

R9 Aerodynamic drag reduction on the application of suction flow control.pdf

by

Submission date: 18-May-2023 10:33AM (UTC+0700)

Submission ID: 2095938107

File name: R9 Aerodynamic drag reduction on the application of suction flow control.pdf (930.87K)

Word count: 3946

Character count: 19428

PAPER • ³ OPEN ACCESS

Aerodynamic drag reduction on the application of suction flow control on vehicle model with varied upstream velocity

⁴
To cite this article: R Tarakka *et al* 2021 *IOP Conf. Ser.: Mater. Sci. Eng.* **1173** 012045

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



ECS **240th ECS Meeting**
Digital Meeting, Oct 10-14, 2021
We are going fully digital!
Attendees register for free!
REGISTER NOW

Aerodynamic drag reduction on the application of suction flow control on vehicle model with varied upstream velocity

R Tarakka^{1*}, N Salam¹, Jalaluddin¹, W Rauf^{1,2} and M Ihsan³

¹ Department of Mechanical Engineering, Faculty of Engineering, Hasanuddin University, Jalan Poros Malino, KM.6, Gowa, 92172, Indonesia

² Department of Mechanical Engineering, Faculty of Engineering, Gorontalo University, Jalan Jenderal Sudirman, Gorontalo 96211, Indonesia

³ Sekolah Tinggi Teknik Baramuli, Pinrang, 91215, Indonesia

*Email: rustan_tarakka@yahoo.com

Abstract. The innovation in the automotive industry must prioritize the issue of energy security and the environment. One aspect of the innovations is regarding the development of more aerodynamic vehicle design. The study of vehicle aerodynamics has been considered as very important, due to its direct relation to the safety and efficiency factors of energy utilization. Applying active control on the onset region of flow separation is one example of the efforts. This study aims to analyse the effect of the application of active control by suction to the aerodynamic drag of the vehicle model through qualitatively observations of flow dynamics and quantitatively pressure fields. The test model is a modified Ahmed body with 35° slant angles and varied flow orientations. This study has been conducted in both numerical computation and experimental testing at a suction velocity of 1.0 m/s and upstream velocity of 11.1 m/s, 13.9 m/s, and 16.7 m/s, respectively. The results have found out that the attachment of active control is proven capable able to delay flow separations, to increase the pressure coefficients on the back wall, and to reduce aerodynamic drag by 10.8487% for computational methods and 10.9748% for experimental methods.

Keywords: flow control, pressure coefficient, aerodynamic drag

1. Introduction

Innovations in the automotive industry must prioritize the issue of the energy crisis and environmental resilience. One aspect of the innovations is focused on developing more aerodynamic vehicle designs. Vehicle aerodynamics is a study related to the pattern of airflow through the surface to affect vehicle performance when driving. This study has been conducted as an effort to improve the safety and efficiency of fuel consumptions [1]. Efficiency on fuel consumptions of 700 to 1000 litres every year can be achieved just by reducing aerodynamic drag by 2-3% [2].

Aerodynamic drag analysis has been supported by visualizing flow characteristics and pressure fields on the rear wall of the vehicle. Airflow patterns describe the separation process occurring at the top edge of the back. The faster the process of flow separation occurs; the aerodynamic drag of the vehicle will



be even greater due to the high intensity of wake formation, which is the main cause of the rearward traction phenomenon.

The pressure field on the back wall also determines the intensity of the aerodynamic drag. The flow separation will generate a reduction in pressure distribution to cause considerable differences of pressure between the front and rear walls of the vehicle, which is a major contributor to aerodynamic drag. One effort to overcoming this problem is to delay separation. The delay of the flow separation process is intended to reduce the intensity of wake formation in the back wall so that the minimization of negative pressure and the reduction of drag can be achieved. Methods that can be carried out as an effort to delay the process of flow separation include the application of active control. The use of active control is considered effective because it involves the addition of external energy in the form of the application of suction techniques in the initial area of the flow separation process. Part of the fluid flow that reaches the separation area will undergo suction resulting in less intense wake formation and subsequently minimizing the phenomenon of rearward traction.

