Journal of Advanced Research in Applied Sciences and Engineering Technology

Journal homepage:
https://semarakilmu.com.my/journals/index.php/applied_sciences_eng_tech/index
ISSN: 2462-1943

Flow Structure Characteristics of the Simplified Compact Car Exposed to Crosswind Effects using CFD

Muhammad Nabil Farhan Kamal[1,*], Izuan Amin Ishak[2], Nofrizalidris Darlis[1], Nurshafinaz Mohd Maruai[3], Rahim Jamian[1], Razlin Abd Rashid[1], NorAfzanizam Samiran[1], Nik Normunira Mat Hassan[1]

[1] Department of Mechanical Engineering Technology, Faculty of Engineering Technology, University Tun Hussein Onn Malaysia, Edu Hub Pagoh, 84600, Muar, Johor, Malaysia
[2] Sustainable Engineering Technology Research Centre, University Tun Hussein Onn Malaysia, Edu Hub Pagoh, 84600, Muar, Malaysia and Centre for Language Studies, University Tun Hussein Onn Malaysia, Batu Pahat, Malaysia
[3] Malaysia Japan International Institute of Technology, Universiti Teknologi Malaysia, Kuala Lumpur, Malaysia

ABSTRACT

Aerodynamic characteristics of a car are important in reducing car accidents caused by wind loading and in lowering fuel economy, continuing to be a major topic of interest. The restrictions of wind tunnel tests and the rising trend of numerical methods have been complied with by past researchers to investigate vehicle aerodynamics computationally. This research aims to analyze comprehensively the effect of crosswinds on a moving vehicle in terms of aerodynamic loadings and flow structures using commercial fluid dynamic software ANSYS FLUENT. This paper will focus on the CFD-based simplified compact car body developed in CATIA V5 by neglecting the external parts such as side mirrors and underbody. The implementation of Standard $k-\varepsilon$ model with the inlet velocity, v is setup to 30.56 m/s and Reynolds number equal to 8.89×10^5 by using numerical analysis for this research. The generic compact car which represents car geometries are expected to influence the aerodynamic characteristics whereas the crosswind angles are increased, it will cause high values for the coefficient of side forces and rolling moments in terms of aerodynamic loads due to the existence of the vortices at the leeward region. At $\Psi = 0°$, the coefficient of side force (C_s) is close to zero and this is predicted because the flow to the body is aligned to the inlet velocity, v.

Keywords:
Aerodynamic forces; Crosswind; Flow structures; CFD; Simplified car; Aerodynamic loads; Safety

Received: 10 June 2022 Revised: 5 September 2022 Accepted: 7 September 2022 Published: 19 Sept. 2022

1. Introduction

The majority of car research focuses on improving aerodynamics and safety for environmental adaptability. The characteristic of a vehicle is determined by cost structure, which differs depending on the size, segment, and other characteristics of the vehicle as MPVs typically have higher operating costs than compact cars. The main factors that affect fuel efficiency are car speed and the aerodynamic design concept [1]. Since years ago, there has been a great interest in the aerodynamics

* *Corresponding author.*
E-mail address: nabilfarhan1910@gmail.com

https://doi.org/10.37934/araset.28.1.5666

of car vehicles. A crosswind is normally considered as the wind that is acting perpendicular to the line and direction of travel [2]. In the area of automotive transportation, there has been a significant increase in community awareness of the safety factors especially the crosswind stability of the vehicle. The crosswind can also cause side slips and rollovers due to the lift force generated [3]. This could be destructive because it could cause the car to lose traction or turn around [4].

Crosswind force on vehicles can be separated into two vector components. The element of headwind or tailwind in the direction of travel and the crosswind component is perpendicular to the former for those elements. Aerodynamic forces and moments that are affected by the crosswind such as wind direction, yaw angles, drag forces, lift forces, side forces, yawing moments, rolling moments, and pitching moments are shown in Figure 1 [5][6]. The crosswind aspects can also affect the safety ride comfort of the vehicle and stability of the car, which can be calculated by using a simple static analysis based on Batista *et al*. [7].

