



# Simulation The Effect of Spoiler on Aerodynamic for Sedan Car

Muhammad Zakie Mohd Zin<sup>1</sup>, Wan Mohd Wardi Wan Abdul Rahman<sup>1\*</sup>

<sup>1</sup>Centre of Automotive and Powertrain Technology, Faculty of Engineering Technology,  
Universiti Tun Hussein Onn Malaysia, 84600 Pagoh, Johor, MALAYSIA

\*Corresponding Author

DOI: <https://doi.org/10.30880/japtt.2022.02.01.001>

Received 20 February 2022 ; Accepted 25 April 2022 ; Available online 19 June 2022

**Abstract:** The goal of this research is to select the parameter of the car body at different velocity with different spoiler and to stimulate and analyse the drag coefficient and lift coefficient of the car body at different velocity with different spoiler. Moreover, aerodynamics in vehicles now play an important role. Aerodynamics affects vehicle performance by changing parameter such as lifting force and drag force, which play a major role in high speeds and fuel economy. The benefit of this research is to reduce drag and prevent undesired lift forces and other causes of aerodynamic instability when travelling at high speeds. For the analysis, the model is created in SOLIDWORKS and then imported into Ansys Workbench and Fluid Flow (Fluent) for further analysis. There are four models of car body in this simulation, which is Model 1, Model 2, Model 3 and Model 4. To collect data on the drag coefficient, lift coefficient, pressure contour and velocity streamline, this simulation was run at five different velocity which is 120 km/h, 140 km/h, 160 km/h, 180 km/h and 200 km/h. In conclusion, spoilers have a significant impact on drag reduction and the prevention of undesired lift forces.

**Keywords:** Aerodynamic, drag coefficient, lift coefficient

## 1. Introduction

The study of the aerodynamics of a road vehicle is known as automotive aerodynamics. One of its primary objectives is to reduce drag and wind noise while also minimising noise emission and preventing undesired lift forces and other causes of aerodynamic instability when travelling at high speeds. In this case, air is also considered to be a fluid. When it comes to some racing vehicle classes, it may also be necessary to generate down force in order to improve traction and thus the ability to corner [1]. The surface or shape of the existing design can be modified to reduce drag forces, preventing undesired lift forces and improve aerodynamics. It will reduce the drag forces and lift forces acting on the vehicle and will hence provide a better fuel economy. Automobile manufacturers began incorporating vehicle body designs that would allow the vehicle to be more streamlined in order to reduce the amount of drag created by the vehicle. Methods of reducing the drag of a vehicle include shaping the rear end, covering the underside of the vehicles and reducing the number of protrusions on the vehicle surface. Optimizing the shape of a vehicle in order to reduce aerodynamic drag can allow vehicle designers to build cars that are more fuel-efficient than their competitors [1].

In order to assess the performance of a vehicle, the aerodynamic elevator is equally important in view of the vehicle's stability. Tractive strength is generated by the moving vehicle. The tractive force tends to be limited by the amount of tangential force passed on by the wheels without spin [2]. The aerodynamic properties of a vehicle have a significant impact on its performance, handling, safety, and comfort, among other characteristics. Extra parts are added to the body, such as rear spoilers, lower front and rear bumpers, air dams, and numerous other aerodynamics aids, in order to direct airflow in a variety of ways and reduce drag while also improving the overall stability of the vehicle [3]. A drag force is the resistance force caused by the body's motion through fluid such as air or water. A drag forces act against the direction

of the incoming flow velocity. About 50-60% of the total fuel energy is lost to defeat the drag force and between 30-40% to overcome road resistance and electricity consumes 10-20%. Thus, aerodynamic drag drop has become one of the major factors in the design of vehicles. Such vehicles are generally identified as unproductive aerodynamically compared to other ground vehicles, due to their large frontal areas and bluff-body forms. Drag is caused by the difference in velocity and pressure between the solid object and the fluid [4]. The objective of this research is to select the parameter of the car body at different velocity with different spoiler and to simulate and analyse the drag coefficient and lift coefficient of the car body at different velocity with different spoiler. Therefore, manufacturers began incorporating vehicle body designs that would allow the vehicle to be more streamlined in order to reduce the amount of drag generated by the vehicle and prevent the vehicle from producing undesired lift forces. To test crashworthiness prospectively, multiple parameters are used including the drag coefficient and lift coefficient of the car body of the vehicle. So, analysing data such as lift coefficient and drag coefficient are needed.

