

19_Distribution_of_fluid_flow_p ressure_through_tandem.pdf

by

Submission date: 18-May-2023 05:05PM (UTC+0700)

Submission ID: 2096140207

File name: 19_Distribution_of_fluid_flow_pressure_through_tandem.pdf (843.75K)

Word count: 3100

Character count: 15494

PAPER • OPEN ACCESS

Distribution of fluid flow pressure through tandem square cylinders with the addition of triangular cylinder as a disturbance object

To cite this article: Syahrir Habiba *et al* 2021 *IOP Conf. Ser.: Earth Environ. Sci.* **841** 012014

View the [article online](#) for updates and enhancements.

You may also like

- 4** - [Dynamic Measurement of Extra Long Stroke Cylinder in the Pneumatic System](#)
Ho Chang, Chou-wei Lan and Liang-Chia Chen
- [Constant-flux discrete heating in a unit aspect-ratio annulus](#)
J.M. Lopez, M. Sankar and Younghae Do
- 8** - [Flow control of a circular cylinder with O-rings](#)
Hee-Chang Lim and Sang-Joon Lee



The Electrochemical Society
Advancing solid state & electrochemical science & technology

242nd ECS Meeting

Oct 9 – 13, 2022 • Atlanta, GA, US

Abstract submission deadline: **April 8, 2022**

Connect. Engage. Champion. Empower. Accelerate.

MOVE SCIENCE FORWARD



Submit your abstract



Distribution of fluid flow pressure through tandem square cylinders with the addition of triangular cylinder as a disturbance object

Syahrir Habiba¹, Nasaruddin Salam¹, Rustan Tarakka^{1*}, Jalaluddin¹,
Muhammad Ihsan²

¹ Department of Mechanical Engineering, Hasanuddin University, Gowa, Indonesia

² Sekolah Tinggi Teknik Baramuli, Pinrang, Indonesia

email: rustan_tarakka@yahoo.com

Abstract. Flow across a square cylinder in a tandem arrangement is a shape characterization that is often used in structural engineering, transportation, and heat exchangers, where wind and water loads are one of the main factors that need to be considered in the design because they are associated with loss of energy. This is a problem faced by various industries in increasing the efficiency and stability of their systems. To reduce this energy loss, flow modification can be carried out in the form of adding disturbance objects that are placed in the front area of the main object, so that the fluid flow can pass through the object without any flow separation and produce a uniform flow after passing through the object. The research objective is focused on analyzing the effect of adding a triangular cylinder as a disturbance object to the characteristics of the flow pattern and the pressure distribution on the walls of the square cylinder in a tandem arrangement. This research used numerical computation methods and was validated through experimental testing at an upstream velocity of 19 m/s. The hydraulic diameter of the disturbance object was determined to be 0.033 m and the distance ratios M/D were 0.05, 0.15, 0.25, 0.35, 0.45, and 0.55, respectively. The results showed that the application of a triangular cylinder as a disturbance object had an effect on the characteristics of the flow pattern and the pressure coefficient, where the lowest pressure coefficient on the front side and the highest pressure coefficient on the side and rear sides were obtained in a model with a distance ratio $M/D=0.45$.

1. Introduction

Flow across a square cylinder in a tandem arrangement is a shape characterization that is often used in structural engineering, transportation, and heat exchangers, where wind and water loads are one of the main factors that need to be considered in the design. When flowing through the characteristic square shape, the fluid will experience separation on the surface wall and the back edge of the object. This will cause a loss of flow energy to move towards the boundary layer, resulting in a decrease in the pressure distribution as well as creating a backward pull force due to a significant difference in pressure between the front and rear walls of the object. This is a problem faced by various industries in increasing the efficiency and stability of their systems. To reduce this energy loss, flow modification can be carried out in the form of adding disturbing objects that are placed in the front area of the main object, so that the fluid flow can pass through the object without any flow separation and produce a uniform flow after passing through the object.

