Progress in Engineering Application and Technology Vol. 3 No. 2 (2022) 642-652

PEAT

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/peat
e-ISSN : 2773-5303

CFD Study on Aerodynamic Effects of a Rear Spoiler on Mitsubishi Lancer 2005

Muhammad Salihin Man[1], Nurhafizzah Hassan[1]*, Izuan Amin Ishak[1]

[1]Department of Mechanical Engineering Technology, Faculty of Engineering Technology,
Universiti Tun Hussein Onn Malaysia, 84600 Pagoh, Johor, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/peat.2022.03.02.061
Received 27 January 2022; Accepted 07 November 2022; Available online 10 December 2022

Abstract: The movement of air around a moving vehicle affects all of its components in some way. Spoilers are typically found on sedans. As a result, by preventing flow separation, the rear spoiler decreases drag. This paper objectives to design an aerodynamic spoiler and to study the effect of spoiler on drag and lift coefficient using Ansys. The flow shift measured for three different height (106mm, 116mm and 126mm) and three different angles (0^o, 6^o and 12^o). The process for analyzing the influence of a spoiler on a vehicle is described using the ANSYS Fluent 19.2 software to do Computational Fluid Dynamics (CFD). According to the flow analysis, the addition of a spoiler virtually reduced the recirculation zone above the rear window, and flow separation was avoided. In the case of passenger vehicles, having more negative force rather than less drag may be more important, as safety is always the main priority. It's crucial to remember that the benefits of a rear spoiler are typically felt only at high speeds. In most cases, a spoiler will have a negative impact on a car's performance, especially at low speeds. The original suggestion for this research is to design the entire vehicle, including the side mirror, door handle, and four wheels. These three elements are equally important when compared to a real vehicle because they can alter the quantitative and qualitative behavior of aerodynamics.

Keywords: Spoiler, Computational Fluid Dynamic, Drag Coefficient, Lift Coefficient

1. Introduction

The exterior shapes of automobiles have evolved over time for a variety of reasons, including safety, comfort, and aesthetic concerns. According to [12], [7], and [2], the implications of these laws on automotive aerodynamics were not a serious worry for many years. A car's performance cannot be judged just on the basis of its horsepower. According to [13], in order to analyze drag and stability, aerodynamic properties must be addressed, and an aerodynamic device such as a rear spoiler is

*Corresponding author: hafizzah@uthm.edu.my

necessary. [3] stated that the improvement in roadway conditions, combined with an increase in the theoretical maximum speed of automobiles, has moved automotive designers' attention to the dynamic characteristics of cars at high speeds. The external shapes of automobiles are often designed to reduce aerodynamic drag. Unfortunately, [6] stated that this action has drawbacks, including car bodywork producing aerodynamic lift forces at high speeds, as well as a reduction in directional stability and safety constraints during fast cornering.According to [1], "wind tunnels are commonly utilized to examine the aerodynamic characteristics of passenger and racing automobiles." According to [4] and [11], computational fluid dynamics (CFD) has recently progressed as a technology to the point where it can compute quantities such as drag and lift for a road vehicle without resorting to wind tunnel testing. The movement of air around a moving vehicle affects all of its components in some way. According to [5], vehicle aerodynamics comprise engine intake and cooling flow, internal ventilation, tire cooling, and total external flow.

Aerodynamics is the study of how things flow through air. Everything that moves through the air, from a rocket to a kite, is affected by aerodynamics. The aerodynamic force may be split into two components: parallel and perpendicular to the relative wind. According to [8] no matter how slowly a car goes through the air it consumes energy. The impact of a vehicle's body moving air out of the way is known as frontal pressure. When millions of air molecules reach the car's front end, they compress, increasing the air pressure in front of it. Air trying to circulate across the front of the vehicle causes frontal pressure. Pressure drags accounts for the majority of overall drag acting on vehicles, according to previous research, especially at high speeds. The design of external aerodynamic attachments to vehicles to boost its aerodynamic performance is used to improve vehicle aerodynamics.

Spoilers are typically found on sedans. As a result, by preventing flow separation, the rear spoiler decreases drag. Flow separation is undesirable because it causes a greater wake and lower pressure on the rear surface, lowering pressure recovery. By using Ansys, it is hope that we can determine the best location for the wing to reduce the low-pressure area and improve performance. This study objectives were to design an aerodynamic spoiler and to study the effect of spoiler on drag and lift coefficient using Ansys. In addition of that to fulfill the objectives of the project, the scope was limited to three. First, the simulations of spoiler of Mitsubishi Lancer 2005 will be done by using Ansys software. Then, the flow shift measured for three different height (106mm, 116mm and 126mm) and three different angles (0^{o}, 6^{o} and 12^{o}). Finally, drag coefficient, and lift coefficient of the Mitsubishi Lancer 2005 will be analyzed.

