Advances in Fluid, Heat and Materials Engineering

Journal homepage:
https://karyailham.com.my/index.php/afhme/index
ISSN: 3083-8134

Modelling of the Effect of Rear Spoiler Angle on the Aerodynamics of Passenger Car

Tan Sook Yin[1], Ishkrizat Taib[1,*], Muhammad Hafizi Husaini[1], Wan Mohama Nordin Wan Ibrahim[1], Muhammad Adib Che Saad[1], Noorhaslina Basar[1] , Muhammad Hazimuddin Halif[1]

[1] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia

ARTICLE INFO

Article history:
Received 17 January 2025
Received in revised form 23 February 2025
Accepted 2 March 2025
Available online 27 March 2025

Keywords:
Rear spoiler; aerodynamic; passenger car

ABSTRACT

The study investigates the impact of propeller blade number on thrust force and efficiency in turbulent flows, a critical factor in the performance of marine and aerial vehicles. Despite advancements in propeller design, the relationship between blade count, thrust generation, and efficiency remains complex and requires deeper understanding. This research aims to elucidate this relationship by conducting a series of computational fluid dynamics (CFD) simulations. Using ANSYS FLUENT, various propeller configurations with different blade counts were modelled and analyzed under turbulent flow conditions. The simulations focused on evaluating thrust force and efficiency metrics across these configurations. Results indicate that increasing the number of blades generally enhances thrust force but may lead to a reduction in efficiency due to increased drag and wake interactions. The optimal blade number was identified, balancing thrust force and efficiency, providing valuable insights for future propeller design. In conclusion, the study highlights the trade-offs between thrust force and efficiency in propeller design, offering a framework for optimizing blade count in various applications, enhancing performance while maintaining energy efficiency.

1. Introduction

Rear spoiler is a design Rear spoiler is a device commonly found on road vehicles for improving their aerodynamic performance [1-3]. This is achieved by reducing the drag force acting on the car, which oppose the car motion. 65 percent of the power generated by the fuel were used to overcome the drag force when the car is at the speed of 110 km/h [4]. To improve the efficiency of fuel consumption, it is necessary to improve the aerodynamic characteristics of a car [5]. In addition to the streamlined design of the car body, a rear spoiler can be added to reduce drag force [6-8]. Spoiler is an add-on device which help to diffuse the unfavourable fluid movement around the car body [9,10]. Besides, rear spoiler also provides additional negative lift, which stabilise the car body when

[*] *Corresponding author.*
E-mail address: iszat@uthm.edu.my

https://doi.org/10.37934/afhme.4.1.3645a

it travels at a high speed. Rear-end spoilers are commonly used in racing car due to effectiveness and design simplicity [11,12].

It is estimated that the aerodynamic drag is a form of resistance when vehicles reach speeds of 80 km/h or higher which will cause the car to have a high tendency to lift [13-15]. To reduce the lift that acted on the rear trunk, a rear spoiler can attach on it to create more high pressure. They act as barriers to airflow, in order to build up higher air pressure in front of the spoiler. The improvements in aerodynamic characteristics can result in significant decrease in driving stability, handling, fuel consumption and overall efficiency [16]. Hence this study aims to study the effect of rear spoiler and the angular effect of spoiler on the aerodynamics of the passenger car in order to find the optimum angle of spoiler with which minimum drag is experienced.

In this study, the optimum angle of rear spoiler will be studied by simulating the air flow through the passenger car model using Ansys Fluent. Several models with different angle of rear spoilers will be simulated and the drag coefficient of each model will be obtained from the simulation and compared. The purpose of this study is to investigate the effect of rear spoiler and the angular effect of spoiler on the aerodynamics of the passenger car. Therefore, the expected results of this study are drag coefficient and lift coefficient of the spoiler and an optimum angle of spoiler with minimum drag experienced.

The flow field around a vehicle are mainly affected by the boundary layer, flow separation, friction drag as well as pressure drag. The flow field can be divided into two regions, which are the region inside and outside the boundary layer. The flow becomes inviscid outside the boundary layer. In the boundary layer surrounding the vehicle, viscosity is dominant and affect the drag of the vehicle. In other words, the boundary layer is one of the major factors in aerodynamic drag.