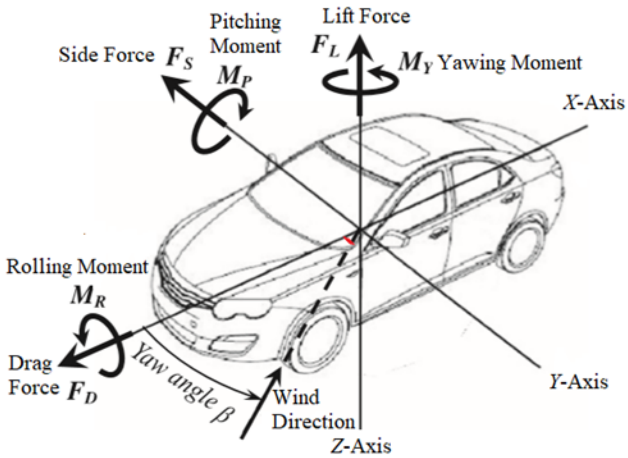


Fig. 1. The aerodynamic forces and direction of the moments are attached to the sedan car [6]

Many researchers have recently investigated the study of flow over a vehicle due to its importance in transportation based on Wang *et al.*, Chaware *et al.*, and others [8]–[12]. In general, further information is required on how the detailed flow structure of the movement and behavior of different car segments influence by the crosswind, which impacts the aerodynamic properties of the vehicle. The different sizes and car segments clearly can enhance the aerodynamic loads of the vehicle. This outcome will increase the facing surface of the car that is subjected to crosswinds. The vehicles that travel faster and feature a variety of designs are created and it appears that the study has raised awareness of crosswind sensitivity as early as the 1950s [13].

The previous findings have explained that the difference in vehicle model geometry is important and varieties of vehicle designs have been used in experimental and numerical research [14]–[17]. Based on Samy *et al.*, the study on the aerodynamic analysis without crosswind involving a Sport Utility Vehicle (SUV) focusing on the drag and lift coefficient [17]. Besides, the research from Firdaus et al., mentioned the important of vehicle aerodynamic analysis of a Multi-Purpose Vehicle influenced the stability of the car during manocurve [16]. The different design of car geometry will be affecting the aerodynamic loads due to the Reynolds number. This paper is more focused on the compact car or known as A-Segment because there is no specific research based on the compact car that is influenced by the crosswind. This paper may assist to recommend all Automotive companies to integrate crosswind stabilization safety features, especially in countries that are affected by higher crosswinds.

The research aims to investigate the flow characteristics of the generic compact car under the influenced of crosswind angles. In this study, the Reynolds numbers of 8.89×10^5 is used based on the dimension of the car and inlet velocity, *v* of 30.56 m/s. This research also inspects the pressure surface contours, Turbulent Kinetic Energy (TKE) of the compact car under influence of crosswind. There are a few sub sections such as the car model, numerical approach, computational domain, boundary conditions, meshes, validation, compares and analyses flow structures including the aerodynamic loads for the current investigation.

2. Methodology

2.1 Car Model

The type of simplified car model used in this paper study is a compact car with specifications between 3500 to 4200 mm in length. The compact car model was imported and modified in CATIA by neglecting the tires, and side mirror and exported in ANSYS Design Modeller as shown in Figure 2.

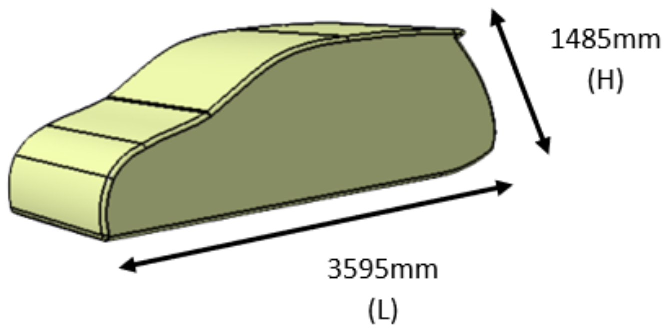


Fig. 2. The compact car model geometry uses to run the numerical simulation

The purpose to neglect some of the external parts is due to the limitations in the simulation technique [18], [19] and to avoid the effect of rotational motion and tires wake [20]. Based on the study by Han *et al.* [21], the simplified Aerodynamic Studien Model (ASMO model) is applied due to the actual model of the car body generally having sophisticated surfaces and part specifics. The complex model will require extensive computation and processing time. Therefore, the implementation of generic and simplified vehicle models has become the choice of many researchers as stated earlier.

