

# Effect of Blowing Flow Control and Front Geometry Towards the Reduction of Aerodynamic Drag on Vehicle Models

*by* Rustan Taraka 07

---

FILE	TOWARDS_THE_REDUCTION_OF_AERODYNAMIC_DRAG_ON_VEHICLE_MODELS.PDF (810.05K)		
TIME SUBMITTED	22-AUG-2019 01:12PM (UTC+0700)	WORD COUNT	4991
SUBMISSION ID	1162261081	CHARACTER COUNT	25796

**Rustan Tarakka**

Assistant Professor  
Hasanuddin University  
Department of Mechanical Engineering  
Indonesia

**Nasaruddin Salam**

Professor  
Hasanuddin University  
Department of Mechanical  
Indonesia

**Jalaluddin**

Associate Professor  
Hasanuddin University  
Department of Mechanical Engineering  
Indonesia

**Muhammad Ihsan**

Lecturer  
Sekolah Tinggi Teknik Baramuli  
Department of Civil Engineering  
Indonesia

## Effect of Blowing Flow Control and Front Geometry Towards the Reduction of Aerodynamic Drag on Vehicle Models

This paper presents analyses of the effect of blowing flow control and variations on front geometry towards the reduction of aerodynamic drag on vehicle models. Blowing flow control is an alternative measure in modifying the onset of flow separation in the boundary layer on the surface of the vehicle. The modification is expected to reduce the dominating influence of the separation area on the total drag. Conducted in computational and experimental approaches, the research investigated the effect of frontal slant angle variations ( $\theta$ ) of 25°, 30° and 35° towards the reduction of aerodynamic drag on vehicle models on the application of blowing flow control with upstream and blowing speed of 16.7 m/s and 0.5 m/s, respectively. Load cells were used in the experimental method to validate the reduction of aerodynamic drag obtained from computational method. It is indicated that the effects of blowing flow control and variations on front geometry are significant in the increasing on pressure coefficients and the reduction of aerodynamic drag on vehicle models. The largest increase on pressure coefficients 8.93% is indicated on the vehicle model with  $\theta=35^\circ$ , while the largest reduction of aerodynamic drag occurred on the same model with the values of 14.81 and 12.54 for computational and experimental methods, respectively.

**Keywords:** active flow control, aerodynamic drag reduction, blowing, front geometry, vehicle model.

### 1. INTRODUCTION

As one of the most preferred types of general vehicles, family car in the form of multi-purpose van (MPV) has both advantages and drawbacks. Of the drawbacks, one typical character is the demand of relatively larger capacity engine which means larger fuel consumption than its more compact counterparts. This type also generally has the basic shape of the bluff body to maximize the space volume of the passenger compartment. In terms of aerodynamics, this form results in larger aerodynamic drag due to the occurrence of enormous flow separation in rear parts of the vehicle body creating higher energy consumption of the vehicle. The aerodynamic drag contributes about half of mechanical energy expenditures of vehicles running at average highway speed of around 55 to 60 mph [1].

Ahmed vehicle model is an extremely simplified bluff-body model frequently used as a benchmark in vehicle aerodynamics research. A number of experimental research [2-6] and numerical studies [7-12] have been performed using the Ahmed model. One of the techniques under development to reduce aerodynamic drag on vehicles and to modify the generation of flow separation in the boundary layer on the surface of the

vehicle which resulted in the generation of a backflow around the vehicle is the active flow control application. Active control strategies involve the addition of energy which aims to control, either in the form of prevention or inhibiting, the occurrence of flow separation which may lead to backflow on the surface of the vehicle without changing the shape of vehicles [13].

Some active control techniques have been developed and focusing on local intervention in wall turbulence dealing with steady blowing or suction [14-22]. Krentel et al. modeled a predictive closed-loop actuation approach for one steady blowing excitation configuration [16]. Harinaldi et al. [23] used a modified or reversed Ahmed body equipped with active flow control by blowing and found that the drag reductions achieved by computational and experimental methods were 13.92% and 11.28%. A review has also been elaborated for methods for the application of flow control on a square back car model [24]. This study aimed to analyze the effect of blowing active control incorporating the variations on front geometry towards the reduction aerodynamic drag on vehicle models. Improved reduction of aerodynamic drag can reduce flow separation which in turns will lead to energy efficiency. Comprehension of aerodynamic drag reductions and pressure coefficients are expected to improve the design method of future vehicles.

### 2. METHODOLOGY

The study investigates drag reduction occurring in a bluff body of van model adapted from Ahmed model, in

Received: November 2018, Accepted: March 2019

Correspondence to: Dr Rustan Tarakka  
Department of Mechanical Engineering,

Jl. Poros Malino, Gowa, Indonesia  
E-mail: rustan\_tarakka@yahoo.com

doi: 10.5937/fmet1903552T

© Faculty of Mechanical Engineering, Belgrade. All rights reserved

FME Transactions (2019) 47, 552-559 552

which flow streams in reversed direction from the original model or in other names, reversed Ahmed model [23, 25-28]. The van model was equipped with an active control by applying blowing techniques. Reversed Ahmed body model was chosen since it is a good representation of typical forms of popular family van produced by car manufacturer. The van model was investigated in both computational method, or CFD, and experimental method.

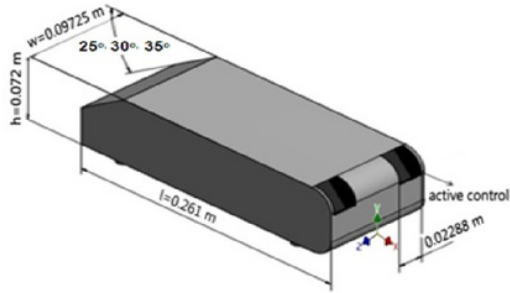


Figure 1. Reversed Ahmed body vehicle model

Figure 1 represents the basic vehicle model employed in the research with a 0.25 geometric ratio of the original Ahmed body model [2]. The vehicle model geometry was defined by its length ( $l=0.261\text{m}$ ), width ( $w=0.09725\text{ m}$ ) and its height ( $h=0.072\text{ m}$ ). In this configuration, the front part of model was inclined with slant angles ( $\theta$ ) of  $25^\circ$ ,  $30^\circ$  and  $35^\circ$  relative to the horizontal reference.