Tomar *et al.,* [17] *s*tudied the influence of the angle of rear spoiler on sedan type car. The sedan car model without spoiler as well as model with rear spoiler at an angle of 0, 10, 20 and 30 degrees were simulated using ANSYS FLUENT. The drag coefficient, C_D and drag force, F_D were obtained and compared. It was found that sedan car with spoiler at an angle of 10 degree yield the lowest drag coefficient, which is 0.619, and hence lowest drag force (471.157 N). On the other hand, drag coefficient and drag force were the greatest when the spoiler was at an angle of 30 degrees, which are 0.695 and 536.74 N respectively. These results showed the importance of installing the spoiler at a correct angle to reduce drag force.

Soares *et al.,* [18] studied the effect of rear end spoiler at different angle on the drag and lift coefficient of vehicle. A simplified, 35% scaled-model driver with smooth underbody, no wheels, and side mirrors is used as the model for both simulation and experimental study. Experiment was carried out in the 8x6 general purpose, closed-return circuit, low speed wind tunnel at Cranfield University, UK. The facility has a closed rectangular test section (2.4 m x 1.8 m) with corner fillets and a breather-slot at the downstream end, ahead of the first diffuser.

2. Methodology

The drag coefficient of several model with different angle of rear spoilers using simulation analysis is to be determined in the current study. The flow chart for this work is shown in Figure 1.

2.1 Flow Chart

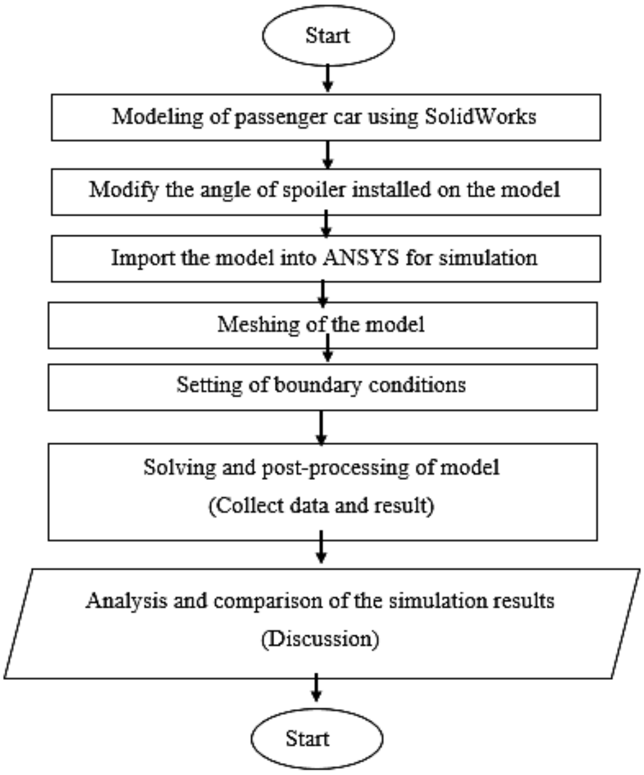


Fig. 1. Flow chart of the aerodynamics of passenger car

2.2 Geometry of the Simplified Passenger Car with the Different Angle of Rear Spoilers

A simplified model of passenger car was modelled using the computational aided design software called SolidWorks (Dassault Systèmes, 2023) as shown in Figure 2. This model has designed based on the model used in research by Sharma *et al.,* [19]. This model has a dimension with a length of 1044mm and width of 389mm as shown in Figure 3. Furthermore, several models of the rear spoilers were modelled and some modification has been made on the angle of rear spoilers as shown in Figure 4.