2.2 Boundary Condition

The computational domain is defined as the area surrounding the 3D car model that defines the flow simulation in the restricted area. This defines the area within which the flow simulation will take place. The effective computational domain is critical for ensuring that the domain is at its optimal size before beginning the meshing process. The following Figure 3 shows the dimension of the computational domain from an isometric view in the simulation case.

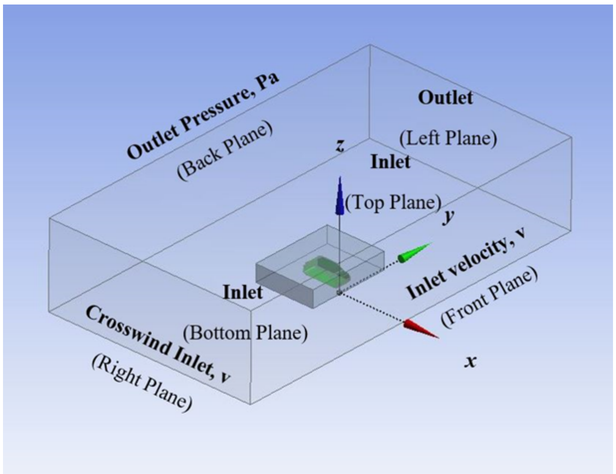


Fig. 3. Overview the computational domain from an isometric view

The computational domain is chosen to extend two vehicle lengths (2*L*) in front of the vehicle and three lengths (3*L*) behind the rear of the vehicle on the *x*-direction. The distance behind the car should be sufficient to allow fluid flow without interfering with the flow of air in the outlet section [20]. The distance of the vehicle from the side is selected to enlarge five vehicle lengths (5*L*) for the both left and right of the car on the *y*-direction. Based on Mansor *et al.* [22], it is critical to ensure that the flow is completely formed and capture the flow structures for the crosswind situation. The Standard $k - \varepsilon$ model is applied with the inlet velocity, *v* is setup to 30.56 m/s due to the Malaysian Institute of Road Safety Research (MIROS) [13] that affected by the topology of Asia's land. The Reynolds number equal to 8.89×10^5 according to the dimension of the compact car as shown in Figure 2 above. The overall computational domain dimensions employed in the study are shown in Table 1 below. The boundary conditions for all of the modifications are created and the conditions are specified in Table 2.

Table 1
Dimension of the Computational domain

Axis-coordinates	Configurations (mm)
$+x$	2*L*
$-x$	3*L*
$+y$	5*L*
$-y$	5*L*
$+z$	2*L*
$-z$	0.172*L*

Table 2
Domain setup and boundary condition

Domain setup and boundary condition	Configuration
Inlet velocity, *v*	30.56 m/s
Outlet Pressure	0 Pascal
Car body wall	No-slip (v_w=0)
Fluid properties	Air
Density	1.225 Kgm^{-3}
Turbulence model	Standard $k - \varepsilon$ model

2.3 Computational Validation Study and Meshing

The car model, enclosure, and boundary conditions are described in great detail in the validation case study. The validation is established by using the Ahmed body model as a vehicle model geometry with the same values of inlet velocity, the number of iterations, and the same turbulence models but various computational domains from previous studies [23], [24]. The time-averaged drag force coefficient, C_d from the numerical analysis is differentiated from the previous researchers by Salahuddin *et. al* [25] and Arnold *et. al* [26]. Tables 3 and 4 illustrate the specific comparison with previous results by applying the various computational domains and the percentage errors of the aerodynamic loads.

