PROSES REVIEW ARTIKEL DI IJMERR

Judul: On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis

1. Submit artikel di IJMERR (10 Juli 2022)

Submission of 19889-1 Article

Participants Edit

Ms. Haylee Lin (haylee)

Rustan Tarakka (rustantarakka)

Messages

Note	From
Dear Sir/Madam The editor(s) of the International Journal of Mechanical Engineering and Robotics Research Herewith, We submit our paper titled "On the aerodynamics of rear of vehicle model with active control by blowing: computational and experimental analysis" authored by Rustan Tarakka, Nasaruddin Salam, Andi Amijoyo Mochtar, Wawan Rauf, and Muhammad Ihsan, to be considered in the International Journal of Mechanical Engineering and Robotics Research. I hope that the paper has a chance to be accepted and to be published subsequently in the esteemed journal and therefore can give contributions to our field. Thank you very much for your kind attention. Sincerely yours, Dr. Rustan Tarakka Dept. of Mechanical Engineering Hasanuddin University	rustantarakka 2022-07-10 03:28 AM

On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis

Rustan Tarakka, Nasaruddin Salam, Andi Amijoyo Mochtar,
Department of Mechanical Engineering, Hasanuddin University, Gowa, Indonesia,
Email: {rustan_tarakka@yahoo.com}{nassalam.unhas@yahoo.co.id}{andijoyo@unhas.ac.id}

Wawan Rauf
Department of Mechanical Engineering, Gorontalo University, Gorontalo, Indonesia
Email: {wawanrauf241193@yahoo.com }

Muhammad Ihsan
Department of Civil Engineering, Institut Sains Kesehatan Bone, Watampone, Indonesia
Email: {muhammadihsan@alumni.ait.asia}

***Abstract*—Aerodynamics related to the generation of drag due to flow separation that occurs at the rear of the vehicle is an important consideration in vehicle design. It includes flow separation, wake area formed, and the pressure, which in this paper is focused on the rear wall of the vehicle. Reduction of pressure can be differed significantly between the front and rear walls of the vehicle. This pressure difference can generate a phenomenon of backward pull and an increase in drag on the vehicle. To minimize back flow as well as to cater increasing pressure on the rear wall of the vehicle can be obtained by applying active control, including attaching blowing apparatus. This study aims to analyze the effect of the application of blowing active control on the aerodynamics on rear part of vehicles, which is represented by a modified Ahmed body, reversed in flow direction and altered dimensions. The research was conducted using a validated numerical simulation method with laboratory experimental testing at an upstream velocity of 16.7 m/s and blowing velocities of 0.5 m/s, 1.0 m/s, and 1.5 m/s, respectively. The results showed that the use of blowing active control was able to reduce aerodynamic drag, with the highest reduction is obtained in the model with a ratio of velocity UBL3/U0=0.09 of 12.187% for the computational approach and 11.556% for the experimental approach.**

***Index Terms*— aerodynamic drag, blowing active control, vehicle model**

I. Introduction

Aerodynamics related to the generation of drag due to flow separation that occurs at the rear of the vehicle is an important consideration in vehicle design. The magnitude of the aerodynamic drag force, works in contrast to the relative motion of a moving object, undergone by the vehicle will affect the vehicles' energy consumption and their stability [1][2]. This opposing movement occurs between the fluid and the surface of a solid object [3]. One approximation on amount of fuel consumption to overcome these adverse aerodynamic drag is about 50-60% [4]. Reduction of the aerodynamic drag by as small as 15% will contribute to 5-7% fuel consumption savings [5].

The aerodynamic drag on a vehicle is extremely related to the flow characteristics and the pressure distribution at the rear part of the vehicle, which are also influenced by flow separation occuring at the upper rear part of vehicles [6]. The design of this part is then very important in the effort of reducing aerodynamic drag [7]. The flow separation will cause backflow and decrease pressure field in the onset area of the separation. The rate of the flow separation will positively increase the extent of wake area which at the same time will reduce the pressure on the back wall area, which results in a significant difference in pressure between the front and rear parts, triggering the backward pull phenomenon [8]. Minimizing negative pressure and its intensity at the rear of the vehicle could reduce the aerodynamic drag of vehicles [4].

Flow engineering is a method that can be used to minimize back flow while increasing the pressure on the rear of the vehicle with a positive effect in delaying separation and reducing the re-circulation zone. It can be obtained by the application of active control such as blowing, or any combination with other forms of control, either active or passive [7] [9] [10] .

An experimental investigation on the influence of continuous blowing on the sharp edge between the roof and the rear window was published by Mestiri et al., based on a steady blowing attached in 25° tangent to the surface of the rear window of the slanted Ahmed body

Footnotes: 8-point Times New Roman font;

Manuscript received July 1, 2012; revised August 1, 2012; accepted September 1, 2012.

model, showing that tangential steady blowing can increase the separated region on the rear window and disturbing the appearance of the counter-rotating longitudinal vortex on the lateral edges of the rear window. The direct flow control was considere as really effective on the re-circulation region at the top of the rear window wall [11].

A study has been conducted on the potential of reduction of the aerodynamic resistance of a city-car prototype (a baseline version of the XAM 2.0) employing air blow and air relief flow control devices embedded into the vehicle's wheel, conducted in a dedicated wind tunnel, with special arrangement to reduce the turbulence of the front wheel and to cope with the air-flow defection at the end of the sides of the vehicle's body. The study also incorporated a CFD analysis to assess the effects of the modifications. By validating the drag optimization, a correlation between wind tunnel and CFD results was obtained, showing anticipated capabilities of CFD analysis and a record-breaking drag coefficient [12].

Tebbiche and Boutoudj have recently published an experimental investigation of the effect of the blowing flow rate and the Reynolds number for several incidences of the airfoil, two inclinations of the Ahmed body rear window, and a range of flow speeds (15-30 m/s) in a subsonic wind tunnel. The results came with interesting improvements in aerodynamic coefficients. For a tilted 20° rear window of Ahmed body model, a reduction in drag of 3.6% has been confirmed on the application of blowing with a minimum $C\mu = 0.28\%$, and a notable 15% reduction was gained by full blowing at maximum $C\mu = 4.79\%$. At the same study, a high-drag regime related to a 30° rear window slant has produced a 19% drag reduction with $C\mu = 5\%$ [13].

Cerutti et al. conducted an experimental investigation on the manipulation of the generation of on a square back car model by four continuously-blowing rectangular slot jets on Particle Image Velocimetry (pPIV) and stereoscopic PIV (sPIV) experiments and revealed a considerable alteration in the maximum drag reduction configuration on the formation of wake as the jets blow, while similar wake structure was best compromised [14].

II. METHOD

The research works on modified Ahmed body, altered in the orientation of the flow and the dimensional ratio to the original Ahmed body, set on 0.17 (1:6). The dimensions of the model are deifned length 174 mm length (l), 48 mm height (h), and 64.83 mm width (w), with 35° slant angle of the model's front geometry. The upper rear side of the vehicle model is designated as the area where the blowing active control is located. The active control is in the form of 5 appertures with a diameter of 7 mm with a distance between apertures of 10.81 mm. Each circle is defined as B_{L1}, B_{L2}, B_{L3}, B_{L4}, and B_{L5}. The tests have been conducted by setting the blowing velocity of each apertures (BL), U_{BL1}=0.5 m/s, U_{BL2}=1.0 m/s, and U_{BL3}=1.5 m/s at the upstream velocity U_0=16.7 m/s. Furthermore, the definition of the ratio of suction velocity to upstream velocity is written U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06, and U_{BL3}/U_0=0.09. The details of the test model are shown in Fig. 1.

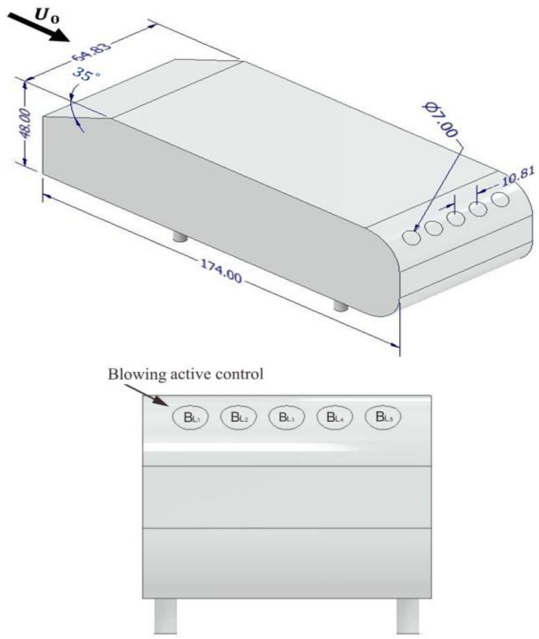


Figure 1. Test model

This research covers the flow characteristics passing through the rear of the vehicle model, the pressure field (C_p), and aerodynamic drag (C_d). Aerodynamic drag (C_d) is investigated by computational and by experimental observations, while for flow characteristics and pressure field (C_p), the investigation is only on numerical simulation. The data collection on pressure distribution is at the rear part of the vehicle model, considering the location is prone for separation of flow and wake to generate, resulting negative pressures contributing up to 80% of the total drag [15]. On the models' transversal axis, the pressure data are taken on 5 different grid lines, where the ratio of the grid width to the model width is defined as z/w=-1/2, z/w=-1/4, z/w=0, z/w=1/4, and z/w=1/2. On the height axis of the model, pressure data have been obtained on 5 grid lines for models without control and 4 grid lines for models with blowing active controls. The ratio of grid height to model height is written as y/h=0.17, y/h=0.33, y/h=0.50, y/h=0.67, and y/h=0.83. Details of the pressure field data collection area are shown in Fig. 2. The pressure field data is written into the pressure coefficient (C_p) value obtained through the use of (1) [16].

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

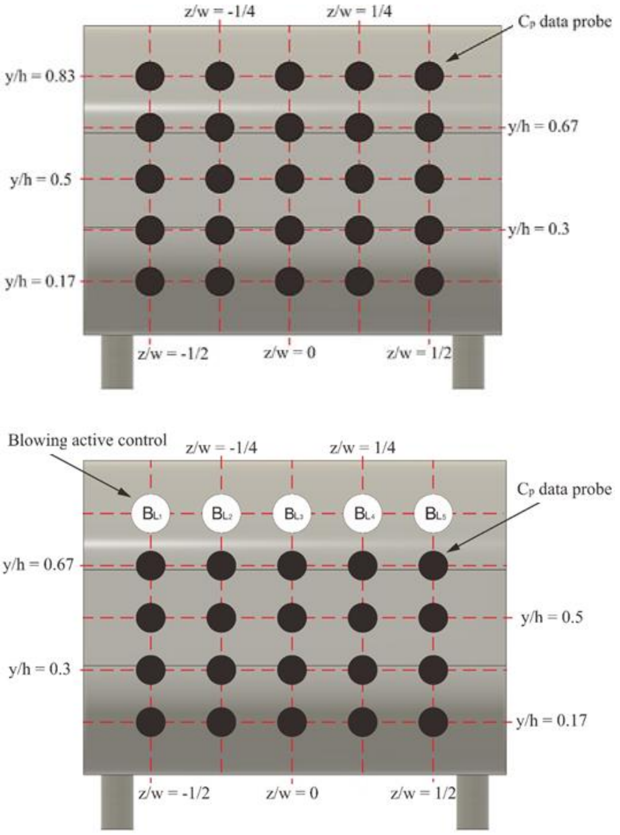


Fig. 2. C_p data probe: (a) without control, (b) with blowing active control

The computational simulation utilizes Fluent 6.3.26 ANSYS computational fluid dynamic software. The vehicle model is designed using Autodesk Inventor™ which is then defined into the computational domain as shown in Fig. 3 and undergone a meshing process using Gambit™ software as shown in Fig. 4. The computational conditions are described in table 1.

TABLE 1
COMPUTATIONAL CONDITIONS

Boundary condition	Type	Value
Fluid properties	Density	1.225 kg/m^3
	Viscosity	1.7894 × 10^{-5} kg/m.s
Model boundary conditions without control	Model	Wall
	Outlet	Pressure outlet
	Inlet	Velocity inlet
	Wall	Wall
Boundary conditions of the model with blowing active controls	Model	Wall
	Outlet	Pressure outlet
	Inlet	Velocity inlet
	Wall	Wall
	U_{BL1} U_{BL2}, U_{BL3}, U_{BL4}, U_{BL5}	Velocity inlet

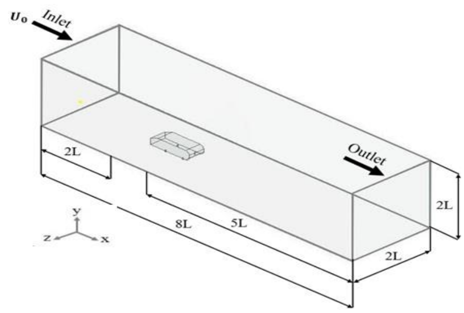


Figure 3. Computational domain

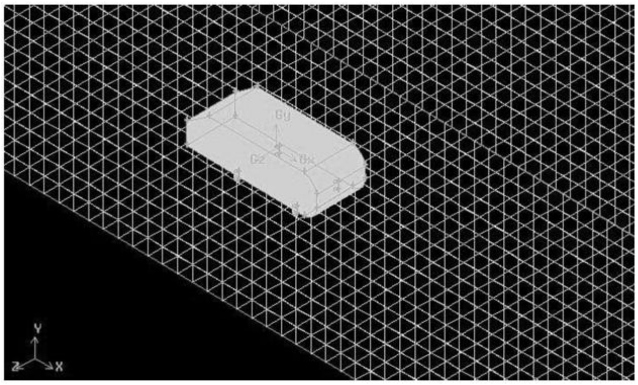


Figure 4. Computational mesh

Experimental testing has utilized the sub-sonic wind tunel facility. The measurement of drag uses a load cell equipped with an Arduino™ device and is directly connected to a computer. The duration of data retrieval is 120 seconds, generating 120 data for each velocity (U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06, and U_{BL3}/U_0=0.09). The 120 data are then averaged to gain drag values for each velocity comparison with a high degree of accuracy. The schematic of experimental setup is depicted in Fig. 5.

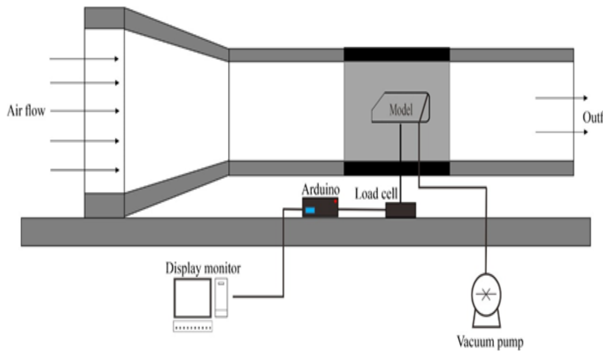


Figure 5. Experimental setup

The drag force is transformed into non-dimensional drag coefficient (C_d) by (2) [17]:

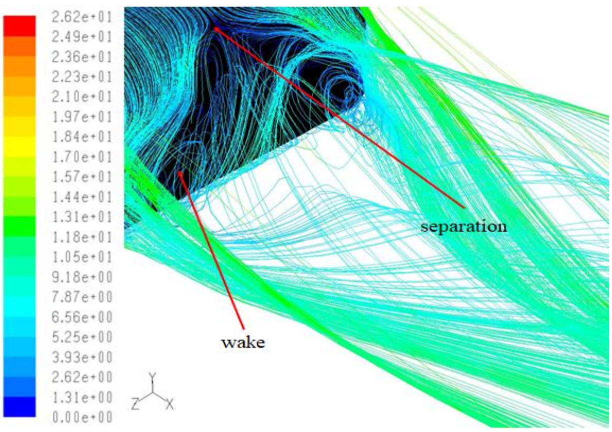
$$C_d = \frac{Fd}{\frac{1}{2}\rho v^2 A} \tag{2}$$

III. Results and discussion

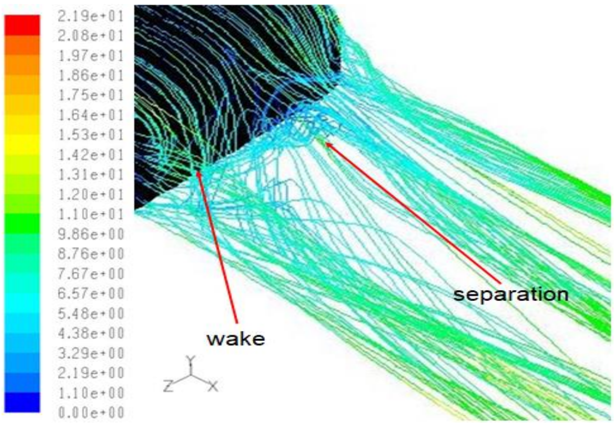
III.1. Flow field

The characteristics of the flow pattern of the model with active control in the ratio of blowing velocity to upstream velocity, U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06 and U_{BL3}/U_0=0.09 as well as models without control are shown in Fig. 6. The model without active control shows a large wake structure caused by flow separation on the upper side of the rear wall of the vehicle model as shown in Fig. 6(a). The fluid, which initially flows regularly, forms backflow and re-circulation areas. This is the main cause of negative pressure, which creates the phenomenon of backward pulling. This phenomenon of re-atraction has been suspected as a major contributor to the extent of aerodynamic drag on the vehicle, besides other presumed factor of longitudinal vortex [18]. Visually, the model without active control has a fairly large longitudinal vortex structure. This is because the fluid loses momentum to move along the rear body due to the adhesion friction force, resulting in a difference in flow velocity between the center and rear side of the vehicle model. This difference in velocity forces the flow on the middle side to move sideways to form a longitudinal vortex.

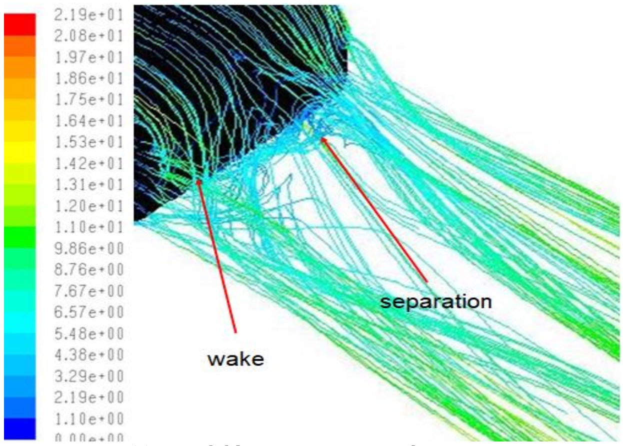
For models with the application of blowing active control at the velocity ratio, U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06 and U_{BL3}/U_0=0.09, as shown in Figs. 6(b), 6(c) and 6(c) respectively, the wake structure is smaller than the one on the model without application of control. This is because there is a delay in flow separation, where the separation process is formed in an area far from the rear wall of the vehicle model so that the intensity of the backflow interacting with the rear wall can be minimized. Besides, the longitudinal vortex that is formed tends to be smaller due to the blowing effect which forces a portion of the flow on the upper wall to move straight towards the downstream area. The model in the velocity ratio U_{BL3}/U_0=0.09 shows smaller wake formation compared to the model at the velocity U_{BL1}/U_0=0.03 and U_{BL2}/U_0=0.06.



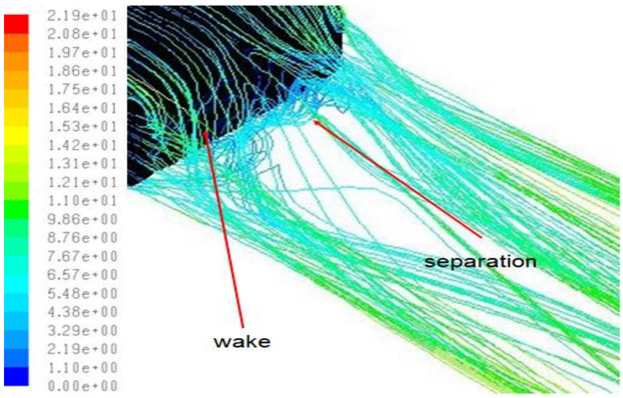
(a) Without control



(b) With blowing active control, U_{BL1}/U_0=0.03



(c) With blowing active control, U_{BL2}/U_0=0.06



(d) With active control, U_{BL3}/U_0=0.09

Figure 6. Flow field

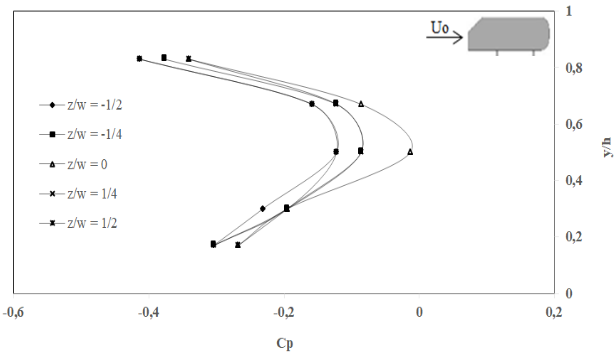
III.2. Pressure distribution

Table 2 shows the comparison of the minimum pressure coefficient for the model without active control and the model with blowing active control at the velocity ratio U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06, and U_{BL3}/U_0=0.09.

