JCF

Journal of Complex Flow, Vol. 6 No. 2 (2024) 21-29

Journal of Complex Flow

Journal homepage: www.fazpublishing.com/jcf

e-ISSN: 2672-7374

Impact of Computational Parameter on Aerodynamics Forces on the Sedan Car Model

Azlan Adam[1,*], Wan Mahafiz Rosni[1], Bukhari Manshoor[2], Arif Mohd Fauzi[2], Ammar Daniel Zulkanain[2], Muhammad Irfan Hishamuddin[2], Lee Ngan Koeng[2], Syed Ahmad Farhan Nabil[2], Muhammed Abdelfattah Sayed Abdelaal[3]

[1] Department of Mechanical Engineering,
Polytechnic Kuching, 93050 Kuching Sarawak, MALAYSIA
[2] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia,
Parit Raja, Batu Pahat, 86400, MALAYSIA
[3] Engineers House Academy,
Manshiet El Bakry Street, Qesm Heliopolis, EGYPT

*Corresponding Author

Received 24 July 2024;
Accepted 13 Oct 2024;
Available online 15 Dec 2024

Abstract: A spoiler is an aerodynamic component designed to reduce drag in automobiles. The main purpose of a rear car spoiler is to enhance the vehicle's grip on the road by reducing aerodynamic drag and increasing stability. Positioned at the rear, it creates a high-pressure zone to counteract the low pressure on the trunk, thereby improving stability. This study aims to investigate the effects of rear spoilers on aerodynamic drag and stability, considering the Malaysian National Speed Limit. Both the sedan vehicle model and the rear spoiler models were created using Computer-Aided Design (CAD) software, SolidWorks and the data was analyzed using Computational Fluid Dynamics (CFD) software to calculate drag and lift forces at speeds of 60 km/h, 90 km/h, and 110 km/h. Certain limits were provided by the design's intricacy. Together with the sedan car, the simulations featured two different rear spoiler designs: the rear wing and the ducktail spoiler. The rear wing significantly increased drag and downforce, but the ducktail spoiler only slightly increased both, according to the results. Rear spoilers also boost downforce. In addition, cars that moved more slowly than those that moved quicker showed more drag. In conclusion, spoilers only help at very high speeds; at lower speeds, they create drag. Spoilers are useful under the Malaysian National Speed restriction, especially on motorways with a 110 km/h speed restriction. The results indicated that a sedan car without a spoiler had a drag coefficient, C_d of 0.5 and lift coefficient, C_l of -1.5. In contrast, a sedan car with a spoiler had a C_d of 0.2 and C_l of -0.2.

Keywords: Aerodynamics, Computational Fluid Dynamics (CFD), Drag coefficient, Lift coefficient

1. Introduction

Aerodynamic performance of vehicles has always been a priority for automobile design experts. The vehicle's aerodynamic efficiency is mostly determined by the drag and lift forces acting on it at maximum speed as it passes through the air. These phenomena appear to negatively affect the performance, handling, stability and fuel efficiency of the vehicle [1]. The design of external aerodynamic car accessories, which may be mounted outside the vehicle, is used to improve aerodynamic efficiency and assist handling and fuel economy on vehicles. For this reason, additional accessories and external design structures are added to the body of the vehicle or car, such as the Rear Wings/Spoilers, Lower Front and Rear Bumpers, Air Dams and plenty more aerodynamic accessories. [2]

These days, cars move so quickly that unintentional accidents are possible. This highlights the need for aerodynamic innovations, including a spoiler that effectively produces lift reduction by generating a controlled stall over the rear wing part. Spoilers are designed to significantly reduce lift and increase drag. In addition to lowering drag and rear axle lift, a rear spoiler can reduce dirt accumulation on the back surface. It is estimated from past studies that aerodynamic drag is the dominant form of resistance when vehicles run at speeds of 80 km/h or more, especially given that 65 per cent of the

*Corresponding author: azlan@poliku.edu.my

power required at 110 km/h is consumed because of overcoming aerodynamic drag [3].

Research in aerodynamics is essential for addressing challenges such as reducing wind noise and drag as vehicle speeds increase. Improved aerodynamic designs directly impact fuel efficiency, making them indispensable in the face of rising fuel costs [4]. Factors such as aerodynamic qualities, vehicle mass, and engine efficiency are carefully assessed when considering vehicle design. Achieving low air drag becomes a prerequisite for minimizing fuel consumption [5].

All drivers operating vehicles on Malaysian motorways, federal, state, and local highways must abide by the Had Laju Kebangsaan (National Speed Limits) restrictions. The National Speed Limits had been enforced on 1 February 1989 following the 1989 National Speed Limits [6]. Although the typical speed limit on motorways is 110 km/h, it may be reduced to 80 or 90 km/h in hazardous hilly terrain, crosswind zones, and congested metropolitan areas. State and federal roadways have a posted speed restriction of 90 km/h; however, during holidays, this may be lowered. In order to estimate the efficacy of an automobile's rear spoiler using (CFD) simulation, these speed constraints will be crucial considerations in this study.

