Volume 6 Issue 1 (March 2026)
DOI 10.24191/jaeds.v6i1.170
eISSN 2805-5756

Journal of Applied Engineering
Design & Simulation

Investigation on the Aerodynamic Drag Reduction of the Frontal Projection of a Car Using Finite Volume Method

Lyeonis Stanley Victor Stanley[1], Nur Hafizah Habideen[2,*], Tajul Afiq Tajul Arus[1], Mohd Faiz Osrin[1] and Mahfuzah Zainudin[2]
[1]School of Engineering, INTI International College Penang, 1-Z Lebuh Bukit Jambul, 11900 Bayan Lepas, Pulau Pinang, Malaysia
[2]Mechanical Engineering Studies, Universiti Teknologi MARA Cawangan Pulau Pinang, Permatang Pauh Campus, 13500 Permatang Pauh, Pulau Pinang, Malaysia.
[*]corresponding author: hafizahhabideen@uitm.edu.my

ABSTRACT

Aerodynamic drag significantly influences vehicle fuel efficiency and energy consumption, particularly at highway operating conditions. While full three-dimensional (3D) simulations are widely employed in automotive aerodynamics, computationally efficient two-dimensional (2D) approaches remain valuable for early-stage design screening. This study investigates the influence of windshield inclination angle on aerodynamic drag characteristics of four simplified vehicle profiles, namely MPV, Hatchback, Sedan, and Sports car configurations, using Computational Fluid Dynamics (CFD) based on the Finite Volume Method (FVM). The models were developed in CATIA V5R20 and simulated using ANSYS Fluent with Reynolds-Averaged Navier–Stokes (RANS) equations employing the standard k–ε turbulence model. The MPV served as the datum model for comparative analysis. Results demonstrate a clear inverse relationship between windshield angle and drag coefficient, where the Sports car configuration achieved the lowest drag coefficient compared to the MPV model. Flow visualization through pressure contours and velocity streamlines confirms improved flow attachment and reduced wake formation for lower windshield angles. Although absolute drag values differ from real 3D vehicle data due to geometric simplifications, the trend agrees with established aerodynamic principles reported in literature. The findings highlight the importance of frontal geometry optimization in early-stage automotive design and validate the applicability of CFD screening for comparative aerodynamic assessment.

Keywords: Aerodynamic drag; CFD; Finite Volume Method; Windshield angle; Vehicle aerodynamics

Nomenclature (Greek symbols towards the end)

A	Frontal projected area of vehicle (m^2)
C_d	Drag coefficient
F_D	Aerodynamic drag force (N)
v	Inlet air velocity (ms^{-1})
ρ	Air density ($kg.m^{-3}$)
Θ	Windshield inclination angle (^{o})
P	Static pressure (Pa)
ε	Turbulent dissipation rate ($m^2.s^{-2}$)

Abbreviations

FVM	Finite volume method
MPV	Multi-purpose vehicle
RANS	Reynolds-averaged Navier Stokes

1.0 INTRODUCTION

Aerodynamic drag has long been recognized as one of the primary resistive forces acting on ground vehicles, particularly at higher velocities where it can account for over 60% of the total propulsive energy expenditure in highway driving conditions [1]. With global emphasis on improving fuel efficiency and reducing vehicular emissions, automotive manufacturers have intensified efforts to optimize vehicle aerodynamics. This is particularly crucial not only for internal combustion engine (ICE) vehicles but also for electric vehicles (EVs), where aerodynamic losses directly affect range and energy efficiency [2-3].

One of the most influential factors contributing to aerodynamic drag is the frontal projection of the vehicle. The geometry of the front-end particularly the windshield angle, hood slope, and overall shape that determines

Received on 15.02.2026
Accepted on 30.04.2026
Published on 05.05.2026

the flow separation point and stagnation pressure, directly affecting the form drag coefficient [4-5]. Previous studies have shown that lower windshield inclination angles and smoother frontal transitions improve flow attachment and reduce wake formation, thereby lowering drag [6-7]. Such findings are particularly relevant for body types such as SUVs and MPVs, which traditionally feature bluff front-end designs associated with higher drag forces [8-9]. However, these studies are often limited to single vehicle configurations or specific design cases, making it difficult to generalize the effect of windshield inclination across different vehicle categories.