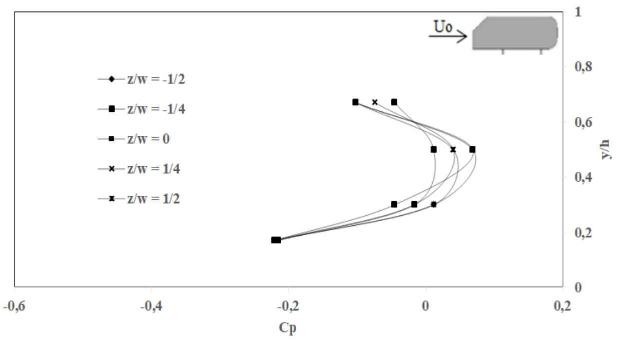
TABLE 2
COMPARISON OF THE MINIMUM PRESSURE COEFFICIENT

z/w	Minimum C_p			
	Without control	With blowing active control, U_{BL}/U_0		
		0.03	0.06	0.09
-1/2	-0.4132	-0.2206	-0.2260	-0.1865
-1/4	-0.3769	-0.2147	-0.2143	-0.1865
0	-0.3405	-0.2206	-0.2143	-0.1865
1/4	-0.3405	-0.2206	-0.2143	-0.1865
1/2	-0.4132	-0.2206	-0.2260	-0.1865
Rate	-0.3769	-0.2194	-0.2189	-0.1865
Increase (%)	-	41.7788	41.9080	50.5248

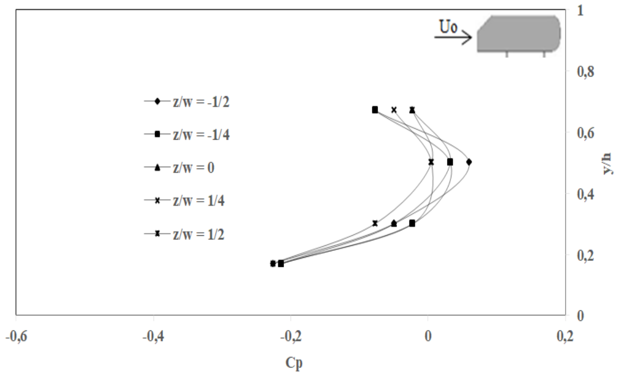
The lowest average minimum pressure coefficient value is found in the model without control at a value of -0.3769. The minimum pressure coefficient is obtained at the ratio of the grid height to the model height y/h=0.83 as shown in Fig. 7(a). This is because the position y/h=0.83 is the starting point for the flow separation. These findings are consistent with an opinion by Anderson revealing that the pressure coefficient is lower at the starting point of flow separation [2]. The values for the respective positions of the ratio of the grid width to the model width z/w=-1/2, z/w=-1/4, z/w=0, z/w=1/4, and z/w=1/2 is written as -0.4132, -0.3769, -0.3405, -0.3405, and -0.4132. These results correlate with the flow pattern characteristics in Fig. 6(a) which shows that the model without active control the model with the largest wake and vortex formation compared to other models.



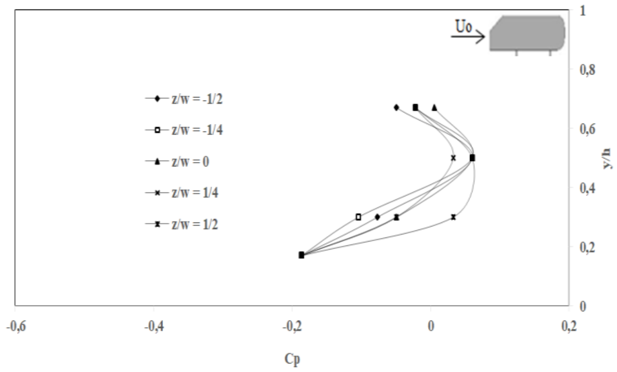
(a) Without control



(b) With blowing active control, U_{BL1}/U_0=0.03



(c) With blowing active control, U_{BL2}/U_0=0.06



(d) With blowing active control, U_{BL3}/U_0=0.09

Figure 7. Pressure distribution

For the model with velocity U_{BL1}/U_0=0.03, an increase in the average minimum pressure coefficient is apparent, as compared to the model without control. The average minimum pressure coefficient of -0.2194 is obtained at the position y/h=0.17 as shown in Fig. 7(b) with an increase in the percentage of 41.7788%. This is caused by backflow on the lower part of the rear of the model. The minimum pressure coefficient value at the z/w position is written respectively z/w=-1/2 of -0.2606, z/w=-1/4 of -0.2147, z/w=0 of -0.2206, z/w=1/4 of -0.2206, and z/w=1/2 of -0.2206.

In addition, the model in the ratio of velocity U_{BL2}/U_0=0.06 also shows an increase in the average minimum pressure coefficient with the increase is higher than the model at the velocity of U_{BL1}/U_0=0.03. This increase of 41.9080% with an average minimum pressure coefficient of -0.2189 is obtained at position y/h=0.17 which is shown in Fig. 7(c), where the value at each position z/w=-1/2, z/w=-1/4, z/w=0, z/w=1/4, and z/w=1/2 are written -0.2260, -0.2143, -0.2143, -0.2143, and -0.2260.

A similar phenomenon also occurs in a model with a velocity of U_{BL3}/U_0=0.09, where the application of blowing active control is able to increase the average minimum pressure coefficient even though the increase is higher than the model at the velocity U_{BL1}/U_0=0.03 and U_{BL2}/U_0=0.06. This increase is 50.5248% with an average minimum pressure coefficient value of -0.1865 obtained at position y/h=0.17 as shown in figure 7(d), where the value for each position of the ratio of the grid width to the model width is written z/w=-1/2 of -0.1865, z/w=-1/4 of -0.1865, z/w=0 of -0.1865, z/w=1/4 of -0.1865 and z/w=1/2 of -0.1865.

In overall the application of suction active control shows an increase in the average minimum pressure coefficient, where the model with the ratio of velocity U_{BL3}/U_0=0.09 is a model with the highest average minimum pressure coefficient increase. This correlates with the characteristics of the flow pattern in Fig. 6(d) which shows that the model at U_{BL3}/U_0=0.09 has the smallest wake formation that is tighter and more regular than other models. The research results confirm previous research which also concluded that the application of active control in the area where flow separation forms initially will give a potential development of the pressure field [19]–[22].

III.3. Aerodynamic drag

The drag coefficient obtained through the computational method is shown in table 3. The highest drag coefficient is found in the model without active control, which is 1.845. For models with the application of active control in various comparisons of blowing velocity to upstream velocity, shows a decrease in the value of the drag coefficient. The drag coefficient is written U_{BL1}/U_0=0.03 of 1.636, U_{BL2}/U_0=0.06 of 1.628, and U_{BL3}/U_0=0.09 of 1.620.

TABLE 3
COMPUTATIONAL METHOD DRAG COEFFICIENT

C_d			
Without control	U_{BL1}/U_0= 0.03	U_{BL2}/U_0= 0.06	U_{BL3}/U_0= 0.09
1.845	1.636	1.628	1.620

In the experimental approach, the highest drag coefficient value was also obtained in the model without active control of 1.763. For models with the application of active control on the ratio of blowing velocity to the upstream velocity, the drag coefficient decreases. The value of the drag coefficient at each speed is written as U_{BL1}/U_0=0.03 at 1.563, U_{BL2}/U_0=0.06 at 1.561, and U_{BL3}/U_0=0.09 at 1.559.

TABLE 4
EXPERIMENTAL METHOD DRAG COEFFICIENT

C_d			
Without control	U_{BL1}/U_0= 0.03	U_{BL2}/U_0= 0.06	U_{BL3}/U_0= 0.09
1.763	1.563	1.561	1.559

Table 5 shows the comparison of the drag coefficient of each model, both computationally and experimental. For the model without active control, the highest drag coefficient is obtained through both computational and experimental approaches with the difference in the drag coefficient of 4.434%. This result correlates with wake structure and the average minimum pressure coefficient value, where the model without active control is a model with the largest wake, vortex, and recirculation zone formation as shown in Fig. 6(a) and the model with the value of the lowest average minimum pressure coefficient as shown in table 2.

From table 5 in the U_{BL1}/U_0=0.03, the application of blowing active control gives optimum results both through computational and experimental methods. The reduction of each method was written 11.328% and 11.321%, respectively. The reduction for the velocity ratio U_{BL2}/U_0=0.06, is written as 11.741% for computation and 11.431% for experimental and for the ratio of velocity U_{BL3}/U_0=0.09, written as 12.187% for the computational approach and 11.556% for the experimental approach.

Overall, the application of blowing active control has a positive effect in the form of reduction of aerodynamic drag as shown in table 5, where the model at U_{BL3}/U_0=0.09 is the model with the highest aerodynamic drag reduction. This correlates with the characteristics of the flow pattern shown in Fig. 6(d), and the average minimum pressure coefficient value in table 2, where the model at U_{BL3}/U_0=0.09. The model with the smallest wake formation has a tighter and more regular flow line and the model with the highest average minimum pressure coefficient increase. The results gives similar fashion as what has been investigated by Harinaldi et al, citing that the application of active control could reduce aerodynamic drag [20].

TABLE 5
COMPARISON OF DRAG COEFFICIENTS

Method	Without control	Ubl_1/U = 0.03	Reduction (%)	Ubl_2/U_0 = 0.06	Reduction (%)	Ubl_3/U_0 = 0.09	Reduction (%)
Computational	1.845	1.636	11.328	1.628	11.741	1.620	12.187
Experimental	1.763	1.563	11.321	1.561	11.431	1.559	11.556
Difference (%)	4.434	4.428	-	4.099	-	3.747	-

IV. CONCLUSIONS

The application of blowing active control has positive effects on the characteristics of the flow pattern, where there is a delay in the flow separation process, and the largest reduction in wake formation is obtained in the model with the ratio of velocity, U_{BL3}/U_0=0.09.

The application of blowing active control also has positive effects on the minimum pressure coefficient, where there is an increase in the minimum pressure coefficient for all blowing velocity. The model with the ratio of velocity, U_{BL3}/U_0=0.09 is a model with an increase in the highest minimum pressure coefficient of 50.5248%. The positive effects are also shown on aerodynamic drag in the form of a reduction in the drag coefficient both computationally and experimentally. The highest reduction is obtained in the model with a ratio of velocity U_{BL3}/U_0=0.09 of 12.187% for the computational approach and 11.556% for the experimental approach.

CONFLICT OF INTEREST

The authors declare no conflict of interest.

AUTHOR CONTRIBUTIONS

Dr. R. Tarakka, the main and corresponding author, organized research promotion and conducted research planning. Dr. N. Salam, conducted research, Dr. A.A. Mochtar, conducted the research, Mr. W. Rauf, conducted experimental research, and Mr. M. Ihsan wrote and translated the manuscript. All authors approved the final version.

ACKNOWLEDGEMENTS

This research is funded by the Hasanuddin University Research and Community Service Institute through the 2021 University Basic Research (PDU) Scheme, Contract No. 915/UN4.22/PT.01.03/20. We extend our gratitude to the Head and Staff of the Fluid Mechanics Laboratory of the Hasanuddin University, Faculty of Engineering.

REFERENCES

[1] R. Tarakka, N. Salam, J. Jalaluddin, and M. Ihsan, "Effect of blowing flow control and front geometry towards the reduction of aerodynamic drag on vehicle models," *FME Transactions*, vol. 47, no. 3, pp. 552–559, 2019, doi: 10.5937/fmet1903552T.

[2] J. Anderson, *Fundamental of Aerodynamics*, 6th ed. Mc Graw Hill, 2017.

[3] P. Kundu, I. Cohen, and D. Dowling, *Fluid Mechanics - 6th Edition*, 6th ed. Academic Press, 2015. Accessed: Nov. 09, 2021. [Online]. Available: https://www.elsevier.com/books/fluid-mechanics/kundu/978-0-12-405935-1

[4] S. M. R. Hassan, T. Islam, M. Ali, and Md. Q. Islam, "Numerical Study on Aerodynamic Drag Reduction of Racing Cars," *Procedia Engineering*, vol. 90, pp. 308–313, Jan. 2014, doi: 10.1016/j.proeng.2014.11.854.

[5] M. Bellman, R. Agarwal, J. Naber, and L. Chusak, "Reducing Energy Consumption of Ground Vehicles by Active Flow Control," Dec. 2010, pp. 785–793. doi: 10.1115/ES2010-90363.

[6] P. Gopal and T. Senthilkumar, "Influence of Wake Characteristics of a Representative Car Model by Delaying Boundary Layer Separation," *Journal of Applied Science and Engineering*, vol. 16, no. 4, pp. 363–374, 2013, doi: 10.6180/jase.2013.16.4.04.

[7] T. Heinemann, M. Springer, H. Lienhart, S. Kniesburges, C. Othmer, and S. Becker, "Active flow control on a 1:4 car model," *Exp Fluids*, vol. 55, no. 5, p. 1738, May 2014, doi: 10.1007/s00348-014-1738-0.

[8] T. B. Hilleman, "Vehicle drag reduction with air scoop vortex impeller and trailing edge surface texture treatment," US7192077B1, Mar. 20, 2007 Accessed: Nov. 09, 2021. [Online]. Available: https://patents.google.com/patent/US7192077B1/en

[9] M. Jahanmiri, "Experimental investigation of drag reduction on ahmed car model using a combination of active flow control methods," *IJE*, vol. 24, no. 4, pp. 403–410, 2011, doi: 10.5829/idosi.ije.2011.24.04a.09.

[10] C. H. Bruneau, E. Creusé, D. Depeyras, P. Gilliéron, and I. Mortazavi, "Coupling active and passive techniques to control the flow past the square back Ahmed body," *Computers and Fluids*, vol. 38, no. 10, p. 1875, 2010, doi: 10.1016/j.compfluid.2010.06.019.

[11] R. Mestiri, A. Ahmed-Bensoltane, L. Keirsbulck, F. Aloui, and L. Labraga, "Active Flow Control at the Rear End of a Generic Car Model Using Steady Blowing," *Journal of Applied Fluid Mechanics*, vol. 7, pp. 565–571, Oct. 2014.

[12] A. Ferraris, H. de C. Pinheiro, A. G. Airale, M. Carello, and D. B. Polato, "City Car Drag Reduction by means of Flow Control Devices," SAE International, Warrendale, PA, SAE Technical Paper 2020-36–0080, Mar. 2021. doi: 10.4271/2020-36-0080.

[13] H. Tebbiche and M. S. Boutoudj, "Active flow control by micro-blowing and effects on aerodynamic performances. Ahmed body and NACA 0015 airfoil," *FMR*, vol. 48, no. 2, 2021, doi: 10.1615/InterJFluidMechRes.2021036842.

[14] J. J. Cerutti, C. Sardu, G. Cafiero, and G. Iuso, "Active Flow Control on a Square-Back Road Vehicle," *Fluids*, vol. 5, no. 2, Art. no. 2, Jun. 2020, doi: 10.3390/fluids5020055.

[15] M. N. Sudin, M. A. Abdullah, S. A. Shamsuddin, F. R. Ramli, and M. Mohd, "Review of Research

on Vehicles Aerodynamic Drag Reduction Methods," *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS*, vol. 14, no. 2, pp. 35–47, 2014.
[16] B. R. Munson, D. F. Young, and T. H. Okiishi, *Fundamentals of fluid mechanics*, 4th ed. John Wiley & Sons Inc, 2002.
[17] Y. A. Çengel and J. M. Cimbala, *Fluid Mechanics: Fundamentals and Applications*. McGraw-Hill Education, 2018.
[18] M. Onorato, A. F. Costelli, and A. Garrone, "Drag Measurement Through Wake Analysis," Feb. 1984, p. 840302. doi: 10.4271/840302.
[19] W. Rauf, R. Tarakka, Jalaluddin, and M. Ihsan, "Effect of Flow Separation Control with Suction Velocity Variation: Study of Flow Characteristics, Pressure Coefficient, and Drag Coefficient," *Universal Journal of Mechanical Engineering*, vol. 8, no. 3, pp. 142–151, May 2020, doi: 10.13189/ujme.2020.080302.
[20] Harinaldi, Budiarso, R. Tarakka, and S. P. Simanungkalit, "Computational Analysis of Active Flow Control to Reduce Aerodynamics Drag on a Van Model," *International Journal of Mechanical & Mechatronics Engineering*, vol. 11, no. 03, pp. 24–30, 2011.
[21] S. Krajnović and J. Fernandes, "Numerical simulation of the flow around a simplified vehicle model with active flow control," *International Journal of Heat and Fluid Flow*, vol. 32, no. 1, pp. 192–200, Feb. 2011, doi: 10.1016/j.ijheatfluidflow.2010.06.007.
[22] R. Tarakka, Jalaluddin, B. Mire, and M. N. Umar, "Effect of turbulence model in computational analysis of active flow control on aerodynamic drag of bluff body van model," *International Journal of Applied Engineering Research*, vol. 10, no. 1, pp. 207–219, 2015.

Authors' information

Rustan Tarakka – is a Lecturer in the Department of Mechanical Engineering, Faculty of Engineering, Hasanuddin University, Makassar, Indonesia. He holds a doctoral degree from University of Indonesia, Jakarta, Indonesia. His research areas are on fluid dynamics and computational fluid dynamics..

Nasaruddin Salam – is a Professor and the Chairman of Fuild Mechanics Laboratory in Department of Mechanical Engineering, Faculty of Engineering, Hasanuddin University Makassar Indonesia. He holds a doctoral degree from Brawijaya University, Malang Indonesia. His research fields include fluid dynamics.

Andi Amijoyo Mochtar –obtained a Doctor of Engineering in Mechanical Engineering in 2016 from Ehime University Japan. He is an Senior Lecturer of Mechanical Engineering of Hasanuddin University, Makassar, Indonesia..

Wawan Rauf – obtained a Master of Engineering in Mechanical Engineering in 2020 from Hasanuddin University. His research areas are on fluid dynamics and computational fluid dynamics.

Muhammad Ihsan –works for Institut Sains Kesehatan Bone, Watampone, Indonesia. He also holds masters degrees in transport engineering from Asian Institute of Technology, Bangkok, Thailand and Universitas Gajah Mada, Yogyakarta, Indonesia.

Word count : 4,310

2. Informasi artikel telah diterima untuk direview oleh IJMERR

[ijmerr] Manuscript ID: IJMERR-6078 - Submission Confirmation ✖

Participants

Ms. Ashley Zhang (ashley)

Rustan Tarakka (rustantarakka)

Messages

Note	From
Dear Rustan Tarakka: Thank you for submitting your manuscript "On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis" to International Journal of Mechanical Engineering and Robotics Research. Submission URL: http://ojs.ejournal.net/index.php/ijmerr/authorDashboard/submission/6078 Before further processing, please confirm that your submission meets the requirements below: 1. **Publication fee:** you support open access publishing, which allows unlimited access to your published paper, and that you will pay the Article Processing Charge (450 USD, http://www.ijmerr.com/index.php?m=content&c=index&a=lists&catid=13), please note that the APC only applies if your paper was accepted after standard peer-review. 2. The submitted paper has not been copyrighted, published, or accepted for publication elsewhere. 3. The submitted paper contains no proprietary material unprotected by patent or patent application. 4. The submitted paper contains no plagiarism/copying and fraudulent data. If your submission meets the above items, please confirm on the submission system in "Pre-Review Discussions" and leave a message to editors. Thank you in advance for your cooperation. We look forward to hearing from you. Ms. Ashley Zhang/Handing Editor ashley.zhang@ejournal.net	ashley 2022-07-11 11:07 AM

3. Informasi artikel telah direview oleh Reviewer IJMERR (8 Agustus 20220)

[ijmerr] Manuscript ID: IJMERR-6078 - Editor Decision - Major Revisions

2022-08-08 03:39 PM

Dear Rustan Tarakka, Salam Nasaruddin, Andi Amijoyo Mochtar, Wawan Rauf, Muhammad Ihsan:

Thank you for submitting your manuscript "On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis" to International Journal of Mechanical Engineering and Robotics Research.

The editorial team had assessed your submission and feel that it has potential for publication, so we would like to invite you to make major revisions for further review.

You can find your manuscript at the following link:
http://ojs.ejournal.net/index.php/ijmerr/authorDashboard/submission/6078

Important notice: Please revise the manuscript according to the reviewers' comments and upload the revised file **within one month. The revisions should be clearly highlighted,** for example using the "Track Changes" function in Microsoft Word, so that changes are easily visible to the editors and reviewers. Please provide a cover letter to explain point-by-point the details of the revisions in the manuscript and **your responses** to the reviewers' comments. (**download author response template**)

As the reviewer have suggested that your manuscript should **undergo extensive English editing**, please address this during revision. We suggest that you have your manuscript checked by a native English-speaking colleague or use a professional English editing service.

Instruction for uploading the revised version can be found at https://docs.pkp.sfu.ca/learning-ojs/en/authoring.

Do not hesitate to contact us if you have any questions regarding the revision of your manuscript.