In the study of vehicle aerodynamics, the coefficients of lift and drag are significant factors. The drag coefficient C_d is a dimensionless variable that quantifies an object's resistance to drag in a fluid environment, such the air. The fluid's density, velocity, and the form, size, and roughness of its surface all have an impact on it. A design that is more aerodynamic and has a lower drag coefficient will accelerate and need less fuel. Minimizing surface drag is important in aerodynamic design to improve vehicle performance, achieved through shaping, smoothing, and employing surface treatments [7]

On the other hand, the lift coefficient C_l is a dimensionless quantity that quantifies the lift force produced by an item within a fluid flow. In the case of cars, lowering the lift coefficient is necessary to keep them stable at high speeds. To counteract the weight of the vehicle and give stability and control, a lift must be produced [8]. Driving circumstances can become hazardous due to high lift, since a vehicle may lose touch with the road. In order to guarantee that the car stays securely planted even at high speeds, rear spoilers are made to reduce the lift coefficient.

With the use of mathematical formulae and flow-based simulation, the effects of airflow over the spoilers may be determined. CFD provides a more detailed and accurate solution compared to theoretical calculations [9]. By calculating using the continuity, energy, and momentum equations, computational approaches provide more accurate predictions faster.

In aerodynamic conditions, separated flow and turbulent body flow can both have a substantial effect on an object's or vehicle's performance. Skin friction drag is increased by turbulent body flow, hence strict design considerations are required to minimize drag and maximize efficiency. On the other hand, divided flow results in increased drag and flow separation pressure, which reduces the vehicle's overall performance and stability. Aerodynamic design techniques such as streamlining, modifying surface contours, and incorporating flow control devices like spoilers or vortex generators are employed to manage both turbulent body flow and flow separation, aiming to improve vehicle performance and control the aerodynamic forces acting on the object [16].

A car's aerodynamic efficiency is greatly enhanced by streamlined flow since the vehicle body is covered in many types of fluid. Reduced drag improves the vehicle's overall handling, stability, peak speed, and fuel efficiency. Automobile manufacturers invest a great deal of time and energy into computational fluid dynamics (CFD) models and wind tunnel testing to ensure optimized flow and increase aerodynamic efficiency, which enhances overall performance on the road.

The overall aerodynamics of a vehicle can be improved by adding aerodynamic components like air dams, deflectors, or carefully sculpted body forms to help reduce stagnation zones. Consequently, to optimize the efficiency of devices or objects, separation bubbles in aerodynamics must be managed. Lessening separation bubbles can be achieved by design strategies such as curving the shape of the object, using flow control surfaces or vortex generators, or flattening surface characteristics. Aerodynamic effectiveness and vehicle performance can be enhanced by minimizing or eliminating flow separation and the resulting separation bubbles [17].

When the airflow around an object becomes turbulent, the fluid particles no longer flow smoothly in parallel layers [18], [19] Rather, they create extremely erratic flow patterns as a result of their random mixing and swirling. Turbulence is often associated with high Reynolds numbers because they indicate that inertial forces prevail over viscous forces in the fluid. Fluid particles revolving around a central axis produce the swirling flow patterns called vortices. These swirling motion regions are Journal of Sustainable and Environment, and they are characterized by a spinning motion or circular flow pattern inside a fluid. As air flows past the surface of an item, it separates from the flow. In the wake, swirling vortices occur because of the fluid's separation and reattachment, which produces high and low-pressure zones [20].

The objective of this study is to investigate the effects of rear spoilers on aerodynamic drag and stability, considering the Malaysian National Speed Limit. Both the sedan vehicle model with and without the rear spoiler models were created using Computer-Aided Design (CAD) software, and the data was analyzed using Computational Fluid Dynamics (CFD) software to calculate drag and lift forces at speeds of 60 km/h, 90 km/h, and 110 km/h. The study includes two rear spoiler designs, the ducktail spoiler and the rear wing, to determine their impact on the aerodynamic performance of the sedan vehicle.

2. Methodology and Simulation Setup

In this study, we utilized CFD software to determine the drag and lift coefficients for a sedan car. A common CFD code may be used to analyze the effects of lighting angles on drag coefficients in three steps. The methodology involved creating a detailed mesh of the car model, setting appropriate boundary conditions, and running simulations to obtain the aerodynamic coefficients. Additionally, a validation test is conducted to compare our results with those obtained using ANSYS methodology, ensuring the accuracy and reliability of our findings. This approach allowed for a comprehensive analysis of the vehicle's aerodynamic performance. Here, the user would provide the simulation's beginning conditions, boundary conditions, and fluid attributes [12-14].

2.1 Governing Equation

Drag (D) describes the resultant force acting in the direction of the upstream velocity, while lift (L) refers to the resultant force perpendicular to the upstream velocity. The resultant forces from these contributions can be broken down into three components: the moment coefficient, the drag coefficient, and the lift coefficient. The aerodynamic drag force can be calculated using the following formula:

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

The lift force acts vertically on the vehicle body. When applied in the positive direction, this force causes the vehicle to lift off the ground. Conversely, if applied in the negative direction, it can result in excessive downward force on the wheels. The formula typically used to describe this force show in Eqn. 2 below. The aerodynamic forces acting on the vehicle are quantified through the drag and lift coefficients. These coefficients are non-dimensional numbers that describe the drag and lift forces relative to the dynamic pressure and reference area.

$$C_l = \frac{F}{\frac{1}{2}\rho V^2 A} \tag{2}$$

The continuity equation ensures mass conservation in a fluid flow and is expressed as in Eqn. 3 while for incompressible flows, where the density ρ is constant, the equation was simplified as in Eqn. 4.