2. Methods

The process for analyzing the influence of a spoiler on a vehicle is detailed in this chapter using the ANSYS Fluent 19.2 software to do Computational Fluid Dynamics (CFD). This procedure has three stages: pre-processing, solver, and post-processing. Initial and boundary conditions are used to characterize almost every problem in CFD. This research project's boundary condition is provided. In order to control how the system functions, many boundary conditions must be assigned in a CFD analysis. A wall boundary condition is without a doubt the most commonly used boundary condition in a CFD analysis, out of the many other types available, including inlet, outlet, wall, and symmetry. The wall boundary condition represents the vehicle's surface in the flow domain. Figure 1 shows the methodology in which study flow was considered in this study.

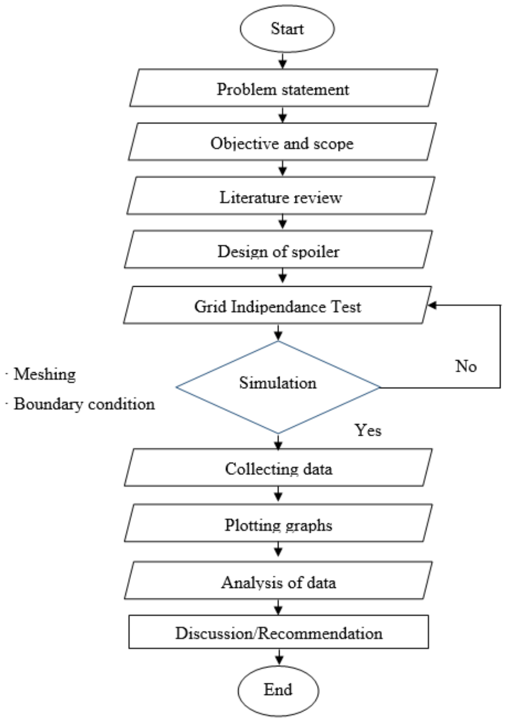


Figure 1: The flowchart of methodology

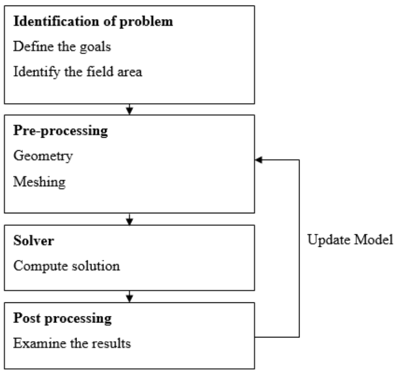


Figure 2: The flowchart of modelling

2.1 Geometry of vehicle and spoiler

The vehicle model is depicted in Figure 3 below. All dimensions are in MMGS (millimeter, gram, second). The vehicle is not a specific or exact replica of the Mitsubishi Lancer 2005; it is simply a mock-up design based on the real. To run the simulation for this research endeavor, the original spoiler design was employed as a model. The spoiler will be altered with three various heights and angles assembled on the automobile. Table 1 and Figure 3 below show the parameters of the spoiler.

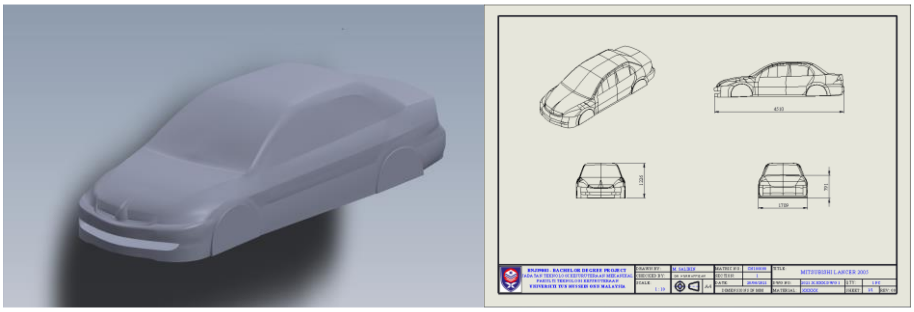


Figure 3: Geometry of vehicle

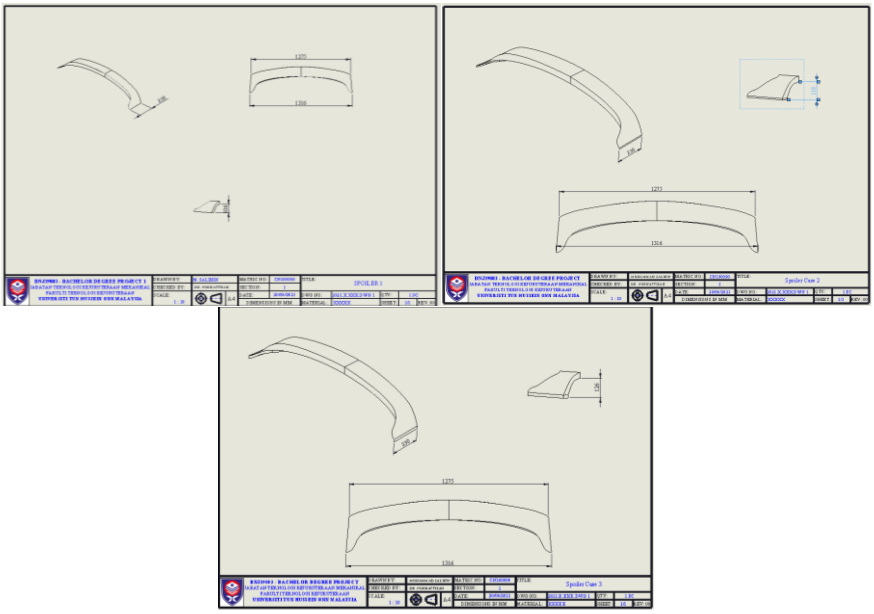


Figure 4: Drawing of spoiler

Table 1: Parameter of spoiler

NO.	Height	Angle
1	106 mm	0°
2	116 mm	0°
3	126 mm	0°
4	126 mm	0°
5	126 mm	6°
6	126 mm	12°