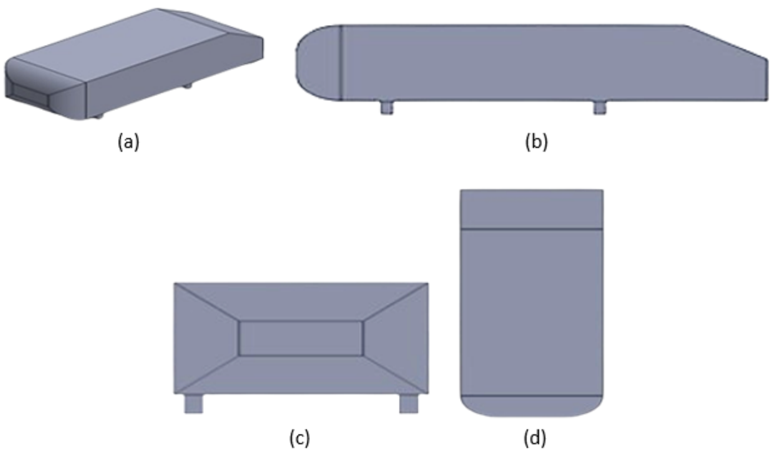


Fig. 2. Passenger car model (a) Isometric view (b) Front view (c) Side view (d) Top view

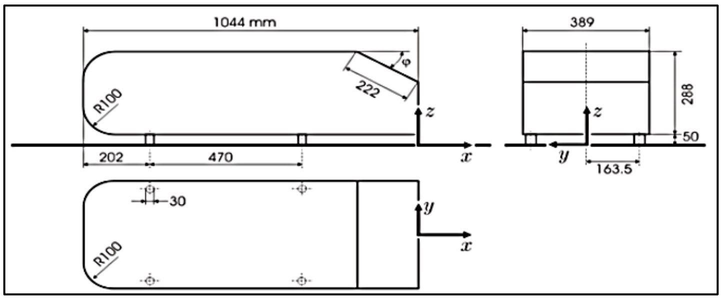
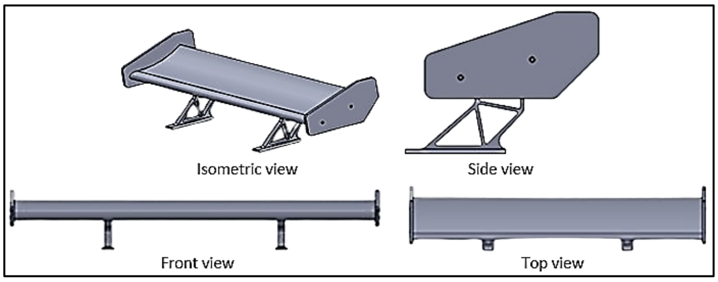
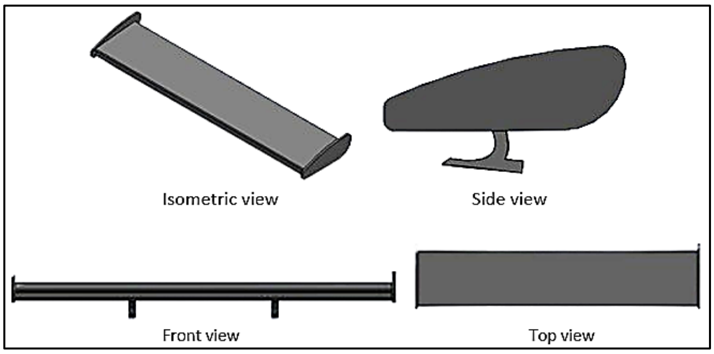


Fig. 3. Geometry of reference Ahmed's [19] model



(a)

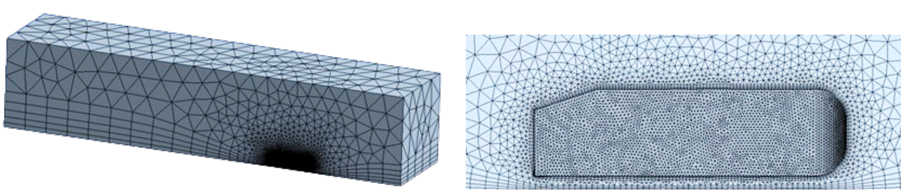


(b)

Fig. 4. Spoiler (a) Design 1 (b) Design 2

2.3 Meshing of the Model

The mesh of model been made towards all body including the air domain with mesh refinement on the car body especially since the flow around the body is the main interest. To ensure the meshing did not affect the results, grid independency test (GIT) was carried out to seek for the optimum element size for meshing [20]. The finalized mesh of Ahmed body had a face sizing of 10 mm element size. Inflation layer was set to program controlled with first aspect ratio. This model had a total of 264650 elements. The meshed body were shown in Figure 5.