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{3}$$

$$\nabla \,.\, \boldsymbol{u} = 0 \tag{4}$$

The Navier-Stokes equations describe the motion of viscous fluid substances and are given by Eqn. 5 and the momentum equation was written as in Eqn. 6 and 7 below.

$$\rho\left(\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u}\,.\nabla)\boldsymbol{u}\right) = -\nabla p + \mu\nabla^2 \tag{5}$$

x-momentum:

$$\rho u\frac{\partial u}{\partial x} + \rho v\frac{\partial u}{\partial y} = -\frac{\partial \hat{p}}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{6}$$

y-momentum:

$$\rho u\frac{\partial v}{\partial x} + \rho v\frac{\partial v}{\partial y} = -\frac{\partial \hat{p}}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{7}$$

Realizable k-ε turbulent model was used and the equation given as in Eqn. 8.

$$\begin{aligned}\frac{\partial}{\partial j}\left(\rho\varepsilon u_j\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 S\varepsilon \\ + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b \\ - C_2\rho\frac{\varepsilon^2}{k+\sqrt{\varepsilon v}} + S_\varepsilon\end{aligned} \tag{8}$$

$$C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09, \\ \sigma_k = 1.0,\ \ \sigma_k = 1.3$$

2.2 Spoiler and CFD Software

A spoiler is an automotive device primarily used to disrupt unfavorable air movement over a moving vehicle, typically referred to as turbulence or drag. While spoilers come in various styles and designs, many drivers install them for additional benefits such as reducing drag force, decreasing lift, and enhancing traction control. The main advantages of adding a spoiler to a car include maintaining traction, increasing fuel efficiency, improving visibility, reducing weight, and enhancing braking stability. At high speeds, cars tend to generate lift; spoilers can create downforce to counteract this lift, thereby providing greater stability.

Computational Fluid Dynamics (CFD) is a set of numerical methods used to approximate solutions to problems involving fluid dynamics and heat transfer. It is a cost-effective way to obtain discrete solutions for real-world fluid problems, achieved at specific points in space and time intervals. This computational analysis is performed using high-speed computers and software like ANSYS Fluent, allowing for virtual simulation-based design and analysis instead of the traditional build-and-test approach [10].

2.3 Geometrical Modelling

The geometry model design for the simulation will be a typical sedan car concept created using CAD (Computer-Aided Design) software, SolidWorks. Both the car and the spoiler will be designed in SolidWorks and analyzed using CFD software, ANSYS Fluent. The dimensions for the sedan car will be based on the Proton S70 model. The original design of the car underwent many changes to become a more common model. Certain parts of the model were left out to simplify the computation and cut expenses. Exhaust, brake, side mirrors, and other tiny pieces are among the items excluded from this list.

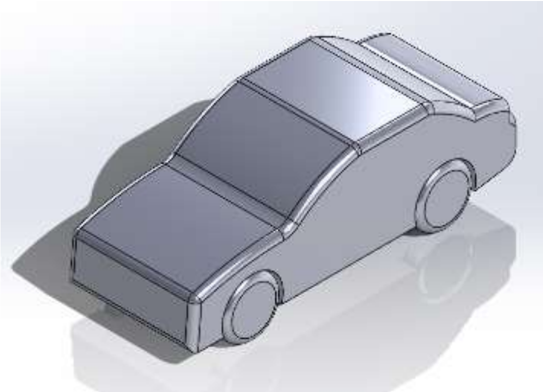


Fig. 1 - The sedan car model without spoiler

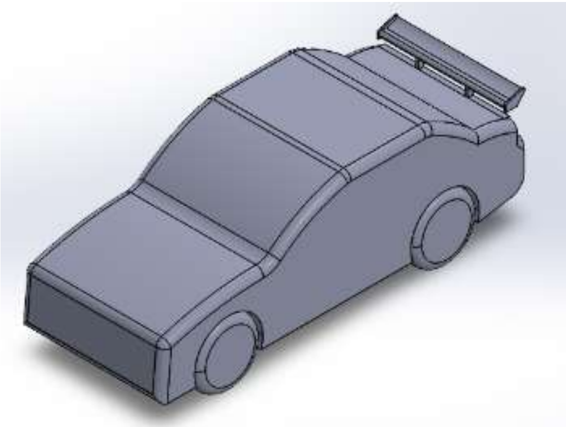


Fig. 2 - The sedan car model with spoiler

Table 1: Design Parameter of the Sedan Vehicle

Vehicle Part	Dimension (mm)
Wheelbase	2627
Length	4602
Width	1809
Height	1466

2.4 Mesh Generation

Once the model was completed, the process for meshing was initiated before the simulation will perform. The meshing process was used a default setting in AnsysFluent and the results for meshing for both models shows in Figure 3 and 4 below.