2.2 Mesh generation and boundary condition

Meshing is defined as the act of dividing a component into a number of components so that when the component is loaded, the load is spread evenly. During loading, the entire structure is divided into several components, each with its unique stiffness. If the model in this study is not meshed, the load distribution will not be uniform, and the results will be erroneous or irregular. Figure 5 shows the mesh generation used in this study.

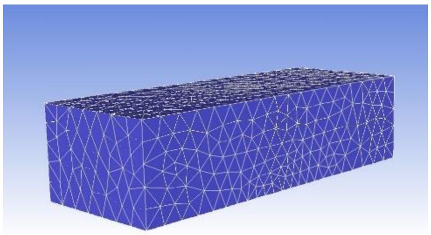


Figure 5: Mesh generated with default setting

This study focuses on the condition in which air flows from the front of the car to the back via the spoiler. Not only that, but it also considers the necessary conditions, such as entrance velocity and wall boundary.

2.3 Grid independence test

Grid independence tests were run in order to optimize the grid size in the simulation. The size of the grid has a significant impact on convergence and expected outcomes. The graph of drag and lift coefficients is confirmed in terms of its number. The drag and lift coefficients for various meshing angles are depicted in Table 2. The graph shows that the trend remains at 6 degrees to 1 degree.

Table 2: Data of drag and lift coefficient with different mesh angle

Degree	18	15	12	9	6	3	1
CD	0.24	0.24	0.24	0.24	0.23	0.23	0.23
Cl	-0.46	-0.46	-0.45	-0.46	-0.46	-0.46	-0.44

3. Results

CFD analysis produces useful engineering data for purposes such as conceptual investigations of novel designs, troubleshooting, complete product development, and redesign. Turbulence was modelled using the realizable k-ε model with non-equilibrium wall functions. The computational results for the following examples are presented and discussed:

- ❖ Case 1: Vehicle model with 106mm height of rear spoiler
- ❖ Case 2: Vehicle model with 116mm height of rear spoiler
- ❖ Case 3: Vehicle model with 126mm height of rear spoiler
- ❖ Case 4: Vehicle model with 0° angle of rear spoiler
- ❖ Case 5: Vehicle model with 6° angle of rear spoiler
- ❖ Case 6: Vehicle model with 12° angle of rear spoiler

For each case, the results were obtained using the same meshing curvature angle, the same k-ε turbulence model, and the same boundary conditions. The free flow velocity was set at 30 m/s (which almost reach the highway speed limit in Malaysia). For the 200 iterations of cases 1, 2, and 3, the second order upwind approach was applied. To achieve the convergence criteria, all residuals have to be smaller than 1e-3. The second order upwind approach was employed for the 120 iterations in cases 4, 5, and 6. The reason for using 120 iterations in these three situations is that the simulation on cases 1, 2, and 3 reveals that the drag and lift coefficients attain convergence at 120 iterations.

3.1 Effects of spoiler based on different height and angle

There was a significant shift in drag force, and the same thing happened in down-force (negative lift-force) over the vehicle body by having a spoiler at the back end, albeit none of the three examples had a significant effect, with the drag coefficient only decreasing to 0.233 as shown in Figure 6. It has been discovered that the drag reduction achieved by having a spoiler at the vehicle's rear end is significantly dependent on the form (design) of the spoiler. Case 3, on the other hand, produced significant down-force (negative lift-force); the lift coefficient for case 3 was lowered to -0.451, while the lift coefficients for cases 1 and 2 were reduced to -0.459 and -0.472, respectively. The drag and lift coefficients for all three situations are shown in Table 3.