Testing through a 2-dimensional numerical simulation approach related to flow dynamics around rectangular configuration cylinders was analyzed using the Boltzman lattice method (LBM) at the Reynolds number $Re=100$ carried out by Islam et al. The main focus of the study was determined on the effect of the distance between the L/D cylinders 0.5, 1.5, 2.5, 4.0, 5.0, 6.0, 7.0, 8.0, 9.0, and 10.0, respectively. The results show that the four different features of each single square cylinder, stable flow, wavy flow, and rotating flow [1].



Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

Published under licence by IOP Publishing Ltd

Salam et al. have studied the flow drage across a square cylinder in tandem arrangement with the addition of an IDB in the form of a circular cylinder. This research used a numerical simulation method using Computational Fluid Dynamics (CFD) FLUENT 6.3.26 and was validated through laboratory experimental testing at Reynolds number $Re=30625$ to 96250 . The ratio of the IDB circular cylinder diameter to the square cylinder diameter (d/D) was varied, respectively, $d/D = 0.08$, $d/D = 0.14$ and $d/D=0.20$. Meanwhile, the ratio between the distance between the cylinders and the diameter of the square cylinder (L/D) was varied at $L/D=0.0$ to 1.0 . The experimental results show that the value of the drag coefficient (C_d) and the pressure coefficient (C_p) decreases with increasing L/D and d/D . Obtained the lowest C_d value of 1.67 or experiencing a reduction of 21.5962% compared to the model without IDB and the lowest C_p of 0.87 or a reduction of 14.7059% compared to the model without IDB at $L/D=0.43$ and $d/D=0.14$ (Salam et al, 2017). Salam et al have also investigated flow separation across three square cylinders in serial and parallel tandem arrangement [2].

Tsutsui and Igarashi conducted research related to the effect of applying a disturbance rod in front of a circular cylinder on aerodynamic drag. The results showed that the Reynolds number and the diameter of the disturbing rods influenced the flow patterns around the model. The highest inhibition reduction was obtained by 63% [3].

Daloglu conducted research related to the relationship between circular cylinders and square cylinders arranged in tandem in an alternating wind tunnel. The distance between cylinders was varied in the ratio S/d to 10 . The results showed that the pressure drop characteristics were influenced by the ratio of the cylinder diameter and the ratio of the distance between the cylinder and the diameter of the circular cylinder (S/d). It is interesting to see that at $S/d=2$ there is a decrease in the smallest pressure value in all treatments for variations in diameter, Reynolds number, and cylinder position [4].

Hasabe et al. conducted a study on 2 square cylinders arranged in tandem with L/D variations of 2 to 5 . Pressure taps were installed at half the circumference of each cylinder. The results showed that the distribution of the average pressure coefficient at each tap position was different for each change in L/D . The average pressure coefficient at the top and back of each cylinder is negative because at that position there is flow separation. The lowest pressure coefficient is obtained at $L/D=4$ [5].

2. Method

The research was carried out through a numerical simulation approach using a fluent CFD 6.3.26 program and validated through experimental laboratory testing. This method has been widely used to display features that are relevant to real objects [6, 7, 8, 9]. In the numerical computation approach, the test model design utilizes the Autodesk Inventor program as shown in Figure 1, and is defined into the computational domain. At this simulation stage, the analysis is focused on the dynamics of the flow which is formed in the form of a velocity pathline and a pressure contour around the test model. The rectangular cylinder is 0.1 m long and 0.1 m high. Meanwhile, the disturbing object has a hydraulic diameter 0.033 m. The ratio of the distance between IDB and square cylinder is varied, respectively written $M/D=0.05$, $M/D=0.15$, $M/D=0.25$, $M/D=0.35$, $M/D=0.45$, and $M/D=0.55$.

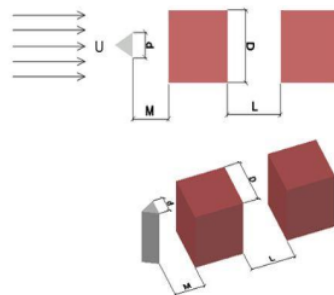


Figure 1. Test model

Furthermore, the test model passes through the meshing process using the Gambit device as well as defining the boundary conditions. The test is carried out at an upstream speed of 19 m/s. The computational conditions are shown in table 1.