Mousa et al investigated of the impact of applying rectangular suction active control placed on the upper side of the back of the vehicle model through a numerical simulation approach (Computational Fluid Dynamics). The test model used is the result of modelling a sport utility vehicle (SUV) with a 1:10 dimension ratio to real vehicles. The results of the study revealed the application of suction active control could reduce aerodynamic drag by 9% [16]. Another study was carried out by Wessen & Thiele who placed active control on the upper end of the Ahmed body model with a 25° angle of attack. The study applied a numerical computation approach with the Large Eddy turbulence model. The results showed a reduction in flow separation, an increase in the distribution of pressure on the rear wall of vehicle models, and obtained an aerodynamic drag reduction of 9.4% [4].

2. Materials and Methods

This study is devoted to assessing the magnitude of the reduction in aerodynamic drag of vehicles that are modelled referring to the Ahmed model by altering the direction of flow orientation and reducing the dimensions of the test model. Vehicle modelling to obtain relevant features related to the fluid dynamics in real vehicle has been widely studied [5, 6, 7, 8, 9, 10, and 11]. The test model has a dimension ratio of 0.17 (1:6) to the actual Ahmed model with a slope of the front geometry angle (θ) of 35°. The dimensions of the test model are written length (l) = 174mm, height (h) = 48mm, and width (w) = 64.83mm. The suction active control applied is in the form of five apertures with a diameter of each circle (\varnothing) 7mm and the distance between apertures (s) of 10.81mm.

The apertures are designated as suction 1 (Sc_1), suction 2 (Sc_2), suction 3 (Sc_3), suction 4 (Sc_4), and suction 5 (Sc_5). The suction velocity (U_{sc}) applied was 1.0 m/s at an upstream velocity (U_0) of 11.1 m/s, 13.9 m/s, and 16.7 m/s. The test model is shown in figure 1.

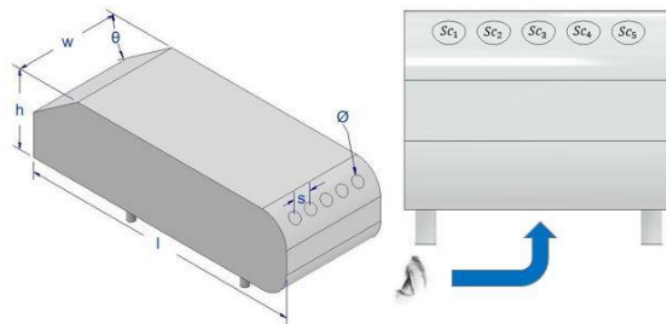


Figure 1. Model test

To support the study of the aerodynamic drag of the vehicle model, fluid flow patterns generated on the rear parts of the model and the pressure field are displayed through the display of the minimum pressure coefficient table (C_p) on the rear wall of the vehicle model through a computational approach. Pressure data retrieval is focused on the rear wall of the vehicle model. The data were taken as many as five lines in the location in the direction of the vertical axis.

Respective locations are defined as $z/w = -1/2$, $z/w = -1/4$, $z/w = 0$, $z/w = 1/4$, and $z/w = 1/2$, where z/w is the ratio of the grid width (z) to the model width (w). For horizontal axis direction, 5-lines pressure field data are obtained ($y/h = 0.83$, $y/h = 0.67$, $y/h = 0.50$, $y/h = 0.33$, and $y/h = 0.17$) on models without active control, whereas 4-lines pressure field data are obtained ($y/h = 0.67$, $y/h = 0.50$, $y/h = 0.33$, and $y/h = 0.17$) on models with applied active control. y/h is the ratio of grid height (y) to model height (h). Figure 2 shows the details of the location of pressure field data retrieval. The value of the pressure field will be displayed in a dimensionless number, the pressure coefficient (C_p) through the application of equation 1 [12].