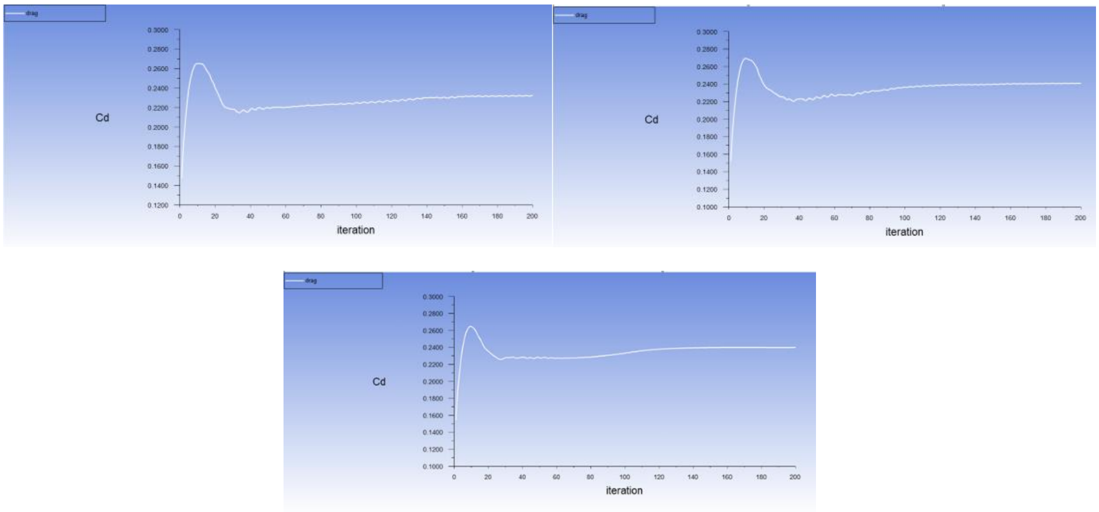


Figure 6: Convergence history of drag coefficient for case 1, 2 and 3

Table 3: Drag and lift coefficients for case 1, 2 and 3

Case	Drag Coefficient C_D	Lift Coefficient C_L
1	0.233	-0.459
2	0.241	-0.472
3	0.239	-0.451

The outcomes for Cases 1, 2, and 3 (in which three different height spoilers were installed to the vehicle's rear end) were compared. Figure 7 depicts the lift coefficient's convergence history.