Table 3
Comparison with existing results

Cases Study	Turbulence Models	Number of Iterations	Number of Grids	Velocity Inlet, *v* (m/s)	Coefficient of Drag Force (C_d)
Numerical Analysis by Salahuddin *et al.* [25]	$k-\varepsilon$ model	1,000	Approximately 583,000	40	0.320
Wind Tunnel by Salahuddin *et al.* [25]	$k-\varepsilon$ model	-	-	40	0.295
Current Study without Inflation	$k-\varepsilon$ model	200	Approximately 2,000,000	40	0.333
Current Study with Inflation	$k-\varepsilon$ model	200	Approximately 3,000,000	40	0.331

Table 4
The percentage errors of the aerodynamic loads with existing studies

Cases Study	Percentages Error for the Coefficient of Drag Force (C_d) compared with Numerical Analysis by Salahuddin *et al.* [25]
Wind Tunnel by Salahuddin *et al.* [25]	10.9 %
Current Study without Inflation	3.9 %
Current Study with Inflation	3.3 %

The current research produced the results nearest to the CFD simulation by Salahuddin *et al.* [25] with only a 3.3 % difference in percentage errors by applying inflation for drag force coefficient based on table 4 above. The inflation mesh is designed by "exaggerating" a triangular surface mesh to create the good quality geometry-aligned elements that capable of resolving boundary layer growth. When analysing boundary conditions accurately, a finer mesh is created close to the wall or the boundary, which is when inflation is applied. The specific of the meshing resolutions with the inflation layers for the compact car are shown in Figure 4. According to the study by M. Lanfrit [27], the maximum percentage of errors that the external flow involved car will tolerate is about 5%. The study by Salahuddin *et al.* [25] produces the least amount of grids, while the current study produces the most grids. The effectiveness of the results for aerodynamic loads and flow structures can be impacted by the number of grids, which is a critical factor.

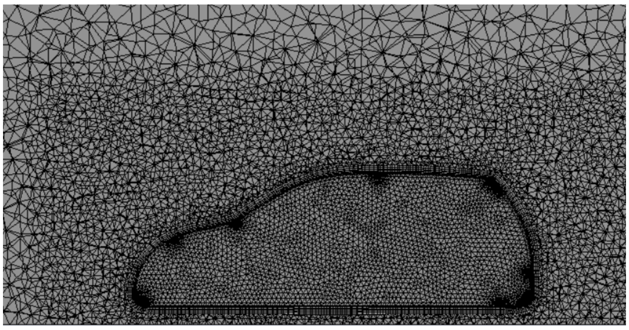


Fig. 4. Side view of compact car with inflation at the middle plane cross-section

3. Results

3.1 Aerodynamic Loads

Figure 5 illustrate the major differences in the value of aerodynamic loads when all cases are simulated to represent the compact car. This is due to the flow field characteristics differing for the no crosswind and crosswind cases. As can be seen, the side force coefficients, C_s, and rolling moment coefficients, C_{Rm} increases in value as the crosswind conditions increase until at Ψ = 60°. The pressure difference between the leeward and windward areas causes this imbalance flow occurs due to the existence of vortices.

The side force coefficients, C_s, and rolling moment coefficients, C_{Rm} are one of the frequent reasons for cars to roll over and sideslip due to the imbalance of pressure on the sides of the vehicle as shown in surface pressure contours in Figures 6 below. The drag coefficient indicates a rising until Ψ = 10° before it approaches zero as the angle increases [28]. The surface area extended by the flow decreases as the yaw angles increase, which causes the viscous friction of the ground. The lift force coefficient, C_l illustrate the steady decrease to the maximum value and rising at Ψ = 60°. The pressure reduction is assumed to generate over the compact car due to the formation of vortices.

3.2 Surface Pressure Contour on Car Body

The difference in the flow field profile develops pressure to emerge. Figures 6 shows surface pressure contour on the windward and leeward areas of the vehicle at different crosswind angles. As can be seen, the flow separation at the front edge of the car moves in the direction of the top and bottom of the vehicle model. As the flow direction increases, the surface pressure is starting to change in its characteristics.

