Aerodynamic Performance Analysis of Open-Wheel Vehicle: Investigation of Wings Installation Under Different Speeds

A Yudianto¹, H A Susanto², A Suyanto², I W Adiyasa¹, A Yudiantoko¹, N A Fauzi³

¹Department of Automotive Engineering Education, Universitas Negeri Yogyakarta, Yogyakarta, Indonesia
²Department of Light Vehicle Engineering, Muhammadiyah Imogiri Vocational High School, Yogyakarta, Indonesia
³Department of Transport Eksploatacja i Mechatronika Samochodów, Politechnika Krakowska, Krakowska, Poland

E-mail: aan.yudianto@uny.ac.id

Abstract. An open-wheel vehicle with manual hybrid engine control with formula-like body shape was developed for a learning media at a vocational high school. This study aims to analyze the airflow of over the vehicle body as the effect of wing installation under various vehicle speeds. The investigations are to observe and evaluate the drag coefficient, drag force, lift coefficient, downforce, pressure distribution, streamlines, and velocity vector condition by emphasizing on the results from Computational Fluid Dynamics (CFD) simulation. The study found that installing the front and rear wings contribute to the decrement of drag coefficient 0.07 points and keeping the drag force relatively constant since the projection area changed. The benefit of having wings installed significantly increases the downforce by 334 N, which is 5.51 times higher than without wings in the speed of 150 km/h. However, the engine cover gives a significant obstacle to airflow, reducing the functionality of the rear wing. As such, the presence of the rear wing does not have a substantial effect on reducing the wake region.

1. Introduction

The evolving vehicle technology and its application are ever-increasing nowadays. The implementation of the automotive field does not only occur in the automotive industries or higher level of education, but it also starts being implemented in career-focused education such as vocational high school. Considering the impact of fuel on the environment, attractive body-shaped vehicle, and the state of the art of the work leads to an initiative in conducting this project. The main purpose of the work is to create a learning media to support the practical knowledge and skills of students and, more broadly, to help in overcoming the environmental problem. The development of hybrid vehicles with some aerodynamic devices is one of the paces that has been done in this work. Many aspects are currently being studied. Investigation of the external body shapes with all the aerodynamic devices attached to it becomes the main concern to investigate in this work.

An external aerodynamic analysis involving external airflow over a vehicle has been becoming an interesting topic to discuss since it could give initial or even real results examination of the vehicle aerodynamics performance. Yet, the limitation of physical and real scale facilities, such as wind tunnel, does not limit the researchers, scholars, or engineers to broaden the knowledge and assess the vehicle design in terms of external flow investigation. The use of numerical simulation could become the most reachable and reliable solution [1], [2], [3]. The use of numerical investigation has been implemented...
in many other kinds of work as well [4]. Moreover, the newly built prototype vehicle, particularly with entirely different shapes and vehicle purposes, might still be questionable due to the lack of the assessment of the identical vehicle of the already studied similar work. Therefore, the study of the vehicle properties is potentially required to ensure the proper performance characteristic of the vehicle in many aspects. The aerodynamics topic has been chosen since the shape of the vehicle in this work is closely similar to the formula open-wheel vehicle, which is relatively famous for its aerodynamic devices.

Numerous research finding in the aerodynamic-related topics demonstrates the essential aspects of the contribution of the research topics being examined. The utilization of a numerical investigation method with software-based analysis and the validation of the result by experimenting on the real-case vehicle have been conducted by many researchers to validate the numerical calculation. Experimental and computational studies on a passenger vehicle have been conducted and the utilization of different yaw angles and two-vehicle platoon became the case to study the drag force and the pressure distribution of the one-fifth scale model of FIAT Linea [5]. The study found that the experimental and numerical coefficient of pressure and coefficient of drag values are in a good agreement by considering different yaw angles and two-vehicle platoon. The investigation on unsteady aerodynamic flow around a simplified square-back road vehicle had also been conducted [6]. The investigation is by employing drag reduction devices. The results showed that recirculation regions at the base are shortened and weakened. The utilization of the devices enormously increased the base pressure and gave the lower drag. Moreover handling and safety performance of a vehicle successfully enhanced the performance related to the vehicle handling and safety of race vehicles by employing active aerodynamic devices [7]. The devices mentioned are the so-called active inverted wings. The detailed analytical study and formulations of a nonlinear model with airfoils were clearly presented. The usage of other aerodynamic devices was studied [8] by examining the effect of underbody diffuser devices on the aerodynamic drag of vehicles in convoy. Different reference values of the diffuser angle were chosen to investigate its effect on the coefficient of drag of different vehicle bodies. Average drag on cases with diffusers found to be lesser than the no diffuser installed.