(a) (b)

Fig. 5. Model (a) Meshed (b) Refined mesh with inflation layer

2.4 Boundary Condition

Boundary conditions were crucial as it limiting the computational domain and ensuring accurate simulation results. The boundary conditions for this simulation were based on prior research where it's been applied to reflect flow behaviour around the model.

2.4.1 Inlet velocity

Velocity inlet was set at the domain side that is opposite to the front side of Ahmed body, as shown by the face highlighted with red in Figure 6. The air flow in the direction from the front of Ahmed body to the rear of the same body.

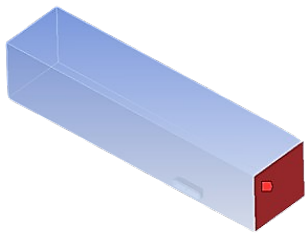


Fig. 6. Velocity inlet

2.4.2 Pressure outlet

Pressure outlet was set at the domain side that is opposite to the rear side of Ahmed body, as shown by the face pointed with a mark in Figure 7.

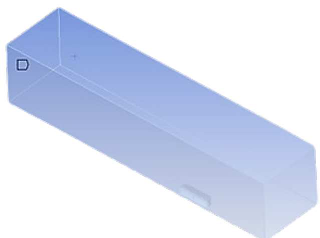


Fig. 7. Pressure outlet

2.4.3 Side walls

Side wall is a symmetrical boundary condition that is commonly employed for symmetrical body in order to reduce the time required for simulation. The model in this case is symmetrical along the x-axis or the long side, as shown by the face highlighted with red in Figure 8.

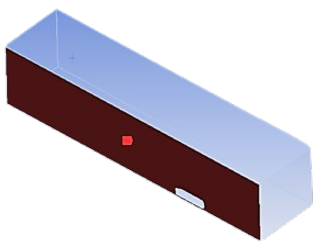


Fig. 8. Symmetrical wall

2.5 Solver Setup

The solution setup for reference model is pressure-based since the air is assumed to be incompressible in this case. Energy equation was on and realizable k-ε model with non-equilibrium wall function were used. Inlet air velocity was set as 40 m/s, with turbulent intensity equal to 1% and turbulence viscosity ratio equal to 10, while outlet turbulence intensity was set to 5% and turbulence viscosity ratio equal to 10. Air density, temperature a viscosity was 1.225 kg/m^3, 288.16 K and 1.7849 x 10^{-5} kg/m^4, respectively, which were default value set by ANSYS. A total of 500 iterations were performed, in which the first 100 iterations used first order upwind for momentum, turbulence kinetic energy and turbulence dissipation rate, and the turbulence viscosity factor was set to 0.8. The remaining 400 iterations used second order upwind for momentum, turbulence kinetic energy and turbulence dissipation rate, and the turbulence viscosity factor was set to 0.95. After setup, hybrid initialization was done before calculations began. The setup was summarized in Table 1.

