JAPTT

Journal homepage: https://publisher.uthm.edu.my/ojs/index.php/japtt/index

Journal of Automotive Powertrain and Transportation Technology

e-ISSN: 2821-286X

CFD Analysis of Rear-Spoilers Effectiveness on Sedan Vehicle in Compliance with Malaysia National Speed Limit

Daniel Syafiq Baharol Maji[1], Norrizal Mustaffa[1*]

[1]Centre of Automotive and Powertrain Technology,
Department of Mechanical Engineering Technology, Faculty of Engineering Technology,
Universiti Tun Hussein Onn Malaysia, Pagoh Campus, KM 1 Jalan Panchor, 84600 Pagoh, Johor, MALAYSIA

*Corresponding Author

DOI: https://doi.org/10.30880/japtt.2022.02.01.003
Received 20 February 2022 ; Accepted 25 April 2022 ; Available online 19 June 2022

Abstract: A spoiler is an aerodynamic component used to decrease drag in automobiles. The primary function of the car rear spoiler is to increase the vehicle's grip on the road by decreasing the aerodynamic drag and increase stability. This rear positioned device built up an area of high pressure to replace the low pressure on the trunk leading to increased stability. The objective of this study is to investigate the effects of rear spoiler on automobile aerodynamic drag and stability in compliance with the Malaysian National Speed Limit. Both the sedan vehicle model and the rear spoiler models were built using CAD (Computer-Aided Design) software. The data was then analyzed in CFD (Computational Fluid Dynamic) software to calculate the drag and lift force acting on the moving sedan car at velocity of 60km/h, 90km/h and 110 km/h. There have been some limitations due to the complexity of the design. Two rear spoiler designs which are the ducktail spoiler and the rear wing were used in the simulation along with the sedan vehicle. The result given by the simulations shows that rear spoilers increase the drag force and the downforce of the car. Rear wing shows a drastic increase in drag and downforce while ducktail spoiler shows a slight increase. The result also shows that slow moving vehicle has higher drag than fast moving vehicle. In summary, spoilers increase drag at low speed and only shows its benefits at high speed. Given the Malaysia National Speed Limit, spoilers may only show its benefit when the vehicle is driven on the expressway since expressways has a speed limit of 110 km/h.

Keywords: Sedan vehicle, rear spoilers, Malaysia national speed limit, CFD, CAD, Drag, Lift

1. Introduction

Vehicle's aerodynamic performance has long been the focus of automotive design engineers. As the vehicle travels through the air, the drag and lift forces on the vehicle at top speed will have a major impact on the vehicle's aerodynamic efficiency. These phenomena appear to negatively affect the performance, handling, stability and fuel efficiency of the vehicle [1]. In order to enhance vehicle aerodynamics, the design of exterior aerodynamic accessories for automobiles that can be installed externally to the vehicle is used to enhance the aerodynamic efficiency and facilitate handling and fuel efficiency. 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].

In modern times, motor vehicles are going so fast that unforeseen accidents caused by such an unrestrained amount of speed happens. This shows the need to invent an aerodynamic wing, a spoiler that creates a highly monitored stall over the wing section at the back of the spoiler, essentially by reducing the lift of that wing section. Spoilers are developed to decrease lift and also increase drag substantially. In addition to reducing drag and lower rear axle lift, the rear spoiler can also help decrease dirt on the rear surface. It is estimated from past studies that aerodynamic drag is the

*Corresponding author: norrizal@uthm.edu.my

dominant form of resistance when vehicles run at speeds of 80 km/h or more, especially given that 65 per cent of the power required at 110 km/h is consumed as a result of overcoming aerodynamic drag [3].

Had Laju Kebangsaan (National Speed Limits) is a set of speed limits applied to all vehicle drivers on Malaysian expressways, federal roads, state roads and municipal roads. The National Speed Limits had been enforced on 1 February 1989 following the 1989 National Speed Limits (Perintah Had Laju Kebangsaan 1989) [4]. The default speed limit for expressways is 110 km/h by default, but may be reduced to 80 or 90 km/h in hazardous mountain roads, cross-wind zones and high-traffic urban areas. Federal roads and state roads have a default speed limit of 90 km/h and may be reduced during festive seasons. These speed limits will be the main variables in this study to simulate the efficiency of a rear spoiler on a car using the Computational Fluid Dynamics simulation software.