Considering many findings in the mentioned aerodynamic-related topic leads the initiative to perform this study. However, the open-wheel vehicle investigated in this study was newly built with highly relatively different shapes of the body and the purpose of the vehicle. This study investigates the aerodynamic aspect of the vehicle, in which the main issue is that whether the installation of aerodynamic components, which are front and rear wings, becomes the best decision for achieving high performance of the vehicle, especially in terms of aerodynamic under different speeds. Considering the rear wing that is actually the component to put a solar cell as the second power source after engine alternator becomes the main consideration to design such a dimension and shape. The vehicle aerodynamic behavior without and with wings installation in the various wind speed conditions is investigated in the following. The aerodynamic parameters in this study are the coefficient of drag, drag force, downforce, pressure distribution, streamlines, and vector velocity condition. The utilization of the Computational Fluid Dynamics method enables to aid of the complicated calculation due to the model complexity and graphically shows the required results in a communicative manner.

2. Vehicle Specification and Description
The vehicle is an open wheeler formula-type vehicle using a hybrid propulsion system with manual-switch-based hybrid control. The gasoline engine is positioned in the rear part of the vehicle, acting as a rear wheel drive with automatic transmission, while the electric brushless DC motors are mounted in both right and left front wheels. The additional power source is obtained by adopting solar cells that are used as an additional electric power source for accessories. These solar cells are mounted in the rear wings of the vehicle. The picture of the vehicle is depicted in Figure 1, and the specification of the vehicle can be seen in Table 1. The front wing is located at the fore-end of the vehicle in front of the front tires. This is designed in such a way so stat it is intended to direct the airflow and generate the downforce at the front part of the vehicle [9]. Since the engine is located at the backside just behind the driver compartment, the engine cover is designed to cover the engine with a part for taking the
fresh air behind the driver’s head indicated with the arrow. As it is already mentioned previously, the rear wing unit is actually a device to attach the solar panel as the additional power resource for the vehicle’s accessories, so that the shape is flat and positioned with certain inclination. This device is also aimed at the aerodynamic device to generate negative lift force at the rear part of the vehicle. A pair of the endplate is also attached at both ends of the solar panel.

![Formula-type open-wheel vehicle](image)

**Figure 1.** Formula-type open-wheel vehicle with the definition of the essential body’s part.

| Component     | Specification                                      |
|---------------|----------------------------------------------------|
| Engine        | Main engine with 1300cc Gasoline                   |
| Transmission  | Automatic Transmission                             |
| Electric Motor| 2 x 5KW Brushless DC motor                         |
| Solar Cells   | 150 Watt mounted at the rear wing                   |
| Batteries     | Lithium-Polymer 72V 63Ah                           |
| Wheelbase     | 2200 mm                                            |
| Wheel Track   | 1485 mm                                            |
| Overall height| 1240 mm                                            |
| Body Materials| Fiberglass                                         |

**Table 1.** Vehicle Specification and Dimension.

3. Model and Simulation Method
A representation of a simplified vehicle model is constructed to reduce the model complexity and simplify the analysis. The model utilized for simulation is the full-scale model. The vehicle model with front and rear wings installed have 3460 mm in overall length, 1720 mm in overall width, and 1280 mm in overall height, while the model of the vehicle without installation of wings has 3050 mm in overall length, 1680 mm in overall width and 1217 mm in overall height. The presence of wheel axels and suspension arm components, both in front and rear of the vehicle, are substituted by solid pipe connecting vehicle body and the wheels. Moreover, the complexity of the engine compartment is simplified by designing a solid body as part of the main body. A continuous solid body also replaces the shape of the driver compartment. The simplified model of the vehicle with and without wings can be seen in Figure 2.