Table 1
Simulation setup selection

Turbulence model	Realizable k-ε	
Near-wall treatment	Non-equilibrium wall function	
Energy equation	On	
Fluid properties		
Type of fluid	Air	
Density (kg/m^3)	1.225	
Viscosity (kg/ms)	1.7894 x 10^{-5}	
Inlet velocity		
Velocity specification method	Magnitude and direction	
Velocity magnitude (m/s)	40	
Turbulent specification method	Intensity and viscosity ratio	
Turbulence intensity	1%	
Turbulence viscosity ratio	10	
Pressure outlet		
Gauge pressure (Pa)	0	
Backflow direction specification method	Normal to boundary	
Turbulent specification method	Intensity and viscosity ratio	
Backflow turbulence intensity	5%	
Backflow turbulence viscosity ratio	10	
Pressure-velocity coupling scheme	Coupled	
Spatial discretization	Least square cell based	
Pressure	Second order	
Momentum	1st 100 iteration	First order upwind
	Remaining 400 iteration	Second order upwind
Turbulence kinetic energy	1st 100 iteration	First order upwind
	Remaining 400 iteration	Second order upwind
Turbulence dissipation rate	1st 100 iteration	First order upwind
	Remaining 400 iteration	Second order upwind
Energy	Second order upwind	
Explicit relaxation factor		
Pressure	0.25	
Momentum	0.25	
Under-relaxation factors		
Turbulence kinetic energy	0.8	
Turbulence dissipation rate	0.8	
Turbulence viscosity	1st 100 iteration	0.8
	Remaining 400 iteration	0.95

3. Results

3.1 Grid Independence Test (GIT)

The first step in the simulation test is to fix the sensitivity of the model grid. This is necessary to determine the acceptable mesh size. The mesh increment of 50, 000 number of nodes was taken approximately to control and ensure that the difference of increment was constant as possible. For each mesh model, set of data was taken. The GIT result is shown in Figure 9.

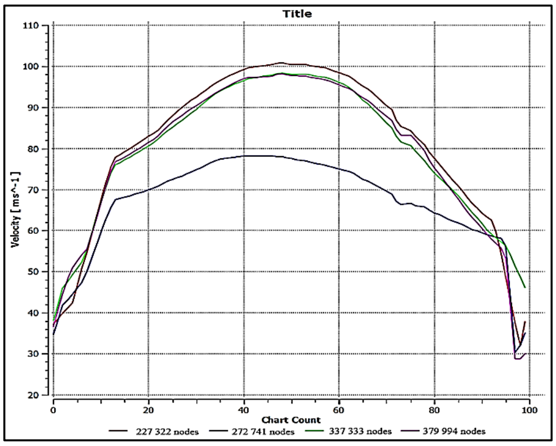


Fig. 9. Results of grid independence test

3.2 Drag and Lift Coefficient for Design 1

The turbulence modelling was performed with the realizable k-ε model using non-equilibrium wall functions. The results of computations for the following cases are presented and discussed; passenger car model with 0°, 10°, 20° and 30° rear-spoiler design. Whenever the air flows through the body, it creates a velocity distribution which results in the aerodynamic loads acting on the moving body of passenger car model. The amount of speed present for the applied speed in different regions surrounding the passenger car model is displays in Figure 10.

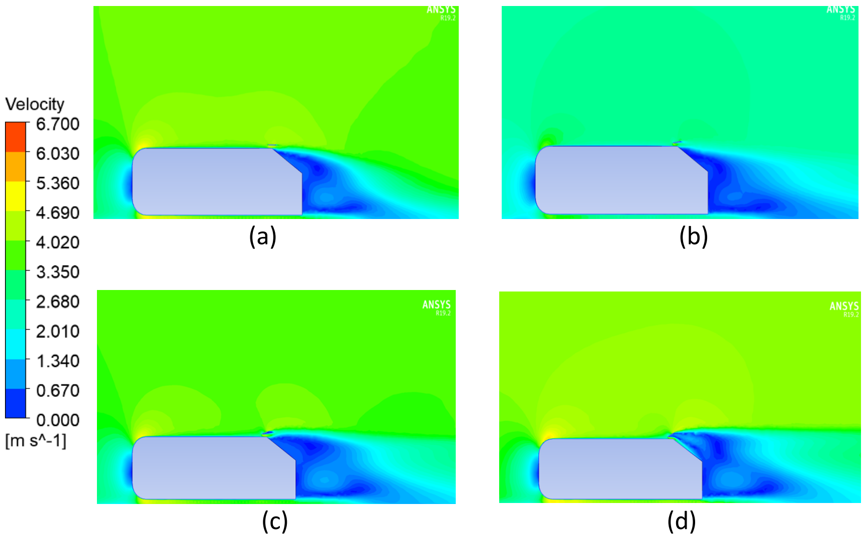


Fig. 10. Velocity distribution of flow in the symmetry plane (a) 0° rear-spoiler design (b) 10° rear-spoiler design (c) 20° rear-spoiler design (d) 30° rear-spoiler design