## 2.0 Materials and Methods

### 2.1 Methodology Flow Chart

Figure 2.1 shows the methodology flowchart that used as a guide to construct this study. The beginning of this project is to get the literature review from another researcher's journal on how to start the project. Draw the aerodynamic car body in SOLIDWORKS and import the file in IGES format file into the Workbench. Edit the model and it imported into explicit dynamic analysis. This simulation of an aerodynamic car solved by using four different types of models, which is Model 1, Model 2, Model 3 and Model 4. The velocity in this simulation is 120 km/h, 140 km/h, 160 km/h, 180 km/h and 200 km/h. The solution that inserted to explicit dynamic analysis is drag coefficient and lift coefficient. Collected the data after the simulation had been solved. After the simulation data have been collect, the result from the simulation aerodynamic car body are compared. The collected data then be analyse and discuss. The last step is to do the conclusion by writing the conclusion and recommendation based on the simulation that have been done.

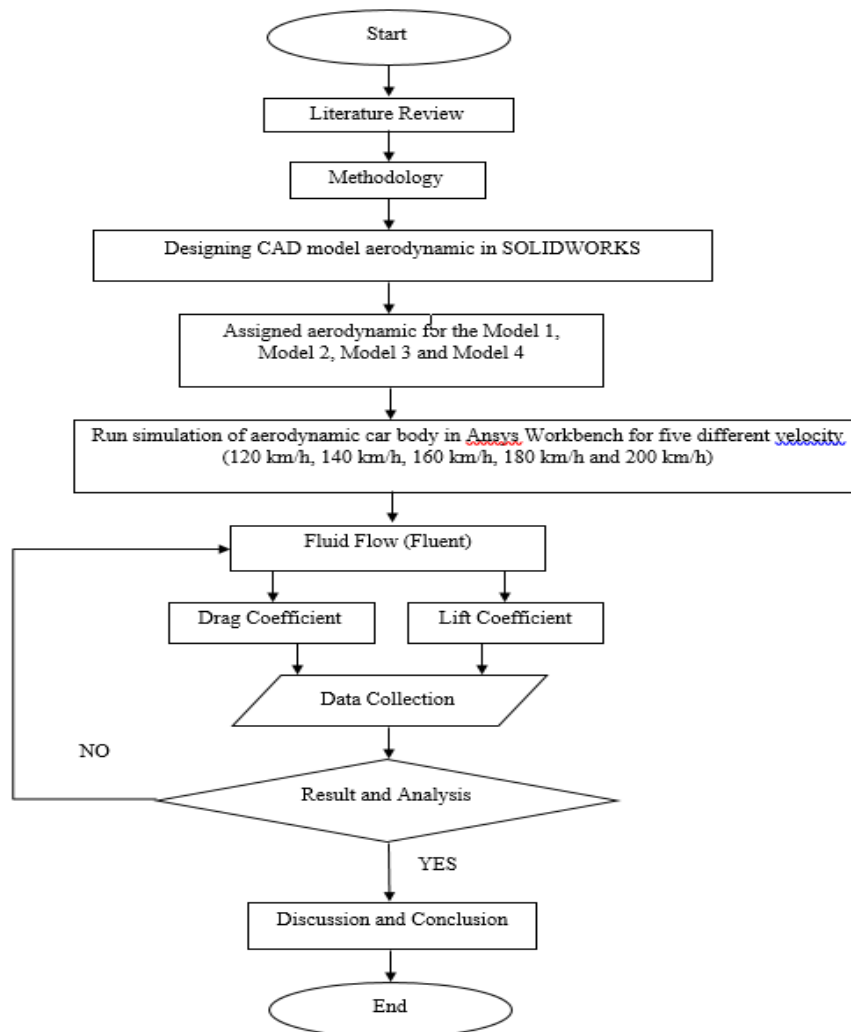


Figure 2.1 - Methodology flow chart

## 2.2 Selection of Parameter

In this study, car body that been design in SOLIDWORKS is use as test panel and collect the data from explicit dynamic analysis ANSYS Workbench simulation.