$$C_p = \frac{P - P_0}{\frac{1}{2} \rho V^2} \tag{1}$$

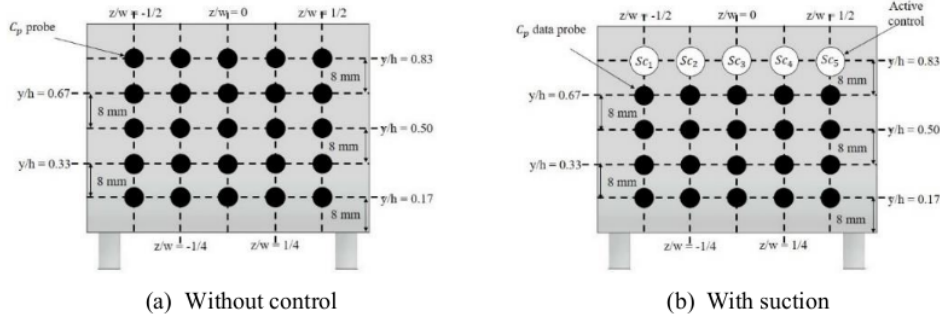


Figure 2. Location for pressure field retrieval

The vehicle model was designed using the Autodesk inventor software as shown in figure 1. Then the model was assigned into the computational domain and the meshed was generated using Gambit software. The computational domain and mesh display are shown in figure 3.

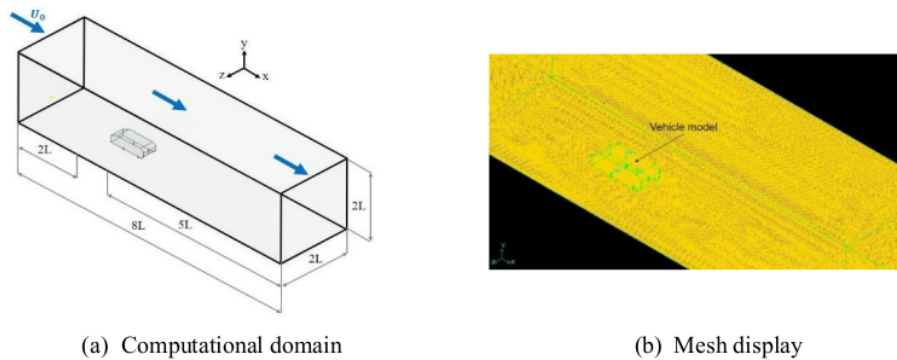


Figure 3. Computational domain and mesh display

The numerical computational results for aerodynamic drag data are validated through the experimental laboratory testing process using the subsonic wind tunnel facility as shown in figure 4. The dimensions of the physical model and numerical model are identical. The measurement of the drag force utilizes a load cell device that is connected to the vehicle model via aluminium support rods. The force received by the vehicle model will be read on the computer display as a drag force (F_d) with newton unit (N). Retrieval of data is determined for 120 seconds to obtain a total of 120 drag data for each upstream velocity on the model with and without suction active control. From the 120 data, the average value is then taken as a comparison of the results obtained through the previous numerical computational approach. The average value of the drag force obtained was then converted into a dimensionless number of drag coefficients (C_d) through the application of equation 2 [12].

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

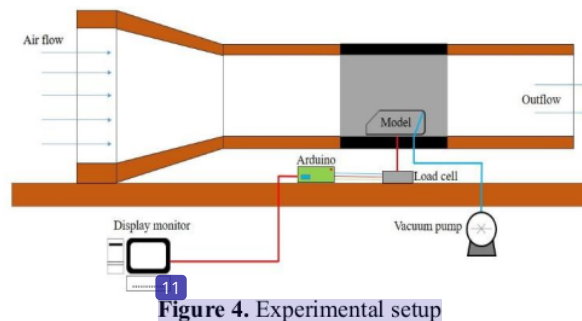


Figure 4. Experimental setup

3. Results and discussion

3.1. Flow field

Comparison of flow models of the model without control and with the application of active control at upstream velocity (U_0) of 11.1 m/s, 13.9 m/s, and 16.7 m/s are shown in figures 5, 6, and 7 respectively. For models without control at upstream velocity, $U_0=11.1$ m/s, 13.9 m/s, and 16.7 m/s as shown in figure 5(a), 6(a), and 7(a) shows large wake formation due to the process of flow separation that occurs right on the upper wall of the rear part of the vehicle model.