Computational Fluid Dynamics (CFD) has become the dominant method for evaluating aerodynamic behaviour, offering a more cost-effective and flexible alternative to wind tunnel testing. Among available numerical methods, the Finite Volume Method (FVM) is most commonly employed in CFD due to its conservative discretization of flow variables across control volumes [10-11]. ANSYS Fluent, a commercial CFD tool, uses FVM and pressure-velocity coupling algorithms such as SIMPLE to accurately simulate complex flow fields around vehicle bodies [12].

Despite the extensive use of full 3D CFD simulations in final-stage automotive design, these models demand substantial computational resources and time, making them less practical for rapid early-stage conceptual screening. Despite existing studies on vehicle aerodynamics, there is a lack of a systematic comparative analysis that isolates the effect of windshield inclination across multiple vehicle categories under identical conditions. This limitation makes it difficult to generalize aerodynamic design guidelines across different vehicle types.

To bridge this research gap, this study investigates the aerodynamic performance of four distinct 2D vehicle profiles which are MPV, Hatchback, Sedan, and Sports car using the Finite Volume Method (FVM) in ANSYS Fluent. The novelty of this work lies in the development of a rapid 2D comparative assessment framework that explicitly correlates windshield steepness with drag coefficient and flow separation behaviour across these four specific configurations under identical boundary conditions. By isolating the windshield angle parameter, this study aims to validate the 2D CFD approach as a reliable, resource-efficient screening tool. The findings provide automotive designers with practical guidelines for optimizing front-end geometries early in the development cycle, significantly reducing the dependency on costly and time-consuming 3D modeling during initial screening.

This study contributes in three key aspects. First, it provides a systematic comparison of windshield inclination effects across four distinct vehicle categories under identical CFD conditions. Second, it establishes a clear relationship between windshield angle and aerodynamic drag behaviour through controlled parameter isolation. Third, it demonstrates that two-dimensional CFD modelling can serve as a reliable and computationally efficient tool for early-stage aerodynamic screening.

2.0 METHODOLOGY

2.1 Two-dimensional (2D) model of car design

The project begins with research and analysis of aerodynamic geometry and shape designs with respect to a car. The two-dimensional (2D) model of car design was constructed using CATIA V5R20 software within the respective fulfilment of aerodynamic design required for a commercial car. The types of cars which are designed and used in this simulation are the Hatchback, Sedan, Multi-purpose vehicle (MPV) and Sports car. 2D vehicle profile for the four types of car is shown in Figure 1 below. Angle of windshield and plane area were determined from the design. Table 1 shows the data of both angle of windshield and plane area for each of the car design.