Ms. Ashley Zhang/Handling Editor
ashley.zhang@ejournal.net

Reviewer A:

Comments to Authors

The authors performed a numerical and experimental research on drag reduction using active control by blowing jets into the flow field. Active flow control is a cutting-edge technology widely studied in aerodynamic drag reduction for automotive vehicle in last decades. Three levels of jet velocity were investigated to determine the influence of the jets on the aerodynamic drag. Pressure recovery was obtained through this control method.

Major issues:

*1. The model that the authors studied is a 35° Ahmed body. This model was introduced by Ahmed SR and widely used in academic research. The critical slant angles correspond to different separations behind the model. However, the authors flipped the orientation of the model by 180°, which eliminates this characteristic. There are neither previous researches to support this alteration, nor any reasons given in this article.

*2. In previous researches, such as Mathieu Roumeas et al(Drag reduction by flow separation control on a car after body) and Wang Bingxin et al(Active flow control on the 25°Ahmed body using a new unsteady jet), it's found that the velocity of the jets(U/U0) or the momentum coefficient (Cμ)should reach a certain level(U/U0=0.4~0.6) to obtain a drag reduction of approximately 12%. In this article, the authors claimed that they obtained a reduction of 12.187% with a ratio of U/U0=0.09. The accuracy of the results is questionable.

3. The explanation of the numerical model should be specified: the turbulent model used to calculate the flow field (RANS or LES), the mesh size and number, et al.

4. In the experiment setup(fig 5), a vacuum pump was used to obtain control flow. However, the vacuum pump was supposed to create suction jet instead of blowing jet. Meanwhile, the model seemed to be suspended in the middle of the test section. Whether or not the ground clearance was considered and this setup was aligned with the numerical configuration.

5. The background reviews in Introduction seemed to be not thorough or comprehensive. More studies could be cited.

Minor issues:

1. Check the spelling mistake, e.g. Page 2 “deifned”, “appertures”.

2. The format of the tables should be unified.

3. The flow field structures(fig 6) should also be demonstrated in 2D planes for better understanding the changes in the flow.

Reviewer B:

Comments to Authors

At the outset I would like to says that paper is very well written and streamline the need to minimizes the fuel related pollution. It is good to see the experimental work is supplemented by computational result.

This paper is a good effort towards controlling the flow separation by placing active control device.

However, there are some points which needs to be clarified as follows.

1. In second last para Cμ is not explained what it is. Kindly give the definition of this coefficient.
2. In the first para of second section at first authors mentions blowing velocity and then suction velocity. In my understanding suction should be changed to blowing.

Apart from these some point needs to be included to assist in understanding of paper.

1. State whether the flow is turbulent and if yes which turbulence modal are used.
2. Mesh details must be included also perform mesh independence study.
3. Justify the domain size. May refer a published paper where comparable domain size was chosen for same problem.
4. In results and discussion flow field should be presented on mid section with clearly showing separation point and wake length. The separation point location and wake length can be correlated with drag.
5. On the numerical framework, the authors should mention the type of turbulence model that they used. Also, they need to mention if they conducted sensitivity studies on the spatial and temporal grids. The introduction part should be enhanced. There are recent studies on the exact same topic that should be discussed briefly.

Reviewer C:

Comments to Authors

This paper analyzed the effect of the application of active blowing control on the aerodynamics of rear part of a 1:6 scaled Ahmed body. Sentences are quite long and complicated, and the structure of many sentences is incorrect, which does not support the understanding of the paper. In addition, there are typos like “appertures” and extra punctuations. Some major issues are as follows:

1. The quality of the figures is very low and should be improved.
2. The difference of the production of pressure between the front and rear walls generates the pressure resistance, hence the sentence “Reduction of pressure ...... an increase in drag on the vehicle” is confusing.
3. Sentence “The magnitude of the aerodynamic drag force, ... and their stability” is confusing. In addition, the citation of reference [3] is to explain the “opposing movement”, which is opposite to the sentence above.
4. Sentence “The rate of the flow separation ...... triggering the backward pull phenomenon”. The increase/decrease of the rate should be described clearly.
5. The meaning of $C\mu$ should be explained.
6. The depth of the apertures should be described.
7. The main subject of this paper is “blowing active control”, but there are two “suction” that showed up in different sentences, which is confusing.
8. The mark of the model’s length is “l” as described in this paper, so the “L” in Figure 3 is confusing. In addition, the boundary condition of the four side walls should be represented in Figure 3.
9. Clear description of meshing parameters is needed. Figure 4 is not enough to reveal the details of meshing. Besides, there may be an extra line between the picture and the words below.
10. The numerical simulation method is not described in detail, such as the turbulence model, solution format, discrete term, etc.
11. There are two “6(c)” in one sentence.
12. There are many mistakes in Figure 6.

(1) The size and position of the vehicle in four pics are different.
(2) The color maps of four pics aren’t the same.
(3) Lacking of coordinate in Fig 6(c) and (d).
(4) Four pics are far from enough to describe the generation of vertices and the influence of blowing at different speeds on the production and development of vertices.
(5) The title of Fig 6(d) is incorrect.

13. Table 2 is incorrect. the Cp of z/w=0 and z/w=1/4 are the same, and the “Rate” is not defined.
14. English of the manuscript is very poor. In the present form, the manuscript may not be evaluation properly. Thus, the manuscript should be evaluated after improving the English.

----

4. Revisi artikel berdasarkan komentar Reviewer IJMERR (7 September 2022)

Participants Edit

Ms. Ashley Zhang (ashley)

Rustan Tarakka (rustantarakka)

Messages	
Note	From
Dear Ms. Ashley Zhang Journal Editor of IJMERR (International Journal of Mechanical Engineering and Robotics Research) ashley.zhang@ejournal.net We have revised our paper "On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis" authored by Rustan Tarakka, Nasaruddin Salam, Andi Amijoyo Mochtar, Wawan Rauf, Muhammad Ihsan, to be considered in the International Journal of Mechanical Engineering and Robotics Research, based on reviewer's comments. All the reviewer's comment are adressed in Comment field in the manuscript. We hope that the revision could fulfill what the reviewers want. Thank you very much. Thank you very much for your kind attention. Sincerely yours, Corresponding author Dr. Rustan Tarakka Dept. of Mechanical Engineering Hasanuddin University rustantarakka, 6078-Manuscript (PDF)-19889-1-2-20220710 - Revised#1.docx	rustantarakka 2022-09-07 11:45 AM

On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis

Rustan Tarakka, Nasaruddin Salam, Andi Amijoyo Mochtar,
Department of Mechanical Engineering, Hasanuddin University, Gowa, Indonesia,
Email: {rustan_tarakka@yahoo.com}{nassalam.unhas@yahoo.co.id}{andijoyo@unhas.ac.id}

Wawan Rauf
Department of Mechanical Engineering, Gorontalo University, Gorontalo, Indonesia
Email: {wawanrauf241193@yahoo.com }

Muhammad Ihsan
Department of Civil Engineering, Sekolah Tinggi Teknik Baramuli, Pinrang, Indonesia
Email: {muhammadihsan@alumni.ait.asia}

***Abstract*—Aerodynamics related to the generation of drag due to flow separation that occurs at the rear of the vehicle is an important consideration in vehicle design. It includes flow separation, wake area formed, and the pressure, which in this paper is focused on the rear wall of the vehicle. Reduction of pressure can be differed significantly between the front and rear walls of the vehicle. This pressure difference can generate a phenomenon of backward pull and an increase in drag on the vehicle. To minimize back flow as well as to cater increasing pressure on the rear wall of the vehicle can be obtained by applying active control, including attaching blowing apparatus. This study aims to analyze the effect of the application of blowing active control on the aerodynamics on rear part of vehicles, which is represented by a modified Ahmed body, reversed in flow direction and altered dimensions. The research was conducted using a validated numerical simulation method with laboratory experimental testing at an upstream velocity of 16.7 m/s and blowing velocities of 0.5 m/s, 1.0 m/s, and 1.5 m/s, respectively. The results showed that the use of blowing active control was able to reduce aerodynamic drag, with the highest reduction is obtained in the model with a ratio of velocity UBL3/U0=0.09 of 12.187% for the computational approach and 11.556% for the experimental approach.**

***Index Terms*— aerodynamic drag, blowing active control, vehicle model**

I. Introduction

Aerodynamics related to the generation of drag due to flow separation that occurs at the rear of the vehicle is an important consideration in vehicle design. The magnitude of the aerodynamic drag force, works in contrast to the relative motion of a moving object, undergone by the vehicle will affect the vehicles' energy consumption and their stability [1][2]. This opposing movement occurs between the fluid and the surface of a solid object [3]. One approximation on amount of fuel consumption to overcome these adverse aerodynamic drag is about 50-60% [4]. Reduction of the aerodynamic drag by as small as 15% will contribute to 5-7% fuel consumption savings [5].

The aerodynamic drag on a vehicle is extremely related to the flow characteristics and the pressure distribution at the rear part of the vehicle, which are also influenced by flow separation occuring at the upper rear part of vehicles [6]. The design of this part is then very important in the effort of reducing aerodynamic drag [7]. The flow separation will cause backflow and decrease pressure field in the onset area of the separation. The rate of the flow separation will positively increase the extent of wake area which at the same time will reduce the pressure on the back wall area, which results in a significant difference in pressure between the front and rear parts, triggering the backward pull phenomenon [8]. Minimizing negative pressure and its intensity at the rear of the vehicle could reduce the aerodynamic drag of vehicles [4].

Flow engineering is a method that can be used to minimize back flow while increasing the pressure on the rear of the vehicle with a positive effect in delaying separation and reducing the re-circulation zone. It can be obtained by the application of active control such as blowing, or any combination with other forms of control, either active or passive [7] [9] [10] .

An experimental investigation on the influence of continuous blowing on the sharp edge between the roof and the rear window was published by Mestiri et al., based on a steady blowing attached in 25° tangent to the surface of the rear window of the slanted Ahmed body

Manuscript received July 1, 2012; revised August 1, 2012; accepted September 1, 2012.
Corresponding author : rustan_tarakka@yahoo.com

model, showing that tangential steady blowing can increase the separated region on the rear window and disturbing the appearance of the counter-rotating longitudinal vortex on the lateral edges of the rear window. The direct flow control was considere as really effective on the re-circulation region at the top of the rear window wall [11].

A study has been conducted on the potential of reduction of the aerodynamic resistance of a city-car prototype (a baseline version of the XAM 2.0) employing air blow and air relief flow control devices embedded into the vehicle's wheel, conducted in a dedicated wind tunnel, with special arrangement to reduce the turbulence of the front wheel and to cope with the air-flow defection at the end of the sides of the vehicle's body. The study also incorporated a CFD analysis to assess the effects of the modifications. By validating the drag optimization, a correlation between wind tunnel and CFD results was obtained, showing anticipated capabilities of CFD analysis and a record-breaking drag coefficient [12].

Tebbiche and Boutoudj have recently published an experimental investigation of the effect of the blowing flow rate and the Reynolds number for several incidences of the airfoil, two inclinations of the Ahmed body rear window, and a range of flow speeds (15-30 m/s) in a subsonic wind tunnel. The results came with interesting improvements in aerodynamic coefficients. For a tilted 20° rear window of Ahmed body model, a reduction in drag of 3.6% has been confirmed on the application of blowing with a minimum blowing intensity Cμ = 0.28%, and a notable15% reduction was gained by full blowing at maximum Cμ = 4.79%. At the same study, a high-drag regimerelated to a 30° rear window slant has produced a 19% drag reduction with Cμ = 5% [13].

Cerutti et al. conducted an experimental investigation on the manipulation of the generation of on a square back car model by four continuously-blowing rectangular slot jets on Particle Image Velocimetry (pPIV) and stereoscopic PIV (sPIV) experiments and revealed a considerable alteration in the maximum drag reduction configuration on the formation of wake as the jets blow, while similar wake structure was best compromised [14].

II. METHOD

The research works on modified Ahmed body, altered in the orientation of the flow and the dimensional ratio to the original Ahmed body, set on 0.17 (1:6). The dimensions of the model are defined length 174 mm length (l), 48 mm height (l), and 64.83 mm width (w), with 35° slant angle of the model's front geometry. The upper rear side of the vehicle model is designated as the area where the blowing active control is located. The active control is in the form of 5 apertures with a diameter of 7 mm with a distance between apertures of 10.81 mm. Each circle is defined as B_{L1}, B_{L2}, B_{L3}, B_{L4}, and B_{L5}. The tests have been conducted by setting the blowing velocity of each apertures (BL), U_{BL1}=0.5 m/s, U_{BL2}=1.0 m/s, and U_{BL3}=1.5 m/s at the upstream velocity U_0=16.7 m/s. Furthermore, the definition of the ratio of blowing velocity to upstream velocity is written U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06, and U_{BL3}/U_0=0.09. The details of the test model are shown in Fig. 1.

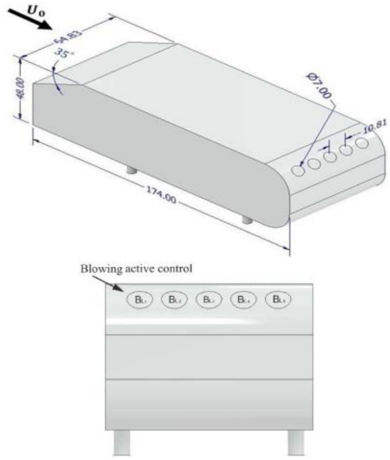


Figure 1. Test model

This research covers the flow characteristics passing through the rear of the vehicle model, the pressure field (C_p), and aerodynamic drag (C_d). Aerodynamic drag (C_d) is investigated by computational and by experimental observations, while for flow characteristics and pressure field (C_p), the investigation is only on numerical simulation using a standard k—epsilon turbulence model with tetra hedral meshing.. The definition of turbulence model and meshing type is based on previous work of Harinaldi et al [15]. The data collection on pressure distribution is at the rear part of the vehicle model, considering the location is prone for separation of flow and wake to generate, resulting negative pressures contributing up to 80% of the total drag [16]. On the models' transversal axis, the pressure data are taken on 5 different grid lines, where the ratio of the grid width to the model width is defined as z/w=-1/2, z/w=-1/4, z/w=0, z/w=1/4, and z/w=1/2. On the height axis of the model, pressure data have been obtained on 5 grid lines for models without control and 4 grid lines for models with blowing active controls. The ratio of grid height to model height is written as y/h=0.17, y/h=0.33, y/h=0.50, y/h=0.67, and y/h=0.83. Details of the pressure field data collection area are shown in Fig. 2. The pressure field data is written into the pressure coefficient (C_p) value obtained through the use of (1) [17].

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

Commented [A8]: Reviewer A point 3, Reviewer B point 1, 2, 5 and Reviewer C point 10. Turbulence model and mesh types have been added to the manuscript. Mesh generated are over 1 million.

Commented [A1]: Reviewer B point 1 and Reviewer C Point 5: The symbol was used by H. Tebbiche and M.S. Boutoudj to represent blowing intensity. The symbol follows the one on their paper.

Commented [A2]: Reviewer A point 5: Authors include 14 references to adequately introduce the topic.

Commented [A3]: Reviewer A point 1. "Deifned" has been change to "defined".

Commented [A4]: Reviewer A point 1: The shape of our test model follows reversed Ahmed body, as the basis of our research. Reversed Ahmed models have been investigated by numerous researcher, including Harinaldi et al, in "Computational Analysis of Active Flow Control to Reduce Aerodynamics Drag on a Van Model".

Commented [i-[5]: Reviewer A point 1. "appertures" has been change to "apertures".

Commented [A6]: Reviewer C point 6: Depth of the holes is not applicable since they are connected to the blower.

Commented [A7]: Reviewer B point 2 dan reviewer C point 7. "Suction" has been changed to "blowing"

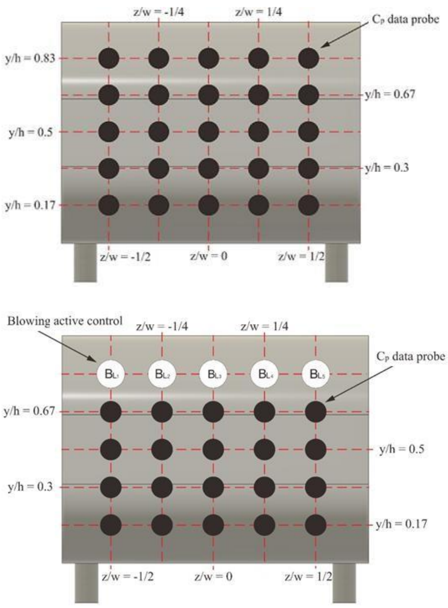


Fig. 2. C_p data probe: (a) without control, (b) with blowing active control

The computational simulation utilizes Fluent 6.3.26 ANSYS computational fluid dynamic software. The vehicle model is designed using Autodesk Inventor™ which is then defined into the computational domain as shown in Fig. 3 and undergone a meshing process using Gambit™ software as shown in Fig. 4. The computational conditions are described in table 1.

TABLE 1
COMPUTATIONAL CONDITIONS

Boundary condition	Type	Value
Fluid properties	Density	1.225 kg/m^3
	Viscosity	1.7894 × 10^{-5} kg/m.s
Model boundary conditions without control	Model	Wall
	Outlet	Pressure outlet
	Inlet	Velocity inlet
	Wall	Wall
Boundary conditions of the model with blowing active controls	Model	Wall
	Outlet	Pressure outlet
	Inlet	Velocity inlet
	Wall	Wall
	U_{BL1} U_{BL2}, U_{BL3}, U_{BL4}, U_{BL5}	Velocity inlet

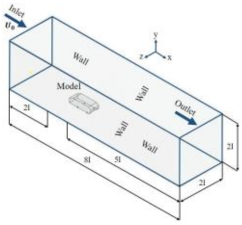


Figure 3. Computational domain

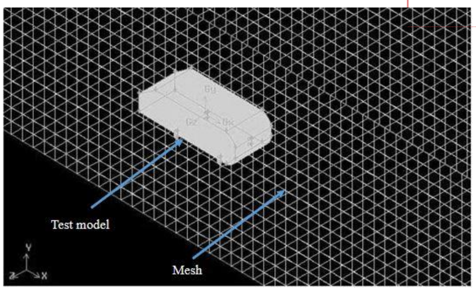


Figure 4. Computational mesh

The experimental testing has utilized the sub-sonic wind tunel facility. The measurement of drag uses a load cell equipped with an Arduino™ device and is directly connected to a computer. The duration of data retrieval is 120 seconds, generating 120 data for each velocity (U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06, and U_{BL3}/U_0=0.09). The 120 data are then averaged to gain drag values for each velocity comparison with a high degree of accuracy. The schematic of experimental setup is depicted in Fig. 5.

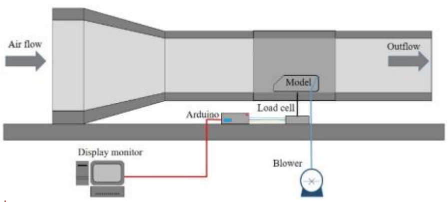


Figure 5. Experimental setup

The drag force is transformed into non-dimensional drag coefficient (C_d) by (2) [18]:

$$C_d = \frac{F_d}{\frac{1}{2}\rho v^2 A} \quad (2)$$

Commented [A9]: Reviewer C point 9 : Notation has been added to mesh display.

Commented [A10]: Reviewer B point 3: the size of computational domain refers to some results, including by Harinaldi et al., Computational Analysis of Active Flow Control to Reduce Aerodynamics Drag on a Van Model.

Commented [A11]: Reviewer C point 8: The capitalized "L" has been change to "l" in figure 3. Boundary conditions of 4 walss have been added and described in Table 1.

Commented [A12]: Reviewer A point 4: Notation has been improved. Boundary conditions of computations and experiments are the same, which are in the middle part of test section, to allow uniform flow.

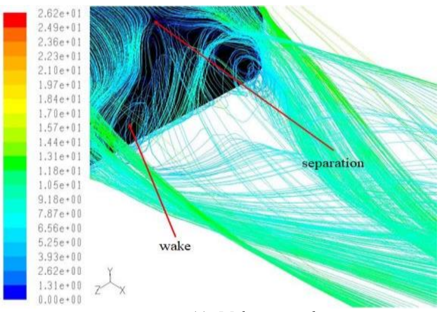
Commented [A13]: Reviewer C point 1 : Some figures has been change to a better resolution.

III. Results and Discussion

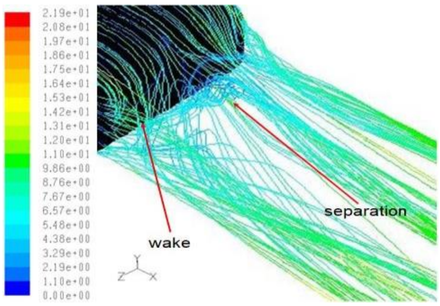
III.1. Flow field

The characteristics of the flow pattern of the model with active control in the ratio of blowing velocity to upstream velocity, $U_{BL1}/U_0=0.03$, $U_{BL2}/U_0=0.06$ and $U_{BL3}/U_0=0.09$ as well as models without control are shown in Fig. 6. The model without active control shows a large wake structure caused by flow separation on the upper side of the rear wall of the vehicle model as shown in Fig. 6(a). The fluid, which initially flows regularly, forms backflow and re-circulation areas. This is the main cause of negative pressure, which creates the phenomenon of backward pulling. This phenomenon of re-atraction has been suspected as a major contributor to the extent of aerodynamic drag on the vehicle, besides other presumed factor of longitudinal vortex [19]. Visually, the model without active control has a fairly large longitudinal vortex structure. This is because the fluid loses momentum to move along the rear body due to the adhesion friction force, resulting in a difference in flow velocity between the center and rear side of the vehicle model. This difference in velocity forces the flow on the middle side to move sideways to form a longitudinal vortex.