At Ψ = 40° and above condition, the low-pressure region on the windward surface zone appears to vanish and move to the leeward surface zone. The pressure area on the windward surface enhances due to the presence of crosswind situations. In general, the pressure acting on the windward surface slightly emerges as the yaw angles increase. It demonstrates that there are larger vortices sizes generated which increases the aerodynamic loads.

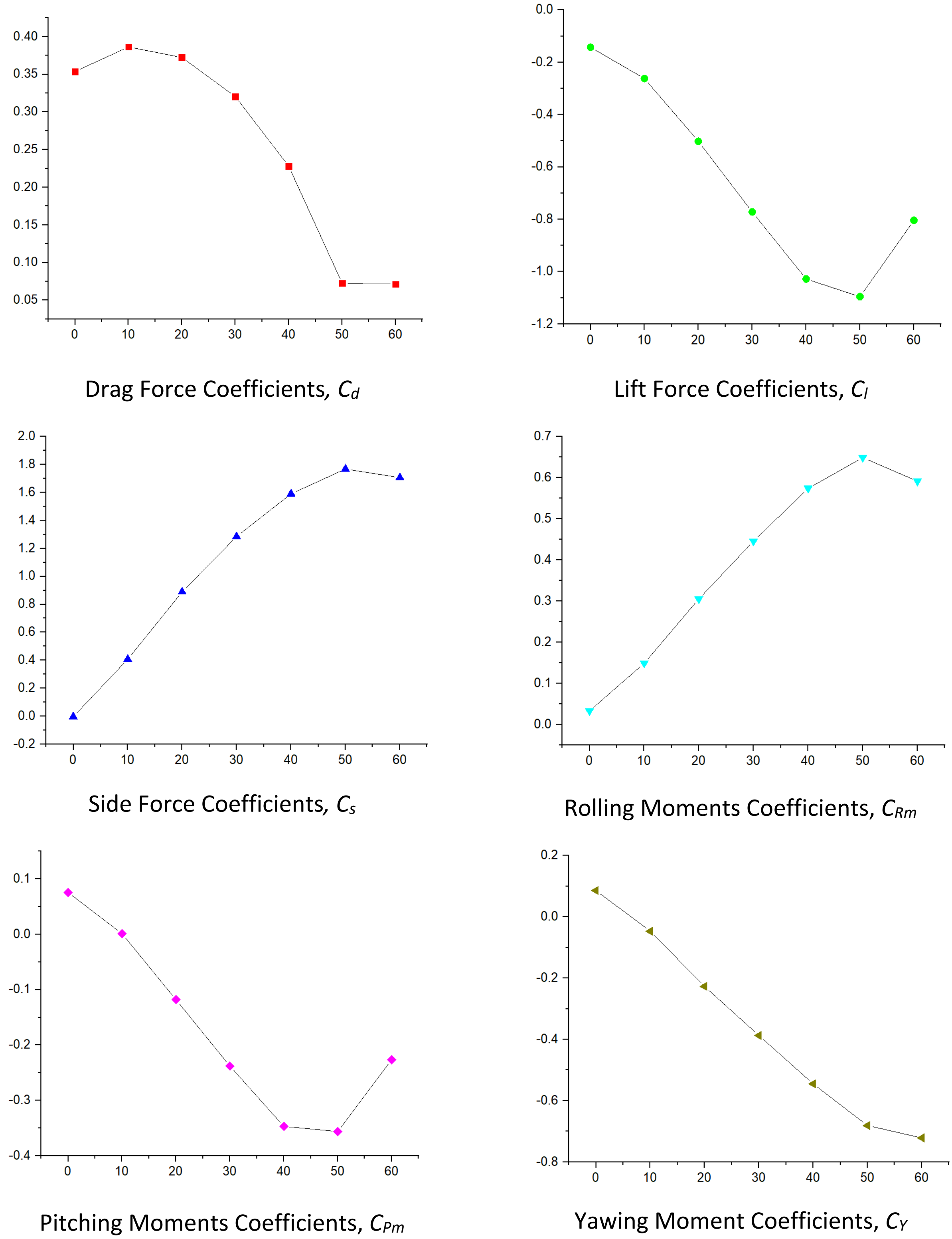


Fig. 5. The aerodynamic loads for forces and moements coefficient compact car with different cases