The effects of air flow over the spoilers can be calculated using flow-based simulation and analytical equations. Computational fluid dynamics (CFD) gives a detailed and accurate solution than when the calculation is done theoretically [5-6]. Computational methods give better predictions in a short time. It basically formulates on the continuity equation energy equation and the momentum equation.

2. Material and Methodology

2.1 Drag and Lift Coefficients

Drag, D is the term given to the resultant force in the direction of the upstream velocity while lift, L is the term given to the resultant force normal to the upstream velocity. The resultant forces by these contributions can be divided into 3 components which are moment, drag and lift coefficient. The aerodynamic drag force is calculated by the following formula:

$$C_D = \frac{D}{\frac{1}{2}\rho V^2 A} \qquad (1)$$

Lift force act vertically on the vehicle body. This particular force causes the vehicle to get lifted in air as applied in the positive direction, whereas it can result in immoderate wheel down force if it is applied in negative direction. The formula usually used to define this force is written as:

$$C_L = \frac{L}{\frac{1}{2}\rho V^2 A} \qquad (2)$$

2.2 Spoilers and CFD Software

A spoiler is an automotive device which is fundamentally used to 'spoil' unfavourable air movement over a moving vehicle body, usually described as turbulence or drag. They are made of unique styles and designs but most drivers use it for alternate reasons such as reducing drag force, reducing lift, and increase traction control. The main benefits of installing spoiler on a car will be maintaining traction, increasing fuel efficiency, added visibility, reduce weight and also increase braking stability. Cars travelling at high speeds tend to create a lift, spoilers can create a down force to counteract the lift and give the car more stability.

CFD (Computational fluid dynamics) can be defined as a set of numerical methods applied to obtain approximate solution of problems of fluid dynamics and heat transfer. CFD is a low-cost method for obtaining a discrete solution for real world fluid problems. This solution is obtained at a finite collection of space points at a certain interval of time. This computational analysis can only be simulated through high-speed computers and the analysis of the flow problem is done using a software called ANSYS fluent [7]. This process is a virtual simulation-based design and analyses instead of the normal build and test method.

2.3 Methodology and CAD Models

The sedan vehicle design that will be used in the simulation will be a typical sedan car concept that will be drawn on a CAD (Computer-Aided Design) software. The design of the car and spoiler will be built in CAD software and analysed in CFD software. The dimension used for the sedan vehicle is based on Proton Wira 1.3 1995 model. As for this simulation, the two spoiler designs that will be used are the ducktail spoiler and the rear wing since these two designs are very common among sedan vehicles. These two spoiler designs are also built in the CAD software. Figure 1 shows the isometric view of the the isometric of ducktail spoiler and the assembly 3D CAD model of vehicle with ducktail spoiler. Figure 2 shows the isometric view of rear wing and the assembly 3D CAD model of vehicle with rear wing.

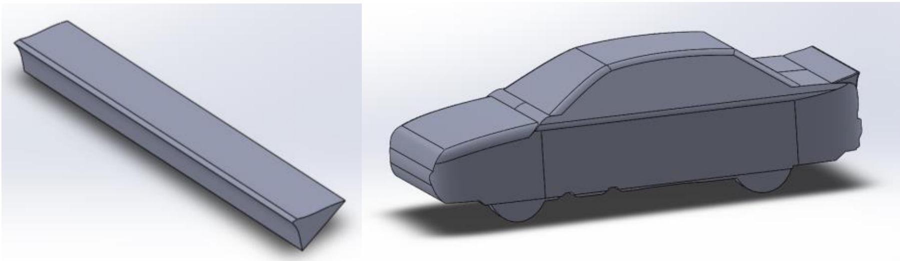


Fig. 1 - The isometric view of ducktail spoiler and the assembly 3D CAD model of vehicle with ducktail spoiler

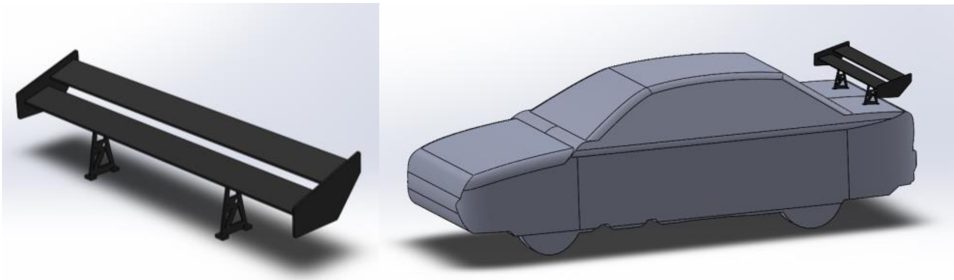


Fig. 2 - The isometric view or rear wing and the assembly 3D CAD model of vehicle with rear wing

2.4 Methodology and CAD Models

The sedan vehicle simulation will be categorized into three different cases. Table 1 shows the simulation for Case1, Case 2 and Case 3.

