Penerbit Akademia Baru

CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

CFD Analysis of Splitter Plate on Bluff Body

Open Access

Abdul Aabid[1], Azmil Afifi[1], Fharukh Ahmed Ghasi Mehaboob Ali[2, 3], Mohammad Nishat Akhtar[4], Sher Afghan Khan[1,*]

[1] Department of Mechanical Engineering, Faculty of Engineering, International Islamic University Malaysia, 53100, Kuala Lumpur, Malaysia
[2] Assistant Professor, Department of Mechanical Engineering, Government Engineering College, Huvinahadagali,Bellary dist Karnataka, India
[3] Research Scholar, Department of Mechanical Engineering, Bearys Institute of Technology Mangalore, Karnataka, India
[4] School of Aerospace Engineering, Universiti Sains Malaysia14300 Nibong Tebal, Penang, Malaysia

ARTICLE INFO

Article history:
Received 19 September 2019
Received in revised form 14 November 2019
Accepted 10 November 2019
Available online 27 November 2019

Keywords:
Splitter plate; bluff body; numerical simulation; CFD; and wind tunnel test

ABSTRACT

The blunt base either in the front or at the rear portion of the vehicles is prevalent and is found in railways where the front and rear portions the coaches are blunt, which results in zero velocity and hence the stagnation pressure, which is undesirable. The aim of this study is to investigate the effect of the splitter plate on the wake control of a bluff body. A rectangular bluff body of dimension 50 mm x 60 mm is controlled by a splitter plate positioned at the front of the bluff body as a passive control technique. The variation of the splitter plate length with respect to the height of the bluff body was considered. In the present study, K-epsilon enhanced wall treatment turbulence model has been used for the validation of the simulation results with the existing experimental data. The experimental results agree with the simulation's results with a significant impact of upstream splitter plate on the drag reduction in a turbulent flow. From the results, it is found that the splitter plate is useful to decrease the drag by controlling the flow field.

1. Introduction

In the field of external aerodynamics and automobile engineering, the subject of drag reduction is a significant problem with various applications. Splitter plates have been widely used in the aerospace industry where it can be found in some aircraft. The basic principle of the splitter plate is by dividing the flow field and diverting the boundary layer that becomes a source of vortex shedding [1]. This is usually away from a body that is being controlled, and in this study, a rectangular bluff body flow filed is controlled by the splitter plate.

From literature, splitter plate is also known as a passive control in the external aerodynamics. The splitter plate controls the flow by reducing the drag, and the passive control divides the flow through the splitter plate and reduces the wake region [2]. A bluff body is characterized by a body that resists the flow of fluid and forms a vast wake region with the stagnation region. Due to this, the value of drag is considerable [3]. This is due to the massive wake that is present on the body. Unlike a

* *Corresponding author.*
E-mail address: sakhan@iium.edu.my (Sher Afghan Khan)

streamlined body, the drag is minimal compared to the bluff body. The difference between a bluff body and a streamlined body is seen mostly on the geometry [4] of the bodies, and the effects of the bodies have been experimented and analyzed by Konstantinidis *et al.,* [3]. Numerical simulation of CD nozzle and effect of microjet control studied thoroughly from the previous authors using k-epsilon turbulence model, and they proved this type of model is suitable for optimizing inlet flows [5-9], the similar model experimentally studied well and also validated some cases with simulations results [10-18] circular cylinder [19], wedge [20,21], rectangular channel [22], and backward-facing step model [23].

One of the significant problems in the field of aerodynamics is to reduce drag. The problem of aerodynamic drag can be treated in many ways; however, the proper and careful study should be made. Various types of experiments had been conducted to understand how fluid flows across an object. It can be done by the flow visualization by smokes or fluids such as water past through an object. The experiment related to this paper was conducted at a high-speed aerodynamics laboratory, Indian Institute of Technology, Kanpur [24]. From his paper, with a splitter plate, the value of drag is reduced when varying the length of the splitter plate; however, this is most effective at zero angles of attack. Meanwhile, from the literature study, it has been found the experimental work has been done for splitter study. Hence, the focus of the present work is to validate the results of the wind tunnel tests and the effect of the splitter plate length and location with the CFD software. Therefore, computational fluid dynamics has been used to visualize the flow, and simulation was done with the help of ANSYS software.

2. Problem Formulation

The fundamental approach and conditions of the experiment were similar to Rathakrishnan [24], as shown in Figure 1. However, the experiment is fully simulated in CFD software. The rectangular bluff body has a dimension of 62 mm x 50 mm with varying lengths of the splitter plate. Unlike in Teksin *et al.,* [25] and Gurram *et al.,* [26], the splitter plates are not flexible. Moreover, the position of the splitter plate is fixed at the center of the bluff body, and the angle of attack is also fixed to 0 degrees. Figure 1 shows the schematic model of the experimental setup prepared by Rathakrishnan [24]. For this paper, the same model is to be used; however, a different set up is made. The wind tunnel modeled for the simulation is of different size. Besides, a 2D model is used to obtain the results.

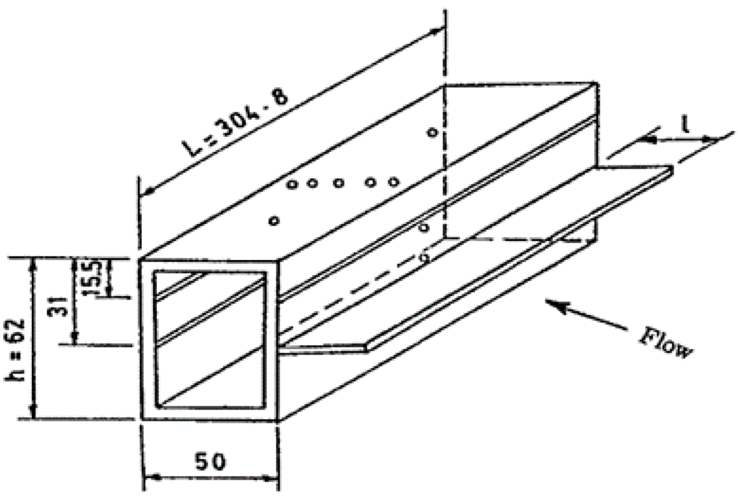


Fig. 1. Experimental setup by Rathakrishnan [24]

3. Finite Volume Method
3.1 Design and Modeling

The shape of the object is taken from the side view in order to design a two-dimensional (2D) model. With this, it is easier to know where the flow will go through and hit the splitter plate. The geometry for 2D is created, as shown in Figure 2. Few constraints have to be made to ensure the geometry is created correctly. The first constraint would be the horizontal lines are created so that it is parallel to the y-axis and the horizontal line is made so that it is parallel to the x-axis. The other constraints would be specific 2 lengths are to be made symmetry with respect to a specific line. As for example, the top and bottom part of the bluff body is both symmetries with respect to x-axis as its center.