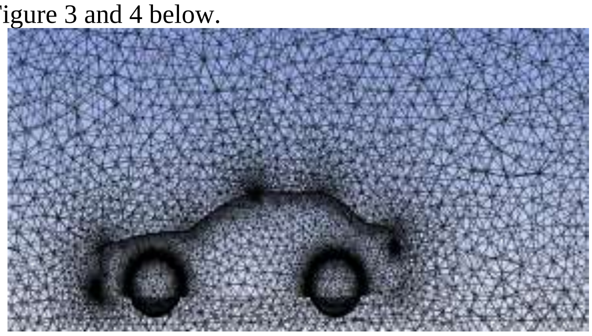


Fig. 3 - Meshing of the sedan car without spoiler

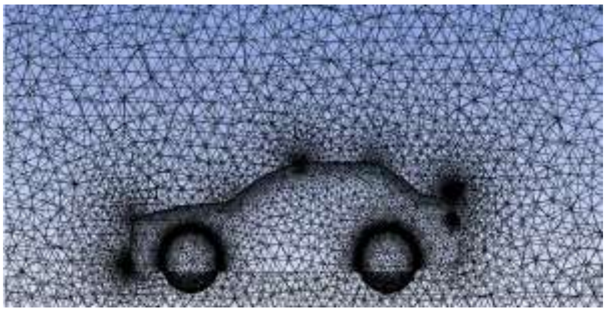


Fig. 4 - Meshing of the sedan car with spoiler

2.5 Boundary Condition

In boundary conditions, the setting was particularly defined as shown in Table 2, to achieve precise simulation outcomes. The inlet boundary condition involved adjusting the velocity inlet across three cases which are 60km/h, 90 km/h, and 110 km/h. The outlet boundary condition was maintained at the default pressure setting. To accurately simulate the interaction with the road, the wall boundary condition was differentiated beneath the car into two specific settings to represent the road's movement relative to the vehicle. This detailed approach to boundary conditions was crucial for capturing the aerodynamic behavior and performance of the sedan model under varying operational scenarios.

Table 2: Boundary Conditions

Boundary	Type	Detail
Inlet	Velocity Inlet	60, 90, 110 km/h
	Turbulent Intensity	5%
	Turbulent Viscosity Ratio	10
Outlet	Pressure	Default
Road	Wall Motion	Moving
	Motion	Translational
Wall	Shear Condition	Specified Shear
Car Body	Stationary Wall	No Slip Shear Condition

2.6 Validation

The numerical simulation used in this work was initially validated against a prior investigation by Ahmed [11]. The simulation model employed in Hassan's study is compared with the outcomes from two distinct turbulence models as part of the validation procedure. The simulation and Ahmed model are the chosen turbulence models for validation as shown in Table 3 below. The comparison between the results by Ahmed and the current simulated was shows in Figure 5. From the results, its shows that the good agreement between the based results from the previous study and the current simulation. Hence, the rest of simulation was used the same setting as the used for the validation.

Table 3: Validation Result

Angle	C_d Ahmed	C_d	Error (%)
0	0.093	0.098	5.37
5	0.063	0.061	3.17
12.5	0.075	0.077	2.60
20	0.066	0.068	3.03
30	0.0377	0.0405	7.43
40	0.044	0.04	9.09
Average	0.063	0.064	5.12

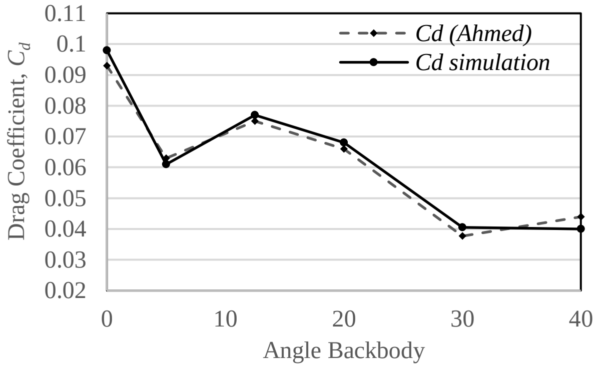


Fig. 5 - Validation Graph

3. Result and Discussion

There are three main parameters that interesting to discuss here from the simulation results. The first parameter was a lift and drag coefficient for both models simulated. It is the main objective here to determine the different of lift and drag coefficient for case of the car with and without the spoiler. The second part of discussion regarding the results were the velocity contour and the streamlines over the model. The detail discussion was in the next following section.

3.1 Lift and Drag Coefficient

Figure 6 shows how the drag coefficient varies with velocity for a vehicle both with and without a spoiler, over speeds ranging from 60 km/h to 110 km/h. The results indicate that, without a spoiler, the drag coefficient remains steady at around 0.5 across all tested speeds, meaning the aerodynamic resistance stays constant regardless of speed. However, when a spoiler is added, the drag coefficient consistently drops to about 0.2, showing a significant reduction in aerodynamic drag. Spoilers are designed to alter airflow to reduce drag and sometimes to increase downforce. The data clearly demonstrate that a vehicle with a spoiler has a much lower drag coefficient compared to one without, emphasizing the spoiler's role in boosting aerodynamic efficiency.

The drop from a drag coefficient of 0.5 to 0.2 is considerable, highlighting the spoiler's effectiveness. Interestingly, for both setups which is with and without a spoiler, the drag coefficient doesn't change with speed, suggesting that the vehicle's aerodynamic properties remain stable within the tested speed range. This is typical for many vehicles in a certain speed range where the airflow conditions are similar.