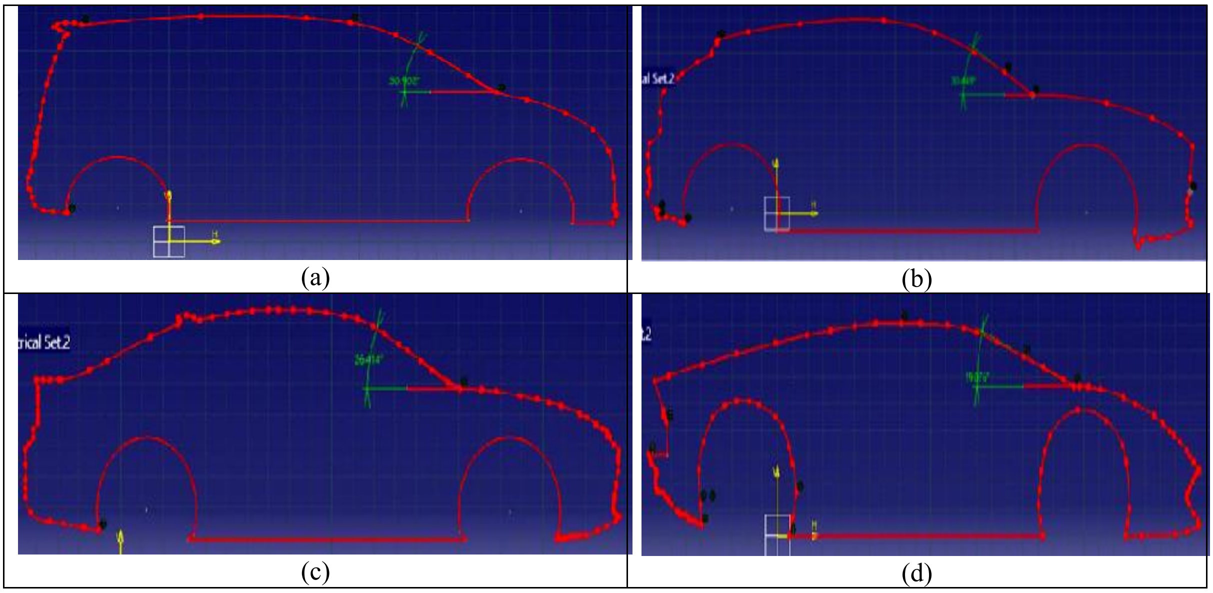

(a) (b) (c) (d)

Figure 1. 2D vehicle profiles of 4 types of car (a) MPV (b) Hatchback (c) Sedan (d) Sports

Table 1: Data of angle of windshield and plan area

Type of Car	Angle of Windshield (°)	2D Plane Area (m^2)
MPV (Datum)	30.902	0.024
Hatchback	30.449	0.040
Sedan	26.414	0.056
Sports	19.076	0.072

2.2 Aerodynamic drag

Aerodynamic is defined as a mechanical force with the motion of a fluid when it interacts with a solid object such as a car and plane [13]. Logically, when any fluid passes through a solid object, it will experience a drag force or the scientific term, resistive force as it is opposing its motion. The drag coefficient, Cd is a dimensionless parameter used to quantify the aerodynamic efficiency of a body. [13]. The formula of drag coefficient is shown below in Equation 1.

$$F_{drag} = \frac{1}{2} C_D A \rho v^2 \qquad \text{(N)} \qquad (1)$$

Form drag or pressure drag is the drag that is due to the separation of boundary layers from the surface and the wake created by that separation. Form drag is highly dependent on the shape of the solid object [14-15]. The equation used for unsteady incompressible fluid is the established Navier-Stokes equations as shown in Equation 2.

$$\frac{\partial u}{\partial t} + \nabla . uu = -\nabla P + \frac{1}{Re} \nabla \mathrm{u}^2 \qquad (\mathrm{N.m^{-3}}) \qquad (2)$$

2.3 Finite Volume Method (FVM) analysis

The FVM is a numerical technique that uses conservation law to evaluate elliptic, parabolic, or hyperbolic partial differential equations that take the form of algebraic equations [16]. One of the most crucial elements of CFD modelling is turbulence model which consists of two main categories, the Large Eddy Simulation (LES) models and Reynolds-averaged Navier-Stokes (RANS) models [17].

All the car designs are simulated using Ansys Workbench Fluent. FVM analysis was done to obtain the required values and data by importing the car designs igs files into Ansys. Datum was set to be MPV car. Turbulence model was set as Reynolds-averaged Navier-Stokes (RANS) model. Type of configuration is pressure-based simulation.

A mesh convergence study was conducted to ensure that the simulation results are independent of grid resolution. Four different mesh densities were evaluated, and the maximum pressure was used as the monitoring parameter. The maximum pressure was selected as the monitoring parameter for mesh convergence as it is directly related to aerodynamic drag behaviour. As shown in Table 2, a significant variation is observed between the coarse and medium mesh (3.10%). However, the difference between the medium and fine mesh is less than 0.1%, indicating that the solution has reached mesh independence. Further refinement to 55,622 elements shows negligible change. Therefore, the mesh with 46,095 elements was selected as the optimal mesh to balance computational cost and accuracy.

For boundary conditions, the inlet was set to a velocity magnitude of 10 m/s, corresponding to a Reynolds number in the order of 10^5 – 10^6 within the turbulent flow regime for external vehicle aerodynamics. Since the drag coefficient for bluff bodies remains relatively insensitive to Reynolds number variations within this regime, the selected velocity is considered sufficient for comparative analysis between models. However, future work should include a broader Reynolds number range to further generalize the findings.