Windward ***Leeward***

At Ψ = 0°,

At Ψ = 10°,

At Ψ = 20°,

At Ψ = 30°,

At Ψ = 40°,

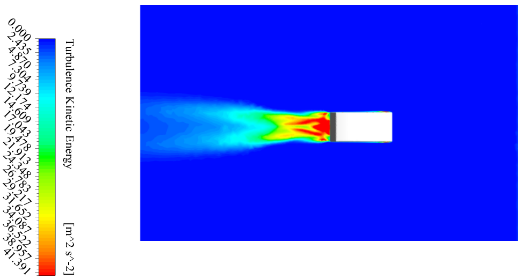
At Ψ = 50°,

At Ψ = 60°,

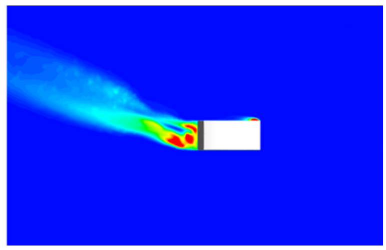
Fig. 6. The surface pressure contour of the compact car with various crosswind angles

3.3 Turbulence Kinetic Energy (TKE)

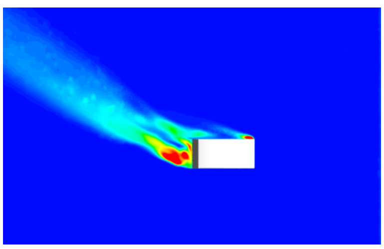
Turbulence Kinetic Energy (TKE) is the energy content of vortices in turbulent flows that denotes magnitude and numerical value. As in Figure 7, the large region amount of turbulence can be identified at the rear and leeward area of the car due to the flow separation. At *Ψ* = 0°, the turbulent kinetic energy (TKE) can be illustrated clearly to rise its highest magnitude at the rear of car segment. The red colours represented as the maximum turbulent kinetic energy (TKE) while the blue colours indicated the lowest values for turbulent kinetic energy (TKE) existence.



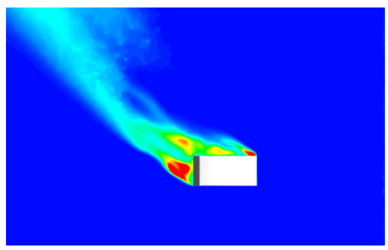
Ψ = 0°



Ψ = 20°



Ψ = 40°



Ψ = 60°

Fig. 7. Comparison of the turbulence kinetic energy for no crosswind and crosswind cases

For crosswind condition from Ψ = 10° to Ψ = 60° as shown in Figure 7, the turbulent kinetic energy (TKE) can be identified clearly moving to the leeward side of the car follow the crosswind magnitude. It can also be observed that there are significant changes of turbulent kinetic energy (TKE) structures and the shape on the leeward region seems to enlarge longer due to the high larger separation. The more kinetic energy is dissipated because there are massive vortices at the back and sides of the compact car. This will influence the stability of vehicle dynamics, exposing it to overturn and sideslip when travelling in a forward direction.

4. Conclusions

In the present work, the aerodynamic forces acting on a simplified compact car exposed to crosswind effects were studied using Computational Fluid Dynamics with performed for a steady viscous fluid flow using the Standard k-ε turbulence model. The computational validation study is made for the size of the enclosure and the percentage of errors for the external flow required car only 3.3 %. The effect of the crosswind angles was determined in terms of aerodynamic loads and flow structures. Observation can be concluded that the side force coefficients, C_s and rolling moments, C_{Rm} increase in values as the crosswind case until at Ψ = 60°. It can be explained that the imbalance flow occurs due to the pressure difference between the leeward and windward zones to the existence of vortices experienced by the car. The results showed that the side force coefficients and rolling moments are significant due to one of the factors for cars to roll over and the sideslip. At crosswind conditions, the low-pressure areas on the windward surface zone start to disappear and move to the leeward surface zone due to the existence of the larger vortices sizes. The results of the present simulation concerning the trends for aerodynamic loads and flow structures were found to be in close agreement with transportation studies results such as lorry and van by the past researchers.