In practical terms, a lower drag coefficient means less aerodynamic resistance, which translates to better fuel efficiency and performance. Vehicles with lower drag require less energy to maintain a given speed, resulting in better fuel economy. For performance cars, a lower drag coefficient can also mean higher top speeds and improved handling due to more efficient airflow.

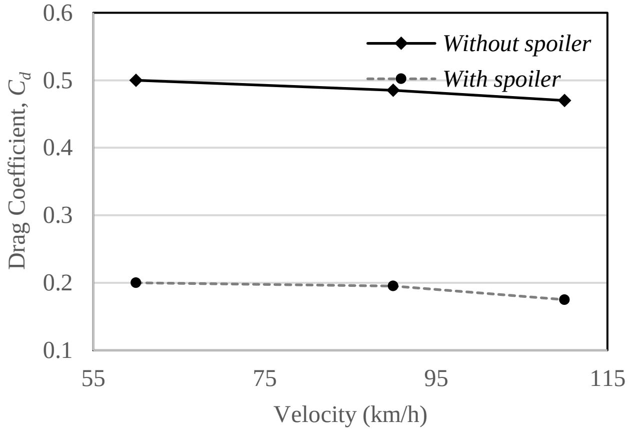


Fig. 6 - Graph of Drag Coefficient vs Velocity

Figure 7 depicts the relationship between the lift coefficient and velocity for a vehicle with and without a spoiler, covering speeds from 60 km/h to 110 km/h. It shows that the lift coefficient for the vehicle without a spoiler is consistently around -1.5 across all speeds. This negative lift coefficient indicates significant downforce, which enhances stability and traction. In contrast, the vehicle with a spoiler has a lift coefficient close to zero, indicating that the spoiler effectively neutralizes the lift force, eliminating both lift and downforce. Spoilers are designed to manage airflow and modify the aerodynamic forces acting on a vehicle. The data shows that while the vehicle without a spoiler experiences consistent downforce, the vehicle with a spoiler maintains a nearly neutral lift coefficient. This demonstrates that the spoiler is effective in counteracting downforce.

Both configurations exhibit a constant lift coefficient across the velocity range, suggesting that the aerodynamic properties of the vehicle remain stable with speed within this range. This stability is typical for vehicles within certain speed ranges, where airflow conditions are relatively consistent. Practically speaking, a vehicle with downforce (negative lift coefficient) has better grip and stability, which is crucial for handling and safety at high speeds. Conversely, a neutral lift coefficient, as seen with the spoiler, can reduce drag and improve fuel efficiency, although it may slightly compromise traction. Therefore, whether to use a spoiler or not depends on whether the priority is stability or aerodynamic efficiency.

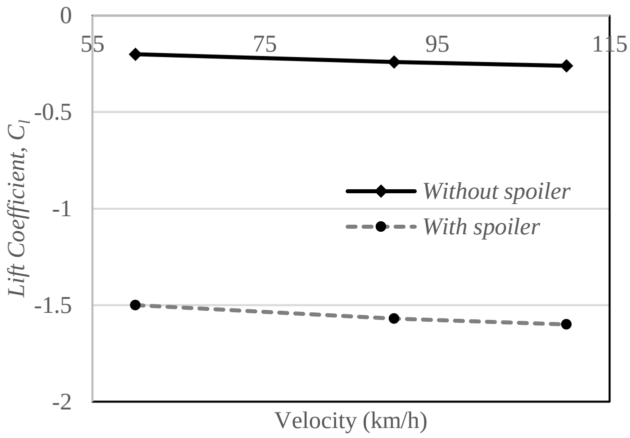


Fig. 7 - Graph of Lift Coefficient vs Velocity

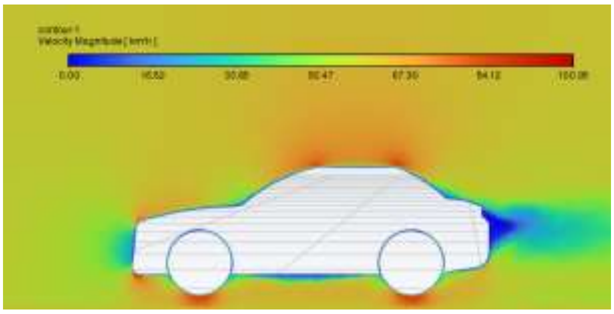
3.2 Velocity Contour

The velocity contour analysis was conducted by examining data from the post-processing phase along the *x*-axis at a position of 0.7 meters. This method provides a detailed view of the velocity distribution at this specific point, offering key insights into the vehicle's aerodynamic performance. Velocity contours, which graphically represent how airflow speed varies across different areas, help identify regions of high and low velocity, as well as zones of turbulence and separation. These are crucial for understanding the aerodynamic forces acting on the vehicle. Focusing on velocity contours at the 0.7-meter mark along the *x*-axis is particularly important because it reveals the impact of design changes, such as adding a spoiler, on the flow field. This precise location helps to understand how airflow interacts with different parts of the vehicle, especially the rear end where changes in lift and drag are most significant.