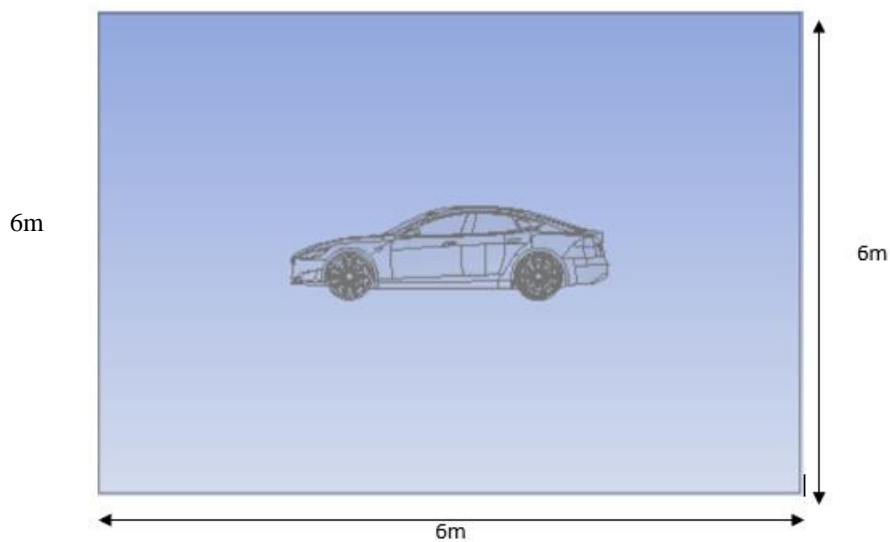
Five different velocity which is 120 km/h, 140 km/h, 160 km/h, 180 km/h and 200 km/h for the four type of models were conducted in order to collect the data from the solution of Fluid Flow (Fluent).

## 2.3 Simulation Software

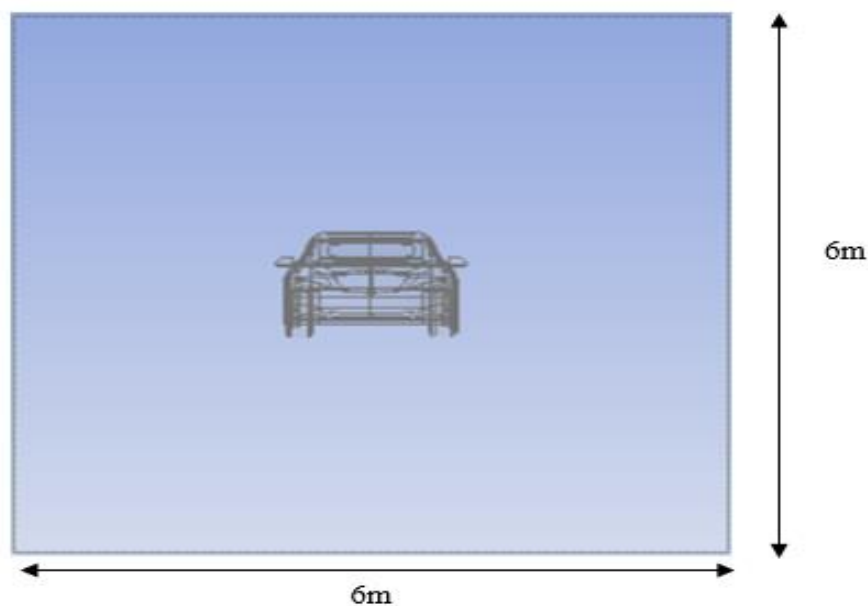
Experimental apparatus is a set of software designed for a particular use. In this experiment, the software that been used are Ansys Workbench 2021 R2 software, SOLIDWORKS software and Fluid Flow (Fluent).