![Simplified model](image)

**Figure 2.** Simplified model (A) with wings and (B) without wings.
The efficiency and the financial aspect make Computational Fluid Dynamic (CFD) a better solution in analyzing the aerodynamic behavior of the vehicle [10]. Moreover, numerical investigation software greatly reduces the production time owing to decrementing the need for costly physical testing and prototyping [11]. Therefore, a numerical investigation is performed as a useful tool to identify the aerodynamic behavior of the vehicle. In this case, 3D drawing software is used for creating the simplified model of the vehicle. CFD simulation software is also utilized in order to perform numerical experimental measurements of the aerodynamic of the model. In order to perform external aerodynamic performance analysis on a vehicle, these three conditions must be satisfied: the volume must be sealed, no internal geometry must be present, and no overlapping or intersection surface should be present [12].

The simulation is performed in a virtual wind tunnel. It is designed with 15 meters length and both 5 meters size in the wide and the height as depicted in Figure 3. The achieved maximum speed of the vehicle in the test drive is 150 km/h in gasoline mode. Thus, in order to study the aerodynamic behavior on the various speeds the wind velocities were set in 30 km/h (8.3 m/s), 60 km/h (16.67 m/s), 90 km/h (25 m/s), 120 km/h (33.3 m/s) and 150 km/h (41.67 m/s). Throughout the simulation, the origin of the x-axis is normal to the front direction of the vehicle, the y-axis is along with the height of the vehicle, and the z-axis is the vehicle side direction.

The mesh continuum is generated in CFD software with a polyhedral mesh having a base size value of 0.5 m. The mesh base size is established at multiple values for the purpose of verifying the result independent of the mesh density [1]. Local mesh setting is applied in the model with a relative minimum size of 5% and a relative size target of 10% with respect to base size value. The physics models are then generated. An assumption is made that the air is incompressible since fluid flow characteristics are desired rather than micro characteristics. The three dimensional and steady gas with the segregated flow and constant density are chosen to perform the simulation. Finally, the turbulent fluid flow with the K-epsilon model is adopted for its good behavior since it is in adverse pressure gradients and in separating flow conditions that are very common situations during the analysis of an external aerodynamic problem [10]. Furthermore, it is also considering its balance of robustness, relative ease of calculation, and accuracy, especially for bluff body problems [1].

![Figure 3. Virtual wind tunnel developed on software with the explanation of face definition.](image)

| Table 2. Surface type. |
|------------------------|
| Component              | Type            |
| Vehicle Model          | No-slip Wall    |
| Inlet Wall             | Velocity Inlet  |
| Outlet Wall            | Pressure Outlet |
| Tunnel and floor       | Slip Wall       |
The boundary-type specifications define the physical and operational characteristics of the model at those topological entries that represent model boundaries [10]. The initial pressure is determined to be a reference gage or zero Pa. The same value is set to the pressure outlet condition. The problem definition stated that the vehicle velocity is defined in five conditions, as mentioned before. These values are then assigned as an initial condition of velocity in the inlet wall surface. The vehicle surface, however, is created as a no-slip wall, and the tunnel is set to slip wall since it simulates the moving tunnel as if it simulates the real condition while the vehicle is running on the road. The surfaces are assigned to the following types, presented in Table 2.

### 4. Result and Discussion

#### 4.1. Drag

The coefficient of drag always depends on the shape of the vehicle body [12]. However, this value explains that the installation of the wings significantly has a correlation in determining the values of the $C_d$, considering:

$$C_d = \frac{D}{\frac{1}{2} \rho V^2 A}$$

the coefficient of drag as the function of vehicle speed $V$ obtained in this experiment, projection area $A$, with the total drag force $D$ [9]. Where the mathematical definition of total drag force $D$ can be rewritten from equation (1) as follow:

$$D = C_d \frac{1}{2} \rho V^2$$

In the first case, the complete vehicle with both front and rear wings are present; the result of the drag coefficient is 0.70. This result of the drag coefficient has been generated automatically by the software by equation (1). However, considering the average value of the drag coefficient of sports vehicles, which have a value between 0.30 – 0.37 [10], the obtained value is relatively much larger than the common vehicle running in the road. In the second case with the vehicle model without wings installation, the vehicle without wings has a drag coefficient of 0.77. However, if we compared to a prototype race car which has coefficient of drag approximately 0.75 [9], the value of drag coefficient of the vehicle with and without wings in this work is very reasonable and acceptable.