Moving on from that, as had been mentioned, the test section for the body was in a wind tunnel; therefore, a rectangular test section is built to match the wind tunnel. With this, a more accurate representation of the 3D model is created for the 2D.

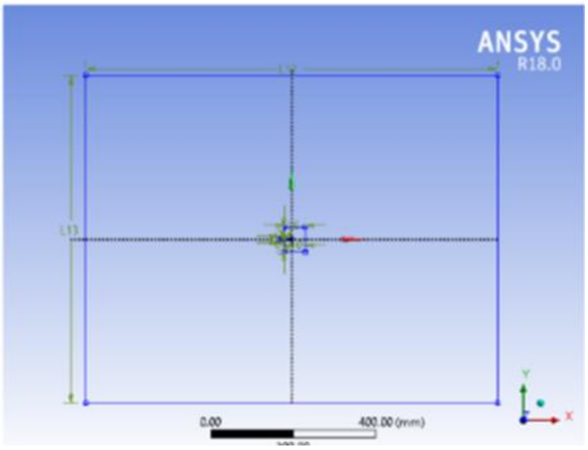


Fig. 2. Two-dimensional finite element model

From Figure 2 above, the rectangular section is already created to match the wind tunnel condition. Therefore the geometry is now ready for surfacing. The surfacing will give the geometry a surface with material. The material of the surface can be changed during the setup step. The surfacing tool is found under the concept of toolbar. Only after surfacing, then the geometry is ready for meshing. The surfacing of the geometry would look as of Figure 3.

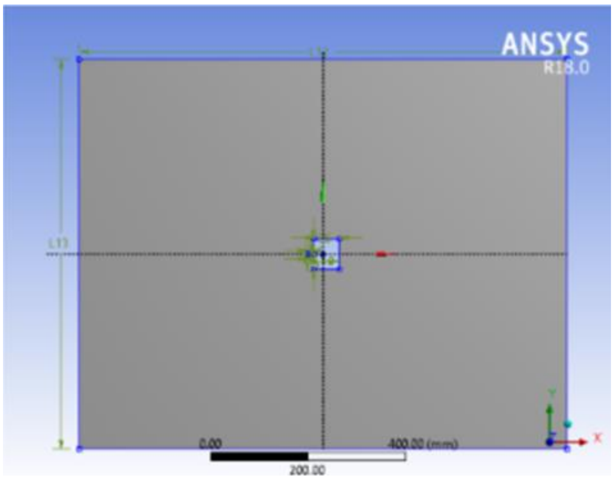


Fig. 3. Geometry with control volume

3.2 Meshing and Boundary Conditions

The crucial part of the simulation is meshing. According to Sadrehaghighi [27], suitable meshes leads to good results. Therefore, careful consideration has been made on the mesh generation. Figure 4 shows the meshing of the bluff body with the splitter plate. It can be observed that the mesh is denser at the body because that is the part of flow where high stresses are present. The meshing from Akhtar *et al.,* [6] also has a high meshed area in the highly stressed area. The element size is defined by several divisions. The meshing surface is made up of triangular meshes, and this is clearly seen in Figure 4 at the zoomed areas. Table 1 illustrates the elements which used mesh the model.

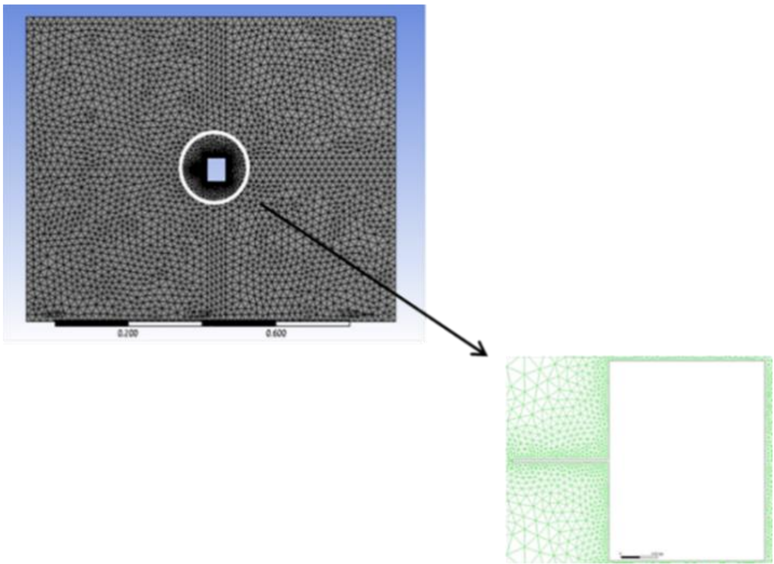


Fig. 4. Two-dimensional FE mesh model close view of the model

Table 1
Number of elements generated for each L/h

L/h	Number of Elements (triangular cells)
0	8128
1/4	8564
1/2	8848
1	9554
3/2	10460
2	9910
3	11500

For the selection of the turbulence model, a proper turbulence model was selected and used, which is also recommended from the previous authors [28-30]. Therefore, in the present work, K-epsilon turbulence model has been used for the simulation with enhanced wall treatment at near-wall function. Apart from that, K-omega is not used as it works well with lower Reynold's number [29,31] for 2D model. However, for a 2D model, an advanced turbulence model is used for better results. Table 2 shows the applied boundary conditions for the two-dimensional model.

