INTERNATIONAL JOURNAL OF INTEGRATED ENGINEERING
ISSN: 2229-838X e-ISSN: 2600-7916

IJIE

Vol. 16 No. 6 (2024) 154-164
https://publisher.uthm.edu.my/ojs/index.php/ijie

Numerical Analysis on the Performance of Aerodynamic Devices on a Car Using CFD

Lingamoorthy s/o Ramachandran[1], Izuan Amin Ishak[1]*, Mohammad Arafat[1], Nor Atiqah Zolpakar[2], Shaiful Fadzil Zainal Abidin[1], Dalila Mohd Harun[1]

[1] *Department of Mechanical Engineering Technology, Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, Bandar Universiti Pagoh, 84600, Muar, Johor, MALAYSIA*
[2] *Faculty of Mechanical & Automotive Engineering Technology, Universiti Malaysia Pahang, 26600 Pekan, Pahang, MALAYSIA*

*Corresponding Author: izuan@uthm.edu.my
DOI: https://doi.org/10.30880/ijie.2024.16.06.015

Article Info

Received: 26 August 2024
Accepted: 14 October 2024
Available online: 4 November 2024

Keywords

Aerodynamic, ANSYS fluent, diffuser, drag reduction, downforce, spoiler

Abstract

Car aerodynamics plays a crucial role in enhancing vehicle performance, efficiency, and stability. This study focuses on evaluating the impact of spoilers and diffusers on drag reduction and downforce generation using ANSYS Fluent CFD software. The k-ε turbulence model is employed for computational efficiency in a steady-state flow simulation for precise mapping of complex geometries. Aerodynamic performance is evaluated at speeds of 70 km/h, 110 km/h, and 150 km/h through aerodynamic forces and flow structure. Results indicate that the diffuser alone achieves the lowest drag coefficient (C_d) of 0.3248, while the spoiler alone yields the highest C_d at 0.4023. The combined use of both devices reduces C_d to 0.3412, a 5.59% improvement over the baseline. Additionally, the combination significantly lowers the lift coefficient (C_l) to -0.3324, enhancing handling and performance. This study concludes that the integrated use of spoilers and diffusers contributes to improved vehicle design for greater efficiency and sustainability.

1. Introduction

This research focuses on the role of aerodynamics in vehicle design, particularly its impact on fuel efficiency, stability, and overall performance. It addresses the challenges of reducing drag and lift, which grow exponentially with speed [1]. To mitigate these effects, devices like spoilers and diffusers are explored for their ability to optimize pressure distribution and minimize turbulence. The objectives include analyzing these devices' impact on drag and lift reduction, visualizing flow structures, and evaluating their performance at various speeds [2]. Using a generic sedan model, the study assesses the effects of these aerodynamic devices under different speed conditions. This research contributes to automotive design advancements by providing insights into enhancing vehicle efficiency, safety, and sustainability. It emphasizes the need for tailored aerodynamic solutions to improve fuel efficiency, stability, and overall performance at different speeds.

Aerodynamic drag significantly impacts the performance and efficiency of road vehicles. The drag experienced by road vehicles is primarily pressure drag, caused by vortex generation at the rear of the vehicle [3]. This pressure drag, which accounts for a significant portion of total drag, is highly dependent on vehicle geometry and can impact fuel consumption [4]. Aerodynamic drag reduction in vehicle development prioritizes fuel efficiency and environmental sustainability [5]. Various aerodynamic devices and techniques have been studied to reduce drag and improve vehicle performance. These include the application of different shape designs [6], flow control rods [7], vortex generators [4], and aerodynamic devices like rear diffusers [8]. Implementing these strategies can lead to drag reduction, enhancing fuel efficiency and overall vehicle aerodynamics. Moreover, by adding a duct at the van's front, effectiveness and stability improved, reducing the drag coefficient [9].

On the other hand, lift force generated on a vehicle due to differences in air pressure can enhance performance by improving stability, but it poses a significant concern for design engineers due to the potential loss of traction at high speeds, leading to severe consequences [10]. Kumar's study on lift force over a car profile reveals that higher-pressure air in front of the windscreen accelerates over it, creating reduced pressure that practically lifts the roof. Additionally, the notch between the back window and the trunk causes a pressure gap, resulting in lift [11]. To counteract lift force, high-performance vehicles, as discussed by Chopade, utilize a diffuser to create downforce, resisting lift by lowering pressure beneath the car and increasing pressure above it [12].

In a study by Kumar et al. [11], it was found that a spoiler mounted at the rear edge of the vehicle without leaving a gap between the spoiler and the vehicle's surface demonstrated superior aerodynamic performance, yielding a drag coefficient of 0.329 and a reduced lift force of 0.106. This outperformed a wing-style spoiler mounted 28m above the surface of the vehicle. Additionally, research by Wardani et al. [13] supports the effectiveness of spoilers in reducing both drag and lift coefficients on sedan-style cars. Table 2.2 illustrates installing a spoiler reduced the drag coefficient and lift coefficient when compared to not installing a spoiler. Diffusers are designed to generate downforce and reduce drag by accelerating airflow beneath a car [14]. Design elements impact the velocity distribution, downforce, and drag of the diffuser, with the diffuser area ratio playing a crucial role in pressure recovery. The Lotus 78 and 79 Formula One vehicles were pioneers in utilizing ground-effect diffusers close to the road surface to effectively generate downforce [15].

Table 1 *The results obtained by installing a spoiler in a sedan type car*

Spoiler model	Drag coefficient reduced (%)	Lift coefficient reduced (%)
Spoiler level 1	Reduced 7.91	Reduced 41.93
Spoiler level 2	Reduced 33.86	Reduced 61.36

2. Methodology

2.1 Geometry Modelling

Geometry modeling is a crucial step in aerodynamic load assessment using ANSYS Fluent for car analysis. The car geometry is constructed with minimal and critical elements in all areas, aligning it with the final production shape. This process involves importing model into ANSYS Fluent and using Solidworks for surface construction. The dimensions and details of the car, as well as the spoiler and diffuser, are outlined in Fig 1 and 2 [16], [17], [18]. The ANSYS Workbench software suite is employed for analyzing the airflow around the car's external shape.