It is necessary to plot the accumulated drag coefficient with respect to the vehicle profile to quantify the local unequal in drag value for every profile shape of the vehicle [13]. Figures 4 and 5 explain the differences in cumulative drag coefficient value for the vehicle with wings and without wings, respectively. The starting value of the drag coefficient for the vehicle with wing increases due to the presence of front wings in the front part of the vehicle, while for the vehicle without wings, the value is rather flat. For both cases, starting from the driver compartment until the area behind the driver’s head location, the values are rather similar before it increases dramatically at around the location of the engine cover. This value differs at the location of the solar panel mounting. It is obvious that the rear wing that also behaves as the solar panel mounting contributes more to the accumulated value of the drag coefficient, as depicted in the following figures.

It is obviously important to realize that, when the fluid flows over the surface, the surface will resist its motion [12]. Moreover, the fuel consumption is directly proportional to the resistance overcome by the engine. The aerodynamic drag force is one of the resistances which are significant in magnitude at the higher speed [14]. This study, therefore, compares the results of the vehicle drag force due to the absence and the presence of the wings. The front wing is designed to have 30 degrees angle of attack. However, the rear wing is actually the solar panel mounted, which is inclined in 12 degrees. The result of the drag force can be observed in Figure 6.

Figure 6 depicts the results of the drag force as the function of vehicle speed. It can be clearly seen that the presence of the wings has only a small effect in the increment of drag force. With the obtained projection area of the vehicle with the wing is 1.37 m², the 982 N of overall vehicle drag force occurs when it is running with the maximum velocity of 150 km/h. Similarly, at the maximum speed with the projection area of the vehicle without a wing is 1.20 m², the obtained overall drag force
is 953 N. Quite similar values of the drag force are obtained at 30 km/h which are respectively 38 N and 39 N for with and without wings.

**Figure 4.** Accumulated drag coefficient plot showing the development of drag starting from the front part of the vehicle profile (right side) traveling rearward to the rear part of the vehicle (left side) with the wings attached, highlighting the step change in the drag in every shape of the vehicle with wings.

**Figure 5.** Accumulated drag coefficient plot showing the development of drag starting from the front part of the vehicle profile (right side) traveling rearward to the rear part of the vehicle (left side) without wings, highlighting the step change in the drag in every shape of the vehicle without wings.

**Figure 6.** Drag force of the vehicle under different speeds for the case of the vehicle with wings and without wings.
4.2. Downforce

The objectives of having aerodynamic components attached to the vehicle are reducing the drag force, creating the downforce, and for better handling due to the side wind. The negative lift that is better known as downforce will be analyzed in the following. The downforce, or aerodynamic grip, works in conjunction with the mechanical grip to improve the acceleration, braking, and cornering speed of the vehicle [15]. Considering the lift coefficient:

\[
C_l = \frac{L}{\frac{1}{2} \rho V^2 A}
\]

where \( L \) is the total lift force, the downforce depends on the vehicle speed and the frontal area of the vehicle [16]. The obtained overall lift coefficient is -0.28 for the vehicle complete with both front and rear wings installed and -0.08 when wings are absent.

The examination of the lift coefficient in every part of body shape is necessary to understand in this case. Figures 7 and 8 show the differences in accumulated lift coefficient along with the profile of the vehicle with wings and without wings. The installation of front wings successfully produces a negative lift indicated by the lower value of the accumulated lift coefficient. However, this value rises as the vehicle profile position is in the area of the front tire before hovering through the surface around the driver’s compartment at giving a peak in the area of the engine cover. The next essential point is that the rear wings effectively generates negative lift force and decrease the value of accumulated lift force. Instead, in the case of the vehicle without wings, the only differences are at the beginning of the shape at the location of the front part of the vehicle and the rear part of the vehicle at the location of the rear wings.