Acknowledgement
This research was financially supported by the Malaysia Ministry of Higher Education (MOHE) through Fundamental Research Grant Scheme (FRGS/1/2020/TK02/UTHM/03/4).

References
[1] Ansari, Abdul Razzaque. "CFD Analysis of Aerodynamic Design of Tata Indica Car." *International Journal of Mechanical Engineering and Technology* 8, no. 3 (2017): 344-355.
[2] Thomas, Dirk. "Lateral stability of high-speed trains at unsteady crosswind." PhD diss., KTH Farkost & Flyg, 2009.
[3] Arafat, Mohammad, and Izuan Amin Ishak. "CFD Analysis of the Flow around Simplified Next-Generation Train Subjected to Crosswinds at Low Yaw Angles." *CFD Letters* 14, no. 3 (2022): 129-139. https://doi.org/10.37934/cfdl.14.3.129139
[4] Pauwelussen, Joop P., Wouter Dalhuijsen, and Menno Merts. "Tyre dynamics, tyre as a vehicle component Part 1: Tyre Handling Performance." *Virtual Education in Rubber Technology, HAN University, Netherlands* (2007).
[5] Ishak, Izuan Amin, Nurshafinaz Maruai, Fadhilah Mohd Sakri, Rahmah Mahmudin, Nor Afzanizam Samiran, Syabillah Sulaiman, Shaiful Fadzil Zainal Abidin, and Nik Normunira Mat Hassan. "Numerical Analysis on the Crosswind Influence Around a Generic Train Moving on Different Bridge Configurations." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 89, no. 2 (2022): 76-98. https://doi.org/10.37934/arfmts.89.2.7698
[6] Zhu, L. D., L. Li, You-Lin Xu, and Q. Zhu. "Wind tunnel investigations of aerodynamic coefficients of road vehicles on bridge deck." *Journal of fluids and structures* 30 (2012): 35-50. https://doi.org/10.1016/j.jfluidstructs.2011.09.002
[7] Batista, Milan, and Marko Perkovič. "A simple static analysis of moving road vehicle under crosswind." *Journal of Wind Engineering and Industrial Aerodynamics* 128 (2014): 105-113. https://doi.org/10.1016/j.jweia.2014.02.009