Table 2
Two-dimensional boundary conditions

Turbulence Model	K-Epsilon
K-epsilon Model	Standard
Near-Wall Treatment	Enhanced Wall Treatment
Fluid	Air, ideal gas, viscosity by Sutherland law
Inlet Velocity	16.18 and 24.8 (m/s)
Solution initialization	Standard, from Inlet

3.3 Mesh Independence Study

To check the quality of mesh of the present work. It is considered three types of mesh types by changing the number of element division or element size (Table 3). To optimize the result, considered only one case of splitter plate length of L/h=1/4. From the results, it has been found that the higher mesh type model (elegant) has quite good results when it is compared to the experimental result of Rathakrishnan [24]. Therefore, for further cases of present work considered the mesh types as elegant.

Table 3
Mesh Check

Mesh Type	Number of Elements	CPU Run Time	CFD Result
Coarse	5,431	13 minutes	0.2215
Medium	6,639	15 minutes	0.2661
Fine	8,564	19 minutes	0.2822

4. Results and Discussion

This section explains the results for a given problem in terms of numerical simulation, which is obtained for all cases of the current study.

4.1 Experimental Validation

Experimental results from Rathakrishnan [24] are observed. The results from the experiment are the change of drag coefficient (C_d) and pressure coefficient (C_p) with the variation of splitter plate length. The graph of C_d and C_p vs. L/h is to be plotted in this paper as well and compared with results from Rathakrishnan [24]. The graphs taken below are only for a single angle of attack of zero degrees with the splitter plate placed at the center of the bluff body. Under this section, the graph of C_d vs. L/h from [24] to be compared with the results achieved in this paper. Figure 5 shows the results of drag coefficient variation with respect to splitter plate length variation from Rathakrishnan [24] and comparison with the present numerical simulations.

Figure 5 indicates that simulation and experimental results show a similar trend of decrease of drag coefficient C_d with a more extended splitter plate. This validates the effect of splitter plate to be able to reduce the drag in the bluff body. However, while comparing the values of the C_d, it is seen that there is a difference in the values of drag coefficients. The differences in values can be attributed due to several factors. The first factor is that the experiment results represent the exact values provided without any errors during the measurements. The experimental results are the measured values at fixed locations, whereas the simulations results are the average values on the entire area.

On the other hand, simulation through CFD represents the ideal situation where, during the initialization, most parameters are set to be for ideal conditions. The simulation is based on a mathematical model that leads to solving the system of equations. However, despite the differences in values of C_d in the results, both results from experiment and simulation can visualize the effect of the splitter plate on a bluff body.

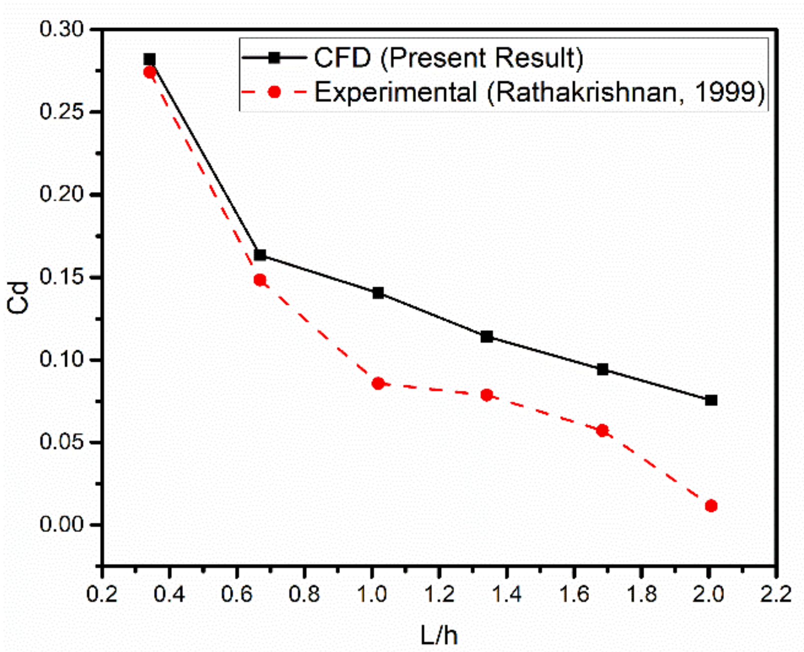
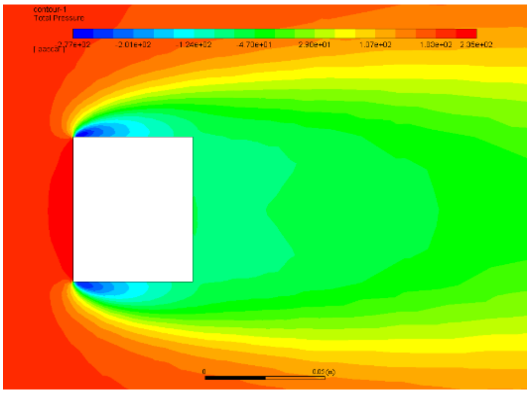


Fig. 5. Graphical validation of the present results

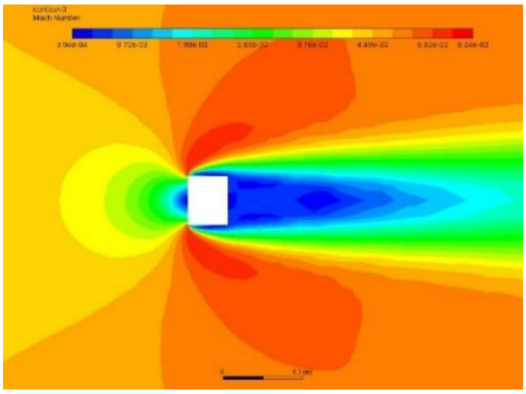
4.2 Without Splitter Plate

Data for the drag coefficient, total pressure variation, velocity variation, and flow visualization are presented in Figure 6 and 7. Flow visualization is presented in the form of contours and vector contours.