In addition, the rear-spoiler influences air flow to spoil the direction of velocity to reduce lift. Adding a rear-spoiler could be considered to make the air slope gentler from the roof to the spoiler. At the rear end of the model, recirculating and backflow of air formed, which disturbed the streamline flow of air and created a separated region. Drag increase as the size separated region increase. From the plots of convergence history, the drag coefficient, C_D and lift coefficient, C_L for passenger car model with spoiler design 1 are shown in Table 2. The negative sign indicates negative lift (downward force).

Table 2
Drag and lift coefficient for passenger car model with spoiler design 1

Angle of spoiler (°)	Drag coefficient, C_D	Lift coefficient, C_L
0	0.32900	0.00982
10	0.33937	-0.04471
20	0.34382	-0.09337
30	0.38018	-0.13887

In comparing the drag and lift coefficients of different rear spoiler angles, the aerodynamic effect of using the rear spoiler can clearly be seen. The drag coefficient showed an increase as the angle of rear-spoiler also increases. While the lift coefficient decreases as the angle of rear-spoiler increases. The high drag experienced on addition of the spoiler is a direct backlash from the high amount of downward force produced.

3.3 Drag and Lift Coefficient for Design 2

The results of computations for the following cases are presented and discussed; passenger car model with 0°, 10°, 20°and 30° rear-spoiler design. The air flows through the body, it creates a velocity distribution which results in the aerodynamic loads acting on the moving body of passenger car model. The amount of speed present for the applied speed in different regions surrounding the passenger car model is displays in Figure 11.

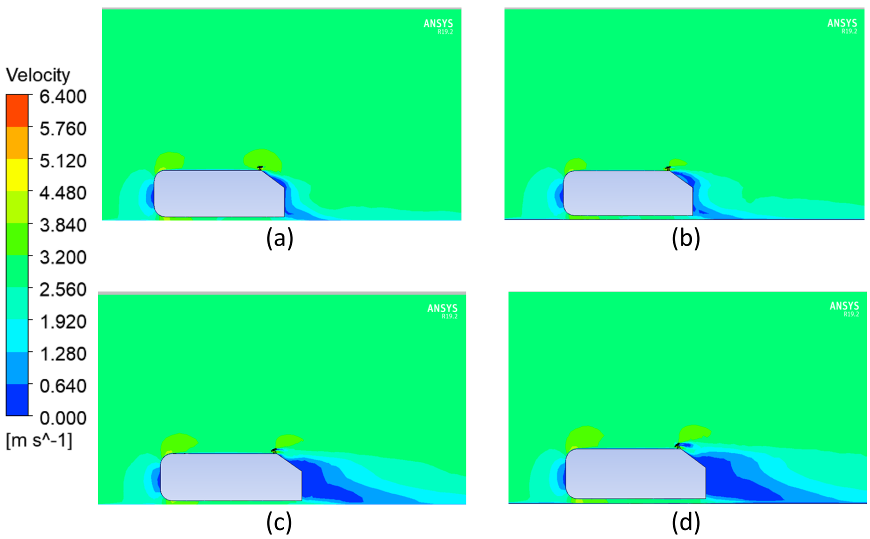


Fig. 11. Velocity distribution of flow in the symmetry plane (a) 0° rear-spoiler design (b) 10° rear-spoiler design (c) 20° rear-spoiler design (d) 30° rear-spoiler design

In addition, the rear-spoiler influences air flow to spoil the direction of velocity to reduce lift. Adding a rear-spoiler could be considered to make the air slope gentler from the roof to the spoiler. At the rear end of the model, recirculating and backflow of air formed, which disturbed the streamline flow of air and created a separated region. Drag increase as the size separated region increase. From the plots of convergence history, the drag coefficient, C_D and lift coefficient, C_L for passenger car model with spoiler design 2 are shown in Table 3. The negative sign indicates negative lift (downward force).