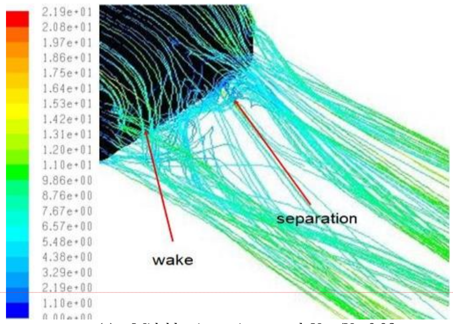
For models with the application of blowing active control at the velocity ratio, $U_{BL1}/U_0=0.03$, $U_{BL2}/U_0=0.06$ and $U_{BL3}/U_0=0.09$, as shown in Figs. 6(b), 6(c) and 6(d) respectively, the wake structure is smaller than the one on the model without application of control. This is because there is a delay in flow separation, where the separation process is formed in an area far from the rear wall of the vehicle model so that the intensity of the backflow interacting with the rear wall can be minimized. Besides, the longitudinal vortex that is formed tends to be smaller due to the blowing effect which forces a portion of the flow on the upper wall to move straight towards the downstream area. The model in the velocity ratio $U_{BL3}/U_0=0.09$ shows smaller wake formation compared to the model at the velocity $U_{BL1}/U_0=0.03$ and $U_{BL2}/U_0=0.06$.



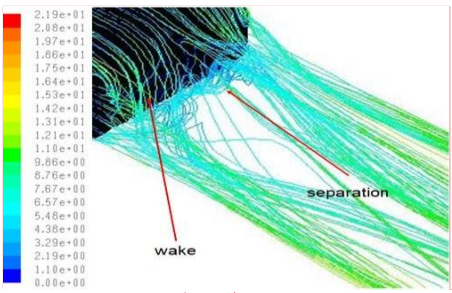
(a) Without control



(b) With blowing active control, $U_{BL1}/U_0=0.03$



(c) With blowing active control, $U_{BL2}/U_0=0.06$



(d) With blowing active control, $U_{BL3}/U_0=0.09$

Figure 6. Flow field

III.2. Pressure distribution

Table 2 shows the comparison of the minimum pressure coefficient for the model without active control and the model with blowing active control at the velocity ratio $U_{BL1}/U_0=0.03$, $U_{BL2}/U_0=0.06$, and $U_{BL3}/U_0=0.09$.

Commented [A14]: Reviewer C point 11: Correction has been done.

Commented [A15]: Reviewer A point 3: Test model is used to compare the characteristics of flow with and wtihout active blowing controls. Since the test model uses active control, 2D representation can not be shown.

Commented [A16]: Reviewer C point 12(5): Correction has been done.

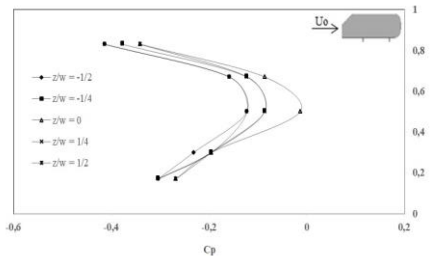
Commented [A17]: Reviewer B Point 4 and Reviewer C point 12: In authors opinion, figures 6 is sufficient to show the onset of separation and wake formation.

TABLE 2

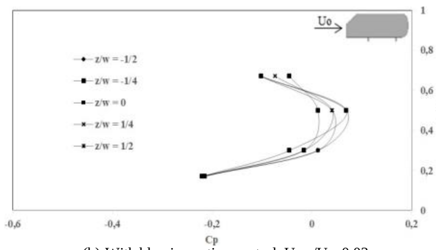
COMPARISON OF THE MINIMUM PRESSURE COEFFICIENT

z/w	Minimum C_p			
	Without control	With blowing active control, U_{BL}/U_0		
		0.03	0.06	0.09
-1/2	-0.4132	-0.2206	-0.2260	-0.1865
-1/4	-0.3769	-0.2147	-0.2143	-0.1865
0	-0.3405	-0.2206	-0.2143	-0.1865
1/4	-0.3405	-0.2206	-0.2143	-0.1865
1/2	-0.4132	-0.2206	-0.2260	-0.1865
Average	-0.3769	-0.2194	-0.2189	-0.1865
Increase (%)	-	41.7788	41.9080	50.5248

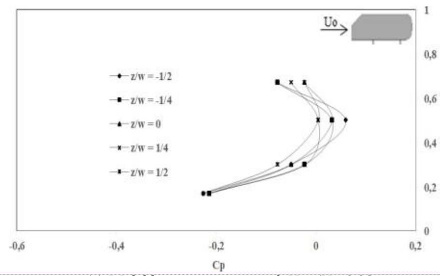
The lowest average minimum pressure coefficient value is found in the model without control at a value of -0.3769. The minimum pressure coefficient is obtained at the ratio of the grid height to the model height y/h=0.83 as shown in Fig. 7(a). This is because the position y/h=0.83 is the starting point for the flow separation. These findings are consistent with an opinion by Anderson revealing that the pressure coefficient is lower at the starting point of flow separation [2]. The values for the respective positions of the ratio of the grid width to the model width z/w=-1/2, z/w=-1/4, z/w=0, z/w=1/4, and z/w=1/2 is written as -0.4132, -0.3769, -0.3405, -0.3405, and -0.4132. These results correlate with the flow pattern characteristics in Fig. 6(a) which shows that the model without active control is the model with the largest wake and vortex formation when compared to those in other models.



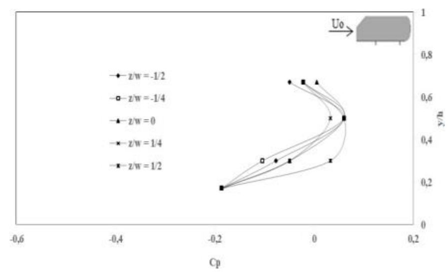
(a) Without control



(b) With blowing active control, U_{BL1}/U_0=0.03



(c) With blowing active control, U_{BL2}/U_0=0.06



(d) With blowing active control, U_{BL3}/U_0=0.09

Figure 7. Pressure distribution

For the model with velocity U_{BL1}/U_0=0.03, an increase in the average minimum pressure coefficient is apparent, as compared to the model without control. The average minimum pressure coefficient of -0.2194 is obtained at the position y/h=0.17 as shown in Fig. 7(b) with an increase in the percentage of 41.7788%. This is caused by backflow on the lower part of the rear of the model. The minimum pressure coefficient value at the z/w position is written respectively z/w=-1/2 of -0.2606, z/w=-1/4 of -0.2147, z/w=0 of -0.2206, z/w=1/4 of -0.2206, and z/w=1/2 of -0.2206.

In addition, the model in the ratio of velocity U_{BL2}/U_0=0.06 also shows an increase in the average minimum pressure coefficient with the increase is higher than the model at the velocity of U_{BL1}/U_0=0.03. This increase of 41.9080% with an average minimum pressure coefficient of -0.2189 is obtained at position y/h=0.17 which is shown in Fig. 7(c), where the value at each position z/w=-1/2, z/w=-1/4, z/w=0, z/w=1/4, and z/w=1/2 are written -0.2260, -0.2143, -0.2143, -0.2143, and -0.2260.

A similar phenomenon also occurs in a model with a velocity of U_{BL3}/U_0=0.09, where the application of blowing active control is able to increase the average minimum pressure coefficient even though the increase is higher than the model at the velocity U_{BL1}/U_0=0.03 and U_{BL2}/U_0=0.06. This increase is 50.5248% with an average minimum pressure coefficient value of -0.1865 obtained at position y/h=0.17 as shown in figure 7(d),

Commented [i-[18]: "Rate" has been changed to "average". The term "rate" is not properly used.

Commented [A19]: Reviewer A point 2 : Correction has been done

Commented [A20]: Reviewer C point 13: The data are correct. The position of z/w=0 dan z/w= ¼ are closely adjacent berdekatan, generating the same value. However, the average pressure coefficient are different.

where the value for each position of the ratio of the grid width to the model width is written z/w=-1/2 of -0.1865, z/w=-1/4 of -0.1865, z/w=0 of -0.1865, z/w=1/4 of -0.1865 and z/w=1/2 of -0.1865.

In overall the application of suction active control shows an increase in the average minimum pressure coefficient, where the model with the ratio of velocity U_{BL3}/U_0=0.09 is a model with the highest average minimum pressure coefficient increase. This correlates with the characteristics of the flow pattern in Fig. 6(d) which shows that the model at U_{BL3}/U_0=0.09 has the smallest wake formation that is tighter and more regular than other models. The research results confirm previous research which also concluded that the application of active control in the area where flow separation forms initially will give a potential development of the pressure field [20]–[22].

III.3. Aerodynamic drag

The drag coefficient obtained through the computational method is shown in table 3. The highest drag coefficient is found in the model without active control, which is 1.845. For models with the application of active control in various comparisons of blowing velocity to upstream velocity, shows a decrease in the value of the drag coefficient. The drag coefficient is written U_{BL1}/U_0=0.03 of 1.636, U_{BL2}/U_0=0.06 of 1.628, and U_{BL3}/U_0=0.09 of 1.620.

TABLE 3

COMPUTATIONAL METHOD DRAG COEFFICIENT

C_d			
Without control	U_{BL1}/U_0= 0.03	U_{BL2}/U_0= 0.06	U_{BL3}/U_0= 0.09
1.845	1.636	1.628	1.620

In the experimental approach, the highest drag coefficient value was also obtained in the model without active control of 1.763. For models with the application of active control on the ratio of blowing velocity to the upstream velocity, the drag coefficient decreases. The value of the drag coefficient at each speed is written as U_{BL1}/U_0=0.03 at 1.563, U_{BL2}/U_0=0.06 at 1.561, and U_{BL3}/U_0=0.09 at 1.559.

TABLE 4

EXPERIMENTAL METHOD DRAG COEFFICIENT

C_d			
Without control	U_{BL1}/U_0= 0.03	U_{BL2}/U_0= 0.06	U_{BL3}/U_0= 0.09
1.763	1.563	1.561	1.559

Table 5 shows the comparison of the drag coefficient of each model, both computationally and experimental. For the model without active control, the highest drag coefficient is obtained through both computational and experimental approaches with the difference in the drag coefficient of 4.434%. This result correlates with wake structure and the average minimum pressure coefficient value, where the model without active control is a model with the largest wake, vortex, and recirculation zone formation as shown in Fig. 6(a) and the model with the value of the lowest average minimum pressure coefficient as shown in table 2.

From table 5 in the U_{BL1}/U_0=0.03, the application of blowing active control gives optimum results both through computational and experimental methods. The reduction of each method was written 11.328% and 11.321%, respectively. The reduction for the velocity ratio U_{BL2}/U_0=0.06, is written as 11.741% for computation and 11.431% for experimental and for the ratio of velocity U_{BL3}/U_0=0.09, written as 12.187% for the computational approach and 11.556% for the experimental approach.

Overall, the application of blowing active control has a positive effect in the form of reduction of aerodynamic drag as shown in table 5, where the model at U_{BL3}/U_0=0.09 is the model with the highest aerodynamic drag reduction. This correlates with the characteristics of the flow pattern shown in Fig. 6(d), and the average minimum pressure coefficient value in table 2, where the model at U_{BL3}/U_0=0.09. The model with the smallest wake formation has a tighter and more regular flow line and the model with the highest average minimum pressure coefficient increase. The results gives similar fashion as what has been investigated by Harinaldi et al, citing that the application of active control could reduce aerodynamic drag [15].

TABLE 5

COMPARISON OF DRAG COEFFICIENTS

Method	Without control	Ubl_1/U = 0.03	Reduction (%)	Ubl_2/U_0 = 0.06	Reduction (%)	Ubl_3/U_0 = 0.09	Reduction (%)
Computational	1.845	1.636	11.328	1.628	11.741	1.620	12.187
Experimental	1.763	1.563	11.321	1.561	11.431	1.559	11.556
Difference (%)	4.434	4.428	-	4.099	-	3.747	-

IV. CONCLUSIONS

The application of blowing active control has positive effects on the characteristics of the flow pattern, where there is a delay in the flow separation process, and the largest reduction in wake formation is obtained in the model with the ratio of velocity, U_{BL3}/U_0=0.09.

The application of blowing active control also has positive effects on the minimum pressure coefficient, where there is an increase in the minimum pressure coefficient for all blowing velocity. The model with the ratio of velocity, U_{BL3}/U_0=0.09 is a model with an increase in the highest minimum pressure coefficient of 50.5248%. The positive effects are also shown on aerodynamic drag in the form of a reduction in the drag coefficient both computationally and experimentally. The highest reduction is obtained in the model with a ratio of velocity U_{BL3}/U_0=0.09 of 12.187% for the computational approach and 11.556% for the experimental approach.

Commented [A21]: Reviewer C point 2 : The research by Mathieu Roumeas et al and Wang Bingxin used the original Ahmed model. Therefore, the characteristics of flow, pressure distribution, and the percentage of drag reduction for any level of velocity will be different. This may emphasis the originality of our research.

CONFLICT OF INTEREST

The authors declare no conflict of interest.

AUTHOR CONTRIBUTIONS

Dr. R. Tarakka, the main and corresponding author, organized research promotion and conducted research planning. Dr. N. Salam, conducted research, Dr. A.A. Mochtar, conducted the research, Mr. W. Rauf, conducted experimental research, and Mr. M. Ihsan wrote and translated the manuscript. All authors approved the final version.

ACKNOWLEDGEMENTS

This research is funded by the Hasanuddin University Research and Community Service Institute through the 2021 University Basic Research (PDU) Scheme, Contract No. 915/UN4.22/PT.01.03/20. We extend our gratitude to the Head and Staff of the Fluid Mechanics Laboratory of the Hasanuddin University, Faculty of Engineering.

REFERENCES

[1] R. Tarakka, N. Salam, J. Jalaluddin, and M. Ihsan, "Effect of blowing flow control and front geometry towards the reduction of aerodynamic drag on vehicle models," *FME Transactions*, vol. 47, no. 3, pp. 552–559, 2019, doi: 10.5937/fmet1903552T.

[2] J. Anderson, *Fundamental of Aerodynamics*, 6th ed. Mc Graw Hill, 2017.

[3] P. Kundu, I. Cohen, and D. Dowling, *Fluid Mechanics - 6th Edition*, 6th ed. Academic Press, 2015. Accessed: Nov. 09, 2021. [Online]. Available: https://www.elsevier.com/books/fluid-mechanics/kundu/978-0-12-405935-1

[4] S. M. R. Hassan, T. Islam, M. Ali, and Md. Q. Islam, "Numerical Study on Aerodynamic Drag Reduction of Racing Cars," *Procedia Engineering*, vol. 90, pp. 308–313, Jan. 2014, doi: 10.1016/j.proeng.2014.11.854.

[5] M. Bellman, R. Agarwal, J. Naber, and L. Chusak, "Reducing Energy Consumption of Ground Vehicles by Active Flow Control," Dec. 2010, pp. 785–793. doi: 10.1115/ES2010-90363.

[6] P. Gopal and T. Senthilkumar, "Influence of Wake Characteristics of a Representative Car Model by Delaying Boundary Layer Separation," *Journal of Applied Science and Engineering*, vol. 16, no. 4, pp. 363–374, 2013, doi: 10.6180/jase.2013.16.4.04.

[7] T. Heinemann, M. Springer, H. Lienhart, S. Kniesburges, C. Othmer, and S. Becker, "Active flow control on a 1:4 car model," *Exp Fluids*, vol. 55, no. 5, p. 1738, May 2014, doi:10.1007/s00348-014-1738-0.

[8] T. B. Hilleman, "Vehicle drag reduction with air scoop vortex impeller and trailing edge surface texture treatment," US7192077B1, Mar. 20, 2007 Accessed: Nov. 09, 2021. [Online]. Available: https://patents.google.com/patent/US7192077B1/en

[9] M. Jahanmiri, "Experimental investigation of drag reduction on ahmed car model using a combination of active flow control methods," *IJE*, vol. 24, no. 4, pp. 403–410, 2011, doi: 10.5829/idosi.ije.2011.24.04a.09.

[10] C. H. Bruneau, E. Creusé, D. Depeyras, P. Gilliéron, and I. Mortazavi, "Coupling active and passive techniques to control the flow past the square back Ahmed body," *Computers and Fluids*, vol. 38, no. 10, p. 1875, 2010, doi: 10.1016/j.compfluid.2010.06.019.

[11] R. Mestiri, A. Ahmed-Bensoltane, L. Keirsbulck, F. Aloui, and L. Labraga, "Active Flow Control at the Rear End of a Generic Car Model Using Steady Blowing," *Journal of Applied Fluid Mechanics*, vol. 7, pp. 565–571, Oct. 2014.

[12] A. Ferraris, H. de C. Pinheiro, A. G. Airale, M. Carello, and D. B. Polato, "City Car Drag Reduction by means of Flow Control Devices," SAE International, Warrendale, PA, SAE Technical Paper 2020-36–0080, Mar. 2021. doi: 10.4271/2020-36-0080.

[13] H. Tebbiche and M. S. Boutoudj, "Active flow control by micro-blowing and effects on aerodynamic performances. Ahmed body and NACA 0015 airfoil," *FMR*, vol. 48, no. 2, 2021, doi: 10.1615/InterJFluidMechRes.2021036842.

[14] J. J. Cerutti, C. Sardu, G. Cafiero, and G. Iuso, "Active Flow Control on a Square-Back Road Vehicle," *Fluids*, vol. 5, no. 2, Art. no. 2, Jun. 2020, doi: 10.3390/fluids5020055.

[15] Harinaldi, Budiarso, R. Tarakka, and S. P. Simanungkalit, "Computational Analysis of Active Flow Control to Reduce Aerodynamics Drag on a Van Model," *International Journal of Mechanical & Mechatronics Engineering*, vol. 11, no. 03, pp. 24–30, 2011.

[16] M. N. Sudin, M. A. Abdullah, S. A. Shamsuddin, F. R. Ramli, and M. Mohd, "Review of Research

on Vehicles Aerodynamic Drag Reduction Methods," *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS*, vol. 14, no. 2, pp. 35–47, 2014.

[17] B. R. Munson, D. F. Young, and T. H. Okiishi, *Fundamentals of fluid mechanics*, 4th ed. John Wiley & Sons Inc, 2002.

[18] Y. A. Çengel and J. M. Cimbala, *Fluid Mechanics: Fundamentals and Applications.* McGraw-Hill Education, 2018.

[19] M. Onorato, A. F. Costelli, and A. Garrone, "Drag Measurement Through Wake Analysis," Feb. 1984, p. 840302. doi: 10.4271/840302.

[20] W. Rauf, R. Tarakka, Jalaluddin, and M. Ihsan, "Effect of Flow Separation Control with Suction Velocity Variation: Study of Flow Characteristics, Pressure Coefficient, and Drag Coefficient," *Universal Journal of Mechanical Engineering*, vol. 8, no. 3, pp. 142–151, May 2020, doi: 10.13189/ujme.2020.080302.

[21] S. Krajnović and J. Fernandes, "Numerical simulation of the flow around a simplified vehicle model with active flow control," *International Journal of Heat and Fluid Flow*, vol. 32, no. 1, pp. 192–200, Feb. 2011, doi: 10.1016/j.ijheatfluidflow.2010.06.007.

[22] R. Tarakka, Jalaluddin, B. Mire, and M. N. Umar, "Effect of turbulence model in computational analysis of active flow control on aerodynamic drag of bluff body van model," *International Journal of Applied Engineering Research*, vol. 10, no. 1, pp. 207–219, 2015.

Authors' information

Rustan Tarakka – is an Assopciate Professor in the Department of Mechanical Engineering, Faculty of Engineering, Hasanuddin University, Makassar, Indonesia. He holds a doctoral degree from University of Indonesia, Jakarta, Indonesia. His research areas are on fluid dynamics and computational fluid dynamics..

Nasaruddin Salam – is a Professor and the Chairman of Fuild Mechanics Laboratory in Department of Mechanical Engineering, Faculty of Engineering, Hasanuddin University Makassar Indonesia. He holds a doctoral degree from Brawijaya University, Malang Indonesia. His research fields include fluid dynamics.

Andi Amijoyo Mochtar –obtained a Doctor of Engineering in Mechanical Engineering in 2016 from Ehime University Japan. He is an Senior Lecturer of Mechanical Engineering of Hasanuddin University, Makassar, Indonesia..

Wawan Rauf – obtained a Master of Engineering in Mechanical Engineering in 2020 from Hasanuddin University. His research areas are on fluid dynamics and computational fluid dynamics.