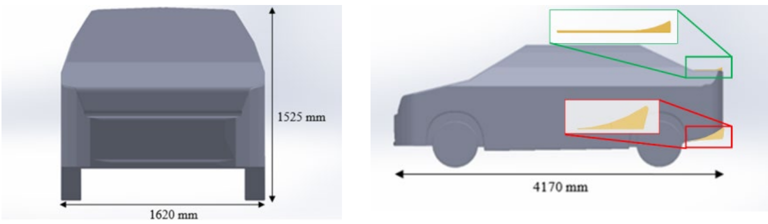
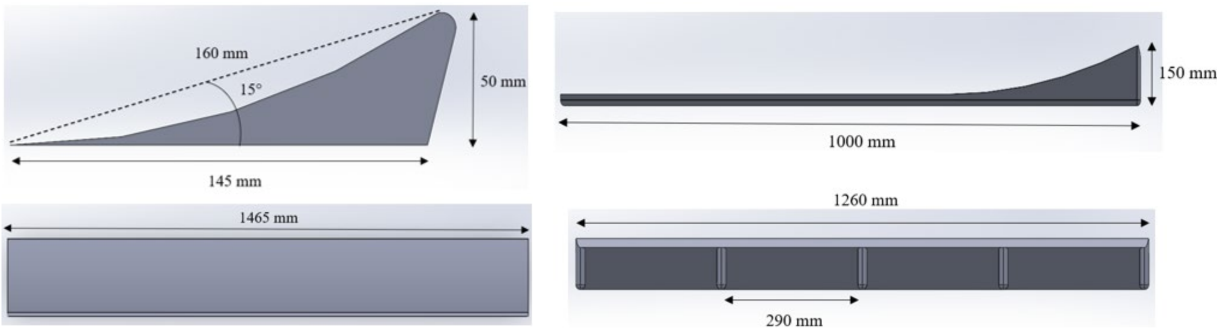


Fig. 1 *Simplified car model with diffuser (green) and spoiler (red)*

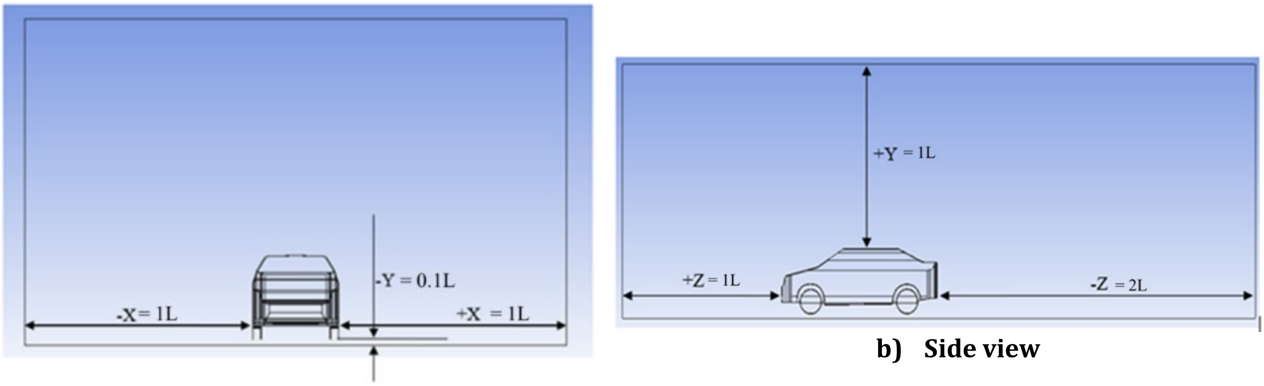


(a) Spoiler **(b) Diffuser**

Fig. 2 *Detail design of (a) Spoiler; and (b) Diffuser*

2.2 Boundary Conditions

To model the flow around the car, appropriate boundary conditions are essential. After positioning the model at its centroid, an enclosure representing the air around the vehicle is created for CFD simulation. Fig 3 visually depicts the enclosure in ANSYS software, highlighting its size dependency on the car model length. The data of enclosure, referencing the vehicle length, with dimensions similar to previous research on drag and lift force over a car's profile [11]. In ANSYS Fluent, boundary conditions, including inlet and outlet conditions and vehicle surface boundary conditions, play a important role in obtaining accurate results for flow simulations around the car [19]. Adjusting the inlet conditions allows for experimentation with the vehicle's speed during analysis, enhancing the precision of the results [15]. Table 3 shows the Data input in Boundary Condition.



b) Side view

a) Front view

Fig. 3 *The enclosure of the car model*

Table 3 *The data input in boundary condition*

Details	Boundary Conditions	Inputs
Inlet	Velocity Inlet	95 km/h (26 m/s)
Outlet	Pressure Outlet	0 Pa (default)
Density	Density	1.225 kg/m^3 (default)
Reference	Temperature	15.01 Celsius (default)
Area	Surface area	1.921654 m^2

2.3 Grid Independent Study

A grid independence study is a technique used to evaluate how mesh density affects computational results, aiming to determine the minimum number of grid points required for solution independence. This study typically involves testing with three different meshes. While two meshes often demonstrate independence, three or more runs may be necessary with coarser initial meshes. The process starts with a coarse grid and gradually refines it until simulation results converge or exhibit differences of less than 1%, ensuring accuracy. Table 4 outlines the parameters employed in the Grid Independence Test, while Fig 4 visually compares the drag coefficient (C_d) for various mesh resolutions: Coarse (Mesh 1), Medium (Mesh 2), and Fine (Mesh 3). Mesh 2, a medium mesh, is selected for this study as it exhibits less than 1% difference compared to the fine Mesh 3. In addition, Mesh 2 would be enough to predict aerodynamic loads considering accuracy and computational cost.