Table 3
Drag and lift coefficient for passenger car model with spoiler design 2

Angle of spoiler (°)	Drag coefficient, C_D	Lift coefficient, C_L
0	0.41382	0.22993
10	0.42507	0.19457
20	0.36186	-0.027678
30	0.36752	-0.014027

4. Conclusions

This study investigated the effect of rear spoiler and the influence of various angular spoiler towards drag coefficient C_d and lift coefficient, C_l on the aerodynamics of passenger car. Two designs of spoiler were tested with various angles of 0°, 10°,20°, 30° for each of the spoiler design. A model without a spoiler has also been tested, and the data served as the baseline. The spoiler angle was found to have a positive influence on the aerodynamic performance of the passenger car model either in different angle. Observation can be concluded that at a particular spoiler height the spoiler that possess smaller angle of wind collision gives lower lift force. This is due to the fact that with smaller angle of wind collision, the spoiler would create smaller recirculation zone behind the rear end of the running vehicle. This implies to higher pressure behind spoiler but lower pressure behind the rear end of the vehicle. Rear spoilers redirect the airflow behind the vehicle & increase the negative lift of the vehicle**.**

Acknowledgement
This research was supported by the Ministry of Higher Education of Malaysia through the Fundamental Research Garat Scheme (FRGS/1/2024/TK10/UTHM/02/6) and through MDR grant (Q686).

References
[1] Deng, Zhaowen, Sijia Yu, Wei Gao, Qiang Yi, and Wei Yu. "Review of effects the rear spoiler aerodynamic analysis on ground vehicle performance." In *Journal of Physics: Conference Series*, 1600, no. 1, p. 012027. IOP Publishing, 2020. https://doi.org/10.1088/1742-6596/1600/1/012027
[2] Nath, Devang S., Prashant Chandra Pujari, Amit Jain, and Vikas Rastogi. "Drag reduction by application of aerodynamic devices in a race car." *Advances in Aerodynamics* 3 (2021): 1-20. https://doi.org/10.1186/s42774-020-00054-7
[3] Cheng, See-Yuan, Kwang-Yhee Chin, and Shuhaimi Mansor. "Experimental study of yaw angle effect on the aerodynamic characteristics of a road vehicle fitted with a rear spoiler." *Journal of Wind Engineering and Industrial Aerodynamics* 184 (2019): 305-312. https://doi.org/10.1016/j.jweia.2018.11.033
[4] Mukut, ANM Mominul Islam, and Mohammad Zoynal Abedin. "Review on aerodynamic drag reduction of vehicles." *International Journal of Engineering Materials and Manufacture* 4, no. 1 (2019): 1-14. https://doi.org/10.26776/ijemm.04.01.2019.01

[5] Ramli, Muhammad Safwan Asyraf, Shamsul Anuar Shamsudin, Zairulazha Zainal, Norasra A. Rahman, and Zulkhairi Zainol Abidin. "Airfoil performance of an active car spoiler." In *International Conference and Exhibition on Sustainable Energy and Advanced Materials*, p. 250-253. Singapore: Springer Nature Singapore, 2021. https://doi.org/10.1007/978-981-19-3179-6_45

[6] Maji, Daniel Syafiq Baharol, and Norrizal Mustaffa. "CFD Analysis of Rear-Spoilers Effectiveness on Sedan Vehicle in Compliance with Malaysia National Speed Limit." *Journal of Automotive Powertrain and Transportation Technology* 2, no. 1 (2022): 26-36. https://doi.org/10.30880/japtt.2022.02.01.003

[7] Kurec, Krzysztof, and Janusz Piechna. "Influence of side spoilers on the aerodynamic properties of a sports car." *Energies* 12, no. 24 (2019): 4697. https://doi.org/10.3390/en12244697

[8] Paul, Akshoy Ranjan, Anuj Jain, and Firoz Alam. "Drag reduction of a passenger car using flow control techniques." *International Journal of Automotive Technology* 20 (2019): 397-410. https://doi.org/10.1007/s12239-019-0039-2

[9] Datta, Basudev, Vaibhav Goel, Shivam Garg, and Inderpreet Singh. "Study of Various Passive Drag Reduction Techniques on External Vehicle Aerodynamics Performance: CFD Based Approach." *International Research Journal of Engineering and Technology (IRJET)* 6, no. 05 (2019): 1851-1871.

