salam, R. Tarakka, Jalaluddin, M. Setiawan, and A. Mahfud, “Characteristics of the coefficient of Resistance to Flow Across Three Square Cylinders in Tandem Seriest and Parallel Configurations.,” in *Prosiding Seminar Nasional Teknik Mesin Politeknik Negeri Jakarta*, 2019, pp. 1244–1251. [in Bahasa]

[15] M. M. Alam, H. Sakamoto, M. Moriya, and K. Takai, “Fluctuating Fluid Forceacting on Two Circular Cylinders in a Tandem Arrangement at a Subcritical Reynolds Number,” *J. Wind Eng. Ind. Aerodyn.*, vol. 91, pp. 139–154, 2003.