Table 1. Computational condition

Fluid properties	Viscosity	0.000018348 kg/ms
	Density	1.186 kg/m ³
Boundary condition	Inlet	Velocity inlet
	Outlet	Pressure outlet
	Wall	Wall
	Test model	Wall

After going through the numerical computation process, the research was continued with validation through laboratory experimental testing including flow and pressure field visualization using the sub-sonic wind tunnel as shown in Figure 2.

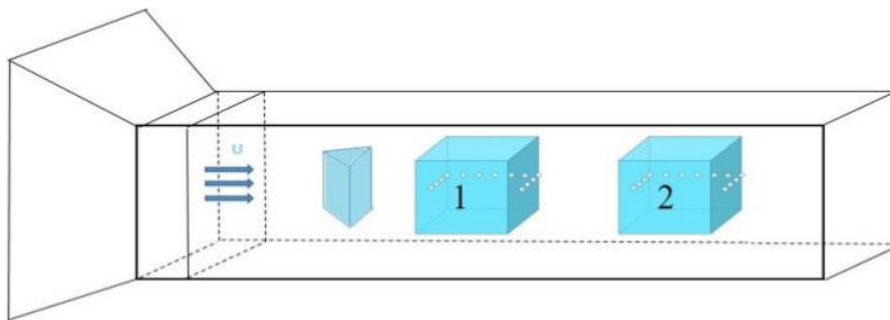


Figure 2. Sub-sonic wind tunnel

Pressure measurement is focused on 25 points to determine the effect of adding a triangular cylinder to the pressure distribution on the surface of the test model. The position of the pressure distribution data collection is shown in Figure 3.

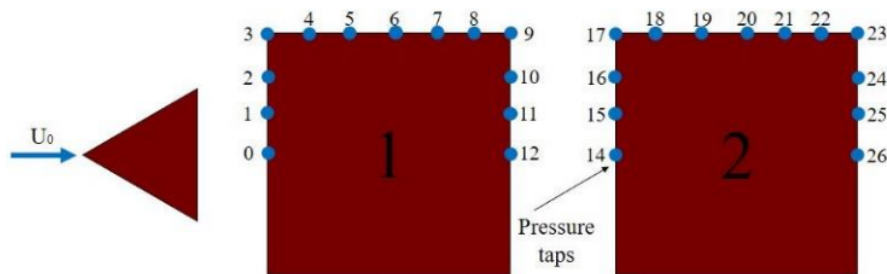


Figure 3. Pressure tap positions

The pressure distribution data obtained is then displayed in dimensionless units through the application of

equation 1 [10].

$$C_p = \frac{P - P_0}{\frac{1}{2} \rho U^2} \quad (1)$$

14

3. Results and discussion

3.1. Flow pattern Characteristics

Comparison of the flow pattern characteristics of the computational approach and flow visualization of a square cylinder in tandem arrangement with the addition of a triangular cylinder as a disturbance at the distance ratio M/D 0.05, 0.15, 0.25, 0.35, 0.45, and 0.55 respectively is shown in Figure 4 and Figure 5. For the front side in distance ratio M/D=0.45 cylinder 1 shows greater flow drag due to the application of the triangular cylinder as a disturbance. So that the fluid pressure on that side is getting smaller.

Besides, all distance ratios M/D show that the separation process occurs on the sides and back of square cylinder 1 and cylinder 2 both computationally and visually. This is because when the flow reaches the side edge of the square cylinder, it will lose energy to move towards the boundary layer. As a result, the flow slows down, where the low-speed flow tends to stick to the wall. A model with a distance ratio of M/D=0.45 has the smallest flow retarding on the side of the square cylinder.

It is interesting to see that the separation process due to low velocity on the side and rear is reduced for square cylinder 2. A significant increase in flow velocity is seen when the fluid reaches the side corner of the first square cylinder, where this increase occurs in all models with a distance ratio M/D. However, this increase in speed is not experienced by cylinder 2. Overall, the model with the ratio of the distance M/D=0.45 is the model with the best flow pattern characteristics when compared to other models.