Flow that initially moves regularly experiences separation when it reaches the rear end of the model. As a result, there is a slowing down of flow on the middle side resulting in backflow. This slowing flow triggers the formation of a longitudinal vortex due to significant speed differences between the centre and side parts of the vehicle model.

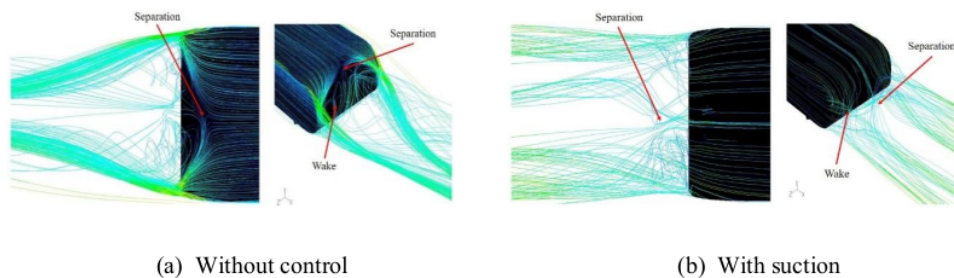


Figure 5. Flow field at upstream velocity, $U_0 = 11.1$ m/s.

In figure 5(b), 6(b), and 7(b) for models with suction active control application, the wake formation is apparently smaller than that without control, due to the delay in the process of flow separation. The separation process tends to move away from the back wall resulting in minimization of backflow. This also causes a decrease in the intensity of longitudinal vortex formation. These results are consistent with the findings of other researcher [13], revealing that the application of suction active control could minimize wake formation. Furthermore, the flow field at upstream velocity of $U_0=13.9$ m/s and $U_0=16.7$ m/s is presented respectively on Fig. 6 and Fig. 7.

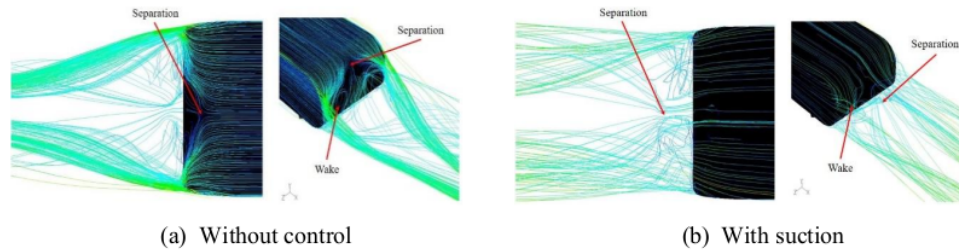


Figure 6. Flow field at upstream velocity, $U_0=13.9$ m/s.

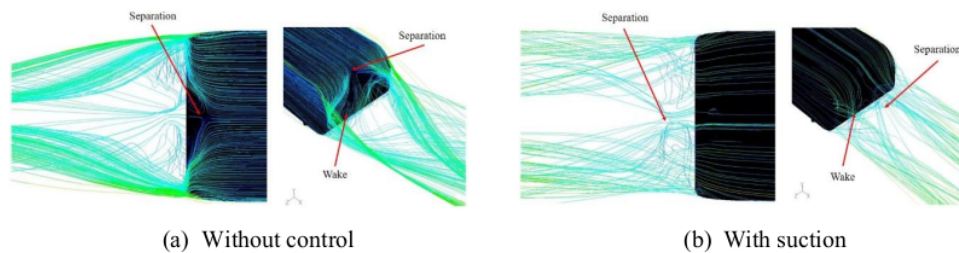


Figure 7. Flow field at upstream velocity, $U_0=16.7$ m/s.

3.2. Pressure field

In table 1, the lowest average minimum pressure coefficient for upstream velocity, $U_0=11.1$ m/s is obtained in the model without control at $C_p=-0.4295$. This minimum pressure coefficient is obtained on the upper rear part ($y/h=0.83$) because in this position the beginning of the flow separation occurs as shown in figure 5(a).