Table 4 *Parameters employed in the grid independence test*

Mesh Type	Coarse	Medium	Fine
	Mesh 1	Mesh 2	Mesh 3
Element size (mm)	1000	500	250
Face sizing (mm)	100	50	25
No of Nodes	1,575,026	3,406,048	6,130,968
No of Elements	299,183	607,476	1,066,693
Drag coefficient (C_d)	0.3680	0.3305	0.3307
Percentage error (%)	11.3469		0.0469

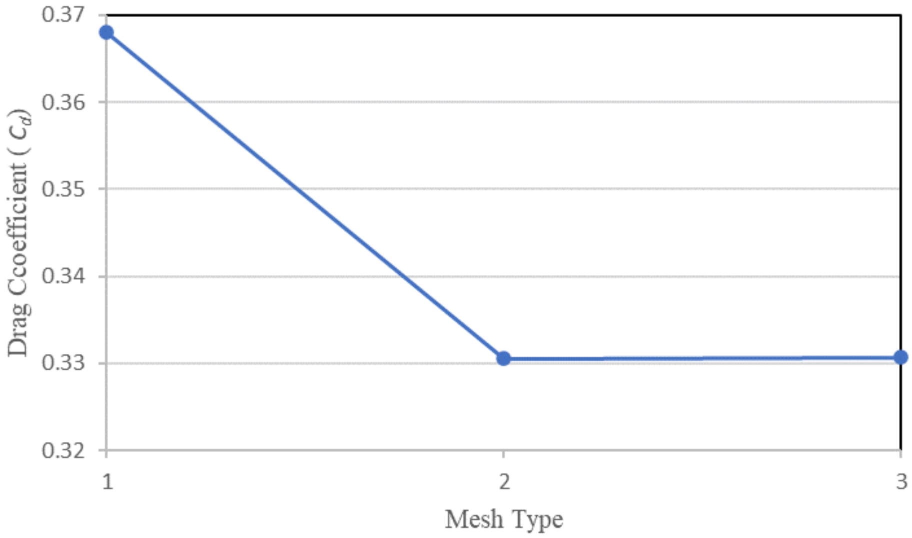


Fig. 4 *Comparison of C_d for different mesh resolutions: coarse (mesh 1), medium (mesh 2), fine (mesh 3)*

3. Results

This section outlines the anticipated outcomes of the project, emphasizing the validation process and subsequent simulations based on parameters established earlier. The primary focus is on presenting and discussing the results of Computational Fluid Dynamics (CFD) analyses that assess the aerodynamic impact of both the spoiler and diffuser on a car. Notably, the discussion integrates a synthesis of results obtained from three different speed conditions for four difference cases where Case A – No device used, Case B- Only spoiler, Case C- Only diffuser, and Case D- both spoiler and diffuser were used.

3.1 Quantitative Results

3.1.1 Drag Force

Fig 5 and Table 5 depict variations in drag force coefficients (C_d) resulting from different configurations of device installations. The lowest C_d of 0.3248 is observed when only the diffuser is installed (Case C), while the highest C_d of 0.4023 occurs with the exclusive installation of the spoiler (Case B). Notably, concurrent installation of both the spoiler and diffuser (Case D) yields a C_d of 0.3412, surpassing the C_d of the car without any devices (Case A). This indicates a 5.59% reduction in C_d compared to the baseline case (Case A, no device), where the C_d is 0.3439.

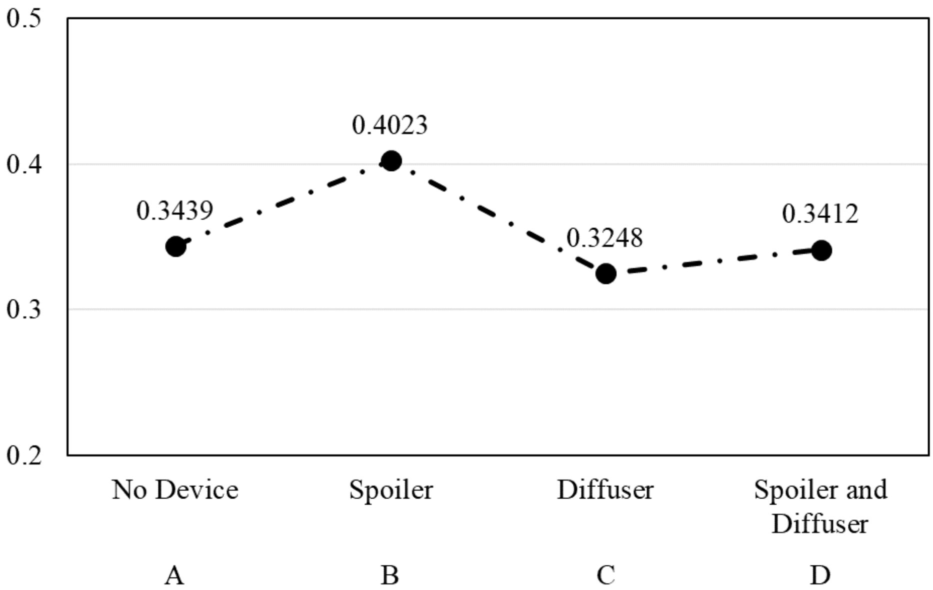


Fig. 5 *Graphical visualization of drag force coefficient (c_d) for different conditions*

Int. Journal of Integrated Engineering Vol. 16 No. 6 (2024) 154-164

Table 5 *Results of drag force coefficient (C_d) for different conditions*

Case	A	B	C	D
Condition	No Device	Spoiler	Diffuser	Spoiler and Diffuser
Drag Force Coefficient (C_d)	0.3439	0.4023	0.3248	0.3412

Fig 6 and Table 6 illustrate variations in drag force coefficients (C_d) for a car equipped with both a spoiler and diffuser across different speed conditions. The C_d remains constant at 0.334 for both low (70 km/h) and medium (110 km/h) speed conditions, mirroring the value observed at 95 km/h (0.3412). However, the C_d peaks at high speeds, particularly at 150 km/h, highlighting the influence of diverse device installations. The increase in car speed leads to elevated dynamic air pressure, resulting in heightened air resistance and an associated rise in the drag coefficient.