The value of viscous drag force and pressure drag force  $F_d$  is denoted in the equation (1).

$$F_d = \int \tau_w \sin \theta dS + \int p \cos \theta dS \quad (1)$$

while, drag coefficient  $C_d$  is expressed in the equation (2).

$$C_d = \int \frac{\tau_w}{\frac{1}{2} \rho V_\infty^2 S} \sin \theta dS + \int \frac{C_p \cos \theta dS}{S} \quad (2)$$

where  $\tau_w = \mu (du/dy)_w$  is the wall shear stress which is assessed from the velocity gradient at the wall and  $C_p = (p-p_\infty)/(\rho V_\infty^2/2)$  is pressure coefficient which is assessed from pressure distribution at the wall.

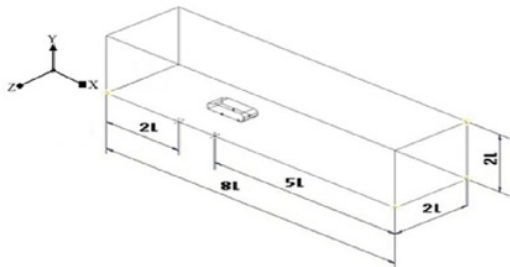


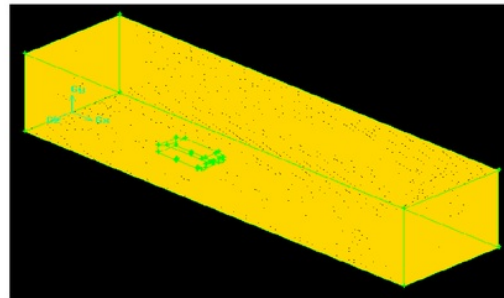
Figure 2. The 3D computational domain

The applied 3D computational domain is as shown in figure 2 denoting dimensions of length ( $L$ )= $8l$ , width ( $W$ )= $2l$ , and height ( $H$ )= $2l$ , where  $l$  is the length of model in  $x$ -axis. This computation approach utilizes

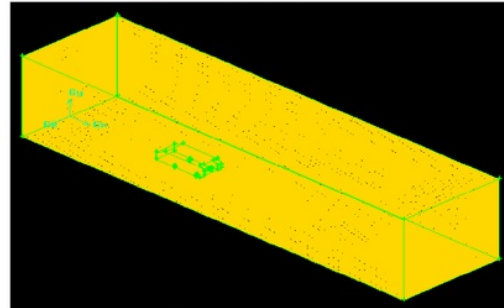
standard  $k$ -epsilon turbulence model as described on equation (3) for kinetic energy and equation (4) for dissipation rate.

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_b - \rho \epsilon + Y_M + S_k \quad (3)$$

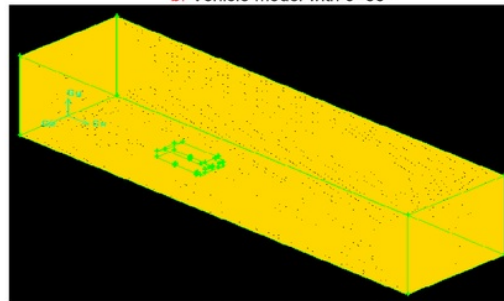
$$\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + c_{1\epsilon} \frac{\epsilon}{k} (P_k + C_{3\epsilon} P_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (4)$$



a. Vehicle model with  $\theta=25^\circ$



b. Vehicle model with  $\theta=30^\circ$



c. Vehicle model with  $\theta=35^\circ$

Figure 3. Meshing on models with blowing control

Figure 3 shows the meshing on respective vehicle models with the application of blowing on the rear part of models. The type of meshing was tetra/hybrid element with hexcore type, where the number of mesh volume for the model with  $\theta=25^\circ$  was 2,321,940, while for models with  $\theta=30^\circ$  and  $\theta=35^\circ$  the number of mesh were 2,274,917 and 2,319,492 respectively. Inlet velocity of  $16.7\text{ m/s}$  is assigned as the boundary condition. Average free stream at upstream region was

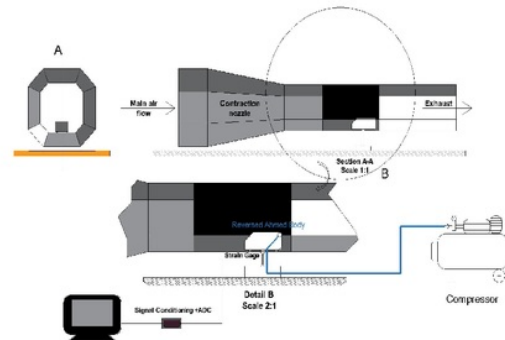
assumed to be in a steady state and uniform condition. The blowing speed was defined in 0.5 m/s. Reynolds number corresponding to the length of the test model was at  $Re = 2.98 \times 10^5$ . The detailed computational conditions are given in table 1.