Muhammad Ihsan –works for Institut Sains Kesehatan Bone, Watampone, Indonesia. He also holds masters degrees in transport engineering from Asian Institute of Technology, Bangkok, Thailand and Universitas Gajah Mada, Yogyakarta, Indonesia.

Word count : 4,380

5. Informasi artikel telah accepted di IJMERR (21 September 2022)

[ijmerr] Manuscript ID: IJMERR-6078 - Editor Decision - Accepted for Publication

2022-09-21 04:11 PM

Dear Rustan Tarakka, Salam Nasaruddin, Andi Amijoyo Mochtar, Wawan Rauf, Muhammad Ihsan,

We are pleased to inform you that the following paper has been officially accepted for publication in International Journal of Mechanical Engineering and Robotics Research.

Manuscript ID: IJMERR-6078
Title: On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis
Submission URL: http://ojs.ejournal.net/index.php/ijmerr/authorDashboard/submission/6078

Your paper will be charged for publishing (450 USD), and the detailed payment information can be found at the end of this email. **If the payment is ready, please send us the payment screenshot and order ID for the record.**

Once the payment is confirmed, We will make the final preparation, and then return the edited manuscript to you for your approval.

Please feel free to email us with any questions.

Ms. Ashley Zhang/Handling Editor
ashley.zhang@ejournal.net

Please proceed with the payment at the following link (No handling fees)
Payment link: http://confsys.iconf.org/online-payment/18130
Terms of Payment: **10 days**
Please make sure you have VISA or Mastered Card Credit Card before clicking this link, and you should also calculate the right amount and pay.

The following information is necessary.

Full Name*:
Email*:
Event Acronym*: IJMERR
Event URL*: http://www.ijmerr.com/
Acceptance/Paper ID*: IJMERR-6078
Paper Title*: On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis

----

6. Informasi pembayaran APC ke IJMERR (27 September 2022)

Payment for article ID-IJMERR-6078

Participants Edit

Ms. Ashley Zhang (ashley)

Rustan Tarakka (rustantarakka)

Messages

Note	From
Dear Ms. Ashley Zhang, Handling editor of IJMERR Thank you for accepting our paper in IJMERR. We have completed the payment for our article ID-IJMERR-6078On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis. Order ID 20220927-18130-104709159591, amounted USD 450. payment link http://confsys.iconf.org/index.php/Payment/show/orderid/20220927-18130-104709159591.html Attached is the screenshot of our payment Sincerely yours, Dr. Rustan Tarakka	rustantarakka 2022-09-27 11:09 AM

Payment info

Order ID	20220927-18130-104709159591
Full Name	Rustan Tarakka
Email	rustan_tarakka@yahoo.com
Event acronym	IJMERR
Event url	http://www.ijmerr.com/
Acceptance/Paper ID	IJMERR-6078
Paper title	On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis
Total	450.00 USD

Payment Success

7. Informasi penerimaan pembayaran APC dari IJMERR (27 September 2022)

[ijmerr] Manuscript ID: IJMERR-6078 - Payment Confirmation ✖

Participants

Ms. Ashley Zhang (ashley)

Rustan Tarakka (rustantarakka)

Messages

Note	From
Dear Rustan Tarakka, Many thanks for your support of open access publishing in International Journal of Mechanical Engineering and Robotics Research. And I will contact you in the coming three days for the final proofreading. Please find below the APC payment confirmation for the following article: Payment of Invoice: IJMERR-6078 Title: On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis Amount Received: 450 USD Order ID: 20220927-18130-104709159591 This is the confirmation of APC payment. No need to reply to this email. Ms. Ashley Zhang/Handling Editor ashley.zhang@ejournal.net	ashley 2022-09-27 03:31 PM

8. Informasi Proofreading dari IJMERR (14 Oktober 2022)

[ijmerr] Manuscript ID: IJMERR-6078 - Proofreading

Participants

Ms. Haylee Lin (haylee)

Ms. Ashley Zhang (ashley)

Rustan Tarakka (rustantarakka)

Messages

Note	From
Dear Authors,	ashley 2022-10-14 03:06 PM

We invite you to proofread your manuscript prior to publication:

Title: On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis

Submission URL:
http://ojs.ejournal.net/index.php/ijmerr/authorDashboard/submission/6078

Please read the following instructions carefully before proofreading:

(1) Download the manuscript from the above link (copyediting menu-copyedited) and upload the final proofed version within **five days**.

(2) Please use Microsoft Word's built-in track changes function to highlight any changes you make or send a comprehensive list of changes in a separate document. Note that this is the *last chance* to make textual changes to the manuscript.

(3) All authors must agree to the final version. Check carefully that authors' names and affiliations are correct, and that funding sources are correctly acknowledged. Incorrect author names or affiliations are picked up by indexing databases, such as Scopus, and can be difficult to correct.

Please note: We have edited your latest submission(revision that submitted in september 7). As there are too many column breaks, section breaks, text boxes, tables, etc. in the manuscript, please check the content and paper flow in the edited version carefully to avoid any errors. The manuscript will be regarded as the final copy after your proofreading and it will be sent to the IJMERR office for publication.

Once proofreading is done, please click on the above link to open the submission system, create a new discussion, and upload the final approved version. (copyediting - add discussion - add journal editor as Participants). After proofreading, final production will be carried out. Once a paper has been published online, we will not accept any corrections or changes to the published version. Changes made later will be published separately via a Correction or Addendum.

In case of any questions regarding final proofreading, please don't hesitate to contact me or the journal editor: Ms. Haylee Lin, haylee.lin@ejournal.net.

Ms. Ashley Zhang
ashley.zhang@ejournal.net

9. Permintaan tambahan waktu proofreading (15 Oktober 2022)

Dear Ms. Ashley Zhang

rustantarakka
2022-10-15 10:45 AM

We are sorry for the delay in the preparation of our manuscript. We are still working on the plagiarism check in Turnitin and its revision, and we just achieved 20% similarity. We hope that it will fulfill our internal requirements, as well as the IJMERR standard.

Regarding the proofreading, we use an external party, and it will be completed in at least 7 working days Therefore we are asking for **an extension of the final submission for 14 days** (two weeks from now).

We hope that this effort can contribute to the quality of our publication.

Sincerely yours,

Dr. Rustan Tarakka

10. Persetujuan waktu tambahan proofreading dari IJMERR (19 Oktober 2022)

Dear Dr. Rustan Tarakka,

Fine, we would be glad to grant an extension. Please proof the final version carefully and return back within the extended deadline.

By the way, Ms. Ashley Zhang was on maternity leave. If you have any questions regarding this paper, please do not hesitate to contact me (haylee.lin@ejournal.net).

Best,
Ms. Haylee Lin
Journal Editor

haylee
2022-10-19 10:30 AM

11. Artikel yang telah diproofreading (29 Oktober 2022)

Dear Ms. Haylee Lin

Thank you for granting us extended time to submit our paper. We have completed the proofreading as well as the plagiarism check for our paper.

We hereby send you the paper, along with the proofreading certificate. Since the final version is the product of the proofreader, we cannot provide track-changes remarks. The changes are mostly in grammar and layout editing, and no changes in the substance of the manuscript.

Sincerely yours,

Dr. Rustan Tarakka

rustantarakka, 6078-Manuscript (Word)-24884-1-9-20221029 Final.docx

rustantarakka, 6078-Manuscript (PDF)-24884-1-9-20221029 Final.pdf

rustantarakka, 6078- Certificate of Proofreading.pdf

rustantarakka
2022-10-29 04:45 PM

On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis

Rustan Tarakka, Nasaruddin Salam, Andi Amijoyo Mochtar,
Department of Mechanical Engineering, Hasanuddin University, Gowa, Indonesia,
Email: {rustan_tarakka@yahoo.com}{nassalam.unhas@yahoo.co.id}{andijoyo@unhas.ac.id}

Wawan Rauf
Department of Mechanical Engineering, Gorontalo University, Gorontalo, Indonesia
Email: {wawanrauf241193@yahoo.com }

Muhammad Ihsan
Department of Civil Engineering, Sekolah Tinggi Teknik Baramuli, Pinrang, Indonesia
Email: {muhammadihsan@alumni.ait.asia}

***Abstract*—Aerodynamics related to the generation of drag due to flow separations that occurs at rear parts of vehicles is an important consideration in vehicle design. It includes flow separation, wake area formed, and the pressure, focusing on the model's rear wall in this paper. The Pressure reductions could differ significantly between front and rear walls of vehicles. This pressure difference can generate a phenomenon of backward pull and an increase in drag on the vehicle. To minimize back flow as well as to cater increasing pressure on vehicles' rear wall can be achieved by applying active control, including attaching blowing apparatus. The paper presents the analysis of the aerodynamic effect of the application of blowing active control on the aerodynamics on rear part of vehicles, which is represented by a modified Ahmed body, reversed in flow direction and altered dimensions. The research was conducted using a validated numerical simulation method with laboratory experiment at an upstream air speed of 16.7 m/s and respective blowing velocities of 0.5 m/s, 1.0 m/s, and 1.5 m/s. Results showed that the application of blowing active control was capable to reduce aerodynamic drag, with the highest drag reduction is achieved in the model with a ratio of velocity UBL3/U0=0.09 of 12.187% for the computational method and 11.556% for the experimental one.**

***Index Terms*— aerodynamic drag, blowing active control, vehicle model**

I. Introduction

Aerodynamics related to the generation of drag due to flow separations at rear parts of vehicles is an important consideration in vehicle design.

Manuscript received July 1, 2012; revised August 1, 2012; accepted September 1, 2012.

Corresponding author : rustan_tarakka@yahoo.com

The magnitudeof the aerodynamic drag force, works in contrast to the relative motion of a moving object, undergone by vehicles will affect vehicles' energy consumption and stability [1][2]. This opposing movement occursbetween the fluid and the surface of a solid object [3]. One approximation on amount of fuel consumption to overcome these adverse aerodynamic drag is about 50- 60% [4]. Reduction of the aerodynamic drag by as small as 15% will contribute to 5-7% fuel consumption savings[5].

The aerodynamic drag on a vehicle is closely related to the flow characteristics and the pressure distribution at the rear part of the vehicle, which are also influenced by flow separations occuring at the upper rear part of vehicles [6]. The design of this part is then very important in the effort of reducing aerodynamic drag [7]. The flow separation is expected cause backflow and decrease pressure field in the onset area of the separation. The rate of the flow separation will positively increase the extent of wake area which at the same time will reduce the pressure on the rear wall area, which results in notable differences in pressure between front and rear parts, triggering the backward pull phenomenon [8]. Minimizing negative pressure, and the intensity, at the rear area of vehicles could decrease the aerodynamic drag [4].

Flow engineering is a method that can be used tominimize back flow and at the same time to increase pressures on the rear of vehicles with a positive effect in delaying separation and reducing the re-circulation zone. It can be obtained by the application of active control such as blowing, or any combination with other forms of control, either active or passive [7] [9] [10] .

An investigation on the effect of continuous blowing on the interface of vehicles' roof and rear window was published by Mestiri et al., based on an experiment on steady blowing attached in 25° tangent to the surface of the slanted rear window of an Ahmed model, showing that tangential steady blowing can produce the separated area on the rear window and disturbing the appearance of the counter-rotating longitudinal vortex on the end side of the rear window. The direct flow control was considere as really effective on the re-circulation area at the upper area of the rear window [11].

A study has been conducted on the potential of aerodynamic drag reductions of a city-car prototype (a baseline version of the XAM 2.0) employingair blow and air relief flow control devices embedded into the vehicle's wheel, conducted in a dedicated wind tunnel, with special arrangement to reduce turbulences of the front wheel, and to cope with the air-flow disturbance at the end of vehicles' side body. The study also incorporated a CFD analysis to evaluate the effects of the modifications. By validating the drag optimization, a correlation of experimental results by wind tunnel results and the CFD results was obtained, showing anticipated capabilities of CFD analysis, as well as a record-breaking result of drag coefficient [12].

Tebbiche and Boutoudj have recently published an experimental investigation of the effect of the various blowing rates and Reynolds numbers for some incidences of airfoils, two angles of Ahmed body's rear windows, and a set of flow velocities (15-30 m/s) in a subsonic wind tunnel. The results came with good improvements on the aerodynamic coefficients. On a 20° tilted rear window of Ahmed body model, 3.6% drag reduction has been confirmed on the application of blowing with a minimum blowing intensity Cμ = 0.28%, and a notable 15% reduction was gained by full blowing at maximum Cμ = 4.79%. At the same study, a high-drag regime related to a 30° tilted rear window has produced drag reduction of 19% at Cμ = 5% [13].

Cerutti et al. conducted an experiment on the manipulation of the generation of on a square-back model by four rectangular jets blowing continuously by means of standart and stereoscopic Particle Image Velocimetry (PIV and sPIV). and revealed a considerable alteration in the maximum drag reduction on the formation of wake as the jets blow, while similar wake structure was best compromised [14].

II. METHOD

The research works on modified Ahmed body, altered in the orientation of the flow and the dimensional ratio to the original Ahmed body, set on 0.17 (1:6). The dimensions of the model are defined length 174 mmlength (l), 48 mm height (l), and 64.83 mm width (w), with 35° slant angle of the model's front geometry. The upper rear side of vehicle models is designated as the area where the blowing active control is located. The active controls present in 5 slots with a diameter of 7 mm with a distance between apertures of 10.81 mm. Each circle is defined as B_{L1}, B_{L2}, B_{L3}, B_{L4}, and B_{L5}. The tests have been conducted by setting the blowing velocity of each apertures (BL), U_{BL1}=0.5 m/s, U_{BL2}=1.0 m/s, and U_{BL3}=1.5 m/s at the upstream speed of U_0=16.7 m/s. Furthermore, the definition of the ratio of blowing velocity to upstream velocity is written U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06, and U_{BL3}/U_0=0.09. T h e m odel setup is detailed in Fig. 1.

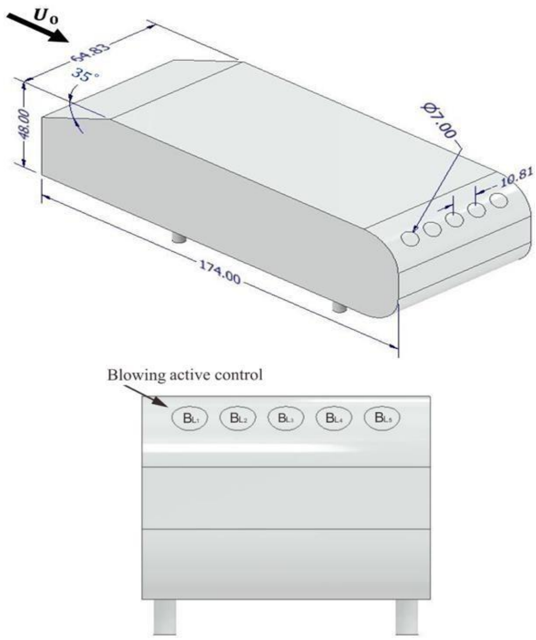


Figure 1. Test model

This research covers the flow characteristics passing through the rear of the model, the pressure field (C_p), and aerodynamic drag (C_d). The aerodynamic drag (C_d) is investigated by computational and by experimental observations, while for flow characteristics and pressure field (C_p), the investigation is only on numerical simulation using a standard k—epsilon turbulence model with tetra hedral meshing.. The definition of turbulence model and meshing type is based on previous work of Harinaldi et al [15]. The data collection on pressure distribution have been conducted at model's rear parts, considering the location is prone for separation of flow and wake to generate, resulting negative pressures contributing up to 80% of the total drag [16]. On the models' transversal axis, the pressure data are taken on 5 different grid lines, where grid width to model width ratios are defined as z/w=-0.5, z/w=-0.25, z/w=0, z/w=0.25, and z/w=0.5. On the height axis of the model, pressure data have been obtained on 5 grid lines for models without control and 4 grid lines for models with blowing active controls. The grid height to model height ratios l (y/h) were written as 0.17, 0.33, 0.50, 0.67, and 0.83. The pressure field data collectionarea are shown in Fig. 2. The pressure field data is written into the pressure coefficient (C_p) value obtained through the use of (1) [17].

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

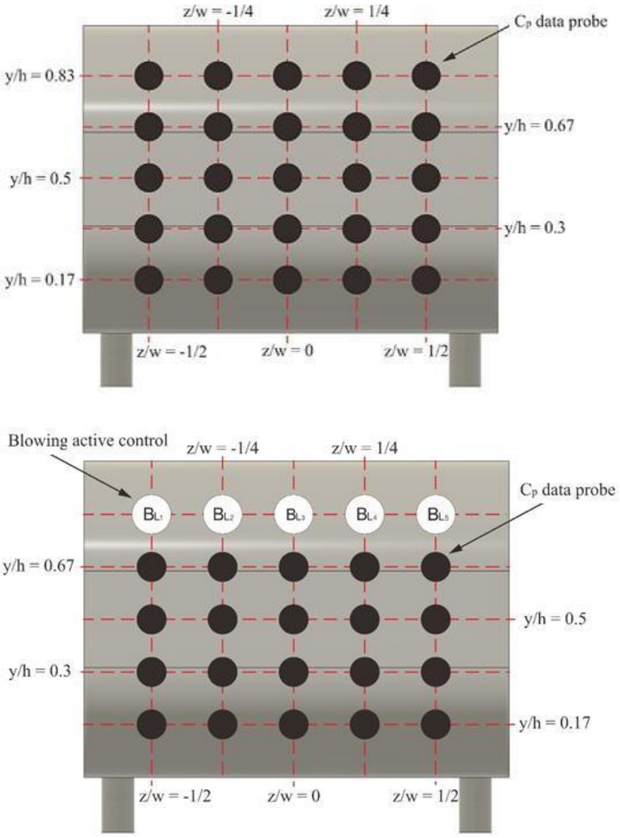


Fig. 2. C_p data probe: (a) without control, (b) with blowing active control

The computational simulation utilizes Fluent 6.3.26 ANSYS computational fluid dynamic software. The vehicle model is designed using Autodesk Inventor™. Fig. 3 and table 1 present the computational domain, while Fig. 4 presents the meshing process in Gambit™ software.

TABLE 1
COMPUTATIONAL CONDITIONS

Boundary condition	Type	Value
Fluid properties	Density	1.225 kg/m^3
	Viscosity	1.7894 × 10^{-5} kg/m.s
Model boundary conditions without control	Model	Wall
	Outlet	Pressure outlet
	Inlet	Velocity inlet
	Wall	Wall
Boundary conditions of the model with blowing active controls	Model	Wall
	Outlet	Pressure outlet
	Inlet	Velocity inlet
	Wall	Wall
	U_{BL1} U_{BL2}, U_{BL3}, U_{BL4}, U_{BL5}	Velocity inlet

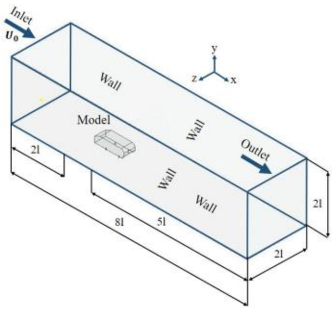


Figure 3. Computational domain

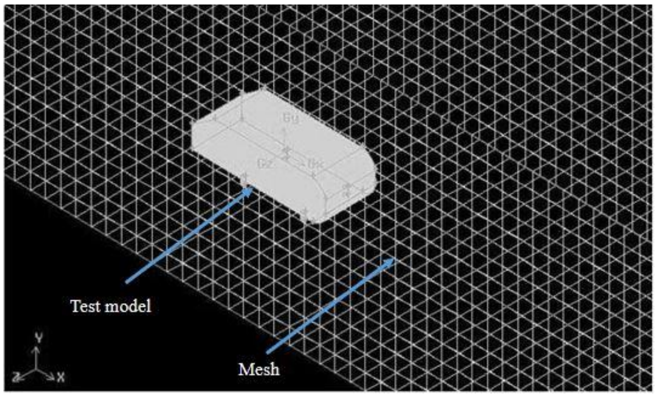


Figure 4. Computational mesh

The experimental testing has utilized the sub-sonic wind tunel facility. The measurement of drag uses a load cell equipped with an Arduino™ device and is directly connected to a computer. The duration of data retrieval is 120 seconds, generating 120 data for each velocity (U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06, and U_{BL3}/U_0=0.09). The 120 data are then averaged to gain drag values for each velocity comparison with a high degree of accuracy. The schematic of experimental setup is depicted in Fig. 5.