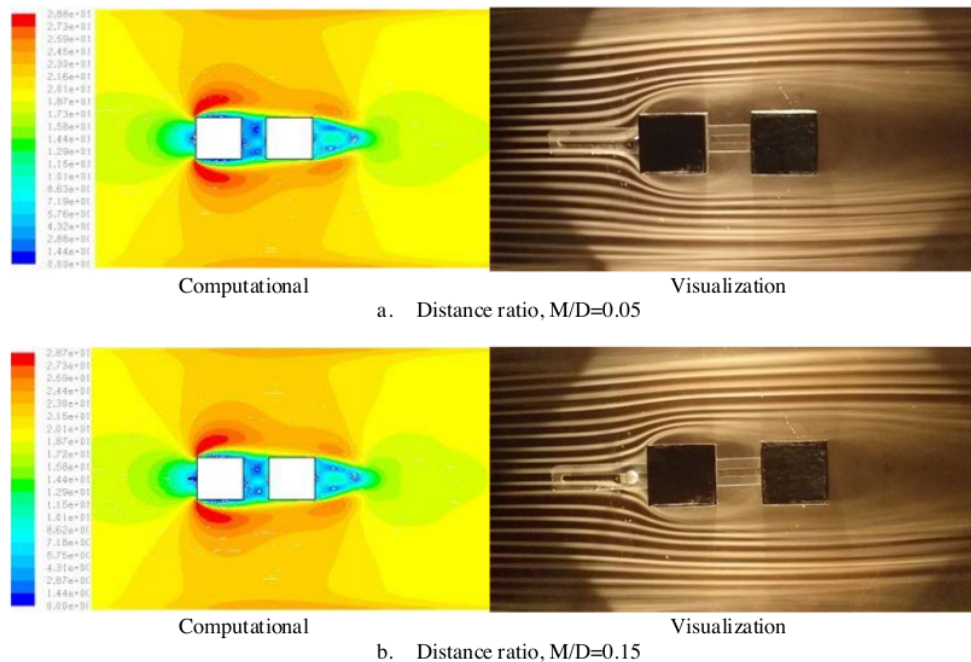
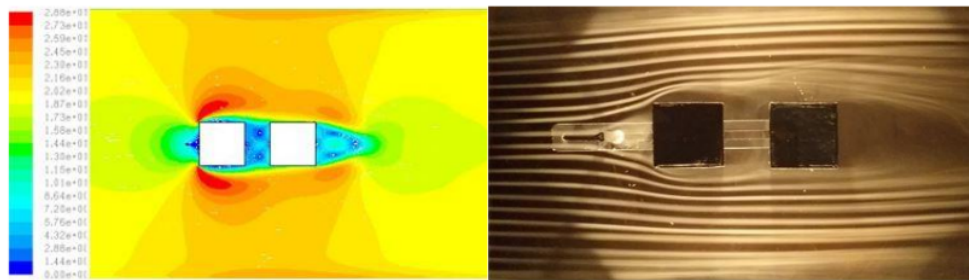


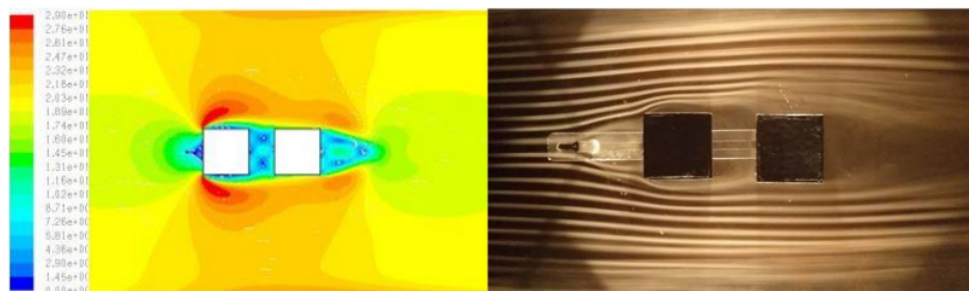
Figure 4. The characteristic flow pattern with upstream velocity, $U_0 = 19$ m/s (a,b)