The solution initialization configuration was set to Hybrid initialization method with Pressure-based type and viscous model used is standard k-epsilon. For solution methods, simple scheme is used. Least squares cell based is used for gradient, second order for pressure, second order upwind for momentum, turbulent kinetic energy and specific dissipation rate.

Table 2: Mesh Independence Study

Mesh Level	Number of Elements	Maximum Pressure (Pa)	Percentage Difference (%)
Coarse	4164	65.55	-
Medium	19485	67.58	3.1
Fine	46095	67.57	0.01
Finer	55622	67.51	0.09

3.0 RESULTS AND DISCUSSION

3.1 Drag coefficient

The simulation results show a clear inverse correlation between windshield angle and drag coefficient (Cd) as presented in Table 3 and Figure 2.

Based on the table and graph above, a relationship between angle of windshield and drag coefficient has been established in which the higher the angle of windshield, the higher the drag coefficient; thus, it produced higher drag force. It is also shown that Sports car has the lowest drag coefficient compared to other types of car.

The MPV, which has the most upright windshield, exhibited the highest drag coefficient of 0.6803. This is attributed to its relatively flat front surface that promotes early boundary layer separation and a larger low-pressure wake. In contrast, the Sports car, with the most sloped windshield (19.076°), achieved the lowest drag coefficient at 0.2268, indicating improved flow attachment and reduced form drag.

These results align with established aerodynamic principles in which vehicles with more streamlined frontal profiles exhibit lower form drag due to smoother pressure recovery and delayed flow separation. The drag values are also consistent with literature-reported ranges, providing validation for the CFD approach.

The obtained results fall within the range reported in the literature from [1] and [18] as shown in Table 4 below.

Table 3: Data of drag coefficient and angle of windshield for 4 types of car

Type of Car	Angle of Windshield (°)	Drag Coefficient (C_d)
MPV (Datum)	30.902	0.6803
Hatchback	30.449	0.4082
Sedan	26.414	0.2915
Sports	19.076	0.2268

Figure 2. Graph of drag coefficient vs angle of windshield

Table 4: Data of drag coefficient from [1] and [18]

Type of Car	Drag Coefficient Range (C_d)
MPV (Datum)	0.60 – 0.80
Hatchback	0.35 – 0.45
Sedan	0.26 – 0.35
Sports	0.20 – 0.30

3.2 Form Drag Contour

The frontal surface of each model experienced high static pressure, particularly near the bumper and windshield base. Pressure dropped sharply past the windshield due to flow separation, forming a wake that contributes to pressure drag. This can be seen clearly in Table 5 and Figure 3.

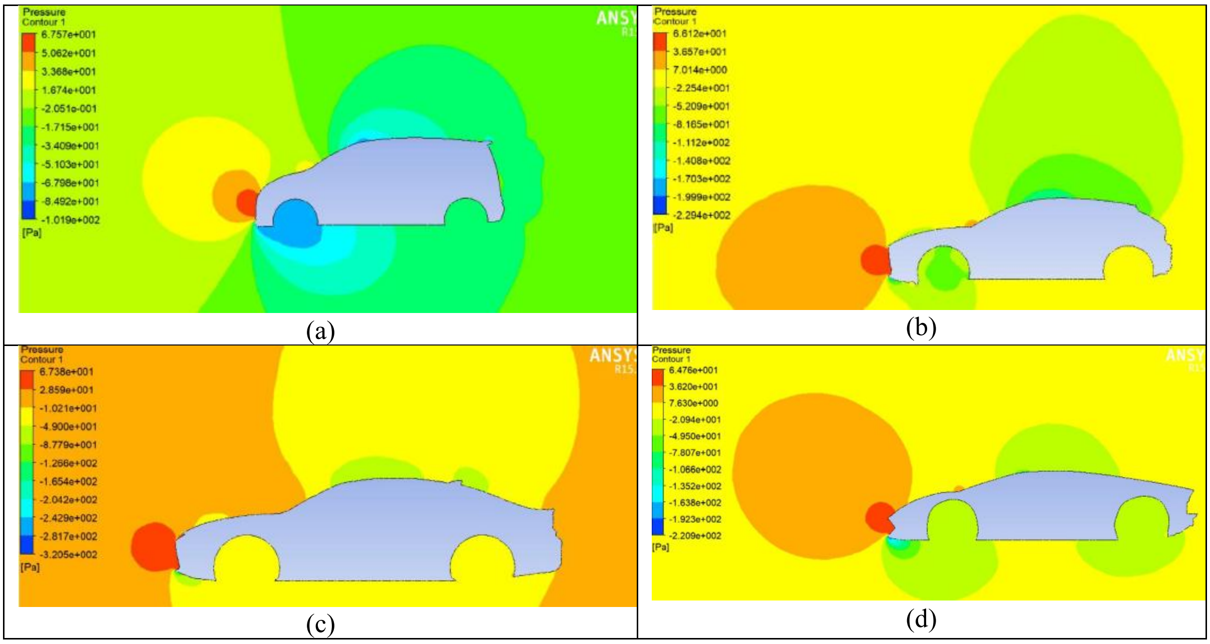
The Sports car displayed the lowest drag and most favourable pressure distribution, likely due to its steep windshield angle and smoother frontal curvature [4].