Table 1. Minimum pressure coefficient on upstream velocity, $U_0=11.1$ m/s

Position	Minimum pressure coefficient, C_p	
	Without control	With suction
$z/w=-1/2$	-0.4516	-0.2939
$z/w=-1/4$	-0.4148	-0.2717
$z/w=0$	-0.4148	-0.2717
$z/w=1/4$	-0.4148	-0.2939
$z/w=1/2$	-0.4516	-0.2939
Rate	-0.4295	-0.2850
Increase in C_p (%)	-	33.6422

Meanwhile, the model with the application of suction control shows 33.6422% increase in the average minimum pressure coefficient of 33.6422%, where the average minimum pressure coefficient value is -0.2850. This is caused by the separation delay where the separation process tends to retreat from the rear wall of the vehicle model. The lowest pressure coefficient value is obtained at position $y/h=0.17$ because of the backflow that appears from the bottom side of the model.

At the upstream velocity, $U_0=13.9$ m/s as shown in table 2 the lowest average minimum pressure coefficient is also obtained on the model without control of at $C_p=-0.4099$. The lowest pressure coefficient value is obtained at $y/h=0.83$ because this position is the beginning of the separation process as shown in figure 6(a). Meanwhile, the model with the application of suction active control shows an increase in the average minimum pressure coefficient of 17.0041%, where the average minimum pressure coefficient is recorder at $C_p=-0.3402$. The minimum pressure coefficient is obtained at position $y/h=0.17$ due to backflow arising from the bottom part of the model.

Table 2. Minimum pressure coefficient on upstream velocity, $U_0 = 13.9$ m/s

Position	Minimum pressure coefficient, C_p	
	Without control	With suction
$z/w=-1/2$	-0.4411	-0.3598
$z/w=-1/4$	-0.4112	-0.3272
$z/w=0$	-0.3747	-0.3272
$z/w=1/4$	-0.3747	-0.3272
$z/w=1/2$	-0.4477	-0.3598
Rate	-0.4099	-0.3402
Increase in C_p (%)	-	17.0041

Table 3 shows that the lowest average minimum pressure coefficient of -0.3769 for upstream velocity, $U_0=16.7$ m/s obtained in the model without control. The minimum pressure coefficient is also found on the direction axis $y/h=0.83$, because in that position the beginning of the flow separation occurs as shown in figure 7(a). For models with the application of suction active control an increase in the average minimum pressure coefficient of 24.2239% was achieved. This is because of the delay in the process of flow separation as shown in figure 7(b).

The minimum pressure coefficient is achieved in the direction of the axis $y/h=0.17$ because it is affected by the backflow that appears from the bottom part of the model. The results obtained in this study, are consistent with previous studies, which revealed that embedding active control on the upper rear part of the vehicle model could increase the basic pressure [14].

Table 3. Minimum pressure coefficient on upstream velocity, $U_0=16.7$ m/s

Position	Minimum pressure coefficient, C_p	
	Without control	With suction
$z/w=-1/2$	-0.4132	-0.3022
$z/w=-1/4$	-0.3769	-0.2841
$z/w=0$	-0.3405	-0.2788
$z/w=1/4$	-0.3405	-0.2788
$z/w=1/2$	-0.4132	-0.2841
Rate	-0.3769	-0.2856
Increase in C_p (%)	-	24.2239

3.3. Aerodynamic drag

3.3.1. Computational method

Both the model without control and the model with suction active control show a decrease in the drag coefficient along with the increase in upstream velocity as shown in figure 8. The drag coefficient for the model without control is greater than the model with the application of suction active control at each upstream velocity. This corresponds to the flow field shown in figures 5, 6, and 7 and the average minimum pressure coefficient shown in tables 1, 2, and 3, where the model without control is the model with the largest wake as well as the model with lowest minimum pressure coefficient at each upstream velocity.

Drag coefficients for models without control at each of the upstream velocity (U_0) 11.1 m/s, 13.9 m/s, and 16.7 m/s of 1.9894, 1.8733, and 1.8486. Meanwhile, for models with a suction active control implementation, drag coefficients are sequentially achieved at 1.7736, 1.7083, and 1.6493 as presented on Fig. 8.