**Figure 7.** Lift development plot indicating a cumulative lift coefficient starting from the front part of the vehicle (right) going through the vehicle’s profile to the rear part of the vehicle (left) with wings.
Figure 8. Lift development plot indicating a cumulative lift coefficient starting from the front part of the vehicle (right) going through the vehicle’s profile to the rear part (left) without wings.

Figure 9. Downforce of the vehicle at different speeds conditions for the vehicle with wings installed and without wings.

Figure 9 compares the results of the vehicle downforce at different values of vehicle speed. It is clear that the presence of the wings gives a significant contribution to the increment of vehicle downforce. At 30 km/h, there is no significant difference shown. However, when the maximum speed is achieved, the vehicle with wings has the maximum downforce of 408 N, and it is 5.5 times higher than the downforce of the vehicle without the installation of the wings.

4.3. Pressure Distribution
The results of total pressure distribution are very useful to understand where the airflow is attached or detached. It is also a better indication of air intake placement. Laminar and turbulent boundary layer flows depend strongly on the pressure distribution, which is imposed by the external flow. For a pressure increase in the flow direction, the boundary layer flow is retarded, especially near the wall (e.g., the surface of the body), and even reversed flow may occur [16]. Figures 10 and 11 show the distribution of the pressure over the vehicle body surface. The highest pressure occurs chiefly in the stagnation point in the front end of the vehicle, the front part of the wheel and the roller bar behind the driver’s head. Since the engine is located in the rear part of the vehicle, there is an air intake placement for engine radiator cooling in the upper part of the roll bar or engine cover, as explained in the previous section. Other parts that reasonable to be criticized are the front part of the wheel and the forebody of the vehicle. It can be observed that in those locations it appears a relatively higher pressure.

Figure 10. Pressure distribution at the maximum speed of the vehicle at 150 km/h indicating every single part of the vehicle’s surface with the value of total pressure imposed on the vehicle’s surface.
As can be seen from Figures 10 and 11, there are regions of high pressure at the engine cover behind the driver’s head. As such, considering the engine location is in the rear part of the vehicle, the decision in designing the engine radiator allowing the airflows in this region is the best choice. Additionally, the higher total pressure also appears at the front end of the vehicle and the front part of the front wheel. The low-pressure spots occur in the edge of the tire, and the bottom side part of the engine cover. With wings are attached, the maximum pressure in some regions of the vehicle is 1153.6 Pa, while for the vehicle without wings installation, the maximum pressure reaches about 1087.7 Pa. The further identification of total pressure distribution in the side, front, and rear parts of the vehicle can be identified on the following figures.

4.4. Streamlines
In order to analyze the air path around the vehicle body and the wake regions, the analysis of the streamlines is performed. Figure 12 and Figure 13 demonstrate how the air flows over the vehicle surface at the speed of 30 km/h and 150 km/h with wings attached. It can be observed that there are two regions of dead water (i.e., the region of wake containing large eddying motion) when the vehicle runs 30 km/h. The first dead-water occurs behind the roll bar or engine cover before the rear wing. The second one is exactly behind the vehicle in the engine compartment. This vortex gives a significant amount of force in the backward direction of the vehicle that creates drag force. However, when the vehicle runs at 150 km/h, the wake behind the engine compartment disappears. It is due to the faster velocity of the airflows over the body. Therefore, it is clear that the wake still occurs in the rear part of the vehicle in the region between the rear wing and the upper part of the engine cover. The engine cover hinders the airflow, making the flow detached and creating a turbulent.

Moreover, after investigating the airflow in the right and left side of the driver’s head, it is clear that there is a wake region as well. Due to the location of the vortex is near the driver’s ears, it potentially creates a noise due to aerodynamic affecting the comfort of the driver. When the vehicle is running at 30 km/h (8.3 m/s) speed, the maximum air velocity through the vehicle part is 10.8 m/s. While the vehicle achieves the maximum speed of 150 km/h (41.67 m/s), the peak of air velocity is 53.78 m/s. This maximum velocity occurs in the top part of the tires, the front body near the dashboard, and the side part of the engine cover.