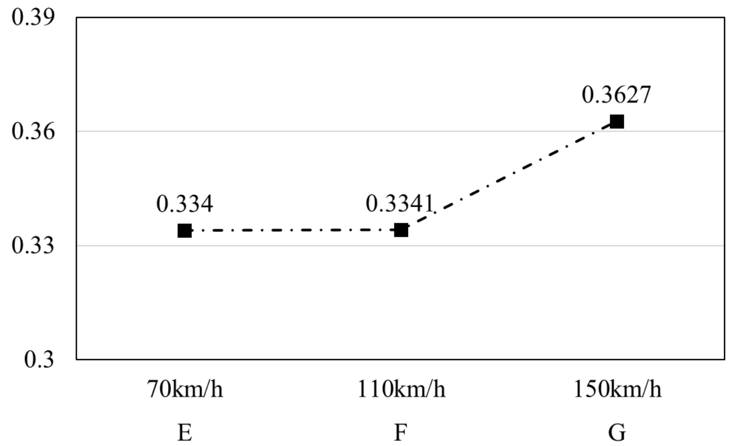


Fig. 6 *Graphical visualisation of drag force coefficient (C_d) for different speed conditions*

Table 6 *Results of drag force coefficient (C_d) for different speed conditions*

Case	E	F	G
Speed	70 km/h	110 km/h	150 km/h
Drag Force Coefficient (C_d)	0.3340	0.3341	0.3627

3.1.2 Lift Force

In Fig 7 and Table 7, lift force coefficients (C_l) variations across different device installations are depicted. The lowest C_l (-0.3496) occurs with the exclusive installation of the spoiler (Case B), while the highest (-0.0727) is observed without any devices (Case A). Concurrent installation of both the spoiler and diffuser (Case D) results in a C_l of -0.3324, showcasing a significant reduction in lift and potential improvements in the car's handling. Notably, the diffuser, while effective, falls short of the spoiler's efficacy in mitigating lift.

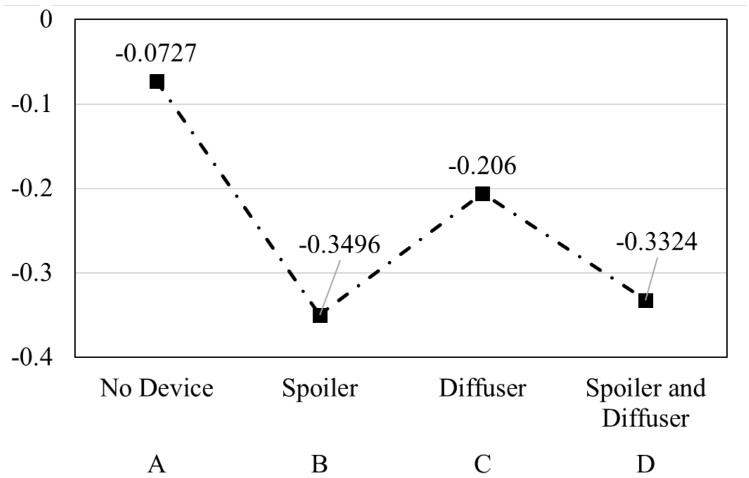


Fig. 7 *Graphical visualization of lift force coefficient (C_l) for different conditions*

Table 7 *Results of lift force coefficient (C_l) for different conditions*

Case	A	B	C	D
Condition	No Device	Spoiler	Diffuser	Spoiler and Diffuser
Lift Force Coefficient (C_l)	-0.0727	-0.3496	-0.2060	-0.3324

Fig 8 and Table 8 show lift force coefficient (C_l) variations for a car with both a spoiler and diffuser (Case D) at different speeds. At 110 km/h, C_l is -0.3256, aligning with the previous study at 95 km/h. At 150 km/h, C_l drops significantly to -0.6162, indicating increased downforce at higher speeds. Conversely, at 70 km/h, C_l is slightly lower at -0.3642 due to reduced dynamic air pressure, resulting in less significant air force for generating downforce.

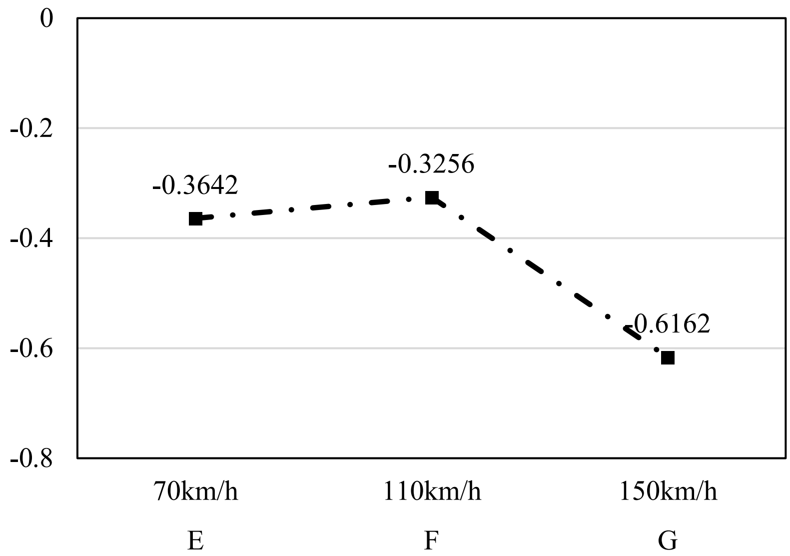


Fig. 8 *Graphical visualisation of lift force coefficient (C_l) for different speed conditions*

Table 8 *Results of lift force coefficient (C_l) for different conditions*

Case	E	F	G
Speed	70 km/h	110 km/h	150 km/h
Lift Force Coefficient (C_l)	-0.3642	-0.3256	-0.6162