**Table 1. Description of computational condition**

Vehicle model	3D, steady state $\theta=25^\circ$ , $\theta=30^\circ$ and $\theta=35^\circ$
Fluid	Air
Fluid properties	Density 1.225 kg/m <sup>3</sup> Viscosity 1.10017894 kg/m-s
Boundary condition without an active flow control	Vehicle model Wall Outlet Pressure outlet Inlet Velocity inlet Wall Wall
Boundary condition with blowing flow control	Vehicle model Wall Outlet Pressure outlet Inlet Velocity inlet Wall Wall Blowing1 Velocity inlet Blowing2 Velocity inlet
Upstream velocity	16.7 m/s
Blowing velocity	0.5 m/s

The tests were conducted in a controlled subsonic wind tunnel supplied with free stream air flow, testing acrylic van model with a 0.25 scale to the original Ahmed body model [2]. The wind tunnel has been calibrated complying manufacturer specification and considering recommendation by several works [29-31]. The van models are classified as models without flow control and models with blowing flow control. The blowing apparatus was configured at the internal part of the body of the model where the flow separation was predicted by computational method to induce a significant drag. The blowing apparatus was operated by using a mini compressor with blowing velocity at 0.5 m/s.

The investigated parameter was the aerodynamic drag force measured by using a load cell. Prior to the main experiments, the load cell was calibrated by using a digital balance. A preliminary measurement was performed to determine the statistical uncertainty of force measurements which was predicted to range at about  $\pm 2\%$ . The setup for the aerodynamic drag force measurement is as shown in figure 4.



**Figure 4. Experimental setup for the aerodynamic drag force measurement**

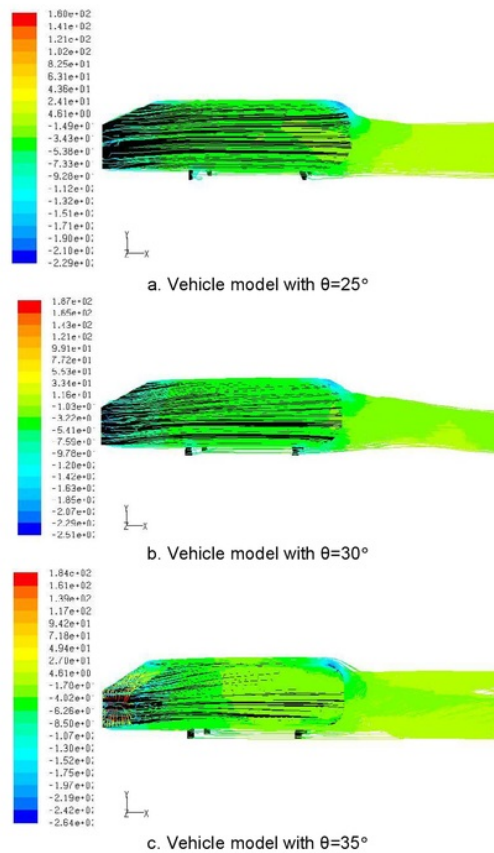
Dimensionless drag coefficient relative to drag force working on the bluff body is defined in the equation (5):

$$C_d = \frac{F_d}{\frac{1}{2} \rho V_\infty^2 S} \quad (5)$$

where,  $\rho$  is air density,  $V_\infty$  is free stream velocity,  $S$  is cross sectional area and  $F_d$  is the total drag force working on vehicle models as measured by the load cell.

### 3. RESULTS AND DISCUSSION

Main results of the research are pressure coefficients and aerodynamic drags which are evaluated in terms of reduction in the application of blowing active control. Figure 5 presents contour pathlines coloured by static pressure for models without active flow control for respective front slant angles.



**Figure 5. Pathlines colored by static pressure without active flow control**

The relationships of flow characteristics and vehicle's geometric parameters are presented in consecutive figures and tables. The first to be discussed is the expression of the distribution of pressure coefficient in the relation to the so-called  $y/h$  ratio which is the ratio of the height of the grid to the height of vehicle model. The second will be the patterns of pressure coefficient distribution on the rear part of vehicle models in

regards with  $z/w$  parameter, the ratio of width of grid to the width of vehicle models. The first one is shown in figure 6 for vehicle models with slant angle variations ( $\theta$ ) of  $25^\circ$ ,  $30^\circ$  and  $35^\circ$  and given upstream velocity of  $16.7 \text{ m/s}$ .

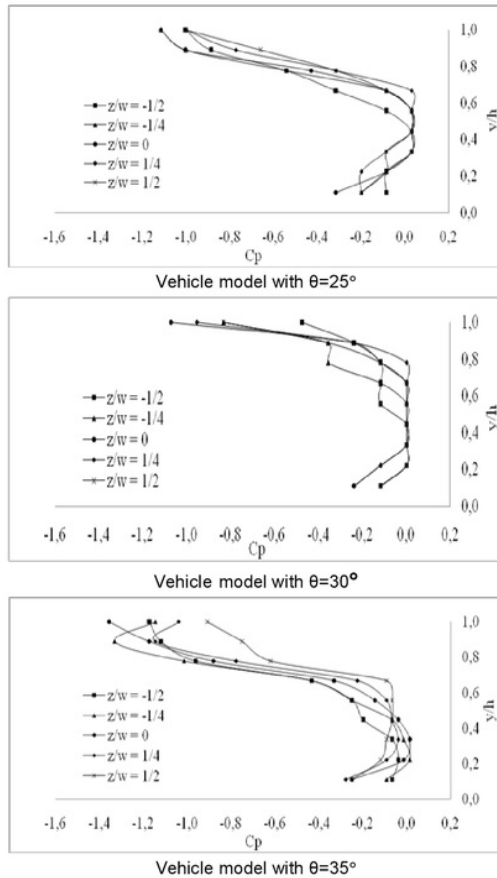


Figure 6. The distribution of pressure coefficients without an active flow control

The minimum values of pressure coefficients on respective test model are presented in table 2 where, as shown in the table 2, the minimum value of pressure coefficients occurs at  $y/h=1$ , specifically on the edge of upper side of rear part of respective vehicle model. The highest pressure coefficient is recorded on the vehicle model with a  $30^\circ$  front slant angle as compared to the coefficients on models with  $25^\circ$  and  $35^\circ$  frontal slant angles. It is expected that a flow separation is likely to occur on the rear part of vehicle models, where the separation could induce back flow and therefore can reduce the pressure coefficient. This evidence is in agreement with the opinion of Anderson et al. [32].