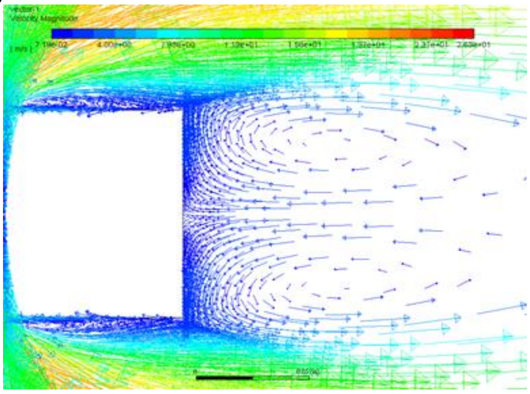
For the preliminary results, it is seen that on a bluff body without flow control is dominated by vortex found at the leading corner of the body, as seen in Figure 6 and 7. The results for this preliminary part corresponds to the results of the experiment from Feuvrier [32]. Apart from that, the pressure variation is high only near the body (Figure 6 and 7). At the side of the body, the pressure has a negative value, and on top of the body and at the back of the body, the pressure is almost zero, and this indicates the formation of the wake region. Besides, at the front of the body, the low-pressure region is also formed, causing the flow to result in high values of drag. According to Roshxot [33], the critical region for the bluff body is near the vortex formation. There is a close relation between drag and wake in order to control the wake region; the drag must be reduced. With the reduction of drag, the wake becomes smaller [34]. From the vector Figure 6 (c) and 7 (c), the zoomed figures clearly show the movement of fluid in the recirculation region, indicating the wake region formation at the back of the bluff body.



(a)

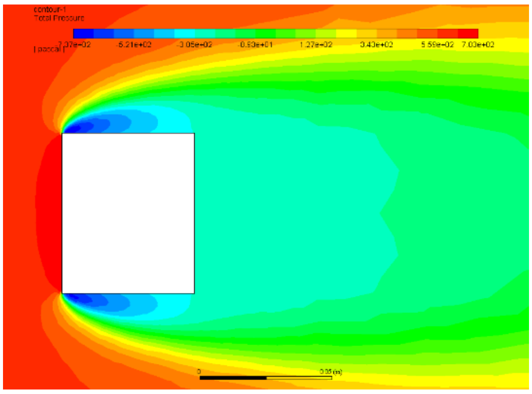


(b)

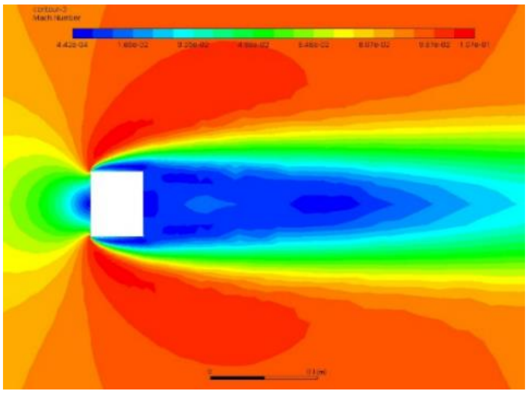


(c)

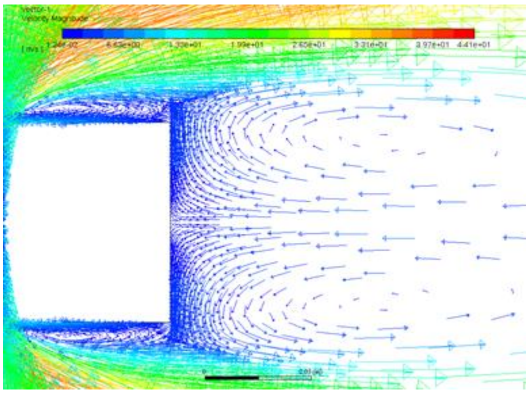
Fig. 6. Velocity = 16.18 m/s (a) total pressure contour (b) velocity contour (c) vector



(a)



(b)



(c)

Fig. 7. Velocity = 24.8 m/s (a) total pressure contour (b) velocity contour (c) vector

4.3 With Splitter Plate

This section illustrates all the results of the pressure and drag coefficient of the defined problem in figure 1.

4.3.1 Pressure coefficient

Results for the pressure coefficient of the rectangular bluff body with a variable length of the splitter plate is presented in Figure 8 and 9. From each figure, they contain 3 different graphs for pressure variation at three different parts of the body. The first part is where the pressure coefficient is taken from the whole body with a splitter plate; the other is the front face and rear face of the bluff body.

From Figure 8 and 9, it is seen that the values of the pressure coefficient for the whole body with a splitter plate increases with more extended splitter plate. The pressure variation remains the same when taken at the front and back face of the bluff body. For the whole body with a splitter plate, the pressure attains negative value and later decreases in magnitude approaching zero until L/h is 0.5, then the value increases in magnitude. The linear increase in pressure can be seen from L/h = 2 to L/h = 3. As for the pressure variation at the rear face is concerned, there is an increase in the magnitude of the pressure coefficient. However, the values are negative. The results for pressure coefficient variation in Rathakrishnan [24] show that the pressure increases until L/h = 1; however, decreases and L/h 1.5 and above there is again increased in the pressure coefficient. With the change of velocity, the effect of splitter does not change. From the figures, the critical L/h can range from 1 to 1.5; this matches a result of experimental work of Rathakrishnan [24], where the author states that a critical L/h is about 1.

4.3.2 Drag coefficient

Results and the effect of splitter plate length variation on the drag coefficient are presented in Figure 10. Three separate graphs are plotted for three different sections of the splitter plate, and they are the splitter plate in the front face of the bluff body and the rear face of the bluff body. This helps the study of drag variation at different parts of the body for analysis. The Figure 10 show that by increasing the length of the splitter plate, the value drag coefficient decreases progressively. When taking the drag coefficient at the back of the body, the effect differs.