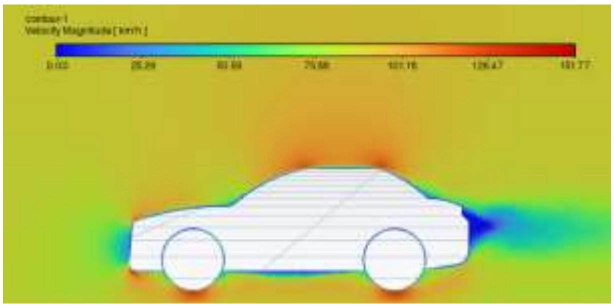
The data from the velocity contours provides a comprehensive picture of the aerodynamic environment, pinpointing areas of optimal airflow and spots that may require design improvements. This information is vital for optimizing vehicle design to achieve better aerodynamic efficiency, improved stability, and enhanced overall performance. Thorough analysis is essential in the iterative design and testing process, ensuring each modification leads to tangible improvements in vehicle dynamics.

Figure 8 and 9 shows the velocity contours around a vehicle at different speeds which are 60 km/h, 90 km/h, and 110 km/h for both with and without a spoiler. These contour maps illustrate how air flows around the vehicle, highlighting areas with varying speeds. For the vehicle without a spoiler, the airflow appears mostly uniform along the body, but there is noticeable turbulence and separation at the rear, especially at higher speeds. At 60 km/h, there is moderate airflow separation behind the vehicle. As the speed increases to 90 km/h and 110 km/h, the separation zones grow, indicating more turbulence and drag, which reduces the vehicle's aerodynamic efficiency. With a spoiler, the airflow pattern changes significantly.

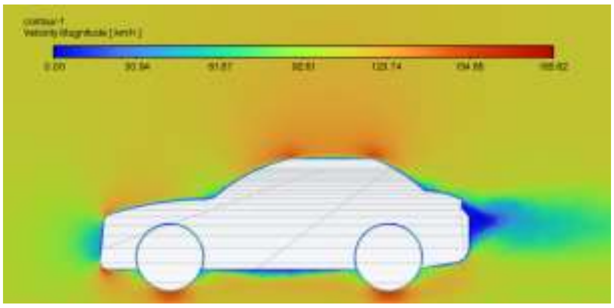
The spoiler helps smooth out the airflow and reduces the size of the separation zones at the rear. At 60 km/h, the airflow stays more attached to the vehicle's surface, with less turbulence compared to the no-spoiler scenario. As speeds increase to 90 km/h and 110 km/h, the benefits of the spoiler become clearer, showing reduced separation and a more streamlined airflow pattern. This decrease in turbulence and drag boosts the vehicle's aerodynamic performance.



60 km/h

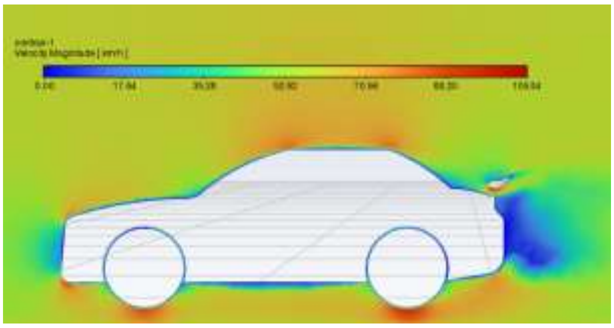


90 km/h



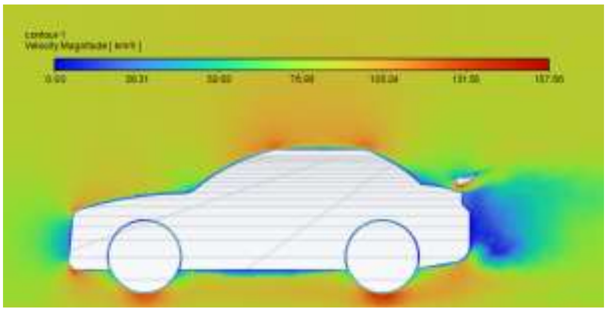
110 km/h

Fig. 8 - Velocity Contour without Spoiler

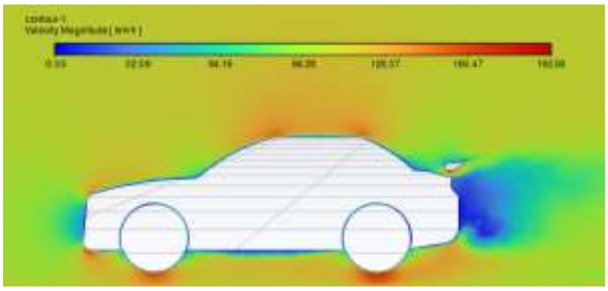


60 km/h

Published by FAZ Publishing
http://www.fazpublishing.com/jcf



90 km/h



110 km/h

Fig. 9 - Velocity Contour with Spoiler

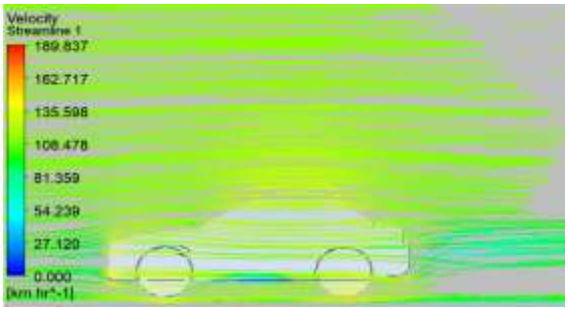
In summary, comparing the velocity contours with and without a spoiler demonstrates the spoiler's effectiveness in managing airflow, reducing turbulence, and enhancing aerodynamic efficiency across different speeds. This analysis highlights the critical role of aerodynamic design elements, like spoilers, in optimizing vehicle performance.