Table 1 - Simulation for Case 1, Case 2 and Case 3

Case	Simulation
1	Sedan vehicle only
2	Sedan vehicle with ducktail spoiler
3	Sedan vehicle with rear wing

The CAD models of case 1, case 2 and case 3 were individually imported into ANSYS Fluent and a virtual air box enclosure which represents the wind tunnel in real life is created around the model as shown in Figure 3. For case 1, case 2 and case 3, the realizable k-epsilon (k-ε) turbulence model was used for the analysis at steady state. K-epsilon (k-ε) turbulence model is the most common model used in CFD to simulate mean flow characteristics for turbulent flow conditions. It is a two-equation model that gives a general description of turbulence by means of two transport equations. The boundary conditions were chosen such that the flow field most closely depicts the real-world scenario of a vehicle moving through air under normal atmospheric conditions. The inlet velocity used is variable according to the National Speed Limit, the gauge pressure was set as 0 Pascal, wall zones were depicted as No slip, the fluid type was set to be air at a density of 1.175 kg/m^3and a kinematic viscosity of 1.827 x 10^(-5) kg/ms. Figure 8 shows the mesh geometry of the virtual wind tunnel.

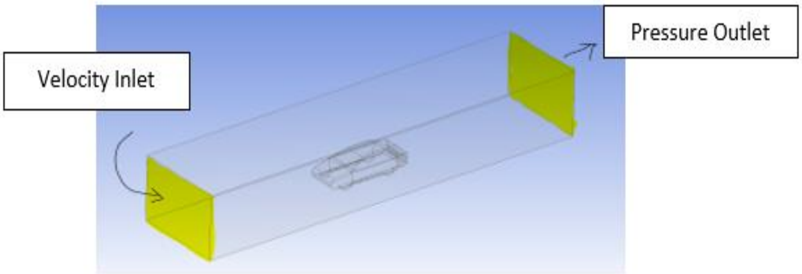


Fig. 3 - Virtual wind tunnel created around the model

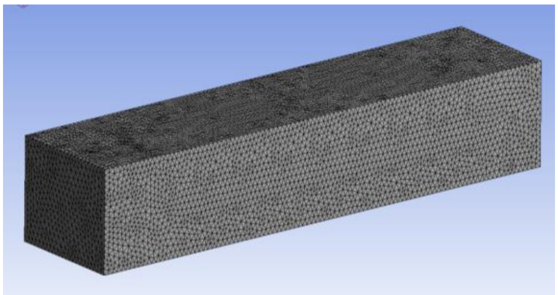


Fig. 4 - Mesh geometry of wind tunnel

Table 2 shows the mesh settings for the generated mesh as in Figure 4. The mesh in Figure 4 was generated with small tweaks with the default settings, sizing and quality.

Table 2 - Mesh geometry settings for the wind tunnel and the vehicle

Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	0.2m
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1.2)
Max Size	0.5m
Mesh Defeaturing	Yes
Defeature Size	Default (1.e-0.003m)
Capture Curvature	Yes
Curvature Min Size	Default (2.e-0.003m)
Curvature Normal Angle	Default (18.0^{o})
Capture Proximity	No
Bounding Box Diagonal	21.1m
Average Surface Area	4.305 m^2
Minimum Edge Length	9.0779e-004 m
Quality	
Check Mesh Quality	Yes, Errors
Target Skewness	Default (0.900000)

Smoothing	High
Mesh Metric	Skewness
Min	4.0702e-004
Max	0.98202
Average	0.23306
Standard Deviation	0.12261
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth	1.2
Inflation Algorithm	Pre
View Advanced Options	No

2.5 Solver Settings in CFD Software

The solver settings that are set up include the type of solver (3D or 2D), the viscous model, the boundary conditions, the solution controls and the reference values. Table 3 shows the solver settings while Table 4 shows Viscous model and Turbulence model used in the simulation. Table 5 shows the boundary condition settings for the simulation. The only variable for the boundary condition is the “Velocity-Inlet” which uses the Malaysia National Speed limit as reference. The speeds are 60 km/h, 90 km/h and 110 km/h. Other than that, Case 1, Case 2 and Case 3 used the same settings and boundary condition.