[8] Wang, Bin, You-Lin Xu, Le-Dong Zhu, and Yong-Le Li. "Crosswind effect studies on road vehicle passing by bridge tower using computational fluid dynamics." *Engineering Applications of Computational Fluid Mechanics* 8, no. 3 (2014): 330-344. https://doi.org/10.1080/19942060.2014.11015519
[9] S. D. Chaware, Y. P. Pawar, M. Tech Automotive, and M. Tech, "Study of Cross Wind for Heavy Vehicles and Analysis over Ahmad body," *Int. J. Eng. Dev. Res.*, 5 (2017): 2321–9939.
[10] Huang, Taiming, Zhengqi Gu, Chengjie Feng, and Wei Zeng. "Transient aerodynamics simulations of a road vehicle in the crosswind condition coupled with the vehicle's motion." *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of automobile engineering* 232, no. 5 (2018): 583-598. https://doi.org/10.1177/0954407017704609
[11] Zhang, Qianwen, Chuqi Su, and Yiping Wang. "Numerical investigation on aerodynamic performance and stability of a sedan under wind–bridge–tunnel road condition." *Alexandria Engineering Journal* 59, no. 5 (2020): 3963-3980. https://doi.org/10.1016/j.aej.2020.07.004
[12] Briskey, Kyle Gary. "Improving caravan design by modelling of airflow." (2013).
[13] Zulhaidi, M. J., MI Mohd Hafzi, S. Rohayu, S. Wong, and MS Ahmad Farhan. "Weather as a Road Safety Hazard in Malaysia–An Overview." *MRev* 3 (2010): 2009.
[14] Vančura, Jan, and Jan Slavík. "16 Crosswind Sensitivity of Road Vehicles."
[15] S. Srinivasarao and V. M. Lakshamaih, "CFD Research on Car Body," *Int. J. Recent Technol. Eng.*, 8 (2019): 1178–1180. https://doi.org/10.35940/ijrte.B1218.0782S319
[16] Azmi, Mohammad Firdaus Mohammed, Mohammad Al Bukhari Marzuki, and Mohd Arzo Abu Bakar. "Vehicle aerodynamics analysis of a multi-purpose vehicle using CFD." *ARPN Journal of Engineering and Applied Sciences* 12, no. 7 (2017): 2345-2350.
[17] Samy, G. S., S. Thirumalai Kumaran, M. Uthayakumar, M. Sivasubramanian, and Krishna Bhagavathi Sankar. "Numerical analysis of drag and lift coefficient of a Sport Utility Vehicle (SUV)." In *Journal of Physics: Conference Series*, vol. 1276, no. 1, p. 012013. IOP Publishing, 2019. https://doi.org/10.1088/1742-6596/1276/1/012013
[18] Vdovin, Alexey. *Investigation of aerodynamic resistance of rotating wheels on passenger cars*. Chalmers Tekniska Hogskola (Sweden), 2013.
[19] Kamal, Muhammad Nabil Farhan, Izuan Amin Ishak, Nofrizalidris Darlis, Daniel Syafiq Baharol Maji, Safra Liyana Sukiman, Razlin Abd Rashid, and Muhamad Asri Azizul. "A Review of Aerodynamics Influence on Various Car Model Geometry through CFD Techniques." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 88, no. 1 (2021): 109-125. https://doi.org/10.37934/arfmts.88.1.109125
[20] Saleh, Z. M., and A. H. Ali. "Numerical Investigation of Drag Reduction Techniques in a Car Model." In *IOP Conference Series: Materials Science and Engineering*, vol. 671, no. 1, p. 012160. IOP Publishing, 2020. https://doi.org/10.1088/1757-899X/671/1/012160
[21] Han, Shan Ling, Zhi Yong Li, Jin Bin Li, and Ru Xing Yu. "Simulation Research on Aerodynamic Characteristics of Vehicle under Steady Crosswind Based on XFlow." In *Applied Mechanics and Materials*, vol. 494, pp. 138-141. Trans Tech Publications Ltd, 2014. https://doi.org/10.4028/www.scientific.net/AMM.494-495.138
[22] Mansor, S., N. A. R. Nik Mohd, and C. W. Chung. "Validation of CFD Modeling and Simulation of a Simplified Automotive Model." In *Applied Mechanics and Materials*, vol. 735, pp. 319-325. Trans Tech Publications Ltd, 2015. https://doi.org/10.4028/www.scientific.net/AMM.735.319
[23] T. Hughes, "CFD Study of Flow over a Simplified Car Using Different Turbulence Models," *Univ. Derby*, vol., no. June, pp. 0–47, 2018.
[24] Strachan, R., R. Knowles, and N. Lawson. "Comparisons between CFD and experimental results for a simplified car model in wall proximity." *Proceedings of the Integrating CFD and Experiments in Aerodynamics* (2005): 5-6.
[25] Khan, Rehan Salahuddin, and Sudhakar Umale. "CFD aerodynamic analysis of Ahmed body." *International Journal of engineering trends and Technology* 18, no. 7 (2014): 301-308. https://doi.org/10.14445/22315381/IJETT-V18P262
[26] Y. Y. Arnold Mukuvare, "CFD Prediction of Separation Flow over a Simplified Car Model," The University of West of England Bristol, 2016.
[27] Lanfrit, Marco. "Best practice guidelines for handling Automotive External Aerodynamics with FLUENT." (2005).
[28] Ishak, Izuan Amin, Mohamed Sukri Mat Ali, Mohamad Fitri Mohd Yakub, and Sheikh Ahmad Zaki Shaikh Salim. "Effect of crosswinds on aerodynamic characteristics around a generic train model." *International Journal of Rail Transportation* 7, no. 1 (2019): 23-54. https://doi.org/10.1080/23248378.2018.1424573