## 2.4 Boundary Condition

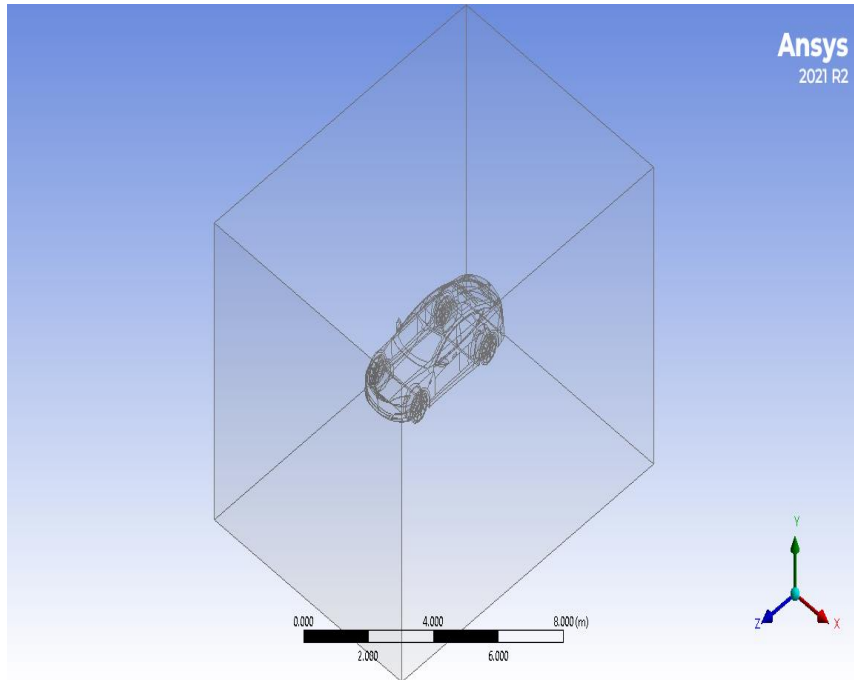
Figure 2.2 and Figure 2.3 illustrates the side view of enclosure and front view of enclosure. Figure 2.4 clearly indicates the boundary conditions for the enclosure surfaces that consists of velocity inlet, velocity inlet top, velocity inlet side, velocity inlet side 2, velocity inlet bottom and pressure outlet. Table 2.1 show boundary condition of this research.