Table 2. The minimum value of pressure coefficients without an active flow control

Vehicle model	Pressure coefficient, $C_p$	$y/h$	$z/w$
$\theta=25^\circ$	-1.1148	1	-1/4 and 0
$\theta=30^\circ$	-1.0716	1	0
$\theta=35^\circ$	-1.3556	1	0

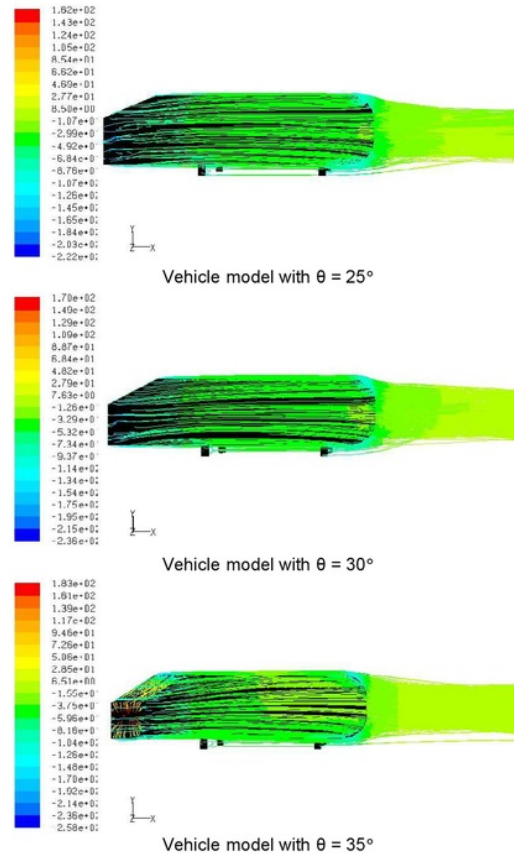


Figure 7. Pathlines colored by static pressure with blowing flow control

The presence of active control in the form of blowing was then evaluated based on the same parameter. Figure 7 presents contour pathlines colored by static pressure for models with the application of blowing active flow control for respective front slant angles. Figure 8 presents the distributions of pressure coefficients for given  $16.7 \text{ m/s}$  upstream velocity  $U_o$  as well as  $0.5 \text{ m/s}$  blowing velocity  $U_{bl}$ ; all were applied on vehicle models with frontal slant angles of  $25^\circ$ ,  $30^\circ$  and  $35^\circ$  respectively.

The figures show that with the application of blowing flow control, the pressure coefficient tends to increase. From around  $y/h=0.6$  to  $y/h=1$ , pressure coefficients start to change in a positive direction. This finding shows that on the upper side of the rear part of respective model, pressure coefficient increases. By the application of blowing flow control, the lower pressure stream, and considering the shape factor and friction of air with model's wall can be reduced therefore the flow separation on the rear part of the test model can be reduced as well. Table 3 summarizes the minimum values of the pressure coefficient distribution on a  $0.5 \text{ m/s}$  given blowing velocity  $U_{bl}$  and a  $16.7 \text{ m/s}$  upstream velocity  $U_o$ . The table also shows that the smallest pressure coefficient distribution was on the model with a  $30^\circ$  front slant angle when compared to the test model with  $25^\circ$  and  $35^\circ$  slant angles.

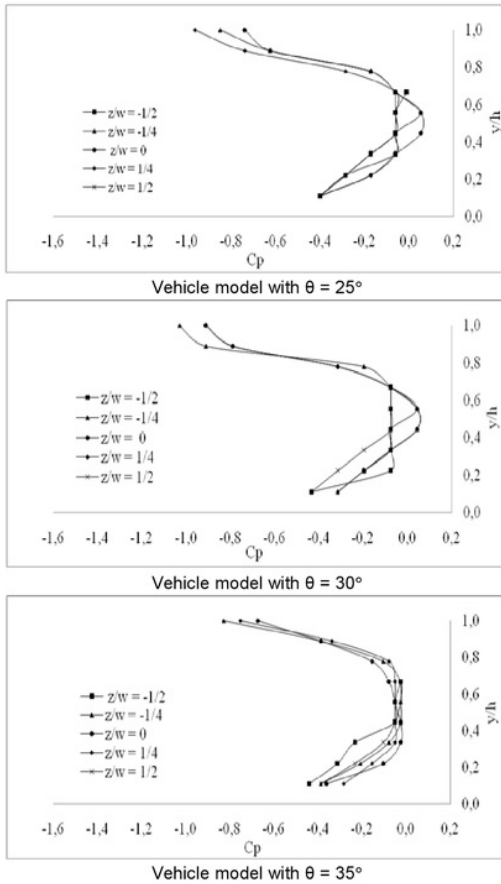


Figure 8. Pressure coefficient distribution with blowing flow control

Table 3. Minimum values of pressure coefficient with blowing flow control

Vehicle model	Pressure coefficient, $C_p$	y/h	z/w
$\theta=25^\circ$	-0.9622	1	1/4
$\theta=30^\circ$	-1.0285	1	-1/4
$\theta=35^\circ$	-0.8279	1	-1/4

The effect of additional active control by blowing with  $U_{bl}=0.5$  m/s blowing speed in the reduction of the flow separation at the rear of each vehicle model is shown in table 4. The reduced flow separation gives an effect in the increase of the pressure coefficients on all vehicle models [33].