Figure 11 indicates that for drag at L/h more than 1, the drag increases with a more extended splitter plate. This shows the wake region formed at the back of the body (rear) is significant with the long splitter plate. The drag coefficient decreases for the front part of the body, and hence, the splitter plate reduces the drag at the front part of the body. However, the drag decreases at the back of the bluff body resulting in the wake region, a low-pressure region. The results for the drag variation show similar trends for both the velocities of the present study.

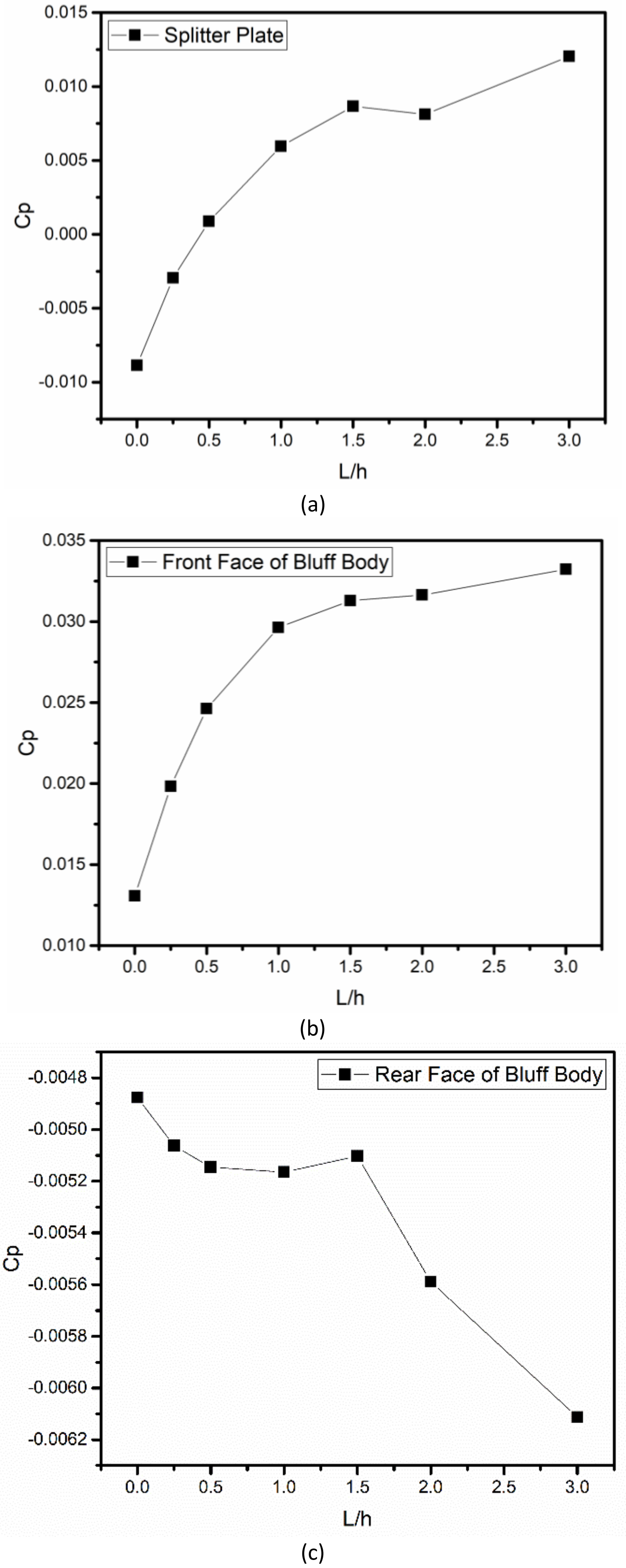
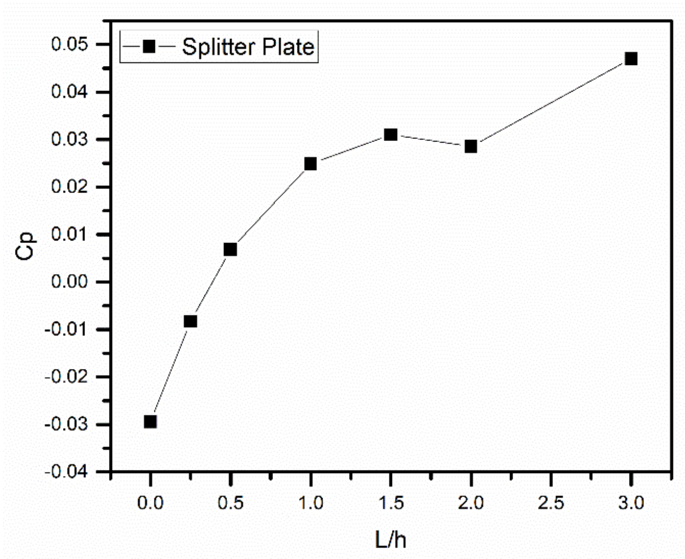
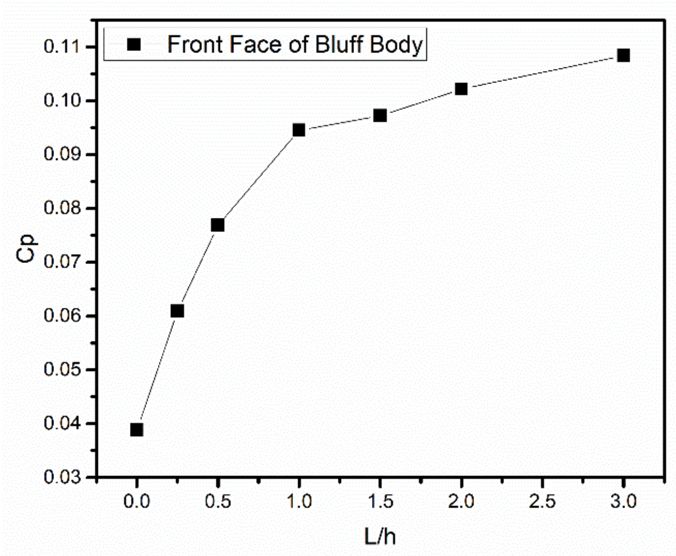