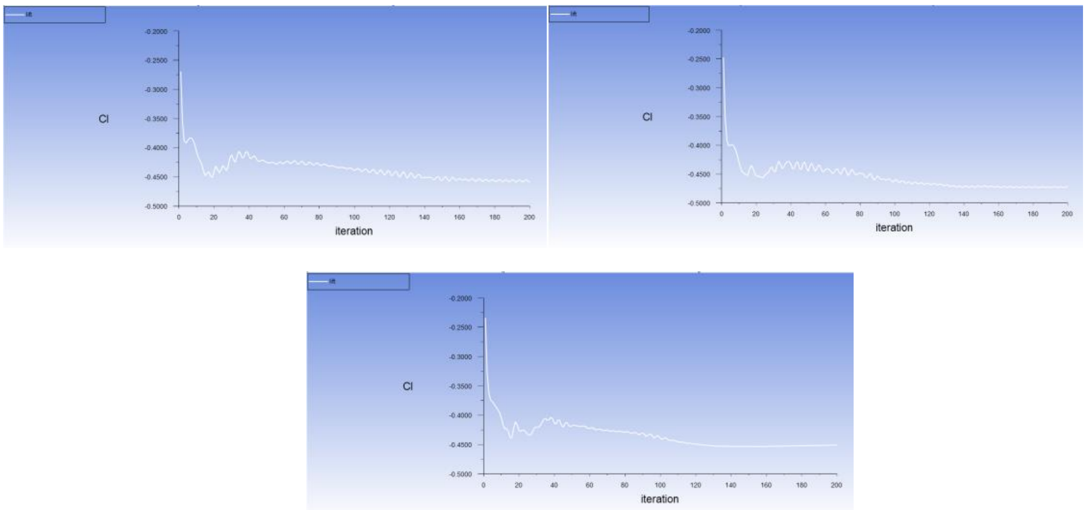


Figure 7: Convergence history of lift coefficient for case 1, 2 and 3

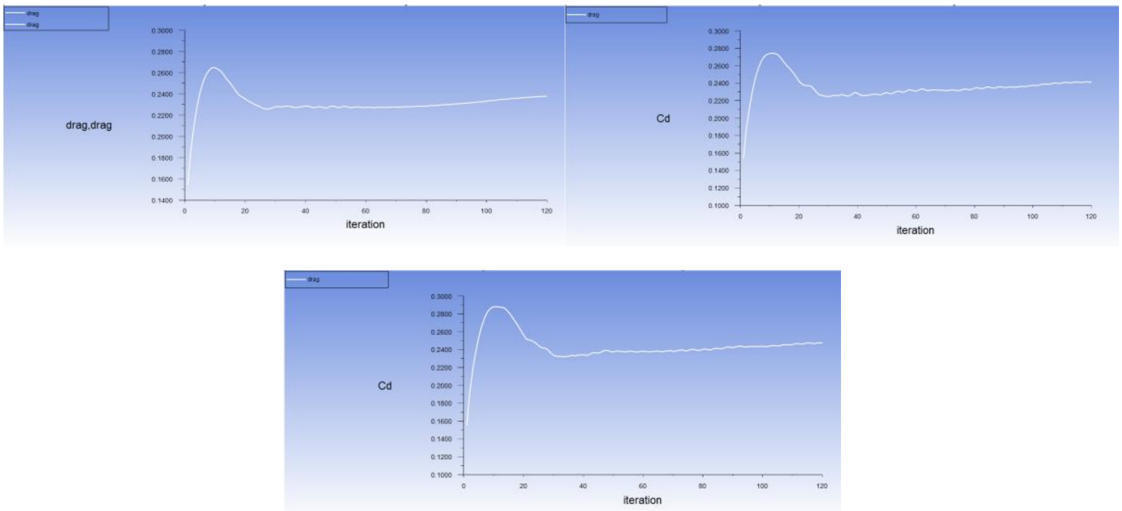


Figure 8: Convergence history of drag coefficient for case 4, 5 and 6

As seen in Figure 8, there was a significant shift in drag force, and the same thing occurred in down-force (negative lift-force) over the vehicle body by having a spoiler at the back end with a different angle. Case 4 created significant down-force (negative lift-force); the lift coefficient for case 4 was reduced to -0.449, while the lift coefficients for cases 5 and 6 were reduced to -0.467 and -0.500, respectively. The convergence history of lift coefficient for case 4, 5 and 6 are shown in Figure 9and the drag and lift coefficients for all three situations are shown in Table 4.

Table 4: Drag and lift coefficients for case 4, 5 and 6

Case	Drag Coefficient C_D	Lift Coefficient C_L
4	0.237	-0.449
5	0.241	-0.467
6	0.247	-0.500

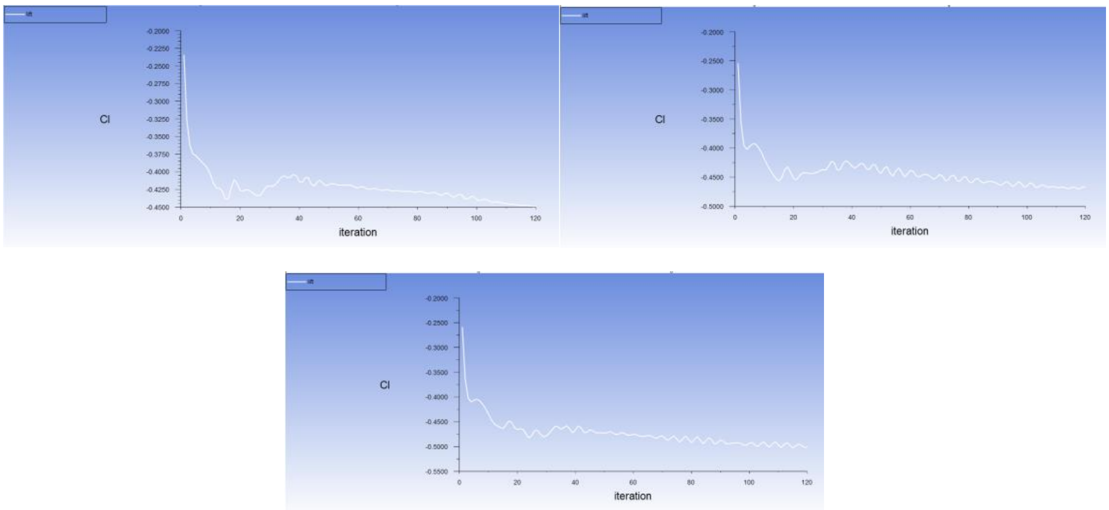


Figure 9: Convergence history of lift coefficient for case 4, 5 and 6

3.2 Velocity flow on contours, streamline and vector

Figures 10 and 11 show velocity contours for a high-speed vehicle in the symmetry plane in each of the six scenarios. The rear ends of the cars with rear spoilers (cases 1 and 3) had massive and complex air swirls. Recirculation zones (cyan blue colour region) were detected behind the vehicle's rear end. In contrast to situations when the flow remains attached to the surface, this state results in a considerable increase in drag. When comparing the examples in the figures, the recirculation zone behind the rear end of the car with spoiler situations (case 1 and case 3) was substantially greater.