Table 3 - Solver settings

Pressure-Velocity Coupling (Case 1, Case 2 and Case 3)	
Scheme	Coupled
Solver	
Gradient	Least Squares Cell Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Turbulent Dissipation Rate	First Order Upwind
Flow Courant Number	200
Explicit Relaxation Factors	
Pressure	0.5
Momentum	0.5
Under-Relaxation Factors	
Turbulent Kinetic Energy	0.8
Turbulent Dissipation Rate	0.8
Turbulent Viscosity	1

Table 4 - Settings for viscous model and turbulence model

Viscous Model (Case 1, Case 2 and Case 3		
Turbulence Model	$k - \varepsilon$ (2 equations)	$k - \omega$ (2 equations)
$\boldsymbol{k - \varepsilon}$ **Model**	Realizable	Shear Stress Transport (SST)
$\boldsymbol{k - \omega}$ **Model**	-	Shear Stress Transport (SST)
Near-wall Treatment	Non-Equilibrium Wall Functions	-

Table 5 - Boundary condition

Boundary Conditions (Case 1, Case 2 and Case 3)		
Velocity Inlet	Magnitude and Direction	60 km/h, 90 km/h, 110 km/h
	Turbulence Specification Method	Intensity and Viscosity ratio
	Turbulence Intensity	1.00%
	Turbulent Viscosity Ratio	10
Pressure Outlet	Gauge Pressure Magnitude	0 Pascal
	Gauge Pressure Direction	Normal to boundary
	Turbulence Specification Method	Intensity and Viscosity ratio
	Backflow Turbulent Intensity	10%
	Backflow Turbulent Viscosity ratio	10
Wall Zones	No slip	
Symmetry	No slip	
Fluid Properties	**Fluid Type**	Air
	Density	ρ = 1.175 kg/m^2
	Kinematic Viscosity	v = 1.8247 x 10^{-5}kg/m. s

3. Results and Data Analysis

3.1 Scaled Residual Convergence

Three speeds of the inlet velocity that were used in the wind tunnel model were 60 km/h, 90 km/h and 110 km/h. The simulation for each case continued with 200 iterations until it reached convergence criteria. The common convergence criteria were having all residuals below 1e-3. However, based on the simulations done, the scaled residuals for the three cases were only close to 1e-3 and not below it. Therefore, the results may contain inaccurate values since no experimental data has been found to compare to the CFD simulation results. Figure 5, Figure 6 and Figure 7 show the plots of scaled residuals for all three different cases respectively.

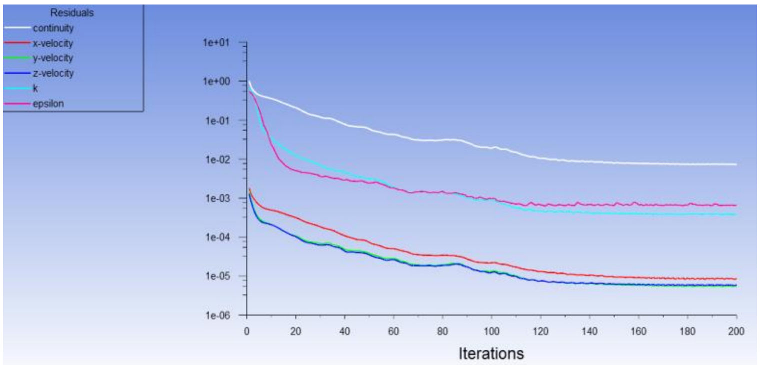


Fig. 5 - Scaled residuals convergence history for Case 1 (Sedan vehicle without rear spoiler)