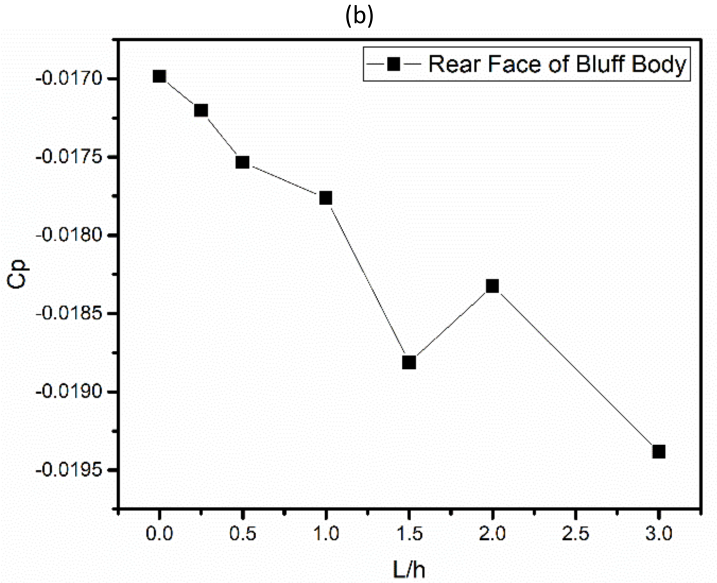
Fig. 8. Pressure coefficient variation at Velocity = 16.18 m/s



(a)

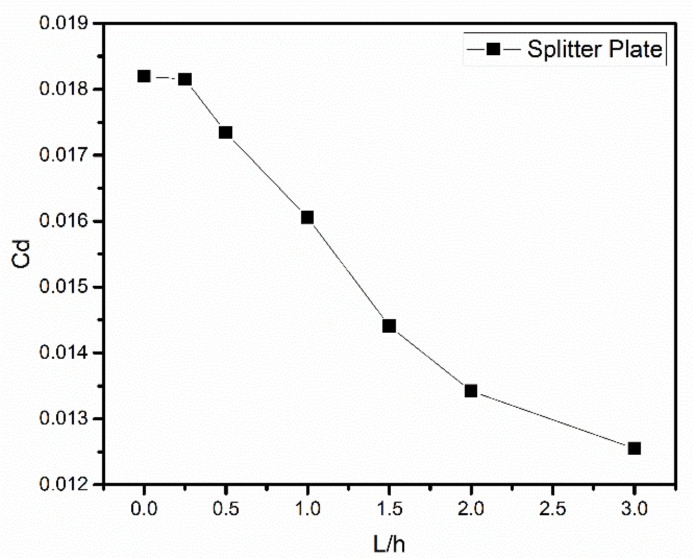


(b)

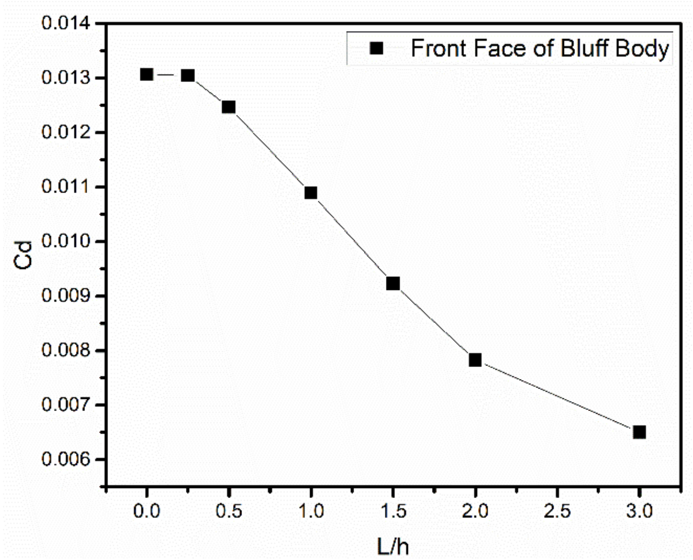


(c)

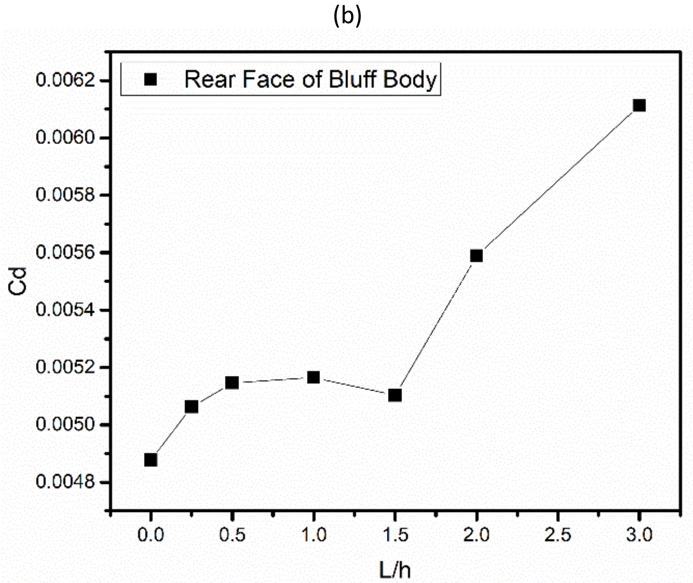
Fig. 9. Pressure coefficient variation at Velocity = 24.8 m/s



(a)



(b)



(c)