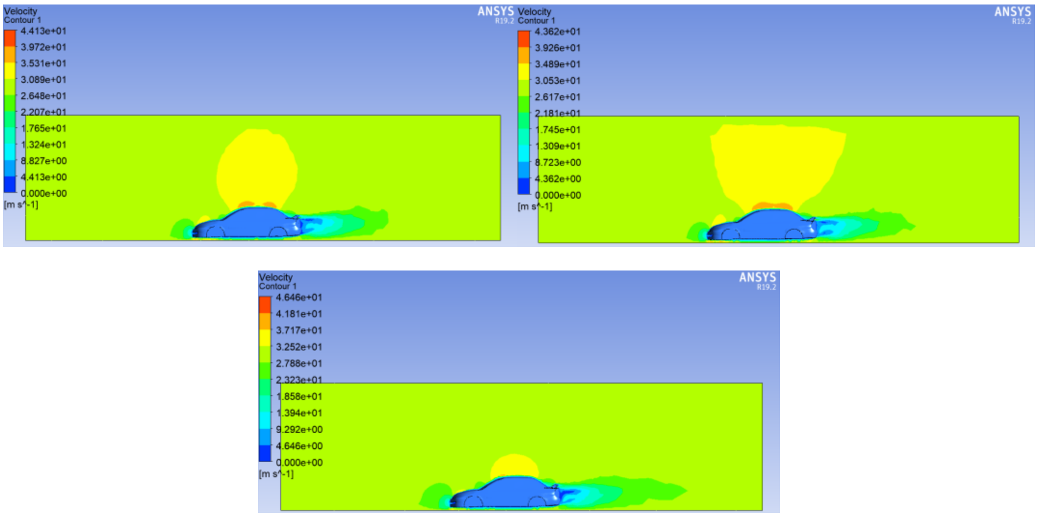


Figure 10: Velocity distribution of flow for case 1, 2 and 3

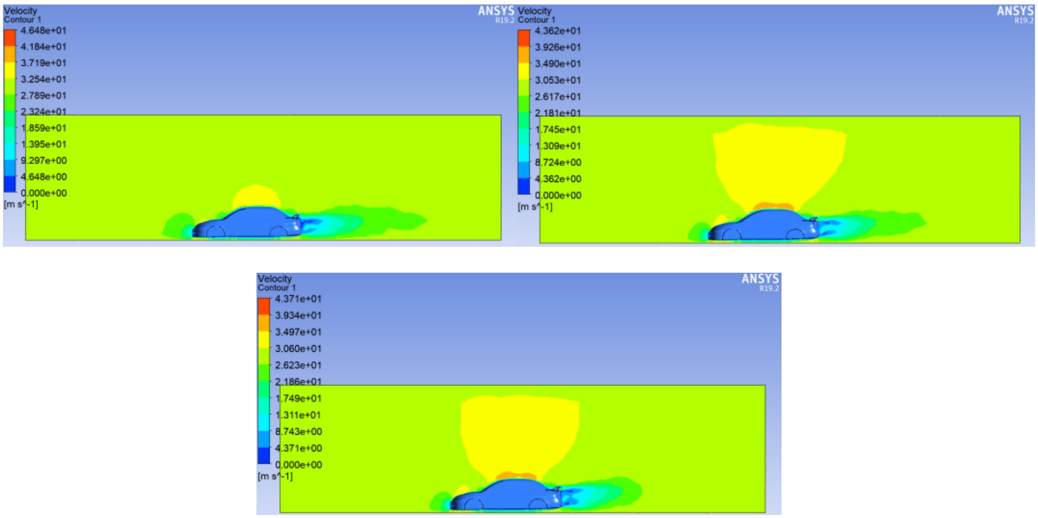


Figure 11: Velocity distribution of flow for case 4, 5 and 6

The rear ends of the cars with rear spoilers (case 6) had massive air swirls. There were recirculation zones (cyan blue colour region) identified behind the vehicle's bottom rear end. When comparing the cases in the figures, the recirculation zone behind the rear bottom end of the automobile with spoiler conditions (case 6) was substantially greater.

Figure 12 clearly depicts the recirculation zones at the vehicle's rear end. The recirculation zone above the rear window was eliminated by using a spoiler. The air gradually slopes above the back window, aiding in its cleanliness. One of the advantages of employing a spoiler is that it keeps the back window clean. When comparing velocity vectors and velocity streamlines of various height and angle spoiler conditions (Figures 12 and 13), the recirculation zone behind the rear end of the vehicle with spoiler is substantially larger.