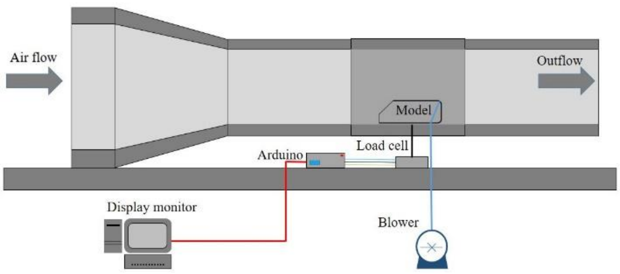


Figure 5. Experimental setup

The drag force is transformed into non-dimensional drag coefficient (C_d) by (2) [18]:

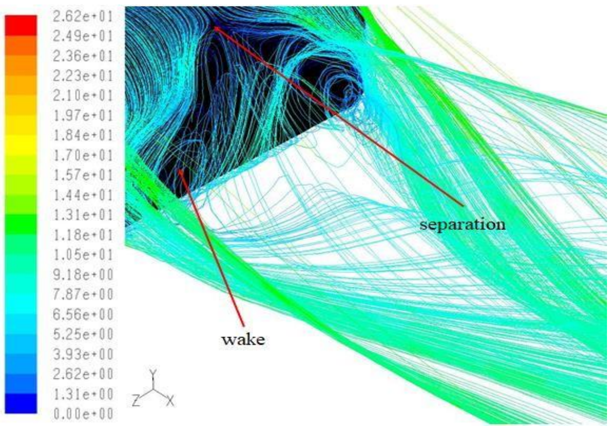
$$C_d = \frac{F_d}{\frac{1}{2}\rho v^2 A} \quad (2)$$

III. RESULTS AND DISCUSSION

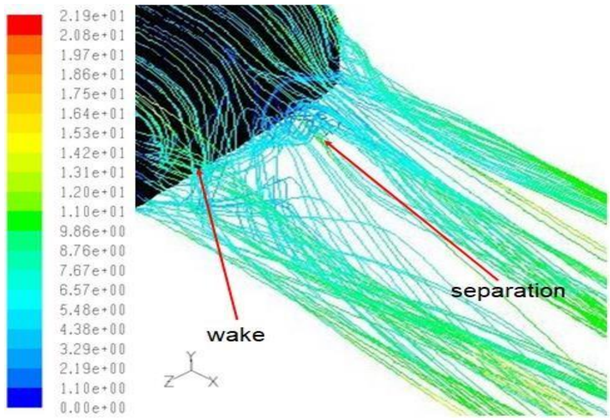
III.1. Flow field

The characteristics of the flow pattern of the test model with active control in the ratio of blowing velocity to upstream velocity, $U_{BL1}/U_0=0.03$, $U_{BL2}/U_0=0.06$ and $U_{BL3}/U_0=0.09$ as well as models without control are shown in Fig. 6. The model without active control shows a large wake structure caused by flow separation on the upper rear side of the rear wall of the test model as shown in Fig. 6(a). Air flow, which initially flows uniformly, forms backflow and re-circulation areas. This is the main cause of negative pressure, which creates the phenomenon of backward pulling. This phenomenon of re-atraction has been suspected as a major contributor to the extent of aerodynamic drag, besides other presumed factor of longitudinal vortex [19]. Visually, the model without active control has a fairly large longitudinal vortex structure. This is because the fluid loses momentum to move along the rear body dueto the adhesion friction force, resulting in differences in flow velocities on the rear side and the center of the vehicles. This difference in speed forces the flowon the middle side to flow sidebound to form a longitudinal vortex.

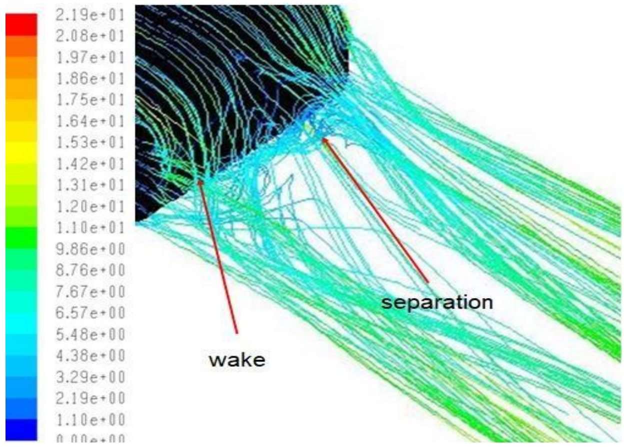
For models with the blowing at the velocity ratio, $U_{BL1}/U_0=0.03$, $U_{BL2}/U_0=0.06$ and $U_{BL3}/U_0=0.09$, as presented in Figs. 6(b), 6(c) and 6(d), the wake structure is smaller than the one on the model without application of control. This is because there is a delay in flow separation, where the separation process is formed in an area far from the rear wall of the model, and minimize the intensity of the backflow interacting with the rear wall. Besides, the longitudinal vortex that is formed tends to be smaller due to the blowing effect which forces a portion of the flow on the upper wall to move straight towards the downstream area. The model in the velocity ratio $U_{BL3}/U_0=0.09$ shows smaller wake formation compared to the model at the velocity $U_{BL1}/U_0=0.03$ and $U_{BL2}/U_0=0.06$.



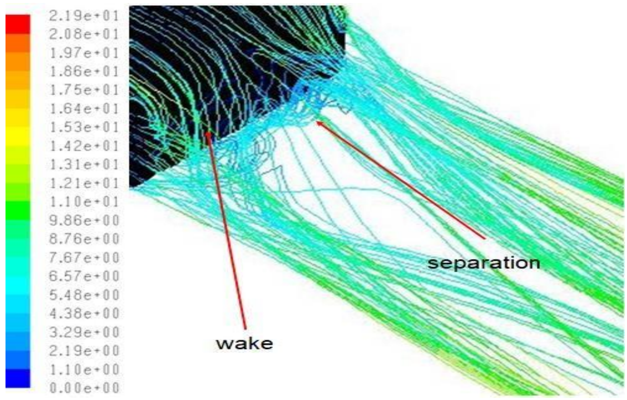
(a) Without control



(b) With blowing active control, $U_{BL1}/U_0=0.03$



(c) With blowing active control, $U_{BL2}/U_0=0.06$



(d) With blowing active control, $U_{BL3}/U_0=0.09$

Figure 6. Flow field

III.2. Pressure distribution

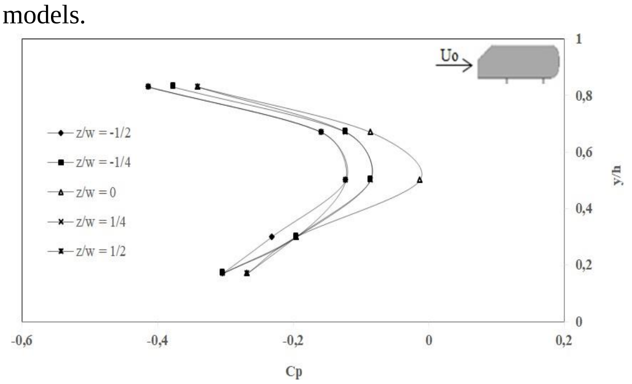
Table 2 shows the comparison of the minimum pressure coefficient for the model without and with blowing at the velocity ratio $U_{BL1}/U_0=0.03$, $U_{BL2}/U_0=0.06$, and $U_{BL3}/U_0=0.09$.

TABLE 2

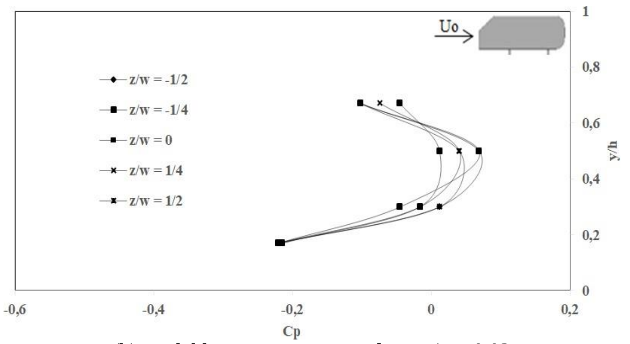
COMPARISON OF THE MINIMUM PRESSURE COEFFICIENT

z/w	Minimum C_p			
	Without control	With blowing active control, U_{BL}/U_0		
		0.03	0.06	0.09
-1/2	-0.4132	-0.2206	-0.2260	-0.1865
-1/4	-0.3769	-0.2147	-0.2143	-0.1865
0	-0.3405	-0.2206	-0.2143	-0.1865
1/4	-0.3405	-0.2206	-0.2143	-0.1865
1/2	-0.4132	-0.2206	-0.2260	-0.1865
Average	-0.3769	-0.2194	-0.2189	-0.1865
Increase (%)	-	41.7788	41.9080	50.5248

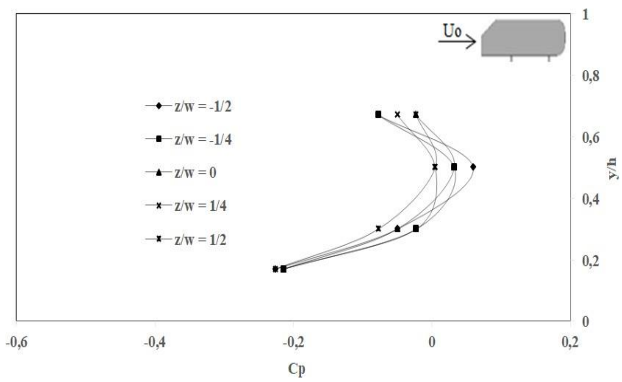
The lowest minimum average pressure coefficient value is found in the model with no blowing controls at a value of -0.3769. The minimum pressure coefficient has been achieved at the grid height to model height ratio y/h=0.83 as shown in Fig. 7(a). This is because the position with y/h=0.83 is the onset point for the flow separation. These findings are consistent with an opinion by Anderson revealing that the pressure coefficient is lower at the onset point of flow separation [2]. The values for the respective positions of the grid width to model width ratios of z/w=-0.5, z/w=-0.25, z/w=0, z/w=0.25, and z/w=0.5 is written as -0.4132, -0.3769, -0.3405, - 0.3405, and -0.4132. These results correlate with the flowpattern characteristics in Fig. 6(a) which shows that the model without active control is the one with the most apparent wake and vortex formation when compared to other models.



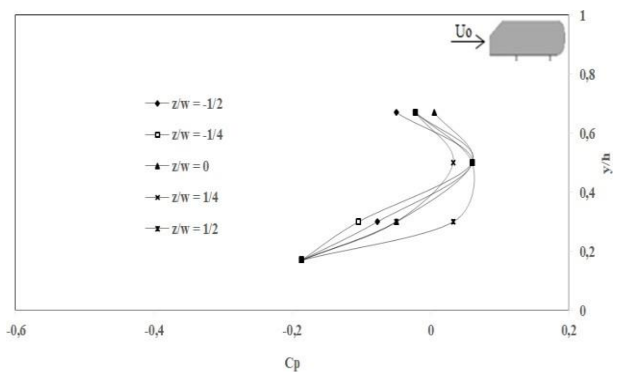
(a) Without control



(b) With blowing active control, U_{BL1}/U_0=0.03



(c) With blowing active control, U_{BL2}/U_0=0.06



(d) With blowing active control, U_{BL3}/U_0=0.09

Figure 7. Pressure distribution

For the model with velocity U_{BL1}/U_0=0.03, an increase in the average minimum pressure coefficient is apparent, as compared to the those in models without blowing. An average minimum pressure coefficient of -0.2194 is obtained at the position y/h=0.17 as shown in Fig. 7(b) with an increase in the percentage of 41.7788%. This is caused by backflow on the lower part of the rear of the test model. Minimum pressure coefficient values at the z/w position are written respectively z/w=-0.5 of -0.2606, z/w=-0.25 of -0.2147, z/w=0 of -0.2206, z/w=0.25 of -0.2206, and z/w=0.5 of -0.2206.

In addition, the model in the ratio of velocity U_{BL2}/U_0=0.06 also indicates increasing average minimum pressure coefficient with the increase is higher than the model at the velocity of U_{BL1}/U_0=0.03. Thisincrease of 41.9080% with an average minimum pressurecoefficient of -0.2189 is obtained at position y/h=0.17 which is shown in Fig. 7(c), where the value at each position z/w=-0.5, z/w=-0.25, z/w=0, z/w=0.25,and z/w=0.5 are written -0.2260, -0.2143, -0.2143, -0.2143, and -0.2260.

A similar phenomenon also occurs in a model with a velocity of U_{BL3}/U_0=0.09, where the application of blowing active control has been capable to increase the average minimum pressure coefficient even though the increase is higher than the model at the velocity U_{BL1}/U_0=0.03 and U_{BL2}/U_0=0.06. This increase is 50.5248% with an average minimum pressure coefficient of -0.1865 obtained at position y/h=0.17 as shown in figure 7(d), where the value for particular positions relative to the ratio of the grid width to the model width is written z/w=-0.5 of -0.1865, z/w=-0.25 of -0.1865, z/w=0 of -0.1865, z/w=0.25 of -0.1865 and z/w=0.5 of -0.1865.

In overall the application of active control indicates increasing average minimum pressure coefficient, where the model with the ratio of velocity U_{BL3}/U_0=0.09 is the model producing the highest increase of the average minimum pressure coefficient. This correlates with the characteristics of the flow shown in Fig. 6(d) where the model at U_{BL3}/U_0=0.09 has thesmallest wake formation that is tighter and more uniform than that of other models. The research results confirm previous research which also concluded that the application ofactive control where flow separation forms initially will give a potential development of the pressurefield [20]–[22].

III.3. Aerodynamic drag

The drag coefficient gained through the computational method is shown in table 3. The highest drag coefficient is found in the model without activecontrol, which is 1.845. Models with the active control in various ratios of blowing velocity to upstream velocity shows a decrease in drag coefficients. The drag coefficient is written U_{BL1}/U_0=0.03 of 1.636, U_{BL2}/U_0=0.06 of 1.628, and U_{BL3}/U_0=0.09 of 1.620.

TABLE 3

COMPUTATIONAL METHOD DRAG COEFFICIENT

C_d			
Without control	U_{BL1}/U_0= 0.03	U_{BL2}/U_0= 0.06	U_{BL3}/U_0= 0.09
1.845	1.636	1.628	1.620

In the experimental approach, the highest drag coefficient value was also achieved in the model without active control of 1.763. Models with active controls on respective ratios of blowing velocity to the upstream velocity give decreases of drag coefficient, for U_{BL1}/U_0=0.03 at 1.563, for U_{BL2}/U_0=0.06 at 1.561, and for U_{BL3}/U_0=0.09 at 1.559.

TABLE 4

EXPERIMENTAL METHOD DRAG COEFFICIENT

C_d			
Without control	U_{BL1}/U_0= 0.03	U_{BL2}/U_0= 0.06	U_{BL3}/U_0= 0.09
1.763	1.563	1.561	1.559

Table 5 shows the comparison of the drag coefficient of each model, both computationally and experimental. The highest drag coefficient recorded through both computational and experimental methods is on model with no blowing controls with a 4.434% difference in drag coefficients. This result correlates to the wake structure, and to the average minimum pressure coefficient value, where the absence of active control creating the apparent wake, vortex, and recirculation zone formation as shown in Fig. 6(a) and the model with the value of the lowest average minimum pressure coefficient as presented in table 2.

From table 5 in the U_{BL1}/U_0=0.03, the application of blowing active control gives optimum results both through computational and experimental methods. The reduction of each method was written 11.328% and 11.321%, respectively. The reduction for the velocity ratio U_{BL2}/U_0=0.06, is written as 11.741% for computation and 11.431% for experimental and for the ratio of velocity U_{BL3}/U_0=0.09, written as 12.187% for the computational method and 11.556% for the experimental one.

Overall, the application of blowing active control has a positive effect in the form of reduction of aerodynamic drag as shown in table 5, where the model at U_{BL3}/U_0=0.09 is the model with the highest aerodynamic drag reduction. This correlates with the flow pattern characteristics shown in Fig. 6(d), and the averageminimum pressure coefficient value in table 2, where the model at U_{BL3}/U_0=0.09. The model with the smallest wake formation shows tighter and more uniform flow lines. The results gives similar fashion as what has been investigated by Harinaldi et al, who showed that active controls could reduce aerodynamic drag [15].

TABLE 5

COMPARISON OF DRAG COEFFICIENTS

Method	Without control	Ubl_1/U = 0.03	Reduction (%)	Ubl_2/U_0 = 0.06	Reduction (%)	Ubl_3/U_0 = 0.09	Reduction (%)
Computational	1.845	1.636	11.328	1.628	11.741	1.620	12.187
Experimental	1.763	1.563	11.321	1.561	11.431	1.559	11.556
Difference (%)	4.434	4.428	-	4.099	-	3.747	-

IV. CONCLUSIONS

The application of blowing active control has given positive effects to the characteristics of flow patterns, showing delays in the flow separation process, and has given considerable reduction in wake formation is obtained in the model with the ratio of velocity, U_{BL3}/U_0=0.09.

The application of blowing active control also has positive effects in increasing the minimum pressure coefficients, for all blowing velocity. The model with the ratio of velocity, U_{BL3}/U_0=0.09 is a model with the highest increase in minimum pressure coefficient of 50.5248%. The positive effects are also shown on aerodynamic drag in the form of a reduction in the drag coefficient both computationally and experimentally. The highest reduction is obtained in the model with a ratio of velocity U_{BL3}/U_0=0.09 of 12.187% for the computational method and 11.556% for the experimental one.

CONFLICT OF INTEREST

The authors declare no conflict of interest.

AUTHOR CONTRIBUTIONS

Dr. R. Tarakka, the main and corresponding author, organized research promotion and conducted research planning. Dr. N. Salam, conducted research, Dr. A.A. Mochtar, conducted the research, Mr. W. Rauf, conducted experimental research, and Mr. M. Ihsan wrote and translated the manuscript. All authors approved the final version.

ACKNOWLEDGEMENTS

This research is funded by the Hasanuddin University Research and Community Service Institute through the 2021 University Basic Research (PDU) Scheme, Contract No. 915/UN4.22/PT.01.03/20. We extend ourgratitude to the Head and Staff of the Fluid Mechanics Laboratory of the Hasanuddin University, Faculty of Engineering.

REFERENCES

[1] R. Tarakka, N. Salam, J. Jalaluddin, and M. Ihsan, "Effect of blowing flow control and front geometry towards the reduction of aerodynamic drag on vehicle models," *FME Transactions*, vol. 47, no. 3, pp. 552–559, 2019, doi: 10.5937/fmet1903552T.

[2] J. Anderson, *Fundamental of Aerodynamics*, 6th ed. Mc Graw Hill, 2017.

[3] P. Kundu, I. Cohen, and D. Dowling, *Fluid Mechanics - 6th Edition*, 6th ed. Academic Press, 2015. Accessed: Nov. 09, 2021. [Online].Available: https://www.elsevier.com/books/fluid-mechanics/kundu/978-0-12-405935-1

[4] S. M. R. Hassan, T. Islam, M. Ali, and Md. Q. Islam, "Numerical Study on Aerodynamic DragReduction of Racing Cars," *Procedia Engineering*,vol. 90, pp. 308–313, Jan. 2014, doi: 10.1016/j.proeng.2014.11.854.

[5] M. Bellman, R. Agarwal, J. Naber, and L. Chusak, "Reducing Energy Consumption of Ground Vehicles by Active Flow Control," Dec. 2010, pp. 785–793. doi: 10.1115/ES2010-90363.

[6] P. Gopal and T. Senthilkumar, "Influence of Wake Characteristics of a Representative Car Model by Delaying Boundary Layer Separation," *Journal of Applied Science and Engineering*, vol. 16, no. 4, pp. 363–374, 2013, doi: 10.6180/jase.2013.16.4.04.

[7] T. Heinemann, M. Springer, H. Lienhart, S. Kniesburges, C. Othmer, and S. Becker, "Active flow control on a 1:4 car model," *Exp Fluids*, vol. 55, no. 5, p. 1738, May 2014, doi:10.1007/s00348-014-1738-0.

[8] T. B. Hilleman, "Vehicle drag reduction with air scoop vortex impeller and trailing edge surface texture treatment," US7192077B1, Mar. 20, 2007 Accessed: Nov. 09, 2021. [Online]. Available: https://patents.google.com/patent/US7192077B1/en

[9] M. Jahanmiri, "Experimental investigation of drag reduction on ahmed car model using a combination of active flow control methods," *IJE*, vol. 24, no. 4, pp. 403–410, 2011, doi: 10.5829/idosi.ije.2011.24.04a.09.

[10] C. H. Bruneau, E. Creusé, D. Depeyras, P.Gilliéron, and I. Mortazavi, "Coupling active and passive techniques to control the flow past the square back Ahmed body," *Computers and Fluids*, vol. 38, no. 10, p. 1875, 2010, doi: 10.1016/j.compfluid.2010.06.019.

[11] R. Mestiri, A. Ahmed-Bensoltane, L. Keirsbulck, F. Aloui, and L. Labraga, "Active Flow Control at the Rear End of a Generic Car Model Using Steady Blowing," *Journal of Applied Fluid Mechanics*,vol. 7, pp. 565–571, Oct. 2014.

[12] A. Ferraris, H. de C. Pinheiro, A. G. Airale, M. Carello, and D. B. Polato, "City Car Drag Reduction by means of Flow Control Devices," SAE International, Warrendale, PA, SAE Technical Paper 2020-36–0080, Mar. 2021. doi: 10.4271/2020-36-0080.

[13] H. Tebbiche and M. S. Boutoudj, "Active flow control by micro-blowing and effects on aerodynamic performances. Ahmed body and NACA 0015 airfoil," *FMR*, vol. 48, no. 2, 2021, doi: 10.1615/InterJFluidMechRes.2021036842.