Figures 14 and 15 explain how the behavior of airflow over the vehicle body with the absence of the front and rear wings at the speed of 30 km/h and 150 km/h. The wake regions still appear in the backside of the vehicle. However, at the speed of 150 km/h, the wake regions are bigger as opposed to 30 km/h. The same phenomenon of dead-water appears as similar to the case with the wings are installed. The maximum air velocity when the vehicle is running at 30 km/h (8.3 m/s) is 10.39 m/s while at 150 km/h (41.67 m/s), the maximum air velocity is 52.03 m/s. The presence of the wings does not substantially contribute to reducing the wake regions when the conditions of the airflows in Figures 12, 13, 14, and 15 are compared. It can be observed that qualitatively, the wake produced by
both conditions are quite similar. It is surprisingly due to the position of the rear wing that the airflow is hindered by the engine cover and are not directly and smoothly attached through the wing.

**Figure 12.** Airflow path over the vehicle surface with wings at 30 km/h depicting the velocity how the air flows start from the front part of the vehicle rearwards.

**Figure 13.** The streamlines showing the air flows through the vehicle at different air velocity in every part of the surface and indicating the vortex region location.

**Figure 14.** Streamlines without wings at 30 km/h with the streamlines color representing the airflow velocity.
4.5. Velocity Vector

Similar to the streamlines plot, it can be similarly seen, yet varying trends, in the velocity vectors. Figures 16 and 17 show the direction of the air flows as a vector. Due to the peak of the engine cover, the highest velocity and the biggest vector inclination with respect to horizontal axis occur. When the rear wing is present, the maximum velocity vector is 10.26 m/s when the vehicle is running at 30 km/s (8.3 m/s). It can also be observed that the maximum velocity vector is approximately 51.83 m/s when the vehicle reaches the speed of 150 km/h (41.67 m/s).

![Figure 16. Velocity vectors with wings at 30 km/h showing the velocity vector at the middle vertical plane of the vehicle.](image1)

![Figure 17. Velocity vectors with wings at 150 km/h showing the velocity vector at the middle vertical plane of the vehicle.](image2)

The interesting phenomenon occurs when the velocity vectors are evaluated in the vehicle without the wings’ installation. Figures 18 and 19 depict the velocity vector in the middle section of the vehicle. It can be evaluated that even without the installation of the wings, the wake regions still greatly in size occur in the rear vehicle. Up to now, it becomes more obvious that the engine cover is the main problem of the occurrence of dead-water, which affects the drag force. The magnitude of velocity vectors determines how streamline in the objects that are observed. The more inclination of the vector, with respect to the direction of the coming flow, the more the friction occurs, affecting the force in the opposite direction of vehicle movement. It, unfortunately, causes the increment value of the drag force.

In both conditions (i.e. with and without wings), as the results of the velocity vector, the shapes of the vehicle that need to be criticized are the front part of the vehicle (forebody), engine cover, and the square back-type of the vehicle. As we can observe, the forebody, which is relatively horizontally shaped, acts as the stagnation point, which makes the velocity vector diverse randomly even in the vertical direction. The engine cover gives a significant contribution to the dead-water behind the
vehicle. Furthermore, the square-back-shaped body of the vehicle also contributes to the wake region produced by the vehicle. This phenomenon primarily due to the detached air flow after the engine cover of the vehicle. That such a big dimension of engine cover undoubtedly gives the prominent cause of this air vortex. Hence, the low-pressure zone is created in this region, which can result in increased drag and instability. The slicing angle and flow separation in the underbody determine the major portion of aerodynamic drag [17]. Concerning this, the shape of the underbody of the vehicle obviously affect the aerodynamic performance as well.

Figure 18. Velocity vectors of the vehicle without wings at 30 km/h.