When the fluid flow moves towards the car and makes contact with the body, high pressure is observed near the front bumper, followed by a pressure drop along the hood due to flow separation [1], [5]. The bottom fluid flow is neglected as this project focuses on the frontal part of the car. From the figure above, we can see that Sports car has the lowest maximum pressure exerted on the frontal part.

A lower minimum pressure indicates more intense suction effects and larger wake formation; however, in the Sports car, the flow remains attached due to its curvature, resulting in reduced drag [5]. These observations highlight that the shape and angle of frontal geometry play a critical role in managing wake structure and reducing drag.

Table 5: Aerodynamic analysis on four type of cars

Type of Car	Drag Coefficient (C_d)	Maximum Pressure (Pa)	Minimum Pressure (Pa)
MPV (Datum)	0.6803	67.57	-67.96
Hatchback	0.4082	66.12	-81.65
Sedan	0.2915	67.38	-49.00
Sports	0.2268	64.76	-49.50



(a) (b) (c) (d)

Figure 3. Form drag contour of 4 types of car (a) MPV (b) Hatchback (c) Sedan (d) Sports

3.3 Velocity Streamline

To visualize the flow behavior around the frontal geometry of each vehicle profile, velocity streamline contours were extracted from the CFD simulations, as shown in Figure 4.

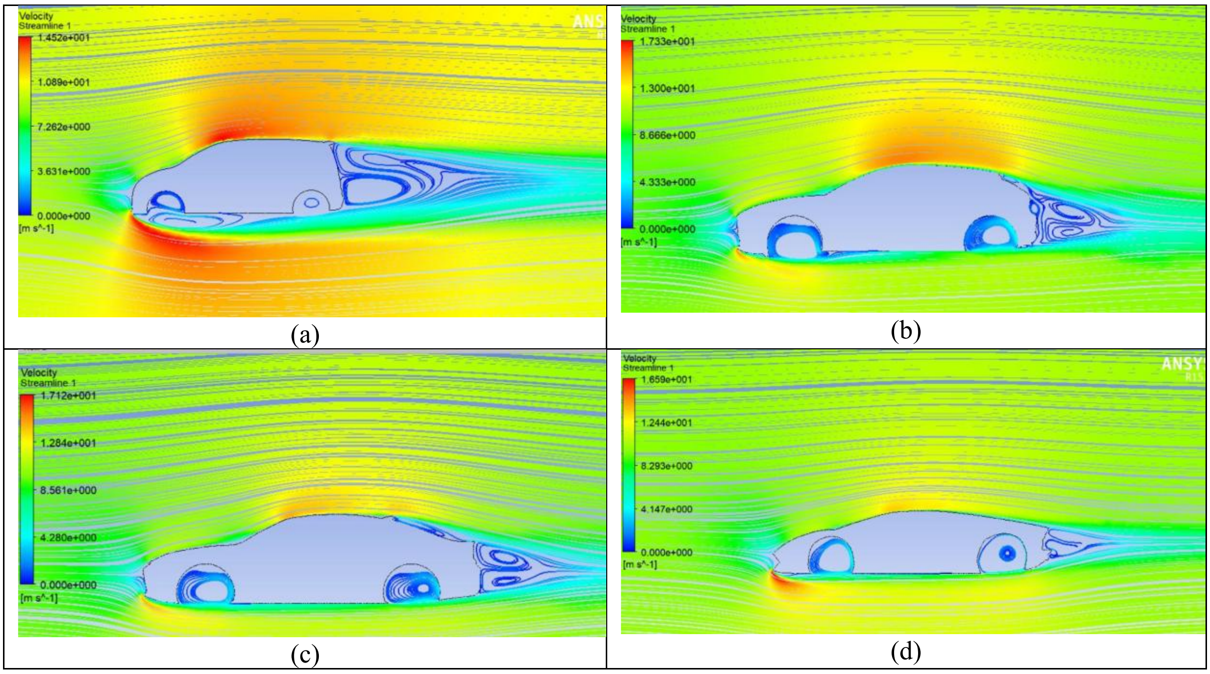
For the MPV (Figure 4a), the upright windshield causes early boundary layer separation, leading to a large recirculation zone near the rear of the roof and a broad wake region, indicative of high form drag [1]. The Hatchback (Figure 4b) shows slightly better flow behaviour, with a reduced but still significant wake, caused by minor improvements in windshield slope. However, its blunt rear design still leads to flow detachment and turbulent recirculation [9]. In contrast, the Sedan (Figure 4c) demonstrates improved streamline attachment along the windshield and roof, resulting in a smaller and more controlled wake region, thereby reducing aerodynamic drag [7]. The Sports car (Figure 4d) displays the most streamlined flow profile, with smooth, continuous streamlines across the windshield and minimal flow separation. Its narrow and symmetric wake region reflects optimal aerodynamic performance [4].