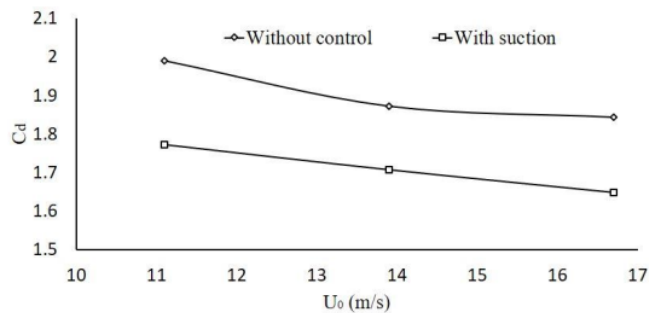


Figure 8. Relationship of drag coefficient and upstream velocity for computational methods

3.3.2. Experimental method

Similar to the computational approach, the experimental approach also shows a decrease in the drag coefficient along with an increase in upstream velocity. The highest drag coefficient is obtained in the model without control for each level of upstream velocity. The drag coefficient values of the model without control for each upstream velocity (U_0) of 11.1 m/s, 13.9 m/s, and 16.7 m/s are obtained at 1.8818, 1.7642, and 1.7628. Drag coefficients for the model with the application of suction active control at each upstream velocity are recorded respectively 1.6753, 1.6353, and 1.5748, refer to Fig. 9.

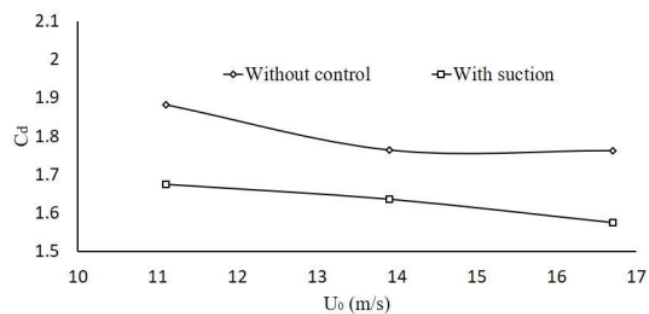


Figure 9. Relationship of drag coefficient to upstream velocity with experimental methods

3.3.3. Reduction ratio

The presence of suction active control in both the computational and experimental approaches at all upstream velocity produced reductions of the drag coefficient as shown in table 4. Reductions for the computational approach at upstream velocity (U_0) 11.1 m/s, 13.9 m/s, and 16.7 m/s are 10.8487%, 8.8079% and 10.5877%, respectively. From experimental approaches, the reductions of drag coefficients were recorded at 10.9748%, 7.3064%, and 10.6648%.

Table 4. Reduction ratio

U_0 (m/s)	Reduction (%)	
	Computational	Experimental
11.1	10.8487	10.9748
13.9	8.8079	7.3064
16.7	10.5877	10.6648

From these results, it was also found out that the highest drag coefficient reduction occurred at an upstream velocity of 11.1 m/s on both computational and experimental results. These results are consistent with previous research, which found out that the active control implementation could reduce the aerodynamic drag of the vehicle model [15-17]. Reduction of drag will improve overall vehicle performance [18].

27 Conclusion

The application of suction active control with a velocity of 1.0 m/s on the back of the vehicle model has a significant impact on the flow field, pressure field, and aerodynamic drag. It was found out that there was a reduction in wake formation due to a delay in the process of flow separation, an increase in the highest-pressure coefficient of 33.6422% at the upstream velocity $U_0=11.1$ m/s. The largest drag coefficient reductions of 10.8487% and 10.9748% were obtained at the same velocity for both the computational and experimental approaches, respectively.

Acknowledgments

This research was funded through the 2020 University Basic Research (PDU) Scheme, managed by the Hasanuddin University Research and Community Service Institute. We extend our gratitude to the Head and Staff of the Fluid Mechanics Laboratory of the Hasanuddin University, Faculty of Engineering.

