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 of 38.93% 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]. Krnel 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 11.11%. 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
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 to the original Ahmed body model [2]. The vehicle model geometry was defined by its length \( l=0.261 \) m, width \( w=0.09725 \) m and its height \( h=0.072 \) m. In this configuration, the front part of model was inclined with slant angles \( \theta \) of 25°, 30° and 35° relative to the horizontal reference.

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

\[
F_d = \int \tau_w \sin \theta dS + \int p \cos \theta dS
\]  

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

\[
C_d = \int \frac{\tau_w}{\frac{1}{2} \rho V_c^2} \sin \theta dS + \int \frac{C_p \cos \theta dS}{S}
\]  

where \( \tau_w = \mu \left( \frac{du}{dy} \right)_{w} \) is the wall shear stress which is assessed from the velocity gradient at the wall and \( C_p = \frac{(p-p_\infty)}{(\rho \frac{V_c}{2})} \) 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.

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho ku_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial k}{\partial x_j}\right] + P_b - \rho_v + \frac{\partial}{\partial x_j}(\rho e u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_e}\right) \frac{\partial e}{\partial x_j}\right] +
\]

\[
+ \frac{c_{1e}}{k} \left( P_b + C_{3e} P_b - C_{2e} P_b \right) - C_{2e} P_b \frac{e^2}{k} + S_e
\]  

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 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 \( \text{Re}=2.98 \times 10^5 \). The detailed computational conditions are given in table 1.

**Table 1. Description of computational condition**

<table>
<thead>
<tr>
<th>Vehicle model</th>
<th>3D, steady state ( \theta=25^\circ ), ( \theta=30^\circ ), and ( \theta=35^\circ )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid</td>
<td>Air</td>
</tr>
<tr>
<td>Fluid properties</td>
<td>Density: 1.225 kg/m(^3)</td>
</tr>
<tr>
<td></td>
<td>Viscosity: 0.00017894 kg/m-s</td>
</tr>
<tr>
<td>Boundary condition without an active flow control</td>
<td>Vehicle model</td>
</tr>
<tr>
<td></td>
<td>Outlet: Pressure outlet</td>
</tr>
<tr>
<td></td>
<td>Inlet: Velocity inlet</td>
</tr>
<tr>
<td></td>
<td>Wall: Wall</td>
</tr>
<tr>
<td>Boundary condition with blowing flow control</td>
<td>Vehicle model</td>
</tr>
<tr>
<td></td>
<td>Outlet: Pressure outlet</td>
</tr>
<tr>
<td></td>
<td>Inlet: Velocity inlet</td>
</tr>
<tr>
<td></td>
<td>Wall: Wall</td>
</tr>
<tr>
<td></td>
<td>Blowing1: Velocity inlet</td>
</tr>
<tr>
<td></td>
<td>Blowing2: Velocity inlet</td>
</tr>
<tr>
<td>Upstream velocity</td>
<td>16.7 m/s</td>
</tr>
<tr>
<td>Blowing velocity</td>
<td>0.5 m/s</td>
</tr>
</tbody>
</table>

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.

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

\[
C_d = \frac{F_d}{\frac{1}{2} \rho V^2 \pi S}
\]

where, \( \rho \) is air density, \( V \) 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 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](image)

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 \( C_p \) 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
(θ) of 25°, 30° and 35° and given upstream velocity of
16.7 m/s.

![Figure 6. The distribution of pressure coefficients without
an active flow control](image)

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 models. The
highest pressure coefficient is recorded on the vehicle
model with a 30° front slant angle as compared to the
coefficients on models with 25° and 35° frontal slant
angles. It is expected that a flow separation is likely to
occur on the rear part of the vehicle models, where the
separation could induce back flow and therefore can
reduce the pressure coefficient. This evidence is in agree-
ment with the opinion of Anderson et al. [32].

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

<table>
<thead>
<tr>
<th>Vehicle model</th>
<th>Pressure coefficient, C_p</th>
<th>y/h</th>
<th>z/w</th>
</tr>
</thead>
<tbody>
<tr>
<td>θ=25°</td>
<td>-1.1148</td>
<td>1</td>
<td>-1/4 and 0</td>
</tr>
<tr>
<td>θ=30°</td>
<td>-1.0716</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>θ=35°</td>
<td>-1.3556</td>
<td>1</td>
<td>0</td>
</tr>
</tbody>
</table>

The presence of active control in the form of blo-
wing 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 coeffi-
cients for given 16.7 m/s upstream velocity U_o as well
as 0.5 m/s blowing velocity U_bl; all were applied on
vehicle models with frontal slant angles of 25°, 30° and
35° 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 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 m/s given
blowing velocity U_bl and a 16.7 m/s upstream velocity
U_o. The table also shows that the smallest pressure
coefficient distribution was on the model with a 30°
front slant angle when compared to the test model with
25° and 35° slant angles.