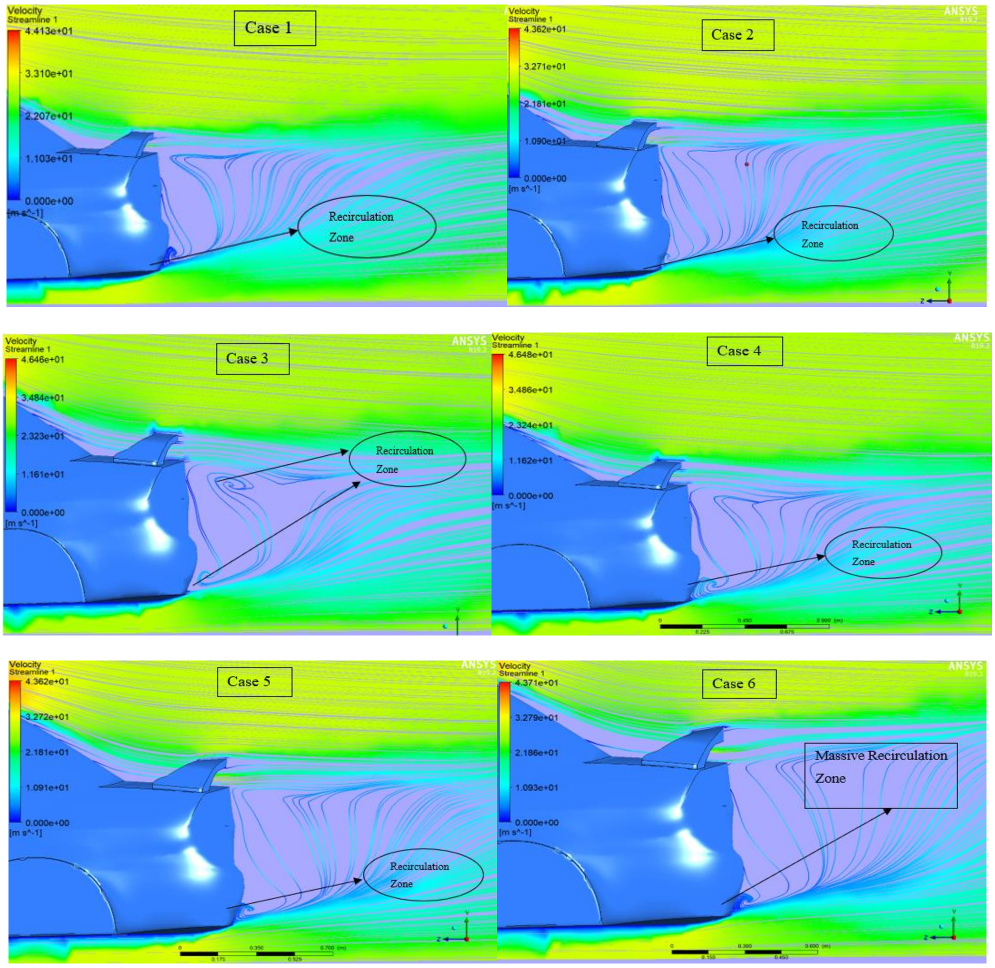
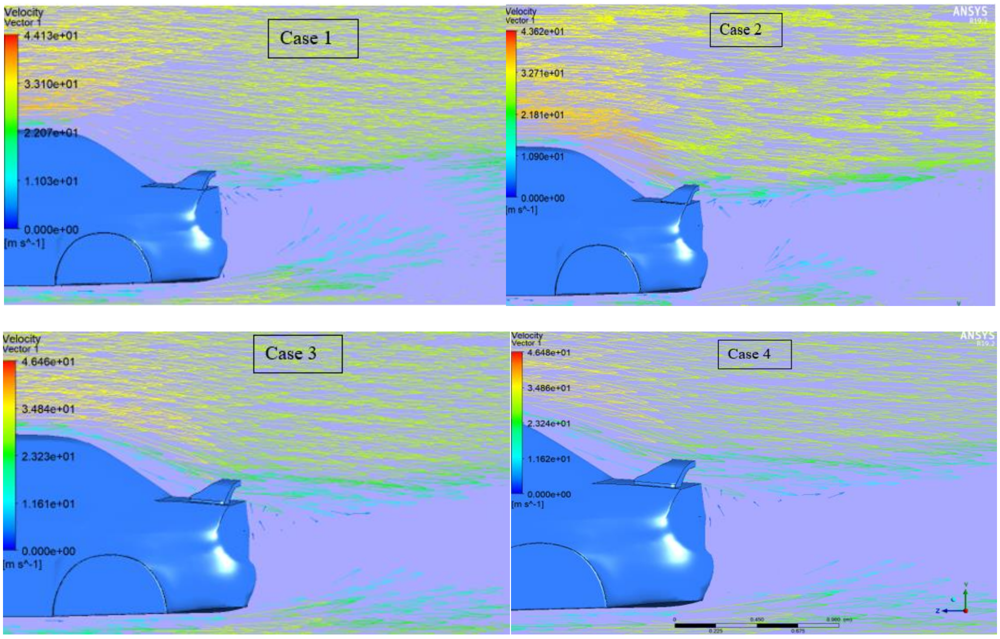


Figure 12: Velocity streamline of flow for all cases



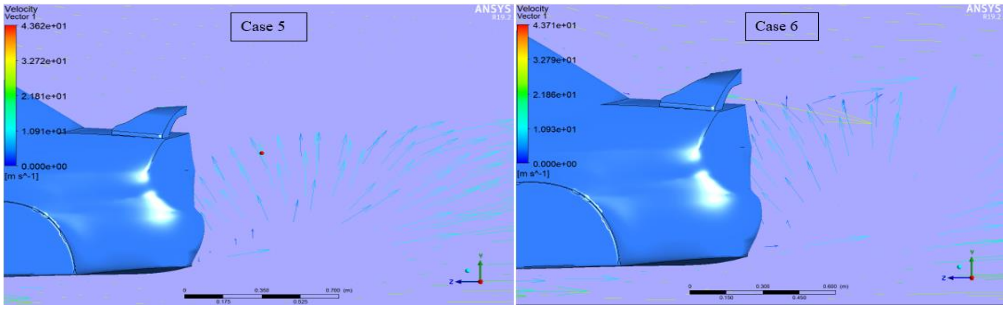


Figure 13: Velocity vector of flow for all cases

When comparing velocity vectors and velocity streamlines of various angles with the same height spoiler circumstances, the recirculation zone behind the rear end of the car with a high angle of spoiler is substantially larger (case 4, 5 and 6). For cases 5 and 6, the velocity vector appears a little aggressive as it shoots upward due to the recirculation zone at the bottom rear end of the vehicle, as seen in Figure 13.

4. Discussions

The drag coefficient (C_D) and lift coefficient (C_L) increase as the spoiler angle increases. There is no discernible alteration in the flow pattern in the front, hood, or beneath the vehicle body. There are, however, subtle differences in the rear bottom end of each case. Even modest changes to the car's performance might have a major impact. According to the flow analysis, the addition of a spoiler virtually reduced the recirculation zone above the rear window, and flow separation was avoided.