3.3 Streamline

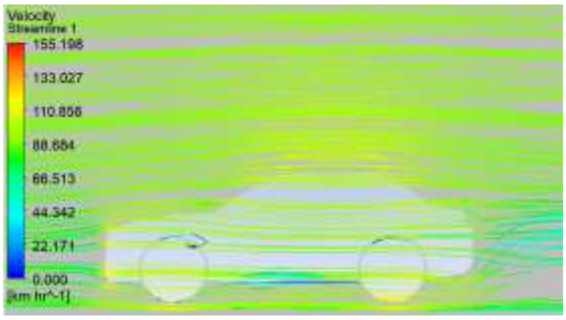
The velocity streamline analysis was carried out by extracting data from the post-processing phase at a specific point along the X-axis, exactly 0.7 meters. This method allows for an in-depth look at the airflow characteristics and behavior at this precise location, offering valuable insights into the vehicle's aerodynamic performance. By examining the streamlines, which trace the paths of fluid particles, engineers can visualize the flow patterns around the vehicle, identifying high-velocity areas, turbulence, and separation zones. Focusing on the streamlines at the 0.7-meter mark along the X-axis is particularly important as it helps reveal the impact of modifications like adding a spoiler on the flow field. This specific location provides a clear understanding of how the airflow interacts with different parts of the vehicle, particularly the rear end, where changes in lift and drag forces are crucial.

The streamline data collected offers a detailed view of the aerodynamic environment, showing smooth airflow regions and areas where turbulence occurs. This information is critical for optimizing vehicle design, aiming for better aerodynamic efficiency, enhanced stability, and overall improved performance. Such a thorough analysis is vital in the iterative design and testing process, ensuring that each modification leads to real improvements in vehicle dynamics.

60 km/h

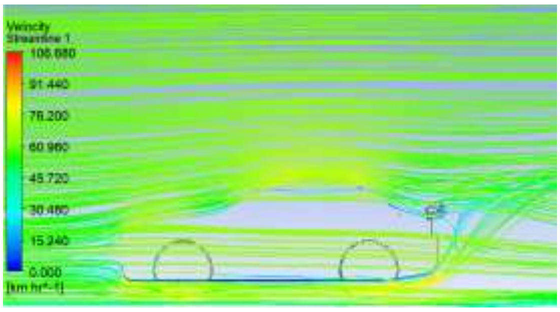


90 km/h

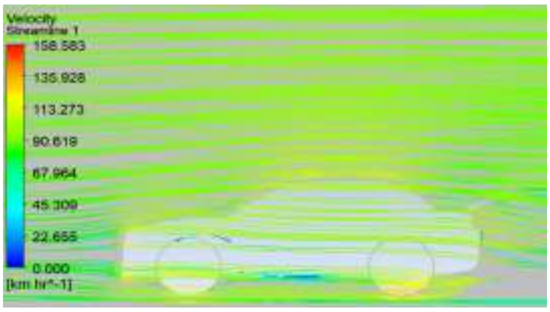


110 km/h

Fig. 10 - Streamline without Spoiler

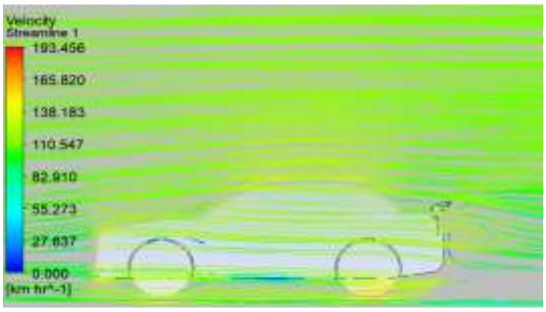


60 km/h



90 km/h

Published by FAZ Publishing
http://www.fazpublishing.com/jcf



110 km/h

Fig. 11 - Streamline with Spoiler

Figure 10 and 11 shows the velocity streamlines around a vehicle at different speeds which are 60 km/h, 90 km/h, and 110 km/h for both with and without a spoiler. These streamlines visually represent how air moves around the vehicle, highlighting areas of smooth flow and turbulence. Without a spoiler, the airflow appears relatively smooth along the vehicle's body at 60 km/h. However, as the speed increases to 90 km/h and 110 km/h, significant turbulence and separation occur behind the vehicle. This results in increased drag and decreases aerodynamic efficiency, as the airflow becomes more chaotic and less streamlined. With a spoiler, the airflow pattern changes. At 60 km/h, the airflow remains attached to the vehicle's surface for longer, with less turbulence compared to the no-spoiler scenario. As the speed increases to 90 km/h and 110 km/h, the benefits of the spoiler become clearer. The streamlines are smoother, and the separation zones are reduced, leading to less turbulence and drag.

In summary, comparing the velocity streamlines with and without a spoiler highlights the spoiler's effectiveness in managing airflow. The spoiler helps maintain a more streamlined flow, reducing turbulence and drag, which enhances the vehicle's aerodynamic performance at various speeds. This analysis emphasizes the critical role of aerodynamic design elements like spoilers in improving vehicle efficiency and stability.