The observed streamline patterns strongly support the drag coefficient trends reported earlier, confirming that steeper windshield angles and streamlined front-end geometry effectively minimize drag by maintaining flow attachment and reducing wake formation. The formula for drag force, F_{drag} can be derived and rearranged to calculate the drag coefficient. The drag force equation can be rearranged to express the drag coefficient, as shown in Equation (3).

$$C_D = \frac{2F_{drag}}{A\rho v^2}$$ (-) (3)

For the datum car design, the datum design and drag coefficient are based on the manufactured Perodua Alza 2009. The drag coefficient, C_d of the Perodua Alza 2009 is 0.33 [9]. Table 6 shows comparison with theoretical calculation.

The comparison shows consistent trends between the present CFD results and reference values, where higher windshield angles correspond to higher drag coefficients.



(a) (b) (c) (d)

Figure 4. Velocity streamlines of 4 types of car (a) MPV (b) Hatchback (c) Sedan (d) Sports

Table 6: Theoretical calculation of drag coefficient

Type of Car	Drag Coefficient (C_d)
MPV (Datum)	0.33
Hatchback	0.32
Sedan	0.24
Sports	0.20

4.0 CONCLUSION

This study successfully demonstrates the role of frontal design geometry in aerodynamic drag reduction using CFD simulations based on the Finite Volume Method. Among the four designs analysed, the Sports car model exhibited the lowest drag coefficient due to its sloped windshield and streamlined front profile. These findings reaffirm that small geometric changes, especially at the windshield and hood region, can substantially influence drag performance. The Sports car configuration demonstrated the lowest drag coefficient (0.2268), while the MPV exhibited the highest (0.6803).

The research highlights CFD as a reliable alternative to physical wind tunnel testing, offering valuable insight during the early design phase. The findings validate the use of simplified CFD screening during early-stage vehicle conceptual design. By focusing on 2D analysis, the main contribution of this study lies in demonstrating the applicability of simplified two-dimensional CFD modelling as an efficient preliminary design tool for aerodynamic assessment. The findings provide automotive designers with practical guidelines for optimizing front-end geometries early in the development cycle, specifically regarding the influence of windshield inclination on aerodynamic drag characteristics. This study provides a computationally efficient method for concept validation, though future work could extend to 3D modelling and real-world validation to enhance accuracy.

ACKNOWLEDGEMENT

This research was conducted without external funding support. The authors acknowledge Inti International College Penang and Mechanical Engineering Studies, Universiti Teknologi MARA for providing computational facilities.