3.2 Qualitative Results

3.2.1 Streamline Superimposed on The Velocity Contour

Fig 9 illustrates the impact of spoiler and diffuser installations on flow patterns around a car, showcasing changes in streamline behavior and velocity contours. In Case A, without devices, streamlines follow a smooth path over the car, creating recirculation and turbulence at the rear. Case B, with a spoiler, deflects streamlines upwards, generating downforce and increased turbulence. Case C, with a diffuser, deflects streamlines downwards, reducing turbulence and drag. Case D, with both devices, deflects streamlines both upwards and downwards, eliminating lift and creating a larger turbulence region. The combined effect creates a substantial low-pressure zone at the rear, enhancing handling and performance, particularly at high speeds. Fig 10 illustrates the impact of varied speed conditions on streamline patterns in Case D, involving both spoiler and diffuser installations. Cases E and F, representing low and medium speeds (70 km/h and 110 km/h), exhibit similar streamline behavior, emphasizing consistent flow characteristics at moderate speeds. In contrast, Case G, representing high speed (150 km/h), displays intensified airflow dynamics with a greater number of streamlines. The comparative analysis reveals subtle recirculation in the upper region and substantial recirculation in the lower rear, contributing to increased drag. Differences in velocity distribution, particularly at the lower rear, lead to heightened pressure and downforce in Case G, effectively enhancing stability and performance at high speeds. This highlights the complex interplay between spoiler and diffuser dynamics, emphasizing the sensitivity of aerodynamic effects to changes in velocity and the pivotal role of speed in shaping car performance.

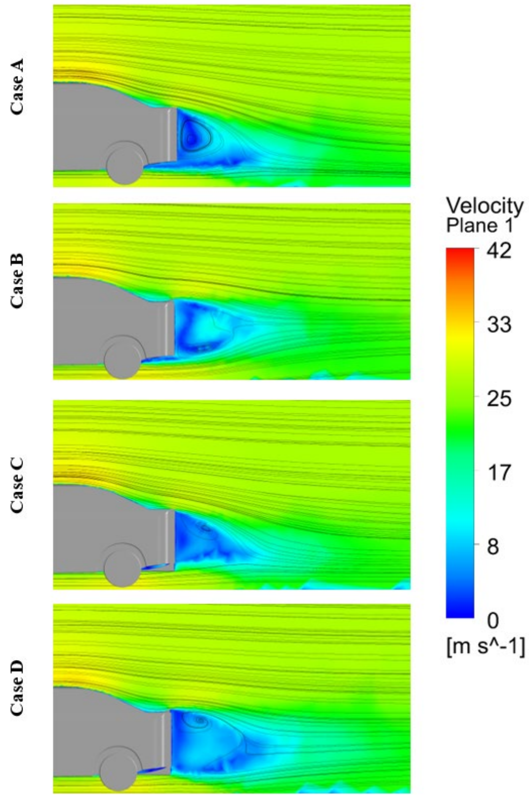


Fig. 9 *Streamline superimposed on the velocity contour for different device installation*

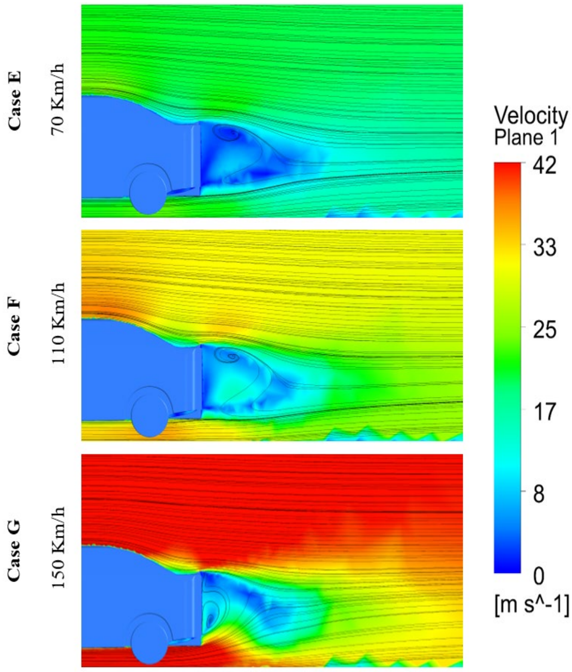


Fig. 10 *Streamline superimposed on the velocity contour for different speed conditions*

3.2.2 Velocity Vector Superimposed on the Velocity Contour

The introduction of velocity vectors alongside the velocity contour enhances the understanding of airflow patterns around the vehicle. As depicted in Fig 11, Case A without a spoiler or diffuser, exhibits uniform velocity vectors, indicating smooth airflow. The absence of aerodynamic devices leads to recirculating airflow, contributing to drag. In Case B, with a spoiler, upward-deflected velocity vectors demonstrate the creation of low pressure above the car, generating lift. Case C, featuring a diffuser, shows downward-deflected vectors, emphasizing the low-pressure region beneath the car, producing lift. Case D, with both spoiler and diffuser, manifests upward and downward deflection, creating large low-pressure regions, significantly enhancing rear velocities and generating substantial lift. The collaborative action of the spoiler and diffuser generates downforce, improving the car's handling, especially at high speeds.

Fig 12 illustrates the velocity contour of a car equipped with a spoiler and diffuser, showcasing accelerated airflow over the car's top and reduced velocity underneath, indicative of downforce creation. The designed downforce enhances handling and stability, particularly at higher speeds. The aerodynamic design minimizes drag, allowing the car to attain higher speeds efficiently. In speed-specific cases (E, F, G), varying velocity vectors and contours demonstrate the correlation between speed, downforce generation, and drag reduction. The effectiveness of the spoiler and diffuser in creating downforce and diminishing drag is consistent across low, medium, and high speeds, highlighting their integral role in optimizing aerodynamic performance.