Table 4. Increasing pressure coefficient with blowing flow control

Vehicle model	Pressure coefficient, $C_p$		Increasing $C_p$ , (%)
	Without active flow control	With blowing	
$\theta=25^\circ$	-1.1148	-0.9622	13.69
$\theta=30^\circ$	-1.0716	-1.0285	4.02
$\theta=35^\circ$	-1.3556	-0.8279	38.93

The largest value of increasing of pressure coefficient with the addition of blowing active control is on the

model with  $35^\circ$  front slant angle gaining a number of percentages of 38.93%.

### 3.1 The computational method

Table 5 presented the aerodynamic drag coefficients for computational approach on the vehicle models with frontal slant angle of  $\theta=25^\circ$ ,  $\theta=30^\circ$  and  $\theta=35^\circ$  both for normal treatment (without flow control) and with the addition blowing flow control with blowing speed  $U_{bl}$  of 0.5 m/s and upstream velocity  $U_o$  of 16.7 m/s.

Table 5. Aerodynamic drag coefficient and aerodynamic drag reduction by computational method

Vehicle model	Aerodynamic drag coefficient, $C_d$		Aerodynamic drag reduction, %
	without active flow control	with blowing	
$\theta=25^\circ$	1.7752	1.5639	11.90
$\theta=30^\circ$	1.6709	1.5699	6.04
$\theta=35^\circ$	1.7556	1.4953	14.81

Table 5 gives information that, in vehicle model without active control, the smallest aerodynamic drag coefficient is obtained in model with a  $30^\circ$  slant angle with the value of 1.6709 while on the models with  $25^\circ$  and  $35^\circ$  front slant angles, aerodynamic drag coefficients were 1.7752 and 1.7556 respectively. It is shown that the aerodynamic drag coefficients on each model were decreasing as the effect of the additional blowing active control while the smallest drag coefficient occurred in the model with  $\theta=35^\circ$  where the value of 1.4953. For models with  $\theta=25^\circ$  and  $\theta=30^\circ$ , the aerodynamic drag coefficients were 1.5639 and 1.5699 respectively.

From the table 5 also, the largest reduction of aerodynamic drag as the effect of blowing flow control occurs on vehicle model with  $\theta$  of  $35^\circ$  on a given 0.5 m/s blowing velocity  $U_{bl}$  and 16.7 m/s upstream velocity  $U_o$  with the reduction percentage was 14.81%. For the vehicle models with slant angles of  $25^\circ$  and  $30^\circ$ , the reduction were 11.90% and 6.04% respectively, as shown in table 5. The aerodynamic drag value on vehicle model with front slant angle  $35^\circ$  is capable to increase pressure coefficient on rear part of vehicle model for up to 38.93% as the effect of blowing flow control. The increasing of pressure coefficient on rear parts of vehicle models can decrease the aerodynamic drag. Results obtained from the research have confirmed the results of other researchers [16, 18, 23, 26, 27] where the application of active control by blowing can reduce aerodynamic drag on vehicle models.

### 3.2 The experimental method

Table 6 presents the aerodynamic drag coefficients obtained from experimental method on three vehicle models, which have similar geometric parameters to the model used for computational method.

Table 6 gives the information that for models without flow control, the smallest aerodynamic drag coefficients occurs in the  $30^\circ$  front slant angle vehicle model with 1.5173 value as for models with  $\theta=25^\circ$  and  $\theta=35^\circ$ ,

the aerodynamic drag coefficients were 1.6237 and 1.5655 respectively. Additional blowing flow control for all models gives reduction of aerodynamic drag coefficients while the smallest aerodynamic drag coefficients occurred in the model with a 35° front slant angle with the value of 1.3747% and 1.4480% as well as 1.4313% for vehicle models with  $\theta=25^\circ$  and  $\theta=30^\circ$ .

**Table 6. Aerodynamic drag coefficient and aerodynamic drag reduction by experimental method**

Vehicle model	Aerodynamic drag coefficient, $C_d$		Aerodynamic drag reduction, %
	without active flow control	with blowing	
$\theta=25^\circ$	1.6237	1.4480	10.82
$\theta=30^\circ$	1.5173	1.4313	5.67
$\theta=35^\circ$	1.5655	1.3747	12.54

From the table 6 also, it is obvious that the smallest reduction of aerodynamic drag by active control with blowing speed  $U_{bl}$  of 0.5 m/s and upstream velocity  $U_o$  of 16.7 m/s occurred on the model with  $\theta=35^\circ$  giving the reduction of 12.54% while on models with  $\theta=25^\circ$  dan  $\theta=30^\circ$  the reductions were 10.82% and 5.67% respectively.

The comparison of aerodynamic drag coefficients for vehicle models with  $\theta=25^\circ$ ,  $\theta=30^\circ$  and  $\theta=35^\circ$  front slant angles obtained from both computational method as well as experimental method are shown on tables 7, 8 and 9 respectively.

Table 7 contains the comparison of aerodynamic drag coefficient for vehicle model with slant angle  $\theta=25^\circ$  from computational and experimental methods. It is shown that the aerodynamic drag coefficients from the two methods on models without flow control differ in about 8.53%, as for models with a 0.5 m/s blowing control, the two methods result a 7.41% difference in the value of aerodynamic drag coefficient. The table 7 also informs that there is a 1.08% slight difference in the reduction of aerodynamic drag coefficient for the two methods.

**Table 7. Aerodynamic drag coefficient ( $C_d$ ) for vehicle model with  $\theta=25^\circ$**

Description	Aerodynamic drag coefficient, ( $C_d$ )		Aerodynamic drag coefficient ( $C_d$ ) reduction (%)
	without active flow control	with blowing	
Computational	1.7752	1.5639	11.90
Experimental	1.6237	1.4480	10.82
Difference	8.53%	7.41%	1.08

Table 8 shows the comparison of aerodynamic drag coefficients from both computational and experimental approaches for vehicle model with slant angle of  $\theta=30^\circ$ . The coefficients of aerodynamic drag of the two methods for models without flow control differ on 9.19% differences. For vehicle models with blowing flow control at blowing speed of 0.5 m/s, the two methods give 8.83% difference in aerodynamic drag coefficient values. The table 8 also shows that the

reduction of aerodynamic drag coefficients between computational and experimental methods differ on only 0.37%.