AUTHORS CONTRIBUTION

Lyeonis Stanley Victor Stanley – CFD simulation, Data curation, Formal analysis, Investigation, Writing – Original Draft.
Nur Hafizah Habideen – Supervision, Conceptualization, Methodology validation, Technical review, Writing – Review & Editing.
Tajul Afiq Tajul Arus – Methodology support, Validation, Technical guidance.
Mohd Faiz Osrin – Project administration, Resources, Review & Editing.
Mahfuzah Zainudin – Data analysis support, Literature review, Manuscript proofreading.

DECLARATION OF COMPETING OF INTEREST

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

REFERENCES

[1] Hucho, W. H. Aerodynamics of Road Vehicles (4th ed.). SAE International. 1998.
[2] Shivam Agarwal, "Aerodynamics of Vehicles," *International Journal of Advanced Research in Engineering and Technology*, vol. 11, no. 11, pp. 1996-2001, 2020.
[3] A, Gurusamy & Ashok, Bragadeshwaran & Mason, Byron. Prediction of Electric Vehicle Driving Range and Performance Characteristics: A Review on Analytical Modeling Strategies With Its Influential Factors and Improvisation Techniques, 2023.
[4] Katz, J. Race Car Aerodynamics: Designing for Speed. Bentley Publishers. 1996.
[5] Essaghouri, Abdellah & Wang, Bo. "CFD analysis on effect of front windshield angle on aerodynamic drag," *IOP Conference Series: Materials Science and Engineering*, vol. 231, 2017.
[6] M. Syafiq, "Computational Fluid Dynamics Analysis of Aerodynamic Characteristics on Overtaking Vehicles in Crosswind Conditions", *Int. J. Automot. Mech. Eng.*, vol. 22, no. 2, pp. 12373–12387, 2025.
[7] M. Z. Mohd Zin, "Simulation The Effect of Spoiler on Aerodynamic for Sedan Car", *Peat*, vol. 3, no. 2, pp. 691–699, 2022.
[8] Saleh, Zahraa & Hamid, Anmar. "Numerical Investigation of Drag Reduction Techniques in a Car Model," *IOP Conference Series Materials Science and Engineering,* vol. 671, 2020.
[9] Morel, T., "Aerodynamic Drag of Bluff Body Shapes Characteristic of Hatch-Back Cars," *SAE Technical Paper*, 1978.
[10] Versteeg, H. K., & Malalasekera, W. "An Introduction to Computational Fluid Dynamics", *The Finite Volume Method (2nd ed.). Pearson Education*. 2007.
[11] Madha, Jowad & Nawar, Anika & Rahman, Md. Automotive Aerodynamic Drag and Lift Analysis using Computational Fluid Dynamics Software. 2023
[12] Moukalled, F., Mangani, L., & Darwish, M. "The Finite Volume Method in Computational Fluid", *Dynamics. Springer*. 2016
[13] Dourmashkin, P. Classical Mechanics. 2022.

[14] Alciatore and Anderson. McGraw-Hill Series In Mechanical Engineering Anderson: Computational Fluid Dynamics: The Basics with Applications. 2006

[15] Gur, O., Mason, W.H., and Schetz, J.A. “Full-Configuration Drag Estimation”. *Journal of Aircraft,* vol. 47, no. 4, pp. 1356–1367, 2010.

[16] Brooks, R.R. “Soft Computing Techniques”. *Distributed Sensor Networks,* pp. 321–333. 2024

[17] Yusof, S.N.A., Asako, Y., Sidik, N.A.C., Mohamed, S.B., and Japar, W.M.A.A. “A Short Review on Rans Turbulence Models”, *CFD Letters*, vol. 12, no. 11, pp. 83–96, 2020.

[18] White, F. M. Fluid Mechanics (7th ed.). McGraw-Hill. 2011.

[19] Autoevolution (2021) Perodua Alza Specs & Photos - 2009, 2010, 2011, 2012, 2013, 2014 - Autoevolution [Online]. Available: <https://www.autoevolution.com/cars/perodua-alza-2009.html#aeng_perodua-alza-2009-15l-4at-103-hp>