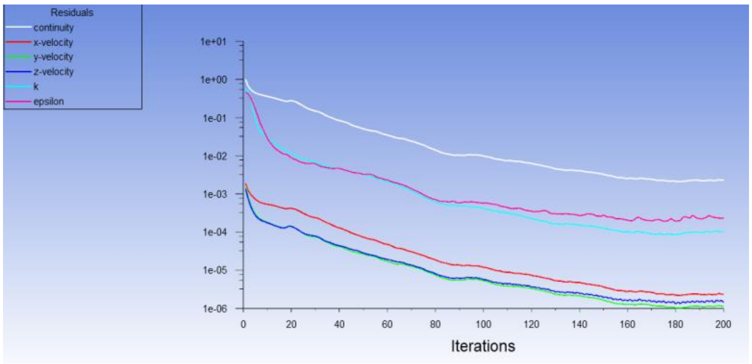


Fig. 6 - Scaled residuals convergence history for Case 2 (Sedan vehicle with ducktail spoiler)

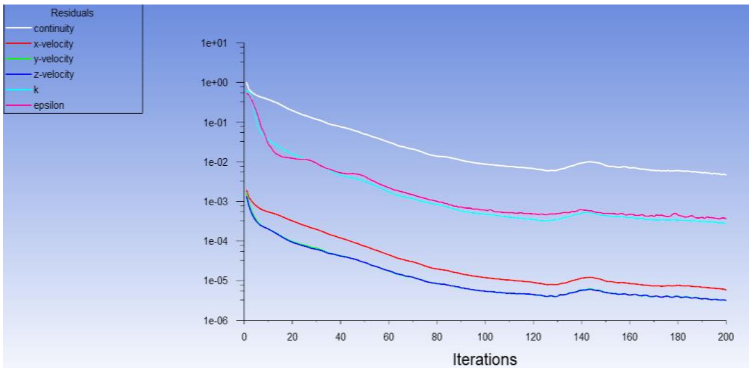
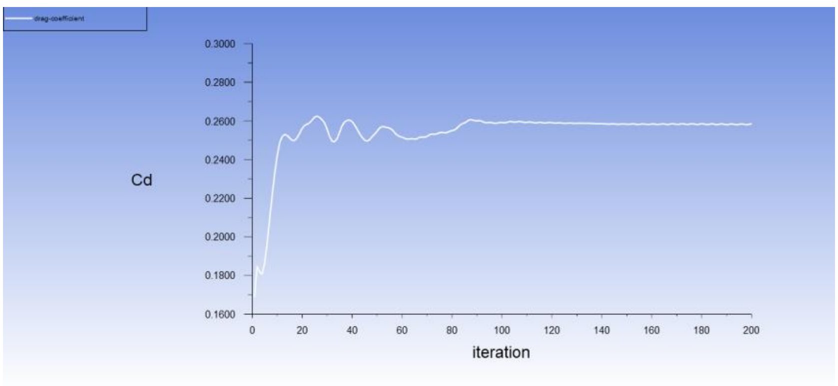


Fig. 7 - Scaled residuals convergence history for Case 3 (Sedan vehicle with rear wing)

As seen in the Figure 5, Figure 6 and Figure 7, the scaled residuals graph shows that the convergence criteria which having all residuals below 1e-3 is not met. However, it is still acceptable since the residuals are close to 1e-3.

3.2 Drag and Lift Coefficients

Figure 8 shows the convergence history of drag coefficient for Case 1, Case 2 and Case 3 at the speed of 110km/h. Each case was then simulated with different speeds which are 60 km/h and 90 km/h. Based on these three cases, the drag coefficient of each cases has converged and changed only by 1% to 2% after the 100th iteration. Figure 9 shows the convergence history of lift coefficient for Case 1, Case 2 and Case 3 respectively. It is observed that the lift coefficient of each case has also converged and changed only by 1% to 2% after the 100th iteration.