**Table 8. Aerodynamic drag coefficient ( $C_d$ ) for vehicle model with  $\theta=30^\circ$**

Description	Aerodynamic drag coefficient, ( $C_d$ )		Aerodynamic drag coefficient ( $C_d$ ) reduction (%)
	without active flow control	with blowing	
Computational	1.6709	1.5699	6.04
Experimental	1.5173	1.4313	5.67
Difference	9.19%	8.83%	0.37

**Table 9. Aerodynamic drag coefficient ( $C_d$ ) for vehicle model with  $\theta=35^\circ$**

Description	Aerodynamic drag coefficient, ( $C_d$ )		Aerodynamic drag coefficient ( $C_d$ ) reduction (%)
	without active flow control	with blowing	
Computational	1.7556	1.4953	14.81
Experiment	1.5655	1.3706	12.54
Difference	10.82%	8.34%	2.27

Aerodynamic drag coefficients obtained from computational and experimental methods for  $\theta=35^\circ$  model are listed in table 9. For models without flow control, the aerodynamic drag coefficients of the two methods have 10.82% differences. On vehicle models with active blowing flow control at blowing speed of 0.5 m/s, the two methods give 8.34% difference in aerodynamic drag coefficient values. The table 9 also shows that the reductions of aerodynamic drag coefficients of the two methods differ about 2.27%.

#### 4. CONCLUSION

Based on the results of pressure coefficients and drag reduction on reversed Ahmed vehicle models with front slant angles ( $\theta$ ) of 25°, 30° and 35° as well as the application of blowing flow control, some conclusion can be drawn. It is obvious that the active flow control by blowing and variations on front geometry provide significant impact to the increasing of pressure coefficients and reduction of aerodynamic drag. The most significant increase on pressure coefficients occurs on vehicle model with the 35° front slant angle, gaining 38.93% value, where pressure coefficients with and without blowing flow control reach -0.8279 and -1.3556, respectively. In terms of aerodynamic drag reduction, vehicle model with the 35° front slant angle also gains the value of 14.81 and 12.54 for computational and experimental methods, while the reductions of aerodynamic drag coefficients of the two approaches have 2.27% difference.

#### 5. ACKNOWLEDGEMENTS

The research was funded by the Ministry of Research, Technology and Higher Education, The Republic of