5. Result Validation Assessment
The results validation assessment adopts the method explained by Slater [18], which consists of examining the iterative convergence, examining the consistency, examining the spatial convergence, assessing the temporal convergence, comparing CFD results to experimental data and examining the model uncertainties. The examination of iterative convergence to the temporal convergence has been performed in every case of the simulations in the software. The utilization of the chosen software has been proven to provide the consistency of the CFD solution [19], [20], [21]. The comparison of CFD results to the experimental data requires to be performed to validate these results. Several previous studies have been conducted to compare the numerical simulation results and the experimental data of similar method and works [5], [6]. The experimental test of the real component has also been made by several previous researchers, even in more detail components and shapes [22], [23], [24]. Even if some insignificant error might occur in the results of experimental data due to the complexity of the related experiments, the entire mentioned previous works resulted that the computational fluids method is agree with the real case condition in the real world. That results demonstrate the accuracy of the results of the numerical simulation conducted in this work.
6. Conclusion
The study can be concluded that installing the front and rear wing contributes to the decrement of the drag coefficient of 0.07 points and keeping the drag force remains constant. This constant value of drag force is mainly due to the values projection area of the vehicle with wings and without wings are not equal. However, the benefits of having wings installed greatly increase the downforce by 334 N, which is 5.51 times higher than such without wings in the speed of 150 km/h. In terms of pressure distribution, a fairly big value takes place in the front part of the tire, and the stagnation point in the forebody of the vehicle occurs. In the rear part of the vehicle, the engine cover (i.e., also in the roll bar part) gives a big obstacle of airflow, making the flow detached and producing a huge amount of vortices in the rear part of the vehicle. It reduces the functionality of the rear wing since the engine cover hinders the airflow through the rear wing. As such, the presence of the rear wing does not have a significant effect in terms of reducing the wake region. The future work is to conduct a comparative study between computational method and real scale wind tunnel measurement. Moreover, it is also planned to perform the optimization of the shape and other properties of aerodynamic devices of the vehicle.

References
[1]  S. Kruepke and S. Kumpaty, “External air flow over a pickup truck: Cfd modeling and investigation,” in ASME International Mechanical Engineering Congress and Exposition, Proceedings (IMECE), 2016, doi: 10.1115/IMECE201665118.
[2]  J. P. De Kock, R. F. Laubscher, S. Kruger, and N. J. Van Rensburg, “Numerical and experimental aerodynamic evaluation of a solar vehicle,” in ASME International Mechanical Engineering Congress and Exposition, Proceedings (IMECE), 2017, doi: 10.1115/IMECE2017-71297.
[3]  S. P. Veluri, C. J. Roy, A. Ahmed, R. Rifki, J. C. Worley, and B. Recktenwald, “Joint computational/experimental aerodynamic study of a simplified tractor/trailer geometry,” J. Fluids Eng. Trans. ASME, 2009, doi: 10.1115/1.3155995.
[4]  A. Yudianto, S. Ghafari, P. Huet, and M. Wakid, “Evaluation of the Temperature Distribution and Structural Deformation of the Car Dashboard Subjected to Direct Sunlight,” in Journal of Physics: Conference Series, 2019, vol. 1273, no. 1, doi: 10.1088/1742-6596/1273/1/012076.
[5]  A. Armagan, Y. Onur, and U. Habib, “Aerodynamic Analysis of a Passenger Car at Yaw Angle and Two-Vehicle Platoon,” J. Fluids Eng. Trans. ASME, 2015, doi: 10.1115/1.4030869.
[6]  B. Khalighi, K. H. Chen, and G. Iaccarino, “Unsteady aerodynamic flow investigation around a simplified square-back road vehicle with drag reduction devices,” J. Fluids Eng. Trans. ASME, 2012, doi: 10.1115/1.4006643.
[7]  F. Diba, A. Barari, and E. Esmailzadeh, “Handling and safety enhancement of race cars using active aerodynamic systems,” Veh. Syst. Dyn., 2014, doi: 10.1080/00423114.2014.930158.
[8]  D. J. S. Aulakh, “Effect of underbody diffuser on the aerodynamic drag of vehicles in convoy,” Cogent Eng., 2016, doi: 10.1080/23311916.2016.1230310.
[9]  K. Joseph, Automotive Aerodynamics. San Diego State University: Wiley, 2016.
[10]  R. E. M. Nasir et al., “Aerodynamics of ARTeC’s PEC 2011 EMo-C car,” in Procedia Engineering, 2012, doi: 10.1016/j.asepro.2012.07.382.
[11]  R. C. Das and M. Riyad, “CFD analysis of passenger vehicle at various angle of rear end spoiler,” in Procedia Engineering, 2017, doi: 10.1016/j.asepro.2017.08.130.
[12]  F. Mariani, C. Poggiani, F. Risi, and L. Scappaticci, “Formula-SAE racing car: Experimental and numerical analysis of the external aerodynamics,” in Energy Procedia, 2015, doi: 10.1016/j.egypro.2015.12.111.
[13]  K. Horrigan, B. Duncan, A. Keating, A. Gupta, and J. Gargoloff, “Aerodynamic simulations of a generic tractor-trailer: Validation and analysis of unsteady aerodynamics,” in SAE Technical Papers, 2008, doi: 10.4271/2008-01-2612.
[14]  C. Chilbule, A. Upadhyay, and Y. Mukkamala, “Analyzing the profile modification of truck-
trailer to prune the aerodynamic drag and its repercussion on fuel consumption,” in *Procedia Engineering*, 2014, doi: 10.1016/j.proeng.2014.12.399.