Computational

Visualization

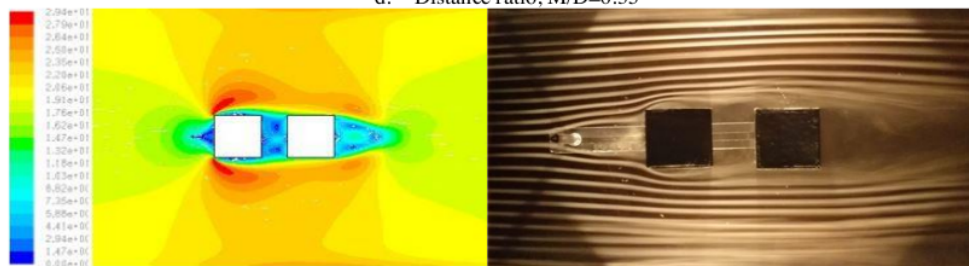
c. Distance ratio, $M/D=0.25$



Computational

Visualization

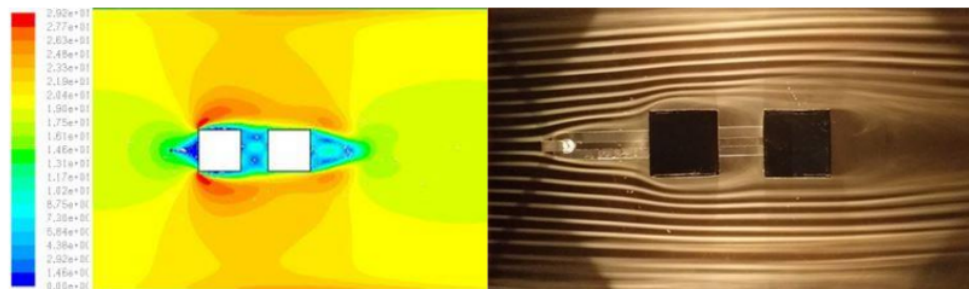
d. Distance ratio, $M/D=0.35$



Computational

Visualization

e. Distance ratio, $M/D=0.45$



Computational

Visualization

f. Distance ratio, $M/D=0.55$

Figure 5. The characteristic flow pattern with upstream velocity, $U_0 = 19$ m/s (c,d,e)

3.2. Pressure distribution

Comparison of computational approach pressure contours at each distance ratio $M/D=0.05$, $M/D=0.15$, $M/D=0.25$, $M/D=0.35$, $M/D=0.45$, and $M/D=0.55$ shown in the Figure 6. Based on the comparison of the M/D distance ratio, the lowest pressure on the front side of cylinder 1 (taping position 0, 1, 2, and 3) and the second cylinder (taping position 14, 15, 16, and 17) is obtained in a model with a distance ratio of $M/D=0.45$.