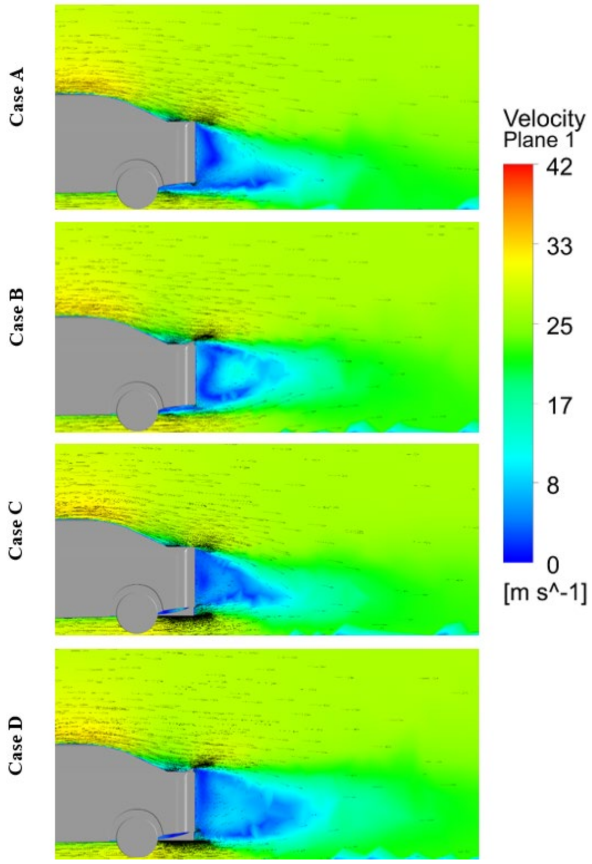


Fig. 11 *Velocity vector superimposed on the velocity contour with different device installation on the car*

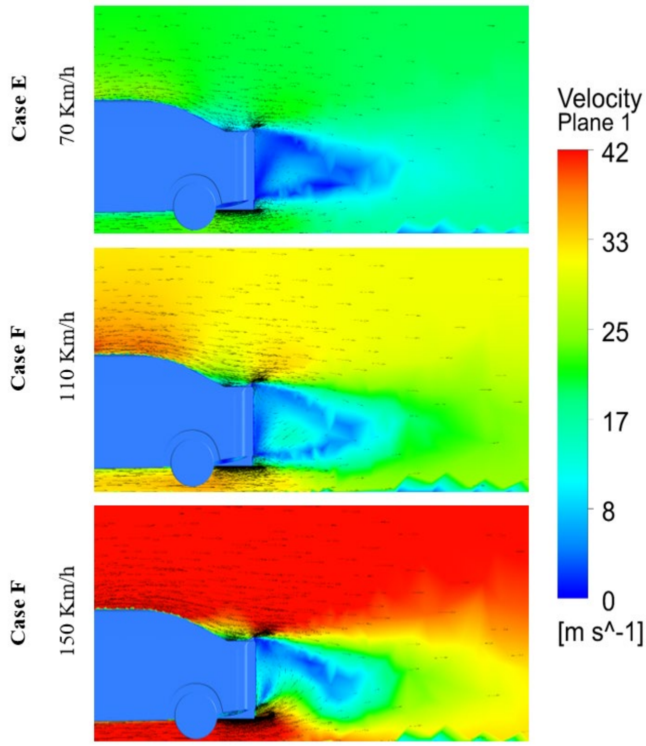


Fig. 12 *Velocity vector superimposed on the velocity contour with different speed conditions*

3.2.3 Pressure Contour on the Surface of the Car

Pressure contours are pivotal in the aerodynamic study of vehicles, offering visual insights into pressure distribution across the car's surface. Analyzing Fig 13, Case A (No spoiler or diffuser) reveals lower pressure on the car's roof and higher pressure on the trunk due to varied airflow speeds. Case B (Spoiler only) elevates pressure on the trunk by altering airflow direction, reducing lift. Case C (Diffuser only) maintains top-view similarity with Case A, but the bottom view showcases reduced pressure under the rear, enhancing airflow speed. Case D (Spoiler and diffuser) combines spoiler-induced high pressure and diffuser-induced low pressure, effectively eliminating lift and enhancing downforce. Together, the spoiler and diffuser create a potent low-pressure zone, reducing drag and increasing overall aerodynamic efficiency. Analyzing Fig 14, at 70 km/h, the pressure contour indicates smooth airflow with no notable downforce or drag. At 110 km/h, increased airflow over the top and slowed airflow underneath suggest the spoiler and diffuser initiating downforce creation, with some drag. At 150 km/h, the pressure contour reflects significantly accelerated top airflow and decelerated airflow underneath, signifying substantial downforce generation by the spoiler and diffuser. Despite heightened drag, the downforce effectively offsets it, enhancing the car's aerodynamic performance, particularly at medium and high speeds. This also supported by previous literature. Studies have shown that rear spoilers and diffusers are effective in reducing drag and increasing downforce at speeds above 80 km/h [20], [21]. The diffuser functions to expand the airflow from underneath the car to the rear, creating a pressure potential that accelerates the flow underneath the vehicle, leading to reduced pressure and enhanced downforce [22].