[15] X. Zhang, W. Toet, and J. Zerihan, “Ground effect aerodynamics of race cars,” *Applied Mechanics Reviews*, 2006, doi: 10.1115/1.2110263.

[16] W.-H. Hucho, *Aerodynamics of Road Vehicles: From Fluid Mechanics to Vehicle Engineering*, 1987.

[17] S. M. Rakibul Hassan, T. Islam, M. Ali, and M. Q. Islam, “Numerical study on aerodynamic drag reduction of racing cars,” in *Procedia Engineering*, 2014, doi: 10.1016/j.proeng.2014.11.854.

[18] J. W. Slater, “NPARC Alliance CFD Verification and Validation Web Site,” 2008. https://www.grc.nasa.gov/WWW/wind/valid/tutorial/valassess.html (accessed May 30, 2020).

[19] J. M. Ortega and K. Salari, “Investigation of a trailer underbody fairing for heavy vehicle aerodynamic drag reduction,” in *SAE Technical Papers*, 2008, doi: 10.4271/2008-01-2601.

[20] T. Driant, L. Remaki, H. Fellouah, S. Moreau, and A. Desrochers, “Aerodynamic study of a tricycle wheel subsystem for drag reduction,” *J. Fluids Eng. Trans. ASME*, 2014, doi: 10.1115/1.4025644.

[21] T. Driant, H. Fellouah, S. Moreau, A. Desrochers, and L. Remaki, “Numerical simulation and wind tunnel measurements on a tricycle wheel sub-system,” *Int. J. Eng. Syst. Model. Simul.*, 2013, doi: 10.1504/IJESMS.2013.052376.

[22] E. Issakhanian, C. J. Elkins, K. P. Lo, and J. K. Eaton, “An experimental study of the flow around a Formula one racing car tire,” *J. Fluids Eng. Trans. ASME*, 2010, doi: 10.1115/1.4001880.

[23] B. Conan, J. Anthoine, and P. Planquart, “Experimental aerodynamic study of a car-type bluff body,” *Exp. Fluids*, 2011, doi: 10.1007/s00348-010-0992-z.

[24] P. Aschwanden, J. Müller, and U. Knörrnschild, “Experimental study on the influence of model motion on the aerodynamic performance of a race car,” in *SAE Technical Papers*, 2006, doi: 10.4271/2006-01-0803.

**Acknowledgment**

The authors like to express our gratitude and appreciation to SMK Muhammadiyah Imogiri for the support of the project funding and facilities, to all the team members of this project, to Politecnico di Torino for complete support, and to Universitas Negeri Yogyakarta as the current affiliation.