## REFERENCES

- [1] Agarwal, R.K. "Sustainable ground transportation – review of technologies, challenges and opportunities", International Journal of Energy and Environment, Vol. 4, No.6, pp.1061-1078, 2013.
- [2] Ahmed, S., Ramm, G., Faltin, G. "Some Salient Features Of The Time-Averaged Ground Vehicle Wake", SAE Technical Paper 840300, 1984.
- [3] Sims-Williams, D., Dominy, R. "Experimental Investigation into Unsteadiness and Instability in Passenger Car Aerodynamics", SAE Technical Paper 980391, 1998.
- [4] Bayraktar, I., Landman, D., Baysal, O. "Experimental and Computational Investigation of Ahmed Body for Ground Vehicle Aerodynamics", SAE Technical Paper 2001-01-2742, 2001.
- [5] Joseph, P., Amandolese, X., Edouard, C., Aider, J. "Flow control using MEMS pulsed micro-jets on the Ahmed body", Experiment in Fluids, Vol. 54, No. 1, 1442, 2013.
- [6] Lienhart, H., Becker, S. "Flow and Turbulence Structure in the Wake of a Simplified Car Model", SAE Technical Paper 2003-01-0656, 2003.
- [7] Han, T. "Computational analysis of three-dimensional turbulent flow around a bluff body in ground proximity", AIAA Journal, Vol. 27, No.9, 1989, pp. 1213-1219.
- [8] Basara, B., Przulj, V., Tibaut, P. "On the Calculation of External Aerodynamics: Industrial Benchmarks", SAE Technical Paper 2001-01-0701, 2001.
- [9] Basara, B. "Numerical simulation of turbulent wakes around a vehicle", in: *ASME Fluid Engineering Division Summer Meeting FEDSM99-7324*. San Francisco, USA, 1999.
- [10] Basara, B., Alajbegovic, A. "Steady state calculations of turbulent flow around Morel body". in: *7<sup>th</sup> International Symposium of Flow Modelling and Turbulence Measurements*, Taiwan, 1998, pp. 1–8
- [11] Gilliéron, P., Chometon, F. "Modeling of stationary three-dimensional separated air flows around an Ahmed reference model", in : *ESAIM, Proceeding of Third International Workshop on Vortex Flows and Related Numerical Methods*, 7, 1999, pp. 173–182.
- [12] Kapadia, S., Roy, S., Vallero, M., Wurtzler, K., Forsythe J. "Detached-Eddy Simulation over a Reference Ahmed Car Model", in: Friedrich, R., Geurts, B.J., Métails, O. (eds) *Direct and Large-Eddy Simulation V*. ERCOFTAC Series, 9. Springer, Dordrecht, 2004, pp 481-488.
- [13] Conan, B., Anthoine, J., Planquart, P. "Experimental aerodynamic study of a car-type bluff body", Experiments in Fluids, Vol. 50, No. 5, pp.1273-1284, 2010.
- [14] Krogstad, P.A., Kourakine, A. "Some effects of localized injection on the turbulence structure in a boundary layer", Physics of Fluids, Vol. 12, No.11, pp. 2990-2999, 2000.
- [15] Park, J., Choi, H. "Effects of uniform blowing through blowing or suction from a spanwise slot on a turbulent boundary layer flow", Physics of Fluids, Vol. 11, No. 10, pp. 3095–3105, 1999.
- [16] Krentel D., Muminovic R., Brunn A., Nitsche W., King R. "Application of Active Flow Control on Generic 3D Car Models". in: King R. (eds) *Active Flow Control II. Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, 108. Springer, Berlin-Heidelberg, Germany, 2010, pp. 223-239.
- [17] Heinemann, T., Springer, M., Lienhart, H. "Active flow control on a 1:4 car model", Experiments in Fluids, Vol. 55, No. 5, 1738, 2014.
- [18] Mestiri, R., Ahmed-Bensoltane, A., Keirsbulck, L., Aloui, F., Labraga, L. "Active Flow Control at the Rear End of a Generic Car Model Using Steady Blowing", Journal of Applied Fluid Mechanics, Vol. 7, No. 4, pp. 565-571, 2014.
- [19] Tounsi, N., Mestiri, R., Keirsbulck, L., Oualli, H., Hanchi, S., Aloui, F. "Experimental Study of Flow Control on Bluff Body using Piezoelectric Actuators" , Journal of Applied Fluid Mechanics , Vol. 9, No. 2, pp. 827-838, 2016.
- [20] Tian, J., Zhang, Y., Zhu, H., Xiao, H. "Aerodynamic drag reduction and flow control of Ahmed body with flaps", Advances in Mechanical Engineering, Vol. 9, No.7, pp. 1–17, 2017.
- [21] Shadmani, S., Mousavi-Nainiyan, S.M., Ghasemiasl, R., Mirzaei, M., Pouryoussefi, S.G. "Experimental Study of Flow Control Over an Ahmed Body Using Plasma Actuator", Mechanics and Mechanical Engineering, Vol. 22, No. 1, pp. 239–251, 2018.
- [22] Prakash, B., Bergada, J.M., Mellibovsky, F. "Three Dimensional Analysis of Ahmed Body Aerodynamic Performance Enhancement using Steady Suction and Blowing Flow Control Techniques", in : *Tenth International Conference on Computational Fluid Dynamics (ICCFD10)*, 2018. Barcelona, Spain.
- [23] Harinaldi, Budiarso, Tarakka, R., Simanungkalit, S.P. "Effect of Active Control by Blowing to Aerodynamic Drag of Bluff Body Van Model", International Journal of Fluid Mechanics Research, Vol. 40, No.4, pp. 312-323, 2013.
- [24] Julian, J., Karim, R.F., Budiarso, Harinaldi, "Review: Flow Control on a Squareback Model" International Review of Aerospace Engineering (IREASE), Vol. 10, No. 4, pp. 230-239, 2017.
- [25] Harinaldi, Budiarso, Warjito, Kosasih, E.A., Tarakka, R., Simanungkalit, S.P. "Active technique by suction to control the flowstructure over a van model", Journal of Engineering and Applied Science, Vol. 7, No. 2, pp.215-222, 2012.

- [26] Harinaldi, Budiarmo, Tarakka, R., Simanungkalit, S.P. "Computational Analysis of Active Flow Control to Reduce Aerodynamics Drag on a Van Model", International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS, Vol. 11, No. 3, pp. 24-30, 2011
- [27] Tarakka, R., Jalaluddin, Mire, B., Umar, M.N. "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.
- [28] Tarakka, R., Salam, N., Jalaluddin, Ihsan, M. "Active Flow Control by Suction on Vehicle Models with Variations on Front Geometry", International Review of Mechanical Engineering, Vol. 12, No. 2, 2018, pp. 128-134.
- [29] Plint and Partners, Manual of Educational Wind Tunnel, England, 1982.
- [30] Ocololjić, G., Damljanović, D., Vuković, Đ., Rašuo, B. "Contemporary Frame of Measurement and Assessment of Wind-Tunnel Flow Quality in a Low-Speed Facility", FME Transactions, Vol. 46, No. 4, pp. 429-442, 2018.
- [31] Ocololjić, G., Damljanović, D., Rašuo, B., Isaković, J. "Testing of a Standard Model in the VTI's Large subsonic Wind-tunnel Facility to Establish Users' Confidence", FME Transactions, Vol. 42, No. 3, pp. 212-218, 2014.
- [32] Anderson, J.D. "Fundamental of Aerodynamics" 3<sup>rd</sup> ed., McGraw-Hill, Singapore, 2001.
- [33] Sphon, A., Gilliéron, P. "Flow separations generated by simplified geometry of an automotive vehicle", in: *IUTAM Symposium: Unsteady separated flows*, 2002, Toulouse, France.

#### NOMENCLATURE

$C_d$	drag coefficient
$C_p$	pressure coefficient
$F_d$	pressure drag force [N]
$h$	height of test model [m]
$l$	length of test model [m]

$\rho$	density [ $\text{kg/m}^3$ ]
$\theta$	front slant angle [ $^\circ$ ]
Re	Reynolds number
$S$	cross section area [ $\text{m}^2$ ]
$\tau_w$	wall shear stress [ $\text{N/m}^2$ ]
$U_{bl}$	blowing velocity [m/s]
$U_o$	upstream velocity [m/s]
$\mu$	viscosity [ $\text{N}\cdot\text{s/m}^2$ ]
$w$	width of test model [m]