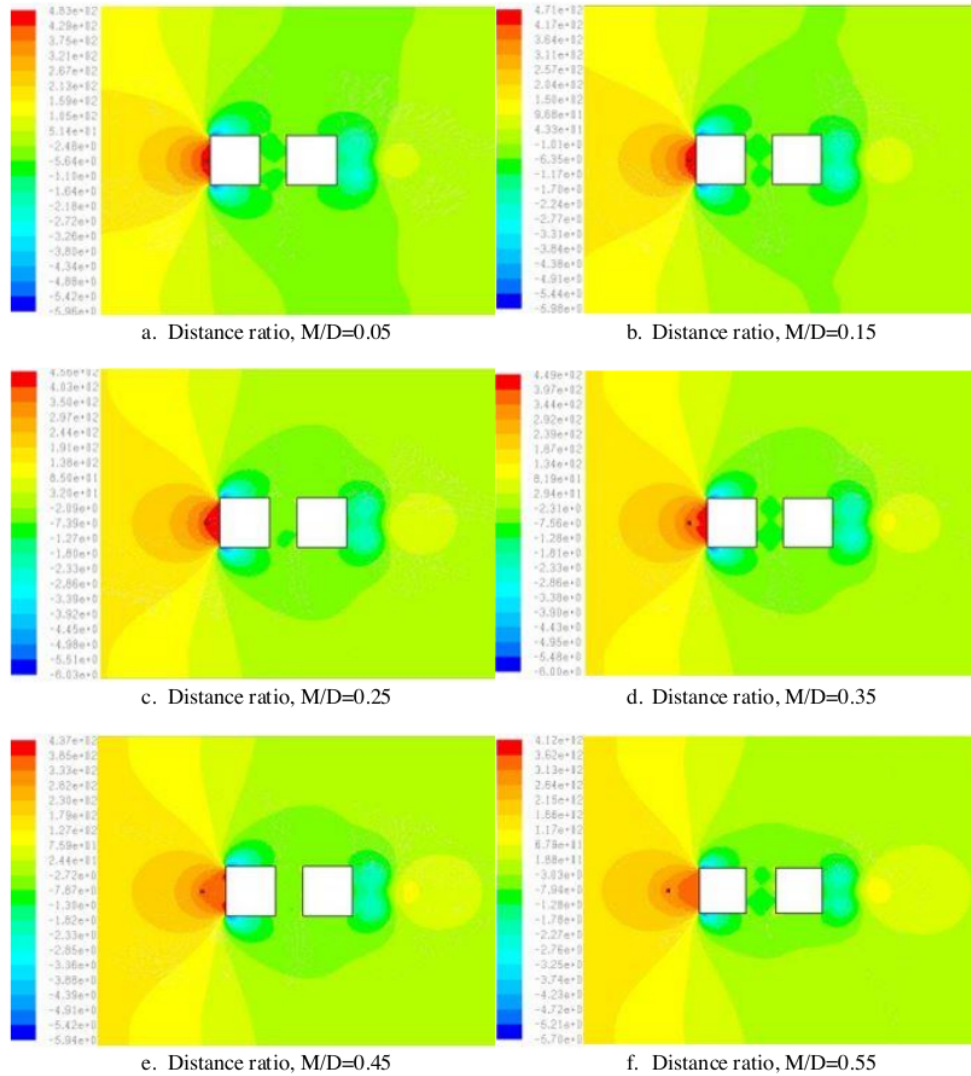


Figure 6. Pressure contour at upstream velocity, $U_0=19$ m/s by computational approach

As for the side, the highest-pressure contour is also obtained on the model with a distance ratio of $M/D=0.45$ for both cylinder 1 and cylinder 2. The same phenomenon also occurs on the backside of cylinder 1 and cylinder 2, where the model with a distance ratio of $M/D=0.45$ has the highest contour pressure compared to other models. This indicates that the flow slowdown is minimized thereby reducing the pressure drop on the back wall.

Comparison of the experimental approach pressure coefficients for each distance ratios, $M/D=0.05$, $M/D=0.15$, $M/D=0.25$, $M/D=0.35$, $M/D=0.45$, and $M/D=0.55$ are shown in the Figure 7. For the front walls of cylinder 1 and cylinder 2 (taping positions 0, 1, 2, 3 and 14, 15, 16, 17) the lowest pressure coefficient is obtained at the ratio of the distance $M/D=0.45$. This is consistent with the characteristics of the flow pattern in Figure 4 and the pressure contour in Figure 5 which shows that the M/D distance ratio 0.45 is a model that has the lowest flow velocity and pressure contour characteristics on the front wall compared to models with other M/D distance ratios.