[14] J. J. Cerutti, C. Sardu, G. Cafiero, and G. Iuso, "Active Flow Control on a Square-Back Road Vehicle," *Fluids*, vol. 5, no. 2, Art. no. 2, Jun. 2020, doi: 10.3390/fluids5020055.

[15] Harinaldi, Budiarso, R. Tarakka, and S. P. Simanungkalit, "Computational Analysis of Active Flow Control to Reduce Aerodynamics Drag on a Van Model," *International Journal of Mechanical & Mechatronics Engineering*, vol. 11, no. 03, pp. 24–30, 2011.

[16] M. N. Sudin, M. A. Abdullah, S. A. Shamsuddin, F. R. Ramli, and M. Mohd, "Review of Research

on Vehicles Aerodynamic Drag Reduction Methods," *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS*, vol. 14, no. 2, pp. 35–47, 2014.
[17] B. R. Munson, D. F. Young, and T. H. Okiishi, *Fundamentals of fluid mechanics*, 4th ed. John Wiley & Sons Inc, 2002.
[18] Y. A. Çengel and J. M. Cimbala, *Fluid Mechanics: Fundamentals and Applications*. McGraw-Hill Education, 2018.
[19] M. Onorato, A. F. Costelli, and A. Garrone, "Drag Measurement Through Wake Analysis," Feb. 1984, p. 840302. doi: 10.4271/840302.
[20] W. Rauf, R. Tarakka, Jalaluddin, and M. Ihsan, "Effect of Flow Separation Control with Suction Velocity Variation: Study of Flow Characteristics, Pressure Coefficient, and Drag Coefficient," *Universal Journal of Mechanical Engineering*, vol. 8, no. 3, pp. 142–151, May 2020, doi: 10.13189/ujme.2020.080302.
[21] S. Krajnović and J. Fernandes, "Numerical simulation of the flow around a simplified vehicle model with active flow control," *International Journal of Heat and Fluid Flow*, vol. 32, no. 1, pp. 192–200, Feb. 2011, doi: 10.1016/j.ijheatfluidflow.2010.06.007.
[22] R. Tarakka, Jalaluddin, B. Mire, and M. N. Umar, "Effect of turbulence model in computational analysis of active flow control on aerodynamic drag of bluff body van model," *International Journal of Applied Engineering Research*, vol. 10, no. 1, pp. 207–219, 2015.

Authors' information

Rustan Tarakka – is an Assopciate Professor in the Department of Mechanical Engineering, Faculty of Engineering, Hasanuddin University, Makassar, Indonesia. He holds a doctoral degree from University of Indonesia, Jakarta, Indonesia. His research areas are on fluid dynamics and computational fluid dynamics..

Nasaruddin Salam – is a Professor and the Chairman of Fuild Mechanics Laboratory in Department of Mechanical Engineering, Faculty of Engineering, Hasanuddin University Makassar Indonesia. He holds a doctoral degree from Brawijaya University, Malang Indonesia. His research fields include fluid dynamics.

Andi Amijoyo Mochtar –obtained a Doctor of Engineering in Mechanical Engineering in 2016 from Ehime University Japan. He is an Senior Lecturer of Mechanical Engineering of Hasanuddin University, Makassar, Indonesia..

Wawan Rauf – obtained a Master of Engineering in Mechanical Engineering in 2020 from Hasanuddin University. His research areas are on fluid dynamics and computational fluid dynamics.

Muhammad Ihsan –works for Institut Sains Kesehatan Bone, Watampone, Indonesia. He also holds masters degrees in transport engineering from Asian Institute of Technology, Bangkok, Thailand and Universitas Gajah Mada, Yogyakarta, Indonesia.

Word count : 4,365

12. Informasi artilel telah diproses untuk dipublish IJMERR (31 Oktober 2022)

[ijmerr] Manuscript ID: IJMERR-6078 - Send to Production

2022-10-31 11:08 AM

Dear Rustan Tarakka, Salam Nasaruddin, Andi Amijoyo Mochtar, Wawan Rauf, Muhammad Ihsan:

The editing of your submission, "On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis," is complete. We are now sending it to production. If you have any questions for the publication schedule, please contact the editorial office at ijmerr@vip.163.com.

Submission URL: http://ojs.ejournal.net/index.php/ijmerr/authorDashboard/submission/6078

Ms. Haylee Lin
haylee.lin@ejournal.net

--
International Journal of Mechanical Engineering and Robotics Research
Website: http://www.ijmerr.com/
Email: ijmerr@vip.163.com

13. Artikel dipublish oleh IJMERR (6 Maret 2023)

[ijmerr] Manuscript ID: IJMERR-6078 - Paper has been published

Participants

Ms. Haylee Lin (haylee)

Rustan Tarakka (rustantarakka)

Messages

Note	From
Dear Authors, We are pleased to inform you that your article "On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis" has been published in "International Journal of Mechanical Engineering and Robotics Research" and is available online: Website link: http://www.ijmerr.com/index.php?m=content&c=index&a=show&catid=220&id=1843 Please take a moment to check that everything is correct. You can reply to the journal editorial office (ijmerr@vip.163.com) if there is a problem. Note that at this stage we will not accept further changes to the manuscript text. Thank you for choosing "International Journal of Mechanical Engineering and Robotics Research" to publish your work, we look forward to receiving further contributions from your research group in the future. Ms. Haylee Lin haylee.lin@ejournal.net -- International Journal of Mechanical Engineering and Robotics Research Website: http://www.ijmerr.com/	haylee 2023-03-06 11:40 AM

On the Aerodynamics of Rear of Vehicle Model with Active Control by Blowing: Computational and Experimental Analysis

Rustan Tarakka [1]*, Nasaruddin Salam [1], Andi Amijoyo Mochtar [1], Wawan Rauf [2], Muhammad Ihsan [3]

[1] Department of Mechanical Engineering, Hasanuddin University, Gowa, Indonesia;
Email:{nassalam.unhas@yahoo.co.id}{andijoyo@unhas.ac.id}
[2] Department of Mechanical Engineering, Gorontalo University, Gorontalo, Indonesia; Email: {wawanrauf241193@yahoo.com }
[3] Department of Civil Engineering, Sekolah Tinggi Teknik Baramuli, Pinrang, Indonesia; Email: {muhammadihsan@alumni.ait.asia}
*Correspondence: rustan_tarakka@yahoo.com

***Abstract*—Aerodynamics related to the generation of drag due to flow separations that occurs at rear parts of vehicles is an important consideration in vehicle design. It includes flow separation, wake formation, and pressures, which, in this paper, are focused on the ones exerted on the model's rear wall. The pressure reductions could differ significantly between vehicles' front and rear walls. This pressure difference can generate a phenomenon of backward pull and an increase in drags. The effort to minimize backflow as well as to cater increasing pressure on vehicles' rear wall can be achieved by applying active control, including attached blowing apparatus. The paper presents the analysis of the effect on the application of blowing active control on the aerodynamics on rear part of vehicles, which is represented by a modified Ahmed body, reversed in flow direction and altered dimensions. The research was conducted using a validated numerical simulation method with laboratory experiments at an upstream air speed of 16.7 m/s and blowing velocities of 0.5 m/s, 1.0 m/s, and 1.5 m/s. The results showed that the application of blowing active control was capable to reduce aerodynamic drag, with the highest decrease achieved in the model with a ratio of velocity UBL3/U0=0.09 of 12.187% for the computational method and 11.556% for the experimental one.**

***Keywords*—aerodynamic drag, blowing active control, vehicle model**

I. Introduction

Aerodynamics related to the generation of drag due to flow separations at rear parts of vehicles is an important consideration in vehicle design. The magnitude of the aerodynamic drag force, works in contrast to the relative motion of a moving object, undergone by vehicles will affect vehicles' energy consumption and stability [1, 2]. This opposing movement usually occurs between the fluid and the surface of a solid object [3]. One approximation on amount of fuel consumption to overcome these adverse aerodynamic drag is about 50-60% [4]. Reduction of the aerodynamic drag by as small as 15% will contribute to 5-7% fuel consumption savings [5].

The aerodynamic drag on a vehicle is closely related to the flow characteristics and the pressure distribution at the rear part, which are also influenced by flow separations occurring at the upper rear part [6]. The design of this part is then very important in the effort of reducing aerodynamic drag [7]. The flow separation is expected to cause backflow and decrease pressure field in the onset area of the separation. The rate of the flow separation tends to increase the extent of the wake area positively and, at the same time, reduce the pressure on the rear wall region. This results in notable differences in the pressure exerted at both front and rear parts, triggering the backward pull phenomenon [8]. The process of minimizing negative pressure, and the intensity, at the rear area of vehicles could decrease the aerodynamic drag [4].

Flow engineering is a method used to minimize backflow and, at the same time, increase pressures exerted on the rear parts. As a results, it positively impacts delaying separation and reducing the re-circulation zone. Furthermore, flow engineering is realized by the application of active controls, such as blowing, with a combination with other forms of control, either active or passive [7], [9], [10].

An investigation carried out on the effect of continuous blowing on the interface of vehicles' roof and rear window was published by Mestiri et al. Based on an experiment performed on steady blowing at 25 °tangent to the surface of the slanted rear window of an Ahmed model, it was proven that tangential steady blowing produced the separated area on the rear window as well as disturbed the appearance of the counter-rotating longitudinal vortex at the end side. Furthermore, the direct flow control was considered effective in the re-circulation area at the upper part of the rear window [11].

Manuscript received September 10, 2022; revised October 4, 2022; accepted November 30, 2022.

doi: 10.18178/ijmerr.12.2.84-90

A study has been carried out on the potentials of aerodynamic drag reductions of a city-car prototype (a baseline version of the XAM 2.0). It employed air blow and relief flow control devices embedded into the vehicle's wheel, in a dedicated wind tunnel, with special arrangement to reduce turbulences at the front wheel, and to cope with the air-flow disturbance at the end of vehicles' side body. The study further incorporated a CFD analysis to evaluate the effects of the modifications. By validating the drag optimization, a correlation of experimental results by wind tunnel results and the CFD results was obtained, showing anticipated capabilities of CFD analysis, as well as a record-breaking outcome in drag coefficients [12].

Tebbiche and Boutoudj have recently published an experimental investigation carried out to determine the effect of the various blowing rates and Reynolds numbers for some incidences of airfoils, two angles of Ahmed body's rear windows, and a set of flow velocities (15-30 m/s) in a subsonic wind tunnel. The results showed improvements in the aerodynamic coefficients. For example, on a 20° tilted rear window of Ahmed body model, 3.6% drag reduction was confirmed due to the application of a minimum blowing intensity Cμ = 0.28%. A notable 15% reduction was gained by maximum Cμ = 4.79%. Additionally, a high-drag regime related to a 30° tilted rear window produced a drag reduction of 19% at Cμ = 5% [13].

Cerutti et al. conducted an experimental research on the manipulation of the generation of on a square-back model by four rectangular jets blowing continuously by means of standart and stereoscopic Particle Image Velocimetry (PIV and sPIV), and revealed a considerable alteration in the maximum drag reduction on the formation of wake as the jets blow, while similar wake structure was best compromised [14].

II. METHOD

The present research focuses on the modified Ahmed body, altered in the orientation of the flow and the dimensional ratio to the original Ahmed body, set at 0.17 (1:6). The dimensions are 174 mm length (l), 48 mm height (l), and 64.83 mm width (w), with 35° slant angle of the model's front geometry. The upper rear side of the vehicle model is designated as the area where the blowing active control is located. The active controls present in 5 slots with a diameter of 7 mm and a distance of 10.81 mm between apertures. Each circle is defined as B_{L1}, B_{L2}, B_{L3}, B_{L4}, and B_{L5}. The tests were carried out by setting the blowing velocity of each apertures (BL), U_{BL1}=0.5 m/s, U_{BL2}=1.0 m/s, and U_{BL3}=1.5 m/s at the upstream speed of U_0=16.7 m/s. The ratios of blowing velocity to upstream velocity are stated as follows U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06, and U_{BL3}/U_0=0.09. Furthermore, a detailed model setup is shown in Fig. 1.

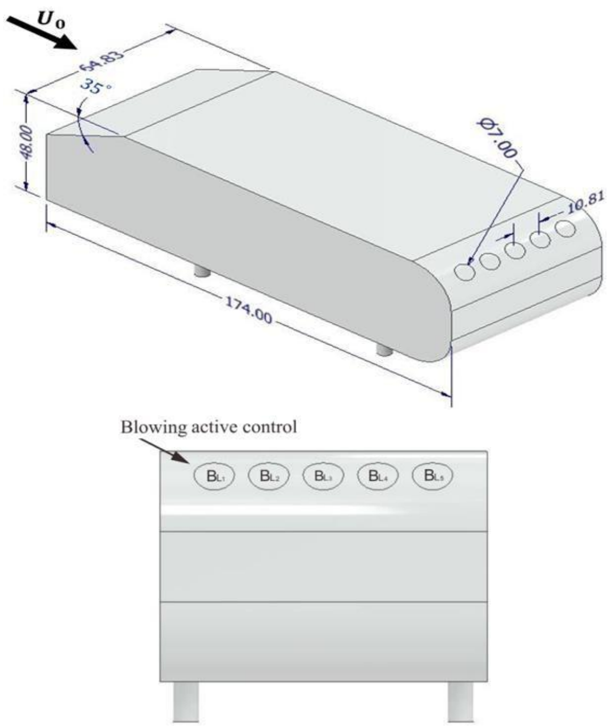


Figure 1. Test model.

This research covers the flow characteristics passing through the model's rear, the pressure field (C_p), and aerodynamic drag (C_d). The aerodynamic drag (C_d) is investigated by computational and by experimental observations. For flow characteristics and pressure field (C_p), the investigations were based only on numerical simulation using a standard k—epsilon turbulence model with tetrahedral meshing. The definition of turbulence model and meshing type is based on previous work of Harinaldi *et al.* [15]. The data collection on pressure distribution have been conducted at model's rear parts, considering the location is prone to the separation of flow and wake to generate, resulting negative pressures contributing up to 80% of the total drag [16]. On the models' transversal axis, the information on pressure are obtained from five different grid lines, where grid-to-model width ratios are defined as z/w=-0.5, z/w=-0.25, z/w=0, z/w=0.25, and z/w=0.5. With respect to model height axis, pressure data have been obtained from five grid lines for models without control and four for those with blowing active controls. The grid-to-model height ratios l (y/h) are as follows 0.17, 0.33, 0.50, 0.67, and 0.83. The pressure field data collection area are shown in Fig. 2 and are incorporated into the pressure coefficient (C_p) value obtained using Eq. (1) [17].

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

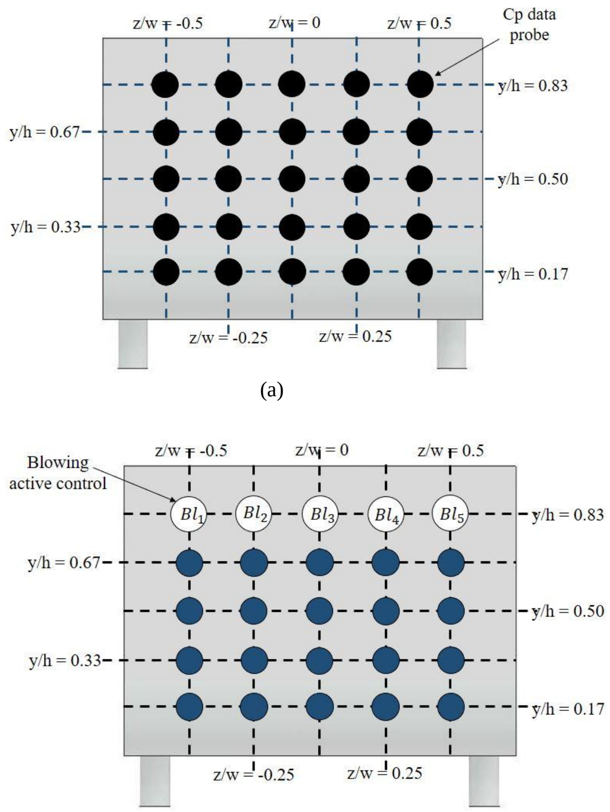


Figure 2. Cp data probe: (a) without control, (b) with blowing active control

The simulation utilized Fluent 6.3.26 ANSYS computational fluid dynamic software. Additionally, the vehicle model is designed using Autodesk Inventor™. Fig. 3 and Table I depict the computational domain, while Fig. 4 presents the meshing process in Gambit™ software.

TABLE I. COMPUTATIONAL CONDITIONS

Boundary condition	Type	Value
Fluid properties	Density	1.225 kg/m^3
	Viscosity	1.7894 × 10^{-5}
Model boundary conditions without control	Model	Wall
	Outlet	Pressure outlet
	Inlet	Velocity inlet
		Wall
Boundary conditions of the model with blowing active controls	Model	Wall
	Outlet	Pressure outlet
	Inlet	Velocity inlet
	Wall	Wall
	U_{BL1} U_{BL2}, U_{BL3}, U_{BL4}, U_{BL5}	Velocity inlet

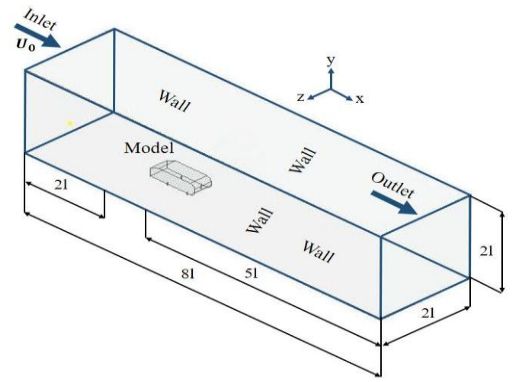


Figure 3. Computational domain.

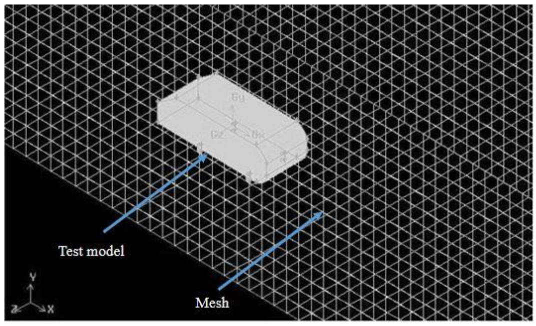


Figure 4. Computational mesh.

The experimental testing has utilized the sub-sonic wind tunnel facility. The measurement of drag used a load cell equipped with an Arduino device directly connected to a computer. The duration of data retrieval is 120 seconds, generating 120 data for each velocity (U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06, and U_{BL3}/U_0=0.09). This information is then averaged to gain drag values for each velocity comparison with high accuracy. The schematic of experimental setup is depicted in Fig. 5.

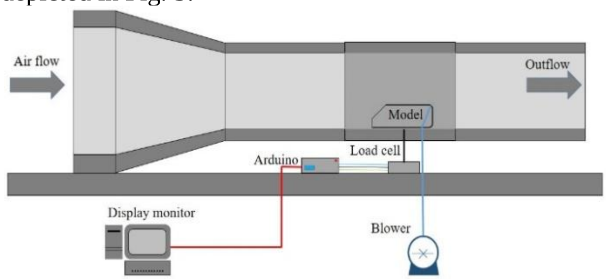


Figure 5. Experimental setup.

The drag force is transformed a into non-dimensional drag coefficient (C_d) by using Eq. (2) [18]:

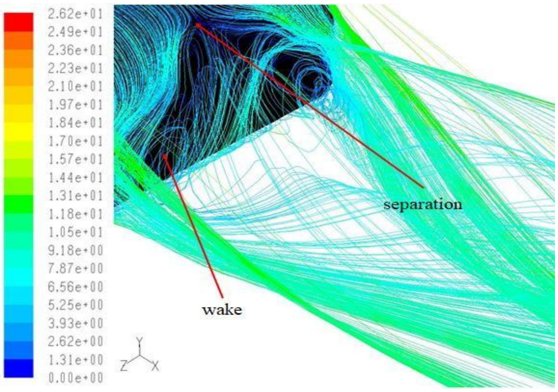
$$C_d = \frac{F_d}{\frac{1}{2}\rho v^2 A} \quad (2)$$

III. RESULTS AND DISCUSSION

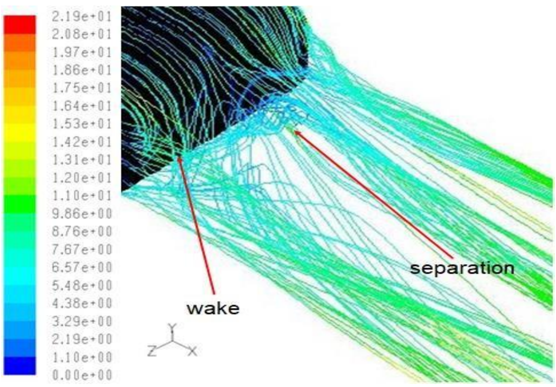
A. Flow Field

The characteristics of the flow pattern of the test model with active control in the ratio of blowing velocity to upstream velocity, U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06 and U_{BL3}/U_0=0.09 as well as those without control are shown in Fig. 6. The model without active control shows a large wake structure caused by the flow separation on the upper rear side as shown in Fig. 6(a). The air flow, which initially flowed uniformly, later formed backflow and re-circulation areas. This is the main cause of negative pressure, which leads to the phenomenon of backward pulling. The re-atraction process has been suspected as a major contributor to the extent of aerodynamic drag, besides other presumed factor of longitudinal vortex [19]. Visually, the model without active control has a fairly large longitudinal vortex structure. This is because the fluid loses its momentum to move along the rear body due to the friction force, thereby resulting in differences in flow velocities on the rear side and the center of the vehicles. This varying speed forces the flow on the middle side to flow sidebound to form a longitudinal vortex.