#### УТИЦАЈ РЕГУЛАЦИЈЕ ПРОТОКА ДУВАЊА И ПРЕДЊЕ ГЕОМЕТРИЈЕ НА СМАЊЕЊЕ АЕРОДИНАМИЧКОГ ОТПОРА КОД МОДЕЛА ВОЗИЛА

Р.Тарака, Н.Салам, Џалалудин, М.Ихсан

Анализира се утицај регулације протока дувања и варијације предње геометрије у циљу редукције аеродинамичког отпора код модела возила. Регулација протока дувања је алтернативна мера за модификацију почетка раздвајања протока у граничном слоју на површини возила. Очекује се да се модификацијом смањи доминантан утицај области раздвајања на укупан отпор. Истраживање извршено нумеричким и експерименталним методама се бави испитивањем утицаја варијација предњег нагибног угла од  $25^\circ$ ,  $30^\circ$  и  $35^\circ$  на редукцију аеродинамичког отпора код модела возила при регулацији протока дувања узлазно и брзине протока од 16,7 м/с односно 0,5 м/с. Мерне ћелије су коришћене код експеримента за евалуацију аеродинамичког отпора израчунатог нумеричком методом. Утврђено је да регулација протока дувања и варијације предње геометрије имају значајан утицај на повећање коефицијената притиска и смањење аеродинамичког отпора код модела возила. Највеће повећање коефицијената притиска од 38,93% је било код модела са нагибним углом од  $35^\circ$ , док је највеће смањење аеродинамичког отпора било на истом моделу: од 14,81 – 12,54 применом нумеричке односно експерименталне методе.

# Effect of Blowing Flow Control and Front Geometry Towards the Reduction of Aerodynamic Drag on Vehicle Models

## ORIGINALITY REPORT

% **16**  
SIMILARITY INDEX

% **11**  
INTERNET SOURCES

% **6**  
PUBLICATIONS

% **5**  
STUDENT PAPERS

## PRIMARY SOURCES

**1** [www.ijens.org](http://www.ijens.org) Internet Source %**4**

**2** [linknovate.com](http://linknovate.com) Internet Source %**2**

**3** [eng.unhas.ac.id](http://eng.unhas.ac.id) Internet Source %**2**

**4** [jafmonline.net](http://jafmonline.net) Internet Source %**1**

**5** [buscompress.com](http://buscompress.com) Internet Source %**1**

**6** Submitted to Swansea Metropolitan University Student Paper %**1**

**7** [ijens.org](http://ijens.org) Internet Source %**1**

**8** Kaushik Dey, Verlekar Rohit, Soni Kumar, Benedict Thomas. "Dynamic analysis of functionally graded shaft", FME Transactions, <%**1**

2019

Publication

- 
- 9 Submitted to Liverpool John Moores University <% 1  
Student Paper
- 
- 10 [www.scielo.org.ar](http://www.scielo.org.ar) <% 1  
Internet Source
- 
- 11 Submitted to Curtin University of Technology <% 1  
Student Paper
- 
- 12 Chand, Ramesh, G.C. Rana, and S.K. Kango. <% 1  
"Effect of variable gravity on thermal instability  
of rotating nanofluid in porous medium", FME  
Transaction, 2015.  
Publication
- 
- 13 Shengli Li, Ru Wan, Dongwei Wang, Pan Guo. <% 1  
"Effect of end plates on transiting test for  
measuring the aerodynamic coefficient of  
structures using wind generated by a moving  
vehicle", Journal of Wind Engineering and  
Industrial Aerodynamics, 2019  
Publication
- 
- 14 Submitted to Chulalongkorn University <% 1  
Student Paper
- 
- 15 [isesp.org](http://isesp.org) <% 1  
Internet Source
- 
- 16 Submitted to University of Brighton <% 1  
Student Paper

17

**Submitted to Universitas Brawijaya**

Student Paper

&lt;% 1

18

Tunay, Tural, Bulent Yaniktepe, and Besir Sahin. "Computational and experimental investigations of the vortical flow structures in the near wake region downstream of the Ahmed vehicle model", Journal of Wind Engineering and Industrial Aerodynamics, 2016.

Publication

&lt;% 1

19

Notes on Numerical Fluid Mechanics and Multidisciplinary Design, 2016.

Publication

&lt;% 1

20

**Submitted to University of Leeds**

Student Paper

&lt;% 1

21

Seeyuan Cheng, Makoto Tsubokura, Takuji Nakashima, Takahide Nouzawa, Takaki Nakamura. "Transient Analysis of Pitching Stability of Sedan-type Vehicle Models Using Large-Eddy Simulation", 28th AIAA Applied Aerodynamics Conference, 2010

Publication

&lt;% 1

22

Zhiqun Yuan, Zhengqi Gu, Yiping Wang, Xiaoqun Huang. "Numerical investigation for the influence of the car underbody on aerodynamic force and flow structure evolution in crosswind", Advances in Mechanical Engineering, 2018

Publication

&lt;% 1

---

23

Submitted to University of Hull

Student Paper

<% 1

---

24

[ira.lib.polyu.edu.hk](http://ira.lib.polyu.edu.hk)

Internet Source

<% 1

---

25

[www.inderscience.com](http://www.inderscience.com)

Internet Source

<% 1

---

26

Submitted to Prince of Songkla University

Student Paper

<% 1

---

27

"Active Flow Control II", Springer Nature, 2010

Publication

<% 1

---

28

B. F. Zhang, Y. Zhou, S. To. "Unsteady flow structures around a high-drag Ahmed body",  
Journal of Fluid Mechanics, 2015

Publication

<% 1

---

29

Sonja Gorjanc, Ema Jurkin. "Web textbook for descriptive geometry", FME Transactions, 2019

Publication

<% 1

---

30

Submitted to Bogazici University

Student Paper

<% 1

---

31

Submitted to University of Technology, Sydney

Student Paper

<% 1

---

32

Optimization and Computational Fluid Dynamics, 2008.

Publication

<% 1

---

---

EXCLUDE QUOTES ON

EXCLUDE  
BIBLIOGRAPHY ON

EXCLUDE MATCHES < 5  
WORDS