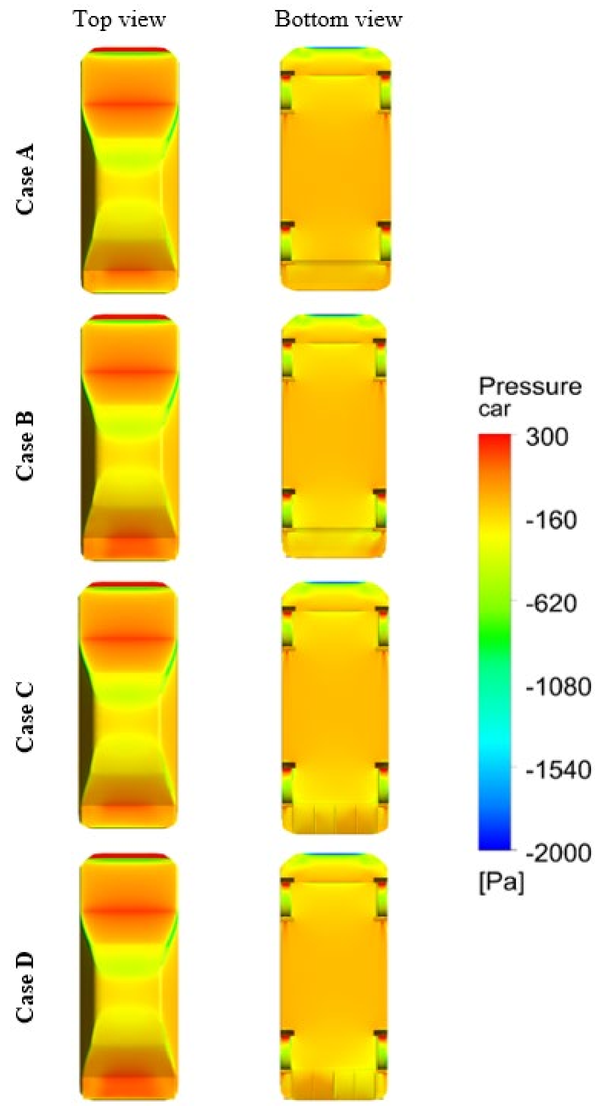


Fig. 13 *Pressure contour on car surface with different device installation*

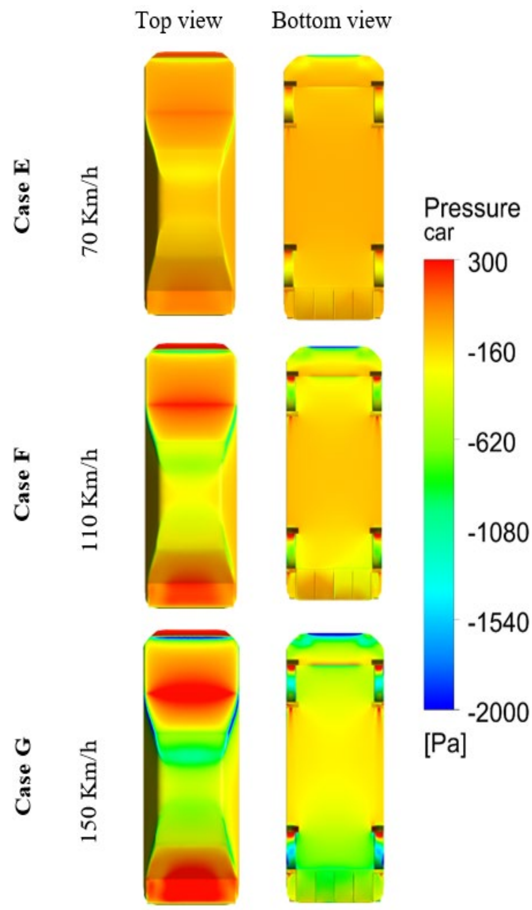


Fig. 14 *Pressure contour on car surface with different speed conditions*

4. Conclusion

In conclusion, this study utilized Computational Fluid Dynamics (CFD) analysis to evaluate the impact of spoilers and diffusers on sedan cars, revealing promising results for enhancing vehicle efficiency. The study systematically analyzed drag and lift forces, demonstrating a potential 5.5% reduction in the drag coefficient, with a 0.7% reduction when both a spoiler and diffuser were used. Flow structure visualizations elucidated how spoilers generate downforce and diffusers decrease pressure, reducing drag and lift forces. Investigation into speed effects confirmed consistent benefits across various speeds, emphasizing reliability in improving aerodynamic efficiency under diverse driving conditions. The research suggests that applying spoilers and diffusers to sedan cars can substantially reduce drag coefficients, enhancing fuel efficiency and reducing emissions. However, it highlights the need for careful consideration of design-specific factors and potential side effects, such as increased lift. To advance practical application, future studies should focus on diverse real-world driving conditions, encompassing urban commuting, highway cruising, and challenging terrains. Evaluating the adaptability of devices like spoilers and diffusers across varied driving environments is crucial for effective implementation. Additionally, for a comprehensive understanding of sustained benefits and practical implications, continuous monitoring of vehicle performance post-implementation through longitudinal studies over extended periods is recommended.

Acknowledgement

This research was supported by Malaysia Ministry of Higher Education (MOHE) through Fundamental Research Grant Scheme (FRGS/1/2022/TK02/UTHM/03/1).

Conflict of Interest

Authors declare that there is no conflict of interest regarding the publication of the paper.

Author Contribution

The authors confirm their contribution to the paper as follows: ***study conception and design:*** *Lingamoorthy s/o Ramachandran, Izuan Amin Ishak;* ***data collection*** *Lingamoorthy s/o Ramachandran, Mohammad Arafat;* ***analysis and interpretation of results*** *Lingamoorthy s/o Ramachandran, Mohammad Arafat;* ***draft manuscript preparation:*** *Lingamoorthy s/o Ramachandran, Izuan Amin Ishak, Nor Atiqah Zolpakar, Shaiful Fadzil Zainal Abidin, Dalila Mohd Harun. All authors reviewed the results and approved the final version of the manuscript.*

References

[1] C. J. Ejeh, G. P. Akhabue, E. A. Boah, and K. K. Tandoh, "Evaluating the influence of unsteady air density to the aerodynamic performance of a fixed wing aircraft at different angle of attack using computational fluid dynamics," *Results Eng.*, vol. 4, p. 100037, Dec. 2019, https://doi.org/10.1016/j.rineng.2019.100037.

[2] M. Bappa, "Aerodynamic Simulation of Vehicle Bodywork Design," 2015, [Online]. Available: https://www.researchgate.net/publication/343968698

[3] Fatima-zahra Hachimy, Ashraf A. Omar, and Omar Elsayed, "The Accuracy of the Numerical Solution in Predicting Ahmed Body Components Drag Coefficients," *CFD Lett.*, vol. 14, no. 5, pp. 24–32, May 2022, https://doi.org/10.37934/cfdl.14.5.2432.

[4] M. P. Ismail, I. A. Ishak, N. A. Samiran, A. F. Mohammad, Z. M. Salleh, and N. Darlis, "CFD Analysis on the Effect of Vortex Generator on Sedan Car using ANSYS Software," *Int. J. Integr. Eng.*, vol. 14, no. 1, pp. 73–83, 2022, https://doi.org/10.30880/ijie.2022.14.01.008.