4. Conclusion

This study successfully employed Computational Fluid Dynamics (CFD) simulations to investigate the aerodynamic characteristics of the Proton car, focusing on drag and lift coefficients and velocity distribution around the vehicle. By utilizing ANSYS Workbench and ANSYS Fluent, the key findings highlight that aerodynamic design significantly influences vehicle efficiency and stability. Through detailed simulations, the study identified potential design optimizations that could reduce drag and improve lift characteristics. These enhancements are crucial for developing more energy-efficient and high-performance vehicles.

Furthermore, the validation of the simulation results against previous studies confirmed the reliability and accuracy of the CFD methods used. This research underscores the importance of aerodynamic design in automotive engineering and demonstrates the effectiveness of CFD simulations in achieving optimal aerodynamic efficiency.

Overall, the study lays a solid foundation for future advancements in automobile aerodynamics, promoting the creation of vehicles with better performance and fuel efficiency.

Acknowledgement

The authors would like to express their gratitude to the Department of Mechanical Engineering, Polytechnic Kuching for the financial support through study leave scheme and the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for supporting the data and technical advice to complete this research.

References

[1] James A. E. "Design of an Aerodynamic Rear Spoiler". Mechanical Engineering Undergraduate Project. University of Agriculture, Makurdi, 2013.

[2] Naveen K., Lalit N.V., Narasimha R. and Sri Ram, Y. "Investigation of drag and Lift Forces over the Profile of Car with Rear Spoiler using CFD", International Journal of Advances in Scientific Research, vol. 1, issue 8, pp 331-339, 2015, doi: 10.7439/ijasr. v1i8.2510.

[3] Muhammad Zaid Nawam, Mohd Afzanizam Mohd Rosli, Zulkifli Mohd Rosli, Nor Azwar. "Simulation Study on the Effect of Rear-Wing Spoiler on the Open Aerodynamic Performance of Sedan Vehicle" Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, vol 49, issue 2, pp 146-154, 2018.

[4] D. H. Didane, N. Rosly, M. F. Zulkafli, and S. S. Shamsudin, "Performance Evaluation of a Novel Vertical Axis Wind Turbine with Coaxial ContraRotating Concept," Renewable Energy, vol. 115, pp. 353–361, 2018, doi: 10.1016/j.renene.2017.08.070.

[5] "Truck Aerodynamics - Enhancing Performance & Fuel Economy." https://www.neuralconcept.com/post/truckaerodynamics-enhancing-performance-fuel-economy

[6] Leduc G., Longer and heavier vehicles, an overview of technical aspects, JRC Scientific and Technical Reports, European Communities, 2009, doi: 10.2790/12649.

[7] C.-W. Pai, "Automobile engineering," 2016.

[8] T. Von Kármán, Aerodynamics: selected topics in the light of their historical development. Courier Corporation, 2004.

[9] Senan Thabet, Thabit Hassan Thabit. "Computational Fluid Dynamics: Science of the Future" International Journal of Research and Engineering, vol. 5, no. 6, June 2018

[10] Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu, "Computational Fluid Dynamics: A Practical Approach", Butterworth-Heinemann; 1st edition, Burlington, MA, November 2007.

[11] S. Hassan, "Aerodynamics Simulation of vehicle Body by using CFD Technology." Master Thesis, Teesside University UK, 2014.

[12] M. A. Q. A. Mukhti, D. H. Didane, M. Ogab, and B. Manshoor, "Computational Fluid Dynamic Simulation Study on NACA 4412 Airfoil with Various Angle of Attacks," Journal of Design for Sustainable and Environment, vol. 3, no. 1, 2021.

[13] H. F. M. Yusri et al., "2D Numerical Simulation of Htype Darrieus Vertical-Axis Wind Turbine (VAWT)," Journal of Design for Sustainable and Environment, vol. 5, no. 1, pp. 11–16, 2023.

[14] P. K. Lek et al., "3D CFD Analysis of Straight and Helical Blades Vertical Axis Wind Turbine," Journal of Design for Sustainable and Environment, vol. 5, no. 1, pp. 22–28, 2023.

[15] C. Rajsinh B. and T. K. Raj R. "Numerical investigation of external flow around the ahmed reference body using computational fluid dynamics," Research Journal of Recent Sciences, vol. 1(9), pp. 1-5, 2012.

[16] "Aerospaceweb.org | Ask Us - Golf Ball Dimples & Drag."https://aerospaceweb.org/question/aerodynamics /q0215 .shtml (accessed Sep. 13, 2023)

[17] R. H. Barnard, Road vehicle aerodynamic design-an introduction. 2001.

[18] T. C. Rudien et al., "Technical Feasibility Analysis of Wind Energy Potentials in two sites of East Malaysia: Santubong and Kudat," Evergreen, vol. 8, no. 2, pp. 271–279, 2021, doi: 10.5109/4480703.

[19] M. Mondal, D. H. Didane, A. Hisseine, I. Ali, and B. Manshoor, "Wind Energy Assessment as a Source of Power Generation in," vol. 3, no. 3, pp. 16–22, 2022.

[20] M. Z. Samsudin, D. H. Didane, S. A. Elshayeb, and B. Manshoor, "Assessment of Solar Energy Potential in Johor, Malaysia," Journal of Design for Sustainable and Environment, vol. 3, no. 1, 2021.