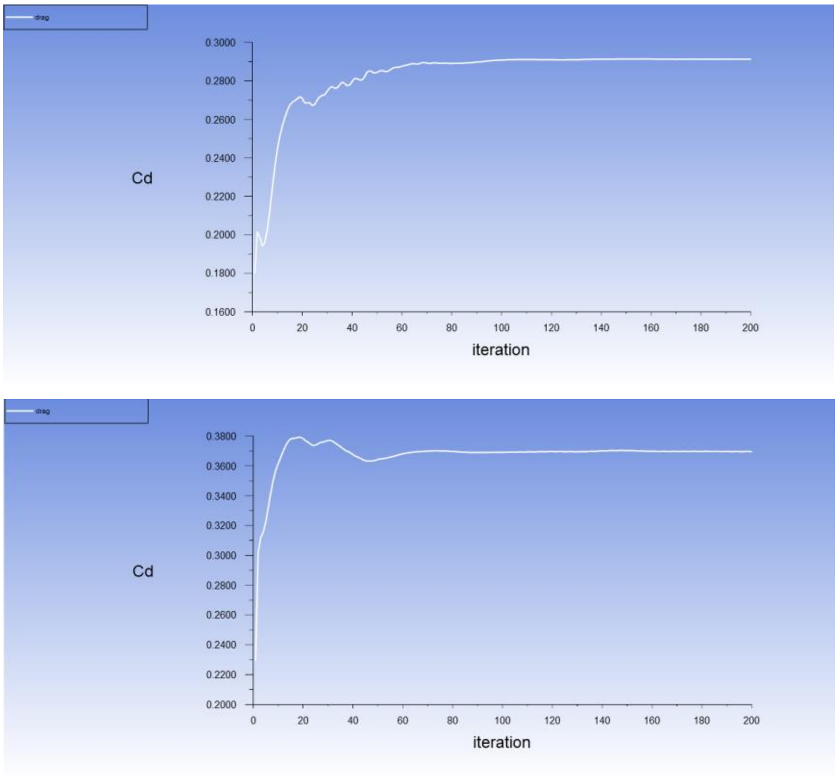
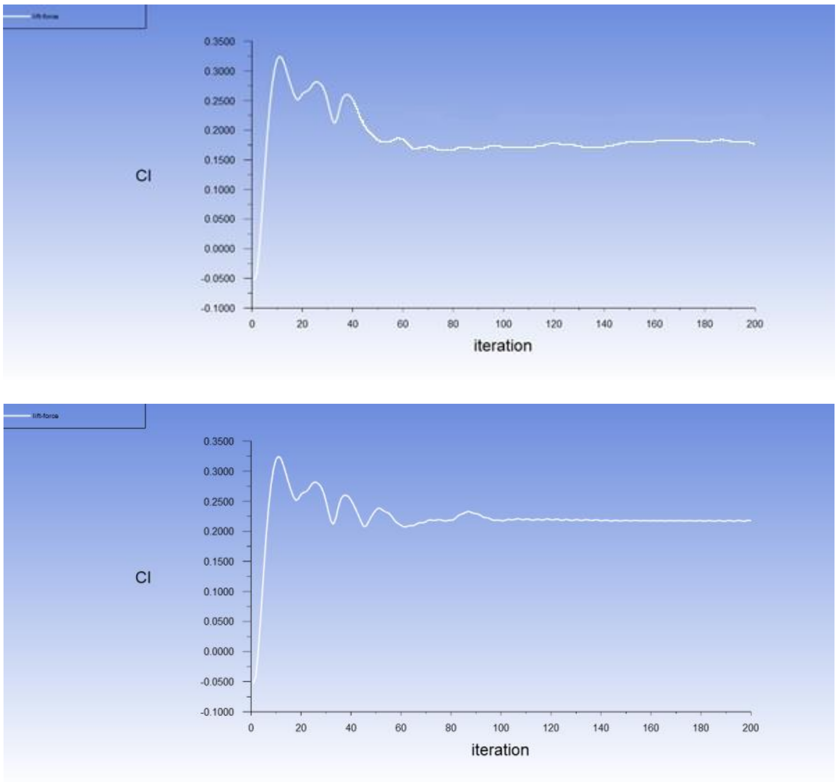