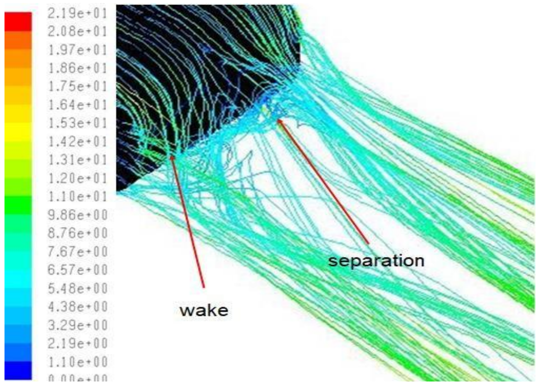
For models with the blowing at the velocity ratio, U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06 and U_{BL3}/U_0=0.09, as presented in Figs. 6(b), 6(c) and 6(d), the wake structure is smaller than the one on the model without application of control. This is because there is a delay in flow separation, where the separation process is formed in an area far from the rear wall of the model, and minimize the intensity of the backflow interacting with the rear wall. The longitudinal vortex formed tends to be smaller due to the blowing effect which forces a portion of the flow on the upper wall to move straight toward the downstream area. The model in the velocity ratio U_{BL3}/U_0=0.09 shows smaller wake formation compared to those at the velocity rations U_{BL1}/U_0=0.03 and U_{BL2}/U_0=0.06.



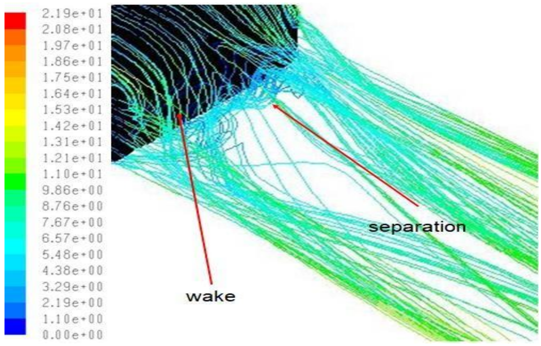
(a) Without control



(b) With blowing active control, UBL1/U0=0.03.



(c) With blowing active control, UBL2/U0=0.06.



(d) With blowing active control, UBL3/U0=0.09.

Figure 6. Flow field.

B. Pressure Distribution

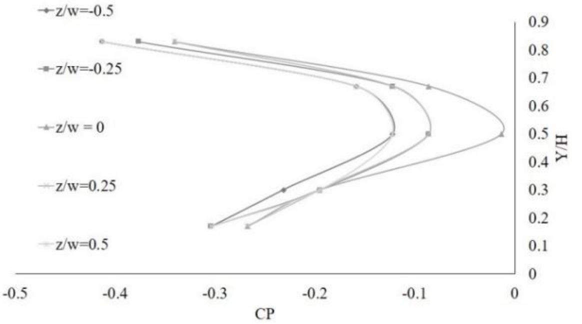
Table II shows compares the minimum pressure coefficient for the model with and without blowing at the velocity ratios of U_{BL1}/U_0=0.03, U_{BL2}/U_0=0.06, and U_{BL3}/U_0=0.09.

TABLE II. COMPARISON OF THE MINIMUM PRESSURE COEFFICIENT.

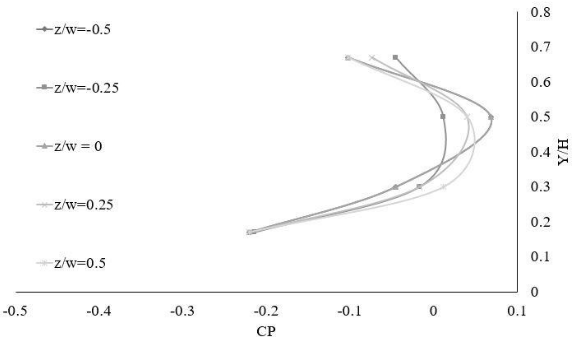
z/w	Minimum C_p			
	Without control	With blowing active control, U_{BL}/U_0		
		0.03	0.06	0.09
−1/2	−0.4132	−0.2206	−0.2260	−0.1865
−1/4	−0.3769	−0.2147	−0.2143	−0.1865
0	−0.3405	−0.2206	−0.2143	−0.1865
1/4	−0.3405	−0.2206	−0.2143	−0.1865
1/2	−0.4132	−0.2206	−0.2260	−0.1865
Average	−0.3769	−0.2194	−0.2189	−0.1865
Increase (%)	-	41.7788	41.9080	50.5248

The lowest minimum average pressure coefficient of -0.3769 was found in the model without blowing controls. It was achieved at the grid- to-model height ratio y/h=0.83 as shown in Fig. 7(a). This is because the position of y/h=0.83 is the point of onset for the flow separation. These findings are consistent with a theory that the pressure coefficient is lower at the onset point of flow separation [2]. The pressure for grid-to-model width ratios of z/w=-0.5, z/w=-0.25, z/w=0, z/w=0.25, and z/w=0.5 are written as -0.4132, -0.3769, -0.3405, -0.3405, and -0.4132.

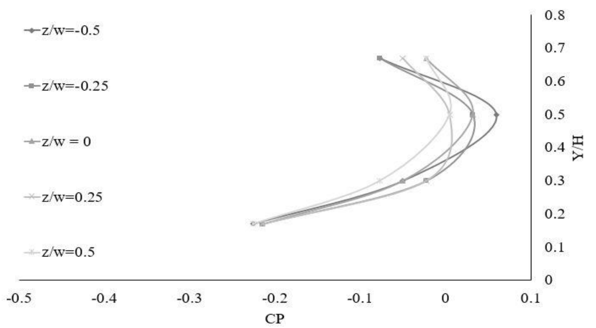
These results correlate with the flow pattern characteristics in Fig. 6(a) which shows that the model without active control is the one with the most apparent wake and vortex formation when compared to other models.



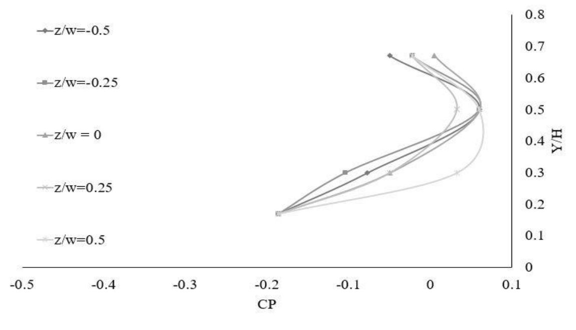
(a). Without control



(b). With blowing active control, UBL1/U0=0.03



(c). With blowing active control, UBL2/U0=0.06



(d). With blowing active control, UBL3/U0=0.09

Figure 7. Pressure distribution.

For the model with velocity U_{BL1}/U_0=0.03, an increase in the average minimum pressure coefficient is apparent compared to the those in models without blowing. An average minimum pressure coefficient of −0.2194 is obtained at the position y/h=0.17 as shown in Fig. 7(b) with an increase in the percentage of 41.7788%. This is caused by backflow on the lower part of the rear of the test model. Minimum pressure coefficient values at the z/w position are written respectively z/w=−0.5 of −0.2606, z/w= −0.25 of −0.2147, z/w=0 of −0.2206, z/w=0.25 of −0.2206, and z/w=0.5 of −0.2206.

Additionally, the model in the ratio of velocity U_{BL2}/U_0=0.06 also indicates an increase in the average minimum pressure coefficient, and this is greater than the one with the velocity ratio of U_{BL1}/U_0=0.03. An increase of 41.9080% with an average minimum pressure coefficient of -0.2189 is obtained at position y/h=0.17 as shown in Fig. 7(c), where the value at each position z/w=−0.5, z/w=−0.25, z/w=0, z/w=0.25, and z/w=0.5 are written −0.2260, −0.2143, −0.2143, −0.2143, and −0.2260.

A similar phenomenon also occurred in a model with the velocity ratio of U_{BL3}/U_0=0.09, where the application of blowing active control has been capable of increasing the average minimum pressure coefficient despite being higher than the ones with velocity ratios U_{BL1}/U_0=0.03 and U_{BL2}/U_0=0.06. This increase is equivalent 50.5248% with an average minimum pressure coefficient of -0.1865 obtained at position y/h=0.17 as shown in Fig. 7(d), where the values of pressure coefficients for particular grid-to-model width ratios are stated as follows z/w=-0.5 of −0.1865, z/w=−0.25 of −0.1865, z/w=0 of −0.1865, z/w=0.25 of −0.1865 and z/w=0.5 of −0.1865.

Generally, the application of active control indicates increasing average minimum pressure coefficient, where the model with the ratio of velocity U_{BL3}/U_0=0.09 is the model producing the highest increase of the average minimum pressure coefficient. This correlates with the characteristics of the flow shown in Fig. 6(d) where the model at U_{BL3}/U_0=0.09 has the smallest wake formation that is tighter and more uniform than that of other models. The research results confirm previous research which also concluded that the application of active control where flow separation formed initially leads to the potential development of the pressure field [20–22].

C. *Aerodynamic Drag*

The drag coefficient gained through the computational method is shown in Table III. The highest value of 1.845 is found in the model without active control. Meanwhile, models with the active control in various ratios of blowing velocity to upstream velocity shows a decrease in drag coefficients. The drag coefficient is written U_{BL1}/U_0=0.03 of 1.636, U_{BL2}/U_0=0.06 of 1.628, and U_{BL3}/U_0=0.09 of 1.620.

TABLE III. COMPUTATIONAL METHOD DRAG COEFFICIENT.

C_d			
Without control	U_{BL1}/U_0= 0.03	U_{BL2}/U_0= 0.06	U_{BL3}/U_0= 0.09
1.845	1.636	1.628	1.620

In the experimental approach, the highest drag coefficient value of 1.763 was also achieved in the model without active control, as shown in Table IV. Models with active controls on respective ratios of blowing velocity to the upstream velocity show decreases of drag coefficient, stated as follows U_{BL1}/U_0=0.03 at 1.563, for U_{BL2}/U_0=0.06 at 1.561, and for U_{BL3}/U_0=0.09 at 1.559.

TABLE IV. EXPERIMENTAL METHOD DRAG COEFFICIENT.

C_d			
Without control	U_{BL1}/U_0= 0.03	U_{BL2}/U_0= 0.06	U_{BL3}/U_0= 0.09
1.763	1.563	1.561	1.559

Table V compares the drag coefficient of each model, both computationally and experimentally. The model without blowing controls recorded the highest drag coefficient through both computational and experimental methods. It had a difference of 4.434% in drag coefficients, which correlates to the wake structure and the average minimum pressure coefficient. The absence of active control creates the apparent wake, vortex, and re-circulation zone formation, as shown in Fig. 6(a). In contrast, the model with the lowest average minimum pressure coefficient is shown in Table II.

Table V shows that the following ratio of U_{BL1}/U_0=0.03, with the application of blowing active control, gives optimum results through computational and experimental methods. The reduction of each method was written 11.328% and 11.321%, respectively. The reduction for the velocity ratio U_{BL2}/U_0=0.06, is written as 11.741% for computation and 11.431% for experiments. For the velocity ratio of U_{BL3}/U_0=0.09, the reduction are written as 12.187% for the computational method and 11.556% for the experimental one.

Overall, the application of blowing active control positively reduces aerodynamic drag, as shown in Table V, where the model at U_{BL3}/U_0=0.09 is the model with the highest aerodynamic drag reduction. This correlates with the flow pattern characteristics shown in Fig. 6(d), and the average minimum pressure coefficient value in Table II, where the model at U_{BL3}/U_0=0.09. The model with the smallest wake formation shows tighter and more uniform flow lines. The results are in similar fashion to that of Harinaldi et al., which proved active controls could reduce aerodynamic drag [15].

TABLE V. COMPARISON OF DRAG COEFFICIENTS.

Method	Without control	Ubl_1/U=0.03	Reduction (%)	Ubl_2/U_0=0.06	Reduction (%)	Ubl_3/U_0=0.09	Reduction (%)
Computational	1.845	1.636	11.328	1.628	11.741	1.620	12.187
Experimental	1.763	1.563	11.321	1.561	11.431	1.559	11.556
Difference (%)	4.434	4.428	-	4.099	-	3.747	-

IV. CONCLUSIONS

The application of blowing active control positively affects the flow pattern characteristics, showing delays in the flow separation process. A considerable reduction in wake formation was obtained in the model with the velocity ratio of U_{BL3}/U_0=0.09.

The application of blowing active control also positively increases the minimum pressure coefficients for all blowing velocities. The model with the velocity ratio of U_{BL3}/U_0=0.09 has the highest increase in minimum pressure coefficient of 50.5248%.

The positive effects are also shown in the form of reduced aerodynamic drag coefficient both computationally and experimentally. The highest reduction was obtained in the model with the velocity ratio of U_{BL3}/U_0=0.09, 12.187% and 11.556% for computational and experimental methods, respectively.

CONFLICT OF INTEREST

The authors declare no conflict of interest.

AUTHOR CONTRIBUTIONS

Dr. R. Tarakka, is the lead and corresponding author, who organized research planning and promotion. Dr. N. Salam, Dr. A.A. Mochtar, and Mr. W. Rauf, conducted the research. Mr. M. Ihsan wrote the manuscript. All authors approved the final version.

FUNDING

The research is supported by the Hasanuddin University Research and Community Service Institute for funding this research through the 2021 University Basic Research (PDU) Scheme, Contract No. 915/UN4.22/PT.01.03/20. The authors are grateful to Hasanuddin University Research and Community Service Institute, and the Head and Staff of the Fluid Mechanics Laboratory of the Hasanuddin University.

REFERENCES

[1] R. Tarakka, N. Salam, J. Jalaluddin, and M. Ihsan, "Effect of blowing flow control and front geometry towards the reduction of aerodynamic drag on vehicle models," *FME Transactions*, vol. 47, no. 3, pp. 552–559, 2019, DOI: 10.5937/fmet1903552T.

[2] J. Anderson, *Fundamental of Aerodynamics*, 6th ed. Mc Graw Hill, 2017.

[3] P. Kundu, I. Cohen, and D. Dowling, *Fluid Mechanics - 6th Edition*, 6th ed. Academic Press, 2015. Accessed: Nov. 09, 2021.

[Online]. Available: https://www.elsevier.com/books/fluid-mechanics/kundu/978-0-12-405935-1

[4] S. M. R. Hassan, T. Islam, M. Ali, and Md. Q. Islam, "Numerical study on aerodynamic drag reduction of racing cars," *Procedia Engineering*, vol. 90, pp. 308–313, Jan. 2014, DOI: 10.1016/j.proeng.2014.11.854.

[5] M. Bellman, R. Agarwal, J. Naber, and L. Chusak, "Reducing energy consumption of ground vehicles by active flow control," Dec. 2010, pp. 785–793. DOI: 10.1115/ES2010-90363.

[6] P. Gopal and T. Senthilkumar, "Influence of wake characteristics of a representative car model by delaying boundary layer separation," *Journal of Applied Science and Engineering*, vol. 16, no. 4, pp. 363–374, 2013,DOI: 10.6180/jase.2013.16.4.04.

[7] T. Heinemann, M. Springer, H. Lienhart, S. Kniesburges, C. Othmer, and S. Becker, "Active flow control on a 1:4 car model," *Exp Fluids*, vol. 55, no. 5, p. 1738, May 2014, DOI: 10.1007/s00348-014-1738-0.

[8] T. B. Hilleman, "Vehicle drag reduction with air scoop vortex impeller and trailing edge surface texture treatment," US7192077B1, Mar. 20, 2007 Accessed: Nov. 09, 2021. [Online]. Available: https://patents.google.com/patent/US7192077B1/e n

[9] M. Jahanmiri, "Experimental investigation of drag reduction on Ahmed car model using a combination of active flow control methods," *IJE*, vol. 24, no. 4, pp. 403–410, 2011, DOI: 10.5829/idosi.ije.2011.24.04a.09.

[10] C. H. Bruneau, E. Creusé, D. Depeyras, P. Gilliéron, and I. Mortazavi, "Coupling active and passive techniques to control the flow past the square back Ahmed body," *Computers and Fluids*, vol. 38, no. 10, p. 1875, 2010, DOI: 10.1016/j.compfluid.2010.06.019.

[11] R. Mestiri, A. Ahmed-Bensoltane, L. Keirsbulck, F. Aloui, and L. Labraga, "Active flow control at the rear end of a generic car model using steady blowing," *Journal of Applied Fluid Mechanics*, vol. 7, pp. 565–571, Oct. 2014.

[12] A. Ferraris, H. de C. Pinheiro, A. G. Airale, M. Carello, and D. B. Polato, "City car drag reduction by means of flow control devices," *SAE International*, Warrendale, PA, SAE Technical Paper 2020-36–0080, Mar. 2021. doi: 10.4271/2020-36-0080.

[13] H. Tebbiche and M. S. Boutoudj, "Active flow control by micro-blowing and effects on aerodynamic performances. Ahmed body and NACA 0015 airfoil," *FMR*, vol. 48, no. 2, 2021, DOI: 10.1615/InterJFluidMechRes.2021036842.

[14] J. J. Cerutti, C. Sardu, G. Cafiero, and G. Iuso, "Active flow control on a square-back road vehicle," *Fluids*, vol. 5, no. 2, Art. no. 2, June 2020, DOI: 10.3390/fluids5020055.

[15] Harinaldi, R. Budiarso, Tarakka, and S. P. Simanungkalit, "Computational analysis of active flow control to reduce aerodynamics drag on a van model," *International Journal of Mechanical & Mechatronics Engineering*, vol. 11, no. 03, pp. 24–30, 2011.

[16] M. N. Sudin, M. A. Abdullah, S. A. Shamsuddin, F. R. Ramli, and M. Mohd, "Review of research on vehicles aerodynamic drag reduction methods," *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS*, vol. 14, no. 2, pp. 35–47, 2014.

[17] B. R. Munson, D. F. Young, and T. H. Okiishi, *Fundamentals of Fluid Mechanics*, 4th ed. John Wiley & Sons Inc, 2002.

[18] Y. A. Çengel and J. M. Cimbala, *Fluid Mechanics: Fundamentals and Applications*. McGraw-Hill Education, 2018.

[19] M. Onorato, A. F. Costelli, and A. Garrone, "Drag measurement through wake analysis," Feb. 1984, p. 840302. DOI: 10.4271/840302.

[20] W. Rauf, R. Tarakka, Jalaluddin, and M. Ihsan, "Effect of flow separation control with suction velocity variation: study of flow characteristics, pressure coefficient, and drag coefficient," *Universal Journal of Mechanical Engineering*, vol. 8, no. 3, pp. 142–151, May 2020, DOI: 10.13189/ujme.2020.080302.

[21] S. Krajnović and J. Fernandes, "Numerical simulation of the flow around a simplified vehicle model with active flow control," *International Journal of Heat and Fluid Flow*, vol. 32, no. 1, pp. 192–200, Feb. 2011, DOI: 10.1016/j.ijheatfluidflow.2010.06.007.

[22] R. Tarakka, Jalaluddin, B. Mire, and M. N. Umar, "Effect of turbulence model in computational analysis of active flow control on aerodynamic drag of bluff body van model," *International Journal of Applied Engineering Research*, vol. 10, no. 1, pp. 207–219, 2015.

Rustan Tarakka – is an Assopciate Professor in the Department of Mechanical Engineering, Faculty of Engineering, Hasanuddin University, Makassar, Indonesia. He holds a doctoral degree from University of Indonesia, Jakarta, Indonesia. His research areas are on fluid dynamics and computational fluid dynamics.

Nasaruddin Salam – is a Professor and the Chairman of Fuild Mechanics Laboratory in Department of Mechanical Engineering, Faculty of Engineering, Hasanuddin University Makassar Indonesia. He holds a doctoral degree from Brawijaya University, Malang Indonesia. His research fields include fluid dynamics.

Andi Amijoyo Mochtar –obtained a Doctor of Engineering in Mechanical Engineering in 2016 from Ehime University Japan. He is an Senior Lecturer of Mechanical Engineering of Hasanuddin University, Makassar, Indonesia..

Wawan Rauf – obtained a Master of Engineering in Mechanical Engineering in 2020 from Hasanuddin University. His research areas are on fluid dynamics and computational fluid dynamics.

Muhammad Ihsan –works for LLDIKTI 9 Sulawesi at Sekolah Tinggi Teknik Baramuli. He holds masters degrees in transport engineering from Asian Institute of Technology, Bangkok, Thailand and Universitas Gajah Mada, Yogyakarta, Indonesia.