![Figure 7. Pathlines colored by static pressure with blowing
flow control](image)
Vehicle model with $\theta = 25^\circ$

Figure 8. Pressure coefficient distribution with blowing flow control

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

<table>
<thead>
<tr>
<th>Vehicle model</th>
<th>Pressure coefficient, $C_p$</th>
<th>$y/h$</th>
<th>$z/w$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\theta=25^\circ$</td>
<td>-0.9622</td>
<td>1</td>
<td>1/4</td>
</tr>
<tr>
<td>$\theta=30^\circ$</td>
<td>-1.0285</td>
<td>1</td>
<td>-1/4</td>
</tr>
<tr>
<td>$\theta=35^\circ$</td>
<td>-0.8279</td>
<td>1</td>
<td>-1/4</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Vehicle model</th>
<th>Pressure coefficient, $C_p$</th>
<th>Increasing $C_p$ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Without active flow control</td>
<td>With blowing</td>
<td></td>
</tr>
<tr>
<td>$\theta=25^\circ$</td>
<td>-1.1148</td>
<td>-0.9622</td>
</tr>
<tr>
<td>$\theta=30^\circ$</td>
<td>-1.0716</td>
<td>-1.0285</td>
</tr>
<tr>
<td>$\theta=35^\circ$</td>
<td>-1.3556</td>
<td>-0.8279</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Vehicle model</th>
<th>Aerodynamic drag coefficient, $C_d$</th>
<th>Aerodynamic drag reduction, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Without active flow control</td>
<td>With blowing</td>
<td></td>
</tr>
<tr>
<td>$\theta=25^\circ$</td>
<td>1.7752</td>
<td>1.5639</td>
</tr>
<tr>
<td>$\theta=30^\circ$</td>
<td>1.6709</td>
<td>1.5699</td>
</tr>
<tr>
<td>$\theta=35^\circ$</td>
<td>1.7556</td>
<td>1.4953</td>
</tr>
</tbody>
</table>

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 coefficient 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
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 θ = 25° and θ = 30°.

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

<table>
<thead>
<tr>
<th>Vehicle model</th>
<th>Aerodynamic drag coefficient, (C\text{d})</th>
<th>Aerodynamic drag reduction, %</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>without active flow control</td>
<td>with blowing</td>
</tr>
<tr>
<td>θ = 25°</td>
<td>1.6237</td>
<td>1.4480</td>
</tr>
<tr>
<td>θ = 30°</td>
<td>1.5173</td>
<td>1.4313</td>
</tr>
<tr>
<td>θ = 35°</td>
<td>1.5655</td>
<td>1.3747</td>
</tr>
</tbody>
</table>

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

The comparison of aerodynamic drag coefficients for vehicle models with θ = 25°, θ = 30° and θ = 35° 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 θ = 25° 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 flow 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\text{d}) for vehicle model with θ=25°

<table>
<thead>
<tr>
<th>Description</th>
<th>Aerodynamic drag coefficient, (C\text{d})</th>
<th>Aerodynamic drag coefficient (C\text{d}) reduction (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>without active flow control</td>
<td>with blowing</td>
</tr>
<tr>
<td>Computational</td>
<td>1.7752</td>
<td>1.5639</td>
</tr>
<tr>
<td>Experimental</td>
<td>1.6237</td>
<td>1.4480</td>
</tr>
<tr>
<td>Difference</td>
<td>8.53%</td>
<td>7.41%</td>
</tr>
</tbody>
</table>

Table 8 shows the comparison of aerodynamic drag coefficients from both computational and experimental approaches for vehicle model with slant angle of θ = 30°. 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\text{d}) for vehicle model with θ=30°

<table>
<thead>
<tr>
<th>Description</th>
<th>Aerodynamic drag coefficient, (C\text{d})</th>
<th>Aerodynamic drag coefficient (C\text{d}) reduction (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>without active flow control</td>
<td>with blowing</td>
</tr>
<tr>
<td>Computational</td>
<td>1.7556</td>
<td>1.4953</td>
</tr>
<tr>
<td>Experimental</td>
<td>1.5655</td>
<td>1.3706</td>
</tr>
<tr>
<td>Difference</td>
<td>10.82%</td>
<td>8.34%</td>
</tr>
</tbody>
</table>

Aerodynamic drag coefficients obtained from computational and experimental methods for θ = 35° 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 (θ) 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.

ACKNOWLEDGEMENTS

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


NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$</td>
<td>density [kg/m$^3$]</td>
</tr>
<tr>
<td>$\theta$</td>
<td>front slant angle [°]</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$S$</td>
<td>cross section area [m$^2$]</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>wall shear stress [N/m$^2$]</td>
</tr>
<tr>
<td>$U_{bl}$</td>
<td>blowing velocity [m/s]</td>
</tr>
<tr>
<td>$U_o$</td>
<td>upstream velocity [m/s]</td>
</tr>
<tr>
<td>$\mu$</td>
<td>viscosity [N.s/m$^2$]</td>
</tr>
<tr>
<td>$w$</td>
<td>width of test model [m]</td>
</tr>
</tbody>
</table>

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