References

- [1] Watkins S and Vano G 2008 *The effect of vehicle spacing on the aerodynamics of a representative car shape* Journal of Wind Engineering and Industrial Aerodynamics. **96** 1232-1239. DOI:10.1016/j.jweia.2007.06.042
- [2] Kumar R, Varshan N and Kannan 2019 *Aerodynamic design optimization of an automobile car using computational fluid dynamics approach* Australian Journal of Mechanical Engineering. DOI:10.1080/14484846.2019.1654963
- [3] Mousa A A, Yadav R and Fischer J *Aerodynamic drag reduction for a generic sport utility vehicle using rear suction* Journal of Engineering Research and Applications. **4** 101-107.
- [4] Wessen E and Thiele F 2009 *Road vehicle drag reduction by combined steady blowing and suction* 39th AIAA Fluid Dynamics Conference. DOI:10.2514/6.2009-4174
- [5] Hassan S M R, Islam T, Ali M and Islam MQ 2014 *Numerical study on aerodynamic drag*

- reduction of racing cars* Procedia Engineering **90** 308-313
DOI:10.1016/j.proeng.2014.11.854
- [6] Kulkarni S, Gandole G, Gaikwad P and Yadav J N 2018 *A review of aerodynamic study for reduction of drag force in cars in order to enhance the performance* International Journal on Emerging Trends in Technology. **5**.
- [7] Meile W, Brenn G, Reppenhagen A, Lechner B and Fuchs A 2010 *Experiments and numerical simulation on the aerodynamics of the Ahmed body* CFD Letters **3** 32-39.
- [8] Nallusamy S P and Mahali P K 2016 *Numerical and experimental investigations of drag force on scaled car model* Thermal Science. **20** 1153-58. DOI: 10.2298/tsci16s4153p
- [9] Tarakka R, Jalaluddin, Mire B and Umar M N 2015 *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. **10** 207-219.
- [10] Fourrie G, Keirsbulck L, Labraga L and Gillieron P 2011 *Bluff-body drag reduction using a deflector* Experiments in Fluids **50(2)** 385-395 DOI:10.1007/s00348-010-0937-6
- [11] Harinaldi, Budiarsa, Tarakka R, and Simanungkalit S P 2011 *Computational analysis of active flow control to reduce aerodynamics drag on a van model* IJMME-IJENS. **11** 24-30.
- [12] Cengel Y A and Cimbala J M 2014 *Fluid mechanics: fundamental and applications* (Mc Graw Hill)
- [13] Harinaldi, Budiarsa, Warjito, Kosasih E A and Tarakka R 2012 *Modification of flow structure over a van model by suction flow control to reduce aerodynamics drag* MAKARA. **16(1)** 15-21. DOI:10.7454/mst.v16i1.1021
- [14] Krajnovic S and Fernandes J 2011 *Numerical simulation of the flow around a simplified vehicle model with active flow control* International Journal of Heat and Fluid Flow. **32(1)** 192-200. DOI:10.1016/j.ijheatfluidflow.2010.06.007
- [15] Lehugeur B and Gillieron P 2006 *Active control of vortex breakdown phenomenon in the wake of a simplified car geometry* ASME 2006 2nd Joint U.S.-European Fluids Engineering Summer Meeting. DOI:10.1115/fedsm2006-98349
- [16] Tarakka R, Salam N, Jalaluddin, Ihsan M 2019 *Effect of blowing flow control and front geometry towards the reduction of aerodynamic drag on vehicle models* FME Transactions. **47(3)** 552-559. DOI:10.5937/fmet1903552T
- [17] Tarakka R, Salam N, Jalaluddin, Ihsan M 2018 *Active Flow Control by Suction on Vehicle Models with Variations on Front Geometry* International Review of Mechanical Engineering. **12(2)** 128-134. DOI:10.15866/ireme.v12i2.13876
- [18] Hyams D G, Sreenivas K, Pankajakshan R, Nicholas S D, and Briley W R 2011 *Computational simulation of model and full scale class 8 truck with drag reduction devices* Computers & Fluids **41(1)** 27-40. DOI:10.1016/j.compfluid.2010.09.015