For the side walls of cylinder 1 and cylinder 2 (taping positions 4, 5, 6, 7, 8 and 18, 19, 20, 21, 22) it shows that the highest-pressure coefficient is also obtained in the model with a distance ratio of $M/D=0.45$. This confirms

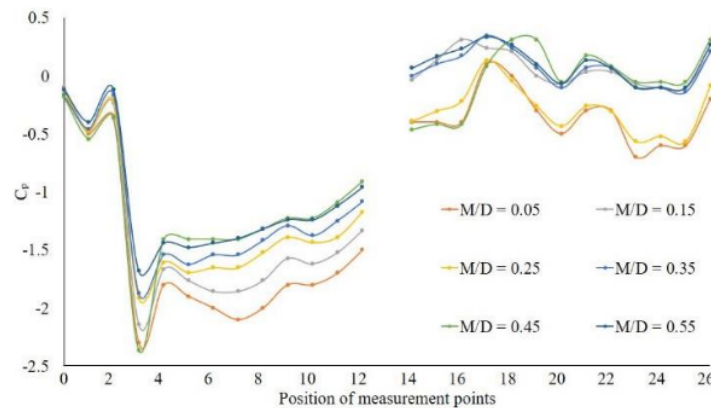


Figure 7. Pressure coefficient at upstream velocity, $U_0=19$ m/s with experimental approach

Overall, it shows that the optimum results based on a review of the characteristics of the flow pattern and pressure distribution are obtained at a distance ratio of $M/D=0.45$. These results are in line with research conducted by Salam et al, which revealed that the distance ratio between square cylinders and disturbance objects (IDB) of 0.43 gives optimum results compared to other models [2].

4. Conclusion

The addition of a triangular cylinder as a disturbance has a significant effect on the characteristics of the flow pattern and the pressure distribution on the front, side, and rear walls of a square cylinder in a tandem arrangement. The flow pattern characteristics show that the lowest flow velocity on the front side of the square cylinder is obtained in the model with a distance ratio of $M/D=0.45$. These results are consistent with the pressure distribution on the front, side, and rear walls of the square cylinder, where the model with a distance ratio of $M/D=0.45$ gives the optimum result compared to other models

Reference

- [1] D. Sumner, S.J. Price, M.P. Paidoussis. "Flow-pattern identification for two staggered circular cylinders in cross-flow". *J. Fluid Mech.*, vol. 411, pp. 263–303, 2000.
- [2] N. Salam, R. Tarakka, Jalaluddin, M. Ihsan. "Flow Separation Across Three Square Cylinders Arranged in Serial and Parallel Tandem Configuration". *International Journal on*

- Engineering Applications*. Vol. 8, pp. 96-106, 2020.
- [3] T. Tsutsui, T. Igarashi. "Drag Reduction of a Circular Cylinder in an Air-Stream". *Journal of Wind Engineering and Industrial Aerodynamics*. vol. 90. pp. 527-541, 2002.
- [4] A. Daloglu. "Pressure Drop in a Channel with Cylinder in Tandem Arrangement". *International Communication in Heat and Mass Transfer*. vol. 35. pp. 76-83, 2008.
- [5] H. Hasebe, K. Watanabe, Y. Watanabe, N.T. Takashi. "Experimental Study On The Flow Field Between Two Square Cylinders In Tandem Arrangement". *The Seventh Asia-Pacific Conference on Wind Engineering (APCWE-VII)*. Taipei, Taiwan, 2009.
- [6] M.M. Alam, H. Sakamoto, M. Moriya, K. Takai. "Fluctuating Fluid Forceacting on Two Circular Cylinders in a Tandem Arrangement at a Subcritical Reynolds Number". *Journal of Wind Engineering and Industrial Aerodynamics*. vol. 91. pp. 139-154, 2003.
- [7] A. Etminan, M. Moosavi, N. Ghaedsharafi N. Characteristics of Aerodynamics Forces Acting on Two Square Cylinders in the Streamwise Direction and its Wake Patterns". *Advances in Control, Chemical Engineering, Civil Engineering and Mechanical Engineering*. pp. 209-217, 2011.
- [8] A. Lankadasu, S. Vengadesan. Interference Effect of Two Equal-Sized Square Cylinders in Tandem Arrangement: with Planar Shear Flow. *International Journal for Numerical Methods in Fluids*. 2007.
- [9] K.S. Kumar, L. Kumaraswamidhas. "Numerical Study on Fluid Flow Characteristics Over the Side-By-Side Square Cylinders at Different Spacing Ratios". *International Review of Mechanical Engineering*, vol. 8. pp. 962-969, 2014.
- [10] Y.A. Cengel, J.M. Cimbala. "Fluid Mechanics: Fundamental and Applications". Third edition. Mc Graw Hill. ISBN 978-0-07-338032-2. 2014.