[10] discovered that the spoiler with the smaller angle of wind impact creates more drag force for a given spoiler height. When the spoiler is set to 3^0, [9] calculated that the angle of flow of air over the spoiler should be 0.59 based on bolero model which is SUV type vehicle. This is because, with a lower angle of impact, the car's rear end generates a smaller recirculation zone behind the vehicle's back end.

5. Conclusion

The aerodynamic lift, drag, and flow parameters of a high-speed (30 m/s) Mitsubishi Lancer 2005 with different spoiler heights and angles were computed. Cases 1 and 2 exhibited an 8 percent increase in drag coefficient, whereas Case 3 showed a 2 percent decrease in drag coefficient. The lift coefficient is likewise roughly the same in these three situations. Case 2's aerodynamic drag is increased from 0.233 to 0.241, an 8 percent increase in drag, while negative lift is increased by lowering the lift coefficient from -0.459 to -0.472, a 13 percent loss in lift. When Figures 11 was compared, it revealed that the recirculation zone above the rear window was practically gone in Case 1. The air was gently inclined above the back window, which helped to keep it clean.

According to the numerical analysis, case 2 provided more negative lift force than cases 1 and 3 but delivered less drag reduction. According to the numerical study, the drag and lift coefficients increase gradually as the angle increases in cases 4, 5, and 6. While case 4, 5, 6 were compared, it can be seen that as the angle of the spoiler increases, so does the recirculation zone at the bottom rear end of the vehicle, resulting in an increase in the lift coefficient. In the case of passenger vehicles, having more negative force rather than less drag may be more important, as safety is always the main priority. It's crucial to remember that the benefits of a rear spoiler are typically felt only at high speeds. In most cases, a spoiler will have a negative impact on a car's performance, especially at low speeds. The vehicle industry has been exploring these negative effects, and some companies have found techniques to mitigate the negative effects of spoilers at low speeds.

Acknowledgement

The authors would like to thank the Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] A. Buljac, H. Kozmar, and I. Džijan, “Aerodynamic performance of the underbody and wings of an open-wheel race car,” Trans. Famena, vol. 40, no. 2, pp. 19–34, Feb. 2016, doi: 10.21278/TOF.40202.

[2] A. P. Akhilesh Singh Tomar and S. S. Anuj Sharma, “CFD Analysis on the Aerodynamic Effects of Spoiler at Different Angle on Car Body,” Blue Eyes Intell. Eng. Sci. Publ., vol. 8, no. 7, May 2019.

[3] C. H. Tsai, L. M. Fu, C. H. Tai, Y. L. Huang, and J. C. Leong, “Computational aero-acoustic analysis of a passenger car with a rear spoiler,” Appl. Math. Model., vol. 33, no. 9, pp. 3661–3673, Sep. 2009, doi: 10.1016/j.apm.2008.12.004.

[4] D. Damjanović, D. Kozak, M. Živić, Ž. Ivandić, and T. Baškarić, “CFD Analysis of Concept Car in Order to Improve Aerodynamics,” Járműipari innováció, Nov. 2010,

[5] J. Katz, “Aerodynamics of Race Cars,” 2005, doi: 10.1146/annurev.fluid.

[6] K. Kurec, M. Remer, J. Broniszewski, P. Bibik, S. Tudruj, and J. Piechna, “Advanced Modeling and Simulation of Vehicle Active Aerodynamic Safety,” J. Adv. Transp., vol. 2019, 2019, doi: 10.1155/2019/7308590.

[7] K. Rajapaksha, C. Kurukulasooriya, M. Herath, “Aerodynamic Analysis of Rear Wings and Rear Spoilers of Passenger Automobiles,” Inst. Eng. Sri Lanka, pp. 515–522, Oct. 2018.

[8] M. Hariharan, E. Harish Babu, S. Kirubakaran, and S. Gopalakrishnan, “Drag Reduction on Passenger Car,” M.Kumarasamy Coll. Eng., vol. 25, no. 6, pp. 4501–4507, May 2021.

[9] M Saiteja, K. Mohan, “CFD analysis and structure of a rear spoiler for Mahindra bolero vehicle,” Int. J. Adv. Res. Sci. Eng. Technol., vol. 6, 2019,

[10] R. C. Das and M. Riyad, “CFD Analysis Of Passenger Vehicle at Various Angle Of Rear End Spoiler,” Procedia Eng., vol. 194, pp. 160–165, Jan. 2017, doi: 10.1016/j.proeng.2017.08.130.

[11] R. Salahuddin Khan and S. Umale, “CFD Aerodynamic Analysis of Ahmed Body,” Int. J. Eng. Trends Technol., vol. 18, 2014.

[12] T. Dominique and J. Gabor, Optimization and Computational Fluid Dynamics. Germany: Springer-Verlag Berlin Heidelberg, 2008.

[13] Z. Deng, S. Yu, W. Gao, Q. Yi, and W. Yu, “Review of effects the rear spoiler aerodynamic analysis on ground vehicle performance,” J. Phys. Conf. Ser., vol. 1600, p. 12027, 2020, doi: 10.1088/1742-6596/1600/1/012027.