Fig. 10. Drag coefficient variation at Velocity = 16.18 m/s

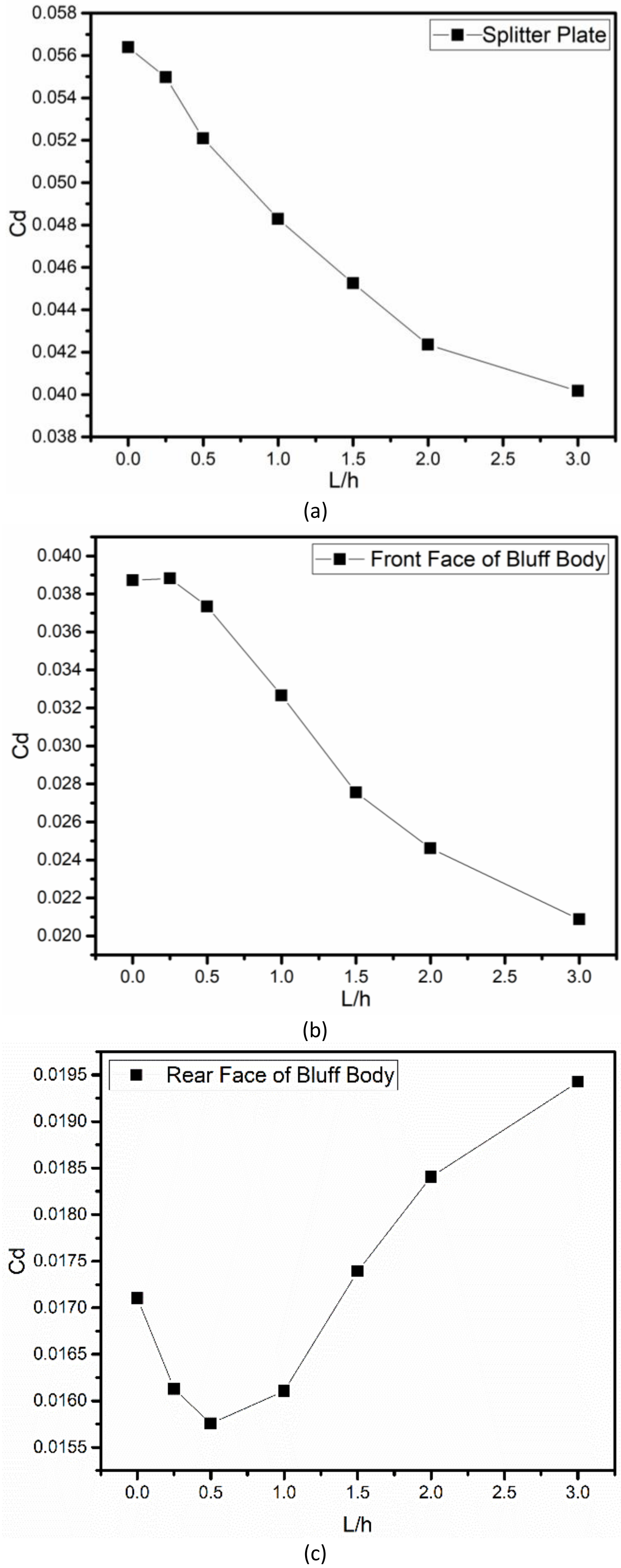


Fig. 11. Drag coefficient variation at Velocity = 24.8 m/s

5. Conclusions

In the present study, the influence of a splitter plate placed on a bluff body in the front as well as at the rear is studied with the help of the CFD method for two different velocities. Experiment results compared using the CFD method, and they are found to be in good agreement. While comparing the simulation results with the experimental data, the results show the variation of pressure values with the length of the splitter plate. However, the drag coefficient is found to be increasing with the length of the splitter plate. The slitter plate can reduce the drag coefficient, but it is only possible for the front face of the bluff body. When looking at the drag coefficient at the rear part of the body, the results are reversed, and this corresponds to the formation of the vortex and an increased wake region. While comparing the simulation results with the wind tunnel experimental values, the effect of splitter plates in reducing the drag is on the similar line with that of Rathakrishnan [24].

References

[1] Irwin, Peter A. "Bluff body aerodynamics in wind engineering." *Journal of Wind Engineering and Industrial Aerodynamics* 96, no. 6-7 (2008): 701-712.

[2] Ogunremi, Ayodele R., and David Sumner. "Flow Around a Surface-Mounted Finite Square Prism With a Splitter Plate." In *ASME 2014 Pressure Vessels and Piping Conference*. American Society of Mechanical Engineers Digital Collection, 2014.

[3] Konstantinidis, Efstathios, and Demetri Bouris. "Bluff body aerodynamics and wake control." *Applied Aerodynamics* (2012): 63.

[4] Koenig, Keith, and Anatol Roshko. "An experimental study of geometrical effects on the drag and flow field of two bluff bodies separated by a gap." *Journal of fluid mechanics* 156 (1985): 167-204.

[5] Aabid, Abdul, Ambreen Khan, Nurul Musfirah Mazlan, Mohd Azmi Ismail, Mohammad Nishat Akhtar, and Sher Afghan Khan. "Numerical Simulation of Suddenly Expanded Flow at Mach 2.2." *International Journal of Engineering and Advanced Technology* 8: 457-462.

[6] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Khan. "Numerical Simulations of a CD Nozzle and the Influence of the Duct Length." *Int. J. Innov. Technol. Explor. Eng* 8, no. 9S2 (2019): 622-630.

[7] Fharukh, Ahmed G M, Abdulrehman A. Alrobaian, Abdul Aabid, and S A Khan. "Numerical Analysis of Convergent-Divergent Nozzle Using Finite Element Method." *International Journal of Mechanical and Production Engineering Research and Development* 8, no.6 (2018): 373–82.

[8] Khan, Sher Afghan, Abdul Aabid, and Maughal Ahmed Ali Baig. "CFD Analysis of CD Nozzle and Effect of Nozzle Pressure Ratio on Pressure and Velocity for Suddenly Expanded Flows." *International Journal of Mechanical and Production Engineering Research and Development* 8 (2018): 1147-1158.

[9] Khan, Sher Afghan, Abdul Aabid, Fharukh Ahmed Mehaboobali Ghasi, Abdulrahman Abdullah Al-Robaian, and Ali Sulaiman Alsagri. "Analysis of area ratio in a CD nozzle with suddenly expanded duct using CFD method." *CFD Letters* 11, no.5 (2019): 61-71.