6 Acknowledgements

We extend our gratitude to the Head and staff of the Fluid Mechanics Laboratory of the Faculty of Engineering, Hasanuddin University, who gave permission and helped carry out this research.

ORIGINALITY REPORT

18%

SIMILARITY INDEX

12%

INTERNET SOURCES

15%

PUBLICATIONS

7%

STUDENT PAPERS

PRIMARY SOURCES

- 1 N Salam, R Tarakka, Jalaluddin, M Ihsan, M A Jimran. "Flow drags across three minibus car models arranged in tandem in four configurations", IOP Conference Series: Materials Science and Engineering, 2021
Publication 3%
- 2 oceanrep.geomar.de
Internet Source 2%
- 3 Yi-Ren Wang, Li-Ping Wu. "Effects of Tuned Mass Damper on Fixed-Fixed 3D Nonlinear String Resting on Nonlinear Elastic Foundation", International Journal of Structural Stability and Dynamics, 2017
Publication 1%
- 4 Ponnusamy Baskar, S Prithivi Rajan, Maggi Saraf. "Nitrogen enriched air for Nox reduction in Diesel engine", IOP Conference Series: Materials Science and Engineering, 2021
Publication 1%
- 5 core.ac.uk

1 %

6

R Tarakka, N Salam, Jalaluddin, W Rauf, M Ihsan. "Aerodynamic drag reduction on the application of suction flow control on vehicle model with varied upstream velocity", IOP Conference Series: Materials Science and Engineering, 2021

Publication

1 %

7

pdfs.semanticscholar.org

Internet Source

1 %

8

E V Tuch. "Influence of auxeticity on the processes of elastic deformation in a single crystal VZhM8 in the direction [011]", Journal of Physics: Conference Series, 2020

Publication

1 %

9

www.slideshare.net

Internet Source

1 %

10

Submitted to Rutgers University, New Brunswick

Student Paper

<1 %

11

acikerisim.gelisim.edu.tr

Internet Source

<1 %

12

Submitted to Higher Education Commission Pakistan

Student Paper

<1 %

13

Submitted to Laredo Community College

Student Paper

<1 %

14

Kim, S.. "Flow-induced vibrations of two circular cylinders in tandem arrangement. Part 1: Characteristics of vibration", Journal of Wind Engineering & Industrial Aerodynamics, 200908

Publication

<1 %

15

Lee, S.-J.. "Reducing the drag on a circular cylinder by upstream installation of a small control rod", Fluid Dynamics Research, 200404

Publication

<1 %

16

www.scipedia.com

Internet Source

<1 %

17

Kim, S.. "Flow-induced vibration of two circular cylinders in tandem arrangement. Part 2: Suppression of vibrations", Journal of Wind Engineering & Industrial Aerodynamics, 200908

Publication

<1 %

18

Triyogi Yuwono, Gunawan Sakti, Fatowil Nur Aulia, Adi Chandra Wijaya. "Improving the performance of Savonius wind turbine by installation of a circular cylinder upstream of returning turbine blade", Alexandria Engineering Journal, 2020

Publication

<1 %

19	kili.railpage.com.au Internet Source	<1 %
20	mafiadoc.com Internet Source	<1 %
21	Mustafa Sarioglu, Yahya Erkan Akansu, Tahir Yavuz. "Control of the Flow Around Square Cylinders at Incidence by Using a Rod", AIAA Journal, 2005 Publication	<1 %
22	M. A. Moussaoui. "Multiple-relaxation-time lattice Boltzmann computation of channel flow past a square cylinder with an upstream control bi-partition", International Journal for Numerical Methods in Fluids, 2009 Publication	<1 %
23	T. Tsutsui, T. Igarashi. "Drag reduction of a circular cylinder in an air-stream", Journal of Wind Engineering and Industrial Aerodynamics, 2002 Publication	<1 %

Exclude quotes On

Exclude matches < 5 words

Exclude bibliography On