[5] I. S. Ali and A. A. Mahmood, "Improvement of Aerodynamics Characteristic of Heavy Trucks," *3rd Int. Conf. Trends Mech. Ind. Eng.*, vol. 7, no. 3, pp. 246–255, 2013, [Online]. Available: https://www.researchgate.net/publication/280551662

[6] O. Elsayed, A. Omar, A. Jeddi, S. Elhessni, and F. Z. Hachimy, "Drag Reduction by Application of Different Shape Designs in a Sport Utility Vehicle," *Int. J. Automot. Mech. Eng.*, vol. 18, no. 3, pp. 8870–8881, 2021, https://doi.org/10.15282/ijame.18.3.2021.03.0680.

[7] C. BAYINDIRLI and M. ÇELİK, "The verify of experimentally drag reduction by flow control rod application on bus model using computational fluid dynamic," *Int. J. Energy Appl. Technol.*, vol. 8, no. 4, pp. 217–221, Dec. 2021, https://doi.org/10.31593/ijeat.1033791.

[8] H. Taiming, Z. Xiaodong, W. Zhongmin, and G. Zhengqi, "Experimental and numerical investigations of the vehicle aerodynamic drag with single-channel rear diffuser," *Proc. Inst. Mech. Eng. Part D J. Automob. Eng.*, vol. 234, no. 8, pp. 2216–2227, Jul. 2020, https://doi.org/10.1177/0954407019893849.

[9] Arvindakarthik K S, Mohan Kumar K K, and Gokul Sangar E, "Drag Reduction in An Automobile by using Aerodynamic Shapes," *Int. J. Eng. Res.*, vol. V4, no. 10, Oct. 2015, https://doi.org/10.17577/IJERTV4IS100393.

[10] Muhammad Nabil Farhan Kamal *et al.*, "A Review of Aerodynamics Influence on Various Car Model Geometry through CFD Techniques," *J. Adv. Res. Fluid Mech. Therm. Sci.*, vol. 88, no. 1, pp. 109–125, Oct. 2021, https://doi.org/10.37934/arfmts.88.1.109125.

[11] N. K. Velagapudi, L. N. K., L. N. V. N. Rao, and S. R. Y., "Investigation of Drag and Lift Forces over the Profile of Car with Rearspoiler using CFD," *Int. J. Adv. Sci. Res.*, vol. 1, no. 8, p. 331, Oct. 2015, https://doi.org/10.7439/ijasr.v1i8.2510.

[12] V. Kshirsagar and J. V Chopade, "Aerodynamics of High Performance Vehicles," *Int. Res. J. Eng. Technol.*, vol. 05, no. 03, pp. 2182–2186, 2018, [Online]. Available: https://www.academia.edu/36833746/Aerodynamics_of_High_Performance_Vehicles

[13] F. Frihdianto, N. S. Wardani, and I. Widiastuti, "Analysis of the use of Spoiler one level and two levels in conditions of Steady against the Drag and Lift Coefficients on a Sedan Type Car by using CFD (Computational Fluid Dynamic)," *J. Mech. Eng. Vocat. Educ.*, vol. 1, no. 2, p. 58, Apr. 2019, https://doi.org/10.20961/jomeve.v1i2.25066.

[14] L. Porcar, "Study of the functioning and importance of diffusers in Formula 1 cars," *Univ. Politècnica Catalunya*, 2020, [Online]. Available: https://upcommons.upc.edu/bitstream/handle/2117/331430/Report_fitxer de consulta.pdf

[15] O. H. Ehirim, K. Knowles, and A. J. Saddington, "A Review of Ground-Effect Diffuser Aerodynamics," *J. Fluids Eng.*, vol. 141, no. 2, Feb. 2019, https://doi.org/10.1115/1.4040501.

[16] Perusahaan Otomobil Kedua Sdn Bhd, "Perodua Bezza," Perodua Malaysia. [Online]. Available: https://www.perodua.com.my/our-models/bezza.html

[17] S. Y. Cheng and S. Mansor, "Rear-roof spoiler effect on the aerodynamic drag performance of a simplified hatchback model," *J. Phys. Conf. Ser.*, vol. 822, no. 1, p. 012008, Apr. 2017, https://doi.org/10.1088/1742-6596/822/1/012008.
[18] S. C. V. Ramana Murty Naidu, V. M. Madhavan, S. Chinta, R. Manikandan, A. Premkumar, and R. Girimurugan, "Analysis of aerodynamic characteristics of car diffuser for dissimilar diffuser angles on Sedan's using CFD," *Mater. Today Proc.*, vol. 92, pp. 240–248, 2023, https://doi.org/10.1016/j.matpr.2023.04.379.
[19] L. W. Chen, I. A. Ishak, M. Arafat, N. Farhan, N. A. Samiran, and A. F. Mohammad, "Analysis of the Aerodynamic Characteristics of Small-Sized Car Vehicles under the Influence of Steady Crosswind," in *International Conference on Applied Science, Technology, and Engineering (ICASTE)*, Jakarta, 2022.
[20] M. Amer, "Experimental Investigation of a Spoiler's Impact on the Flow Pattern of a High-Speed Sport Car," *J. Min. Mech. Eng.*, vol. 1, no. 2, Aug. 2020, https://doi.org/10.32474/JOMME.2020.01.000110.
[21] S. W. Lee and H. L. Kim, "Numerical Study of Active Aerodynamic Control via Flow Discharge on a High-Camber Rear Spoiler of a Road Vehicle," *Appl. Sci.*, vol. 9, no. 22, p. 4783, Nov. 2019, https://doi.org/10.3390/app9224783.
[22] D. S. Nath, P. C. Pujari, A. Jain, and V. Rastogi, "Drag reduction by application of aerodynamic devices in a race car," *Adv. Aerodyn.*, vol. 3, no. 1, p. 4, Dec. 2021, https://doi.org/10.1186/s42774-020-00054-7.