[10] Faheem, Mohammed, Mohammed Kareemullah, Abdul Aabid, Imran Mokashi, and S. A. Khan. "Experiment on of Nozzle Flow with Sudden Expansion at Mach 1.1." *International Journal of Recent Technology and Engineering* 8, no. 2S8(2019): 1769–1175.

[11] Khan, S. A., Abdul Aabid, Imran Mokashi, and Zaheer Ahmed. "Effect of Micro Jet Control on the Flow Filed of the Duct at Mach 1.5." *International Journal of Recent Technology and Engineering* 8, no. 2S8 (2019): 1758–1162.

[12] Khan, S. A., Zaheer Ahmed, Abdul Aabid, and Imran Mokashi. "Experimental Research on Flow Development and Control Effectiveness in the Duct at High Speed." *International Journal of Recent Technology and Engineering (IJRTE)* 8, no. 2S8 (2019): 1763–1768.

[13] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Khan. "Effects of Micro jets on the Flow Field of the Duct with Sudden Expansion." *International Journal of Innovative Technology and Exploring Engineering (IJITEE)* 8, no. 9S2 (2019): 636–640.

[14] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Khan. "Numerical Simulations of a CD Nozzle and the Influence of the Duct Length." *International Journal of Innovative Technology and Exploring Engineering (IJITEE)* 8, no. 9S2 (2019): 622-630.

[15] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Khan. "Control of CD Nozzle Flow Using Microjets at Mach 2.1." *International Journal of Innovative Technology and Exploring Engineering (IJITEE)* 8, no. 9S2 (2019): 631–635.

[16] Khan, S A, Imran Mokashi, Abdul Aabid, and Mohammed Faheem. "Experimental Research on Wall Pressure Distribution in C-D Nozzle at Mach Number 1.1 for Area Ratio 3.24." *International Journal of Recent Technology and Engineering* 8, no. 2S3 (2019): 971–975.
[17] Azami, Muhammed Hanafi, Mohammed Faheem, Abdul Aabid, Imran Mokashi, and S A Khan. "Experimental Research of Wall Pressure Distribution and Effect of Micro Jet at Mach." *International Journal of Recent Technology and Engineering* 8, no. 2S3 (2019): 1000–1003.
[18] Azami, Muhammed Hanafi, Mohammed Faheem, Abdul Aabid, Imran Mokashi, and S A Khan. "Inspection of Supersonic Flows in a CD Nozzle Using Experimental Method." *International Journal of Recent Technology and Engineering* 8, no. 2S3 (2019): 996–999.
[19] Sajali, Muhammad Fahmi Mohd, Syed Ashfaq, Abdul Aabid, and Sher Afghan Khan. "Simulation of Effect of Various Distances between Front and Rear Body on Drag of a Non-Circular Cylinder." *CFD Letters* 11, no. 5 (2019): 37–49.
[20] Khan, Sher Afghan, Abdul Aabid, and Ahmed Saleel C. "CFD Simulation with Analytical and Theoretical Validation of Different Flow Parameters for the Wedge at Supersonic Mach Number." *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS* 19, no. 1 (2019): 170–177.
[21] Khan, Sher Afghan, Abdul Aabid, Imran Mokashi, Abdulrahman Abudllah Al-Robaian, and Ali Sulaiman Alsagri. "Optimization of Two-Dimensional Wedge Flow Field at Supersonic Mach Number." *CFD Letters* 11, no. 5 (2019): 80–97.
[22] Jamil, M. M., M. I. Adamu, T. R. Ibrahim, and G. A. Hashim. "Numerical study of separation length of flow through rectangular channel with baffle plates." *Journal of Advanced Research Design* 7, no. 1 (2015): 19-33.
[23] Jehad, D. G., G. A. Hashim, A. K. Zarzoor, and CS Nor Azwadi. "Numerical Study of Turbulent Flow over Backward-Facing Step with Different Turbulence Models." *Journal of Advanced Research Design* 4, no. 1 (2015): 20-27.
[24] Rathakrishnan, E. "Effect of splitter plate on bluff body drag." *AIAA journal* 37, no. 9 (1999): 1125-1126.
[25] Teksin, S., and S. Yayla. "Effects of Flexible Splitter Plate in the Wake of a Cylindrical Body." *Journal of Applied Fluid Mechanics* 9, no. 6 (2016): 3053–3059.
[26] Gurram, Harika. "Effect of the Wake Structure on Drag Coefficient for a Thin Flexible Wire." PhD diss., Florida Institute of Technology, 2017.
[27] Sadrehaghighi, Ideen. "Mesh Generation in CFD." *CFD Open Series* (2019): 1–301.
[28] El-behery, Samy M, and Mofreh H Hamed. "A Comparative Study of Turbulence Models Performance for Turbulent Flow in a Planar Asymmetric Diffuser." *International Journal of Aerospace and Mechanical Engineering* 3, no. 5 (2009): 580–591.
[29] Mishra, Pankaj, and K. R. Aharwal. "A review on selection of turbulence model for CFD analysis of air flow within a cold storage." In *IOP Conference Series: Materials Science and Engineering*, vol. 402, no. 1, p. 012145. IOP Publishing, 2018.
[30] Soe, Thet Mon, and S. Khaing. "Comparison of turbulence models for computational fluid dynamics simulation of wind flow on Cluster of Buildings in Mandalay." *International Journal of Scientific and Research Publications* 7, no. 8 (2017): 337-350.
[31] Sadrehaghighi, Ideen. "Turbulence Modeling -A Review." *CFD Open Series*, (2019): 1–49.
[32] Feuvrier, A., N. Mazellier, and A. Kourta. "Self-adaptive control of a bluff body wake by means of porous flaps." *International Journal of Engineering Systems Modelling and Simulation 47* 5, no. 1-3 (2013): 57-67.
[33] Anatol Roshko. "On the wake and drag of bluff bodies." *Journal of the aeronautical sciences* 22, no. 2 (1955): 124-132.
[34] Nedić, J., Bharath Ganapathisubramani, and John Christos Vassilicos. "Drag and near wake characteristics of flat plates normal to the flow with fractal edge geometries." *Fluid Dynamics Research* 45, no. 6 (2013): 061406.