Fig. 8 - Drag coefficient (C_D) convergence history of Case 1, Case 2 and Case 3



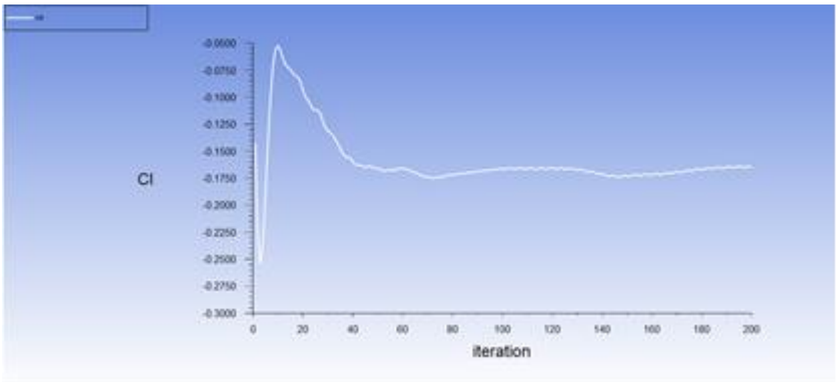


Fig. 9 - Lift coefficient (C_L) convergence history of Case 1, Case 2 and Case 3

As seen from the graph in Figure 8 and Figure 9, it can be analysed that using a ducktail spoiler (Case 2) give a small increase in drag coefficient and slight increase to the downforce (lift coefficient) of the vehicle compared to Case 1. Drag coefficient increased 10.7% while lift coefficient increased by 11.7%. On the other hand, Case 3 has generated an increase in drag coefficient which is 0.3695 (35.1% increment) compared to Case 1 with 0.2590 drag coefficient. The lift coefficient of Case 3 has a remarkable increase of 93.5% compared to Case 1. The vehicle itself (Case 1) without any spoiler had a lift coefficient of -0.2176 which was referred as the reference point. Based on these simulation results, Case 3 proved to be the most efficient in increasing down force compared to Case 2. However, Case 3 also generates a significant amount of drag to the moving vehicle. Table 6, Table 7 and Table 8 show the tabulated data of drag and lift coefficients of Case 1, Case 2 and Case 3 respectively with velocity as the variable.

Table 6 - Drag and lift coefficients for Case 1

Speed (km/h)	Drag coefficient	Lift coefficient (down force)
60	0.2614	-0.2130
90	0.2590	-0.2160
110	0.2590	-0.2176

Table 7 - Drag and lift coefficients for Case 2

Speed (km/h)	Drag coefficient	Lift coefficient (down force)
60	0.2922	-0.1923
90	0.2894	-0.1924
110	0.2885	-0.1935

Table 8: Drag and lift coefficients for Case 3

Speed (km/h)	Drag coefficient	Lift coefficient (down force)
60	0.3727	0.1712
90	0.3702	0.1665
110	0.3695	0.1647

Based on the tabulated results, it has been found that having a spoiler at the rear-end of the sedan vehicle increases its drag and lift significantly.

3.3 Velocity Streamline Analysis

The flow of velocity streamline for a moving sedan car at the symmetry plane for all 3 cases are shown in Figure 10, Figure 11 and Figure 12. The velocity used in the figures is 110 km/h which is the national speed limit for Malaysian Expressway.