[10] Arulshri, K. P., S. Selva Kumar, R. Nesalingam, and Senthi Kumar. "Cfd analysis of automobile rear dynamic spoiler." *International Journal of Aquatic Science* 12, no. 03 (2021): 2.

[11] Singh, Pawan, Vibhanshu Chhettri, and Nitin Kumar Gupta. "Computational Analysis of Aerodynamics Characteristics of High-Speed Moving Vehicle." In *Characterization, Testing, Measurement, and Metrology*, pp. 125-137. CRC Press, 2020. https://doi.org/10.1201/9780429298073-8

[12] Rashid, Razlin Abd, Mohamad Muhaimeen Mohd Hazman, Izuan Amin Ishak, Nor Afzanizam Samiran, Nik Normunira Mat Hassan, and Zuliazura Salleh. "Study on aerodynamic drag effect of a rear spoiler on a passenger car using CFD." In *AIP Conference Proceedings*, 2530, no. 1. AIP Publishing, 2023. https://doi.org/10.1063/5.0120955

[13] Piechna, Janusz. "A review of active aerodynamic systems for road vehicles." *Energies* 14, no. 23 (2021): 7887. https://doi.org/10.3390/en14237887

[14] Palanivendhan, M., J. Chandradass, C. Saravanan, Jennifer Philip, and R. Sharan. "Reduction in aerodynamic drag acting on a commercial vehicle by using a dimpled surface." *Materials Today: Proceedings* 45 (2021): 7072-7078. https://doi.org/10.1016/j.matpr.2021.01.884

[15] Brandt, Adam, Henrik Berg, Michael Bolzon, and Linda Josefsson. "The effects of wheel design on the aerodynamic drag of passenger vehicles." *SAE International Journal of Advances and Current Practices in Mobility* 1, no. 2019-01-0662 (2019): 1279-1299. https://doi.org/10.4271/2019-01-0662

[16] Stojanovic, Nadica, Oday I. Abdullah, Ivan Grujic, Jasna Glisovic, and Ali Belhocine. "The influence of spoiler on the aerodynamic performances and longitudinal stability of the passenger car under high speed condition." *Journal of Visualization* 26, no. 1 (2023): 97-112. https://doi.org/10.1007/s12650-022-00867-2

[17] Tomar, Akhilesh Singh, Anuj Prajapati, Anuj Sharma, and Shubham Shrivastava. "CFD analysis on the aerodynamic effects of spoiler at different angle on car body." *International Journal of Innovative Technology and Exploring Engineering (IJITEE)* 8, no. 7 (2019): 2845-2848.

[18] Soares, Renan Francisco, Andrew Knowles, Sergio Goñalons Olives, Kevin Garry, and Jennifer Holt. *On the aerodynamics of an enclosed-wheel racing car: an assessment and proposal of add-on devices for a fourth, high-performance configuration of the DrivAer model*. No. 2018-01-0725. SAE Technical Paper, 2018. https://doi.org/10.4271/2018-01-0725

[19] Sharma, Sagar, Saurabh Banga, Rohit Singh Dungriyal, Mohammad Zunaid, Naushad Ahmad Ansari, and Suresh Lal. "CFD analysis and optimization of geometrical modifications of Ahmed body." *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN* (2015): 2278-1684.

[20] Smith, W. Spencer, D. Adam Lazzarato, and Jacques Carette. "State of the practice for mesh generation and mesh processing software." *Advances in Engineering Software* 100 (2016): 53-71. https://doi.org/10.1016/j.advengsoft.2016.06.008