R9 Aerodynamic drag reduction on the application of suction flow control.pdf

ORIGINALITY REPORT

15%

SIMILARITY INDEX

13%

INTERNET SOURCES

14%

PUBLICATIONS

8%

STUDENT PAPERS

PRIMARY SOURCES

1	Submitted to Universitas Sebelas Maret Student Paper	5%
2	pure-oai.bham.ac.uk Internet Source	2%
3	repository.lppm.unila.ac.id Internet Source	1%
4	hdl.handle.net Internet Source	1%
5	www.ijettjournal.org Internet Source	1%
6	Raffaele Volpe, Philippe Devinant, Azeddine Kourta. "Experimental characterization of the unsteady natural wake of the full-scale square back Ahmed body: flow bi-stability and spectral analysis", Experiments in Fluids, 2015 Publication	1%
7	theses.whiterose.ac.uk Internet Source	1%

8

link.springer.com

Internet Source

<1 %

9

Nasaruddin Salam, Rustan Tarakka, Jalaluddin Jalaluddin, Muhammad Ihsan. "Flow Separation Across Three Square Cylinders Arranged in Serial and Parallel Tandem Configuration", International Journal on Engineering Applications (IREA), 2020

Publication

<1 %

10

www.jafmonline.net

Internet Source

<1 %

11

www.researchgate.net

Internet Source

<1 %

12

Arslan Rehmat, Fayyaz, Salem Bashmal, Muddassar Sharif, Sikandar Khan. "Numerical Modeling of the Shape Optimization for a Commercial Car by Decreasing Drag and Increasing Stability", Arabian Journal for Science and Engineering, 2023

Publication

<1 %

13

Nasaruddin Salam, Rustan Tarakka, Radiansyah, Muhammad Ihsan Mukrim. "Analysis of the effect of rear-window tilt on aerodynamic drags of sedan vehicle model", AIP Publishing, 2023

Publication

<1 %

14 Notes on Numerical Fluid Mechanics and Multidisciplinary Design, 2012. <1 %
Publication

15 era.library.ualberta.ca <1 %
Internet Source

16 www.mdpi.com <1 %
Internet Source

17 Christine M. Scotti, Jorge Jimenez, Satish C. Muluk, Ender A. Finol. "Wall stress and flow dynamics in abdominal aortic aneurysms: finite element analysis vs. fluid-structure interaction", Computer Methods in Biomechanics and Biomedical Engineering, 2008 <1 %
Publication

18 Erik Wassen, Frank Thiele. "Simulation of Active Separation Control on a Generic Vehicle", 5th Flow Control Conference, 2010 <1 %
Publication

19 Mohammad Saeedi, Ali Tarokh, Kefu Liu. "Effectiveness of tail devices for wake control of road heavy vehicles", International Journal of Aerodynamics, 2021 <1 %
Publication

20 Z M Saleh, A H Ali. "Numerical Investigation of Drag Reduction Techniques in a Car Model", <1 %

IOP Conference Series: Materials Science and Engineering, 2020

Publication

21

ir.library.dc-uoit.ca

Internet Source

<1 %

22

ira.lib.polyu.edu.hk

Internet Source

<1 %

23

lintar.untar.ac.id

Internet Source

<1 %

24

www.issr-journals.org

Internet Source

<1 %

25

"Progress in Hybrid RANS-LES Modelling",
Springer Science and Business Media LLC,
2015

Publication

<1 %

26

N Salam, R Tarakka, Jalaluddin, M Ihsan, M A Jimran. "Flow drags across three minibus car models arranged in tandem in four configurations", IOP Conference Series: Materials Science and Engineering, 2021

Publication

<1 %

27

Rustan Tarakka, Jalaluddin, Nasaruddin Salam, Muhsyafir, Muhammad Ihsan Mukrim. "Effects of the additions of fin and suction on aerodynamic drags of vehicle model", AIP Publishing, 2023

Publication

<1 %

Exclude quotes On

Exclude matches < 5 words

Exclude bibliography On