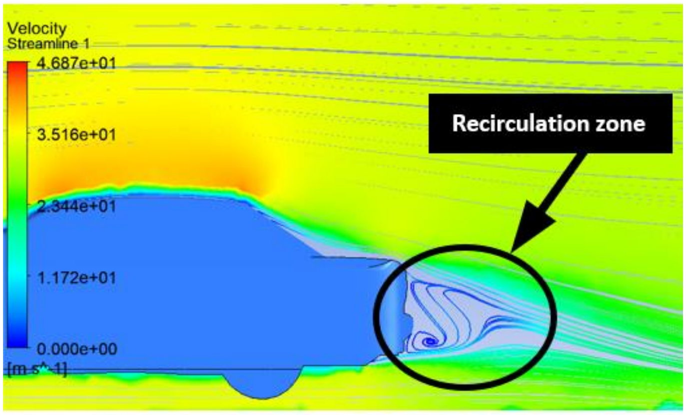


Fig. 10 - The flow of velocity streamlines in the symmetry plane for Case 1

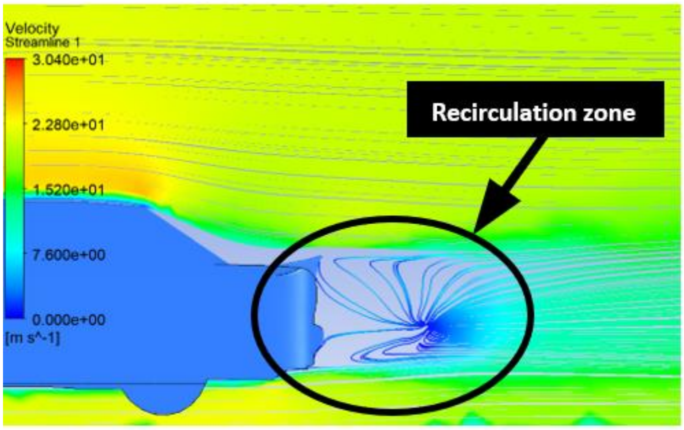


Fig. 11 - The flow of velocity streamlines in the symmetry plane for Case 2

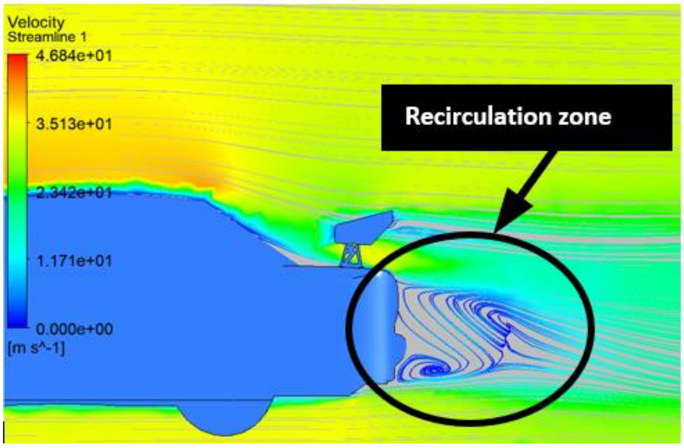


Fig. 12 - The flow of velocity streamlines in the symmetry plane for Case 3

Based on Figure 10, Figure 11 and Figure 12, it can be seen that there were recirculation zones behind the rear end of the vehicle. Comparing these three figures with the naked eye, the recirculation zone of Case 3 and Case 2 are clearly larger than the recirculation zone of Case 1. There are more swirls in recirculation zone of Case 3 compared to Case 2 and Case 1. These swirls somehow affect the drag force of the vehicle and thus increase the drag coefficient. As can be seen from the tabulated results, the drag force of a moving vehicle increased even at low speed. It can be concluded that driving a vehicle with rear spoiler at lower speeds would only increase unnecessary drag to the vehicle.

4. Conclusion

The drag force, lift force and flow characteristics of a sedan vehicle, with and without a spoiler, at the variable speed of 60 km/h, 90 km/h and 110 km/h were numerically investigated. However, due to lack of converged solution, time and CPU consuming for each simulation, low mesh resolution settings are used throughout the simulation. It is known that higher mesh resolution leads to more accurate results. The use of rear spoilers proved to be effective in increasing down force. It is known that having down force (negative lift force) generates the following advantages:

i. Increase cornering force
ii. Increase stability of vehicle at high speed
iii. Improves traction
iv. Improves braking performance

Rear spoiler also increases the drag of the sedan vehicle. However, having more down force (negative lift force) can be more important than having less drag since the main priority is to drive safer. This fact should be kept in mind that achieving the benefits of rear spoiler are usually only appear at high speeds. A spoiler may negatively impact the vehicle performance at low speeds. Therefore, the objectives of this study are achieved. Based on the results obtained through the simulation, it is clearly indicated that rear spoilers, which in this case are the ducktail spoiler and rear wing, increase the sedan vehicle drag and lift (down force). The results show a remarkable down force acting on moving vehicles with rear spoilers. The benefits of rear spoiler do not emerge at low speed, but instead increase the drag of the vehicle even at low speed. This will reduce the aerodynamic performance of the vehicle as well as fuel consumption. Therefore, the use of rear spoiler is only significant for vehicle moving in expressways since the expressways have a speed limit of 110 km/h. Spoiler only negatively impact the sedan vehicle performance if it is driven in federal roads and town areas which have the speed limit of only 60 km/h and 90 km/h.

Acknowledgement

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

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

[5] Senan Thabet, Thabit Hassan Thabit. "Computational Fluid Dynamics: Science of the Future" International Journal of Research and Engineering, vol. 5, no. 6, June 2018 [Online]. Available: https://www.researchgate.net/publication/326238568_Computational_Fluid_Dynamics_Science_of_the_Future . [Accessed September 5., 2020].

[6] Xu-xia Hu, Eric T.T. Wong. “A Numerical Study On Rear-spoiler Of Passenger Vehicle”, World Academy of Science, Engineering and Technology International Journal of Mechanical and Mechatronics Engineering vol. 5, no. 9, 2011, doi: doi.org/10.5281/zenodo.1075368.

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