**Fig. 2.2 - Side view of the enclosure**



**Fig. 2.3 - Front view of the enclosure**



**Fig. 2.4 - Boundary conditions for the enclosure surfaces**

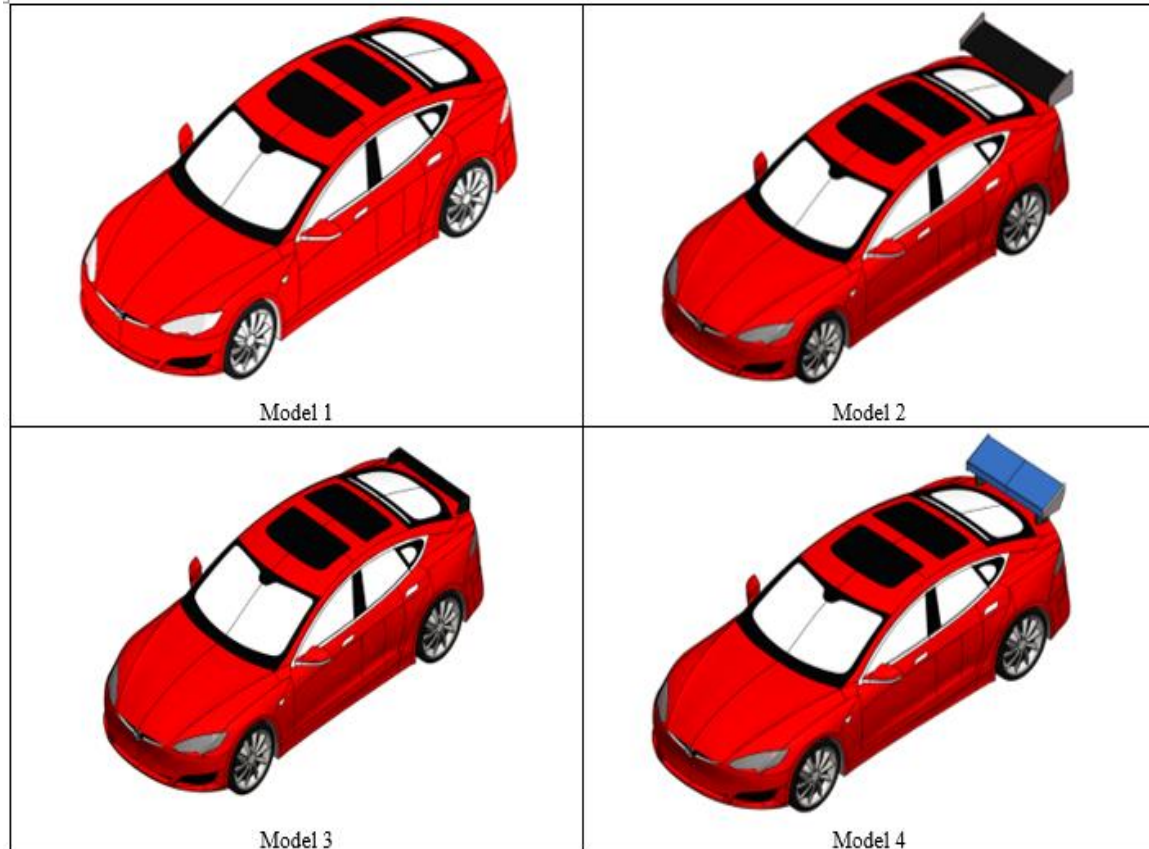
**Table 2.1 - Boundary condition**

Boundary	Value
Car Body Material	Alumina and carbon steel
Width Dimension	2189 mm
Length Dimension	4980 mm
Height Dimension	1440 mm
Body Interaction	Frictionless
Velocity Inlet Condition	120 km/h, 140 km/h, 160 km/h, 180 km/h and 200 km/h
Pressure Outlet Condition	Pressure (0 Pa)
Wall Condition	Stationery and no slip
Weight	2250 kg

## 2.5 Model Design

SOLIDWORKS software was used to create the model. Model 1, Model 2, Model 3 and Model 4 are the four types of car bodies that have been designed. Isometric view of Model 1, Model 2, Model 2 and Model 4 are shown in Table 2.2.

**Table 2.2 - Isometric view of the model**



## 2.6 Material

In Ansys Workbench, a alumina material has been assigned to a car body model and carbon steel for spoiler. Alumina and carbon steel material is obtained from general material data sources in SOLIDWORKS, which is a database of engineering data.

## 2.7 Simulation Procedure

For analysis using the finite element method, the following experimental procedure may be used. First, open Ansys Workbench and then choose fluid flow (fluent) from the analysis systems. For the first aerodynamic car body, choose Model 1. Next, the simulation model is designed in SOLIDWORKS. The dimension for the model is 2189 mm x 4980 mm x 1440 mm. The drawing then been saved in IGES format. Import the model's IGES file into the geometry programme. Next, generate enclosure of each side which is 3m. Mesh generation can be performed after a material has been assigned to the model. Meshing is an important part of the analysis process, and it should be carried out on the vehicle's structure. Meshing is the process of dividing a created model into a number of divisions or elements, each of which is made up of nodes, in order to simplify it. By using a meshing process, we can determine the efficiency and effectiveness of any analysis. In setup setting, velocity are inserted. The velocity are defined by component. The component of velocity is Z-axis. The velocity for the first simulation is 120 km/h. In the analysis, fixed support is applied to the wall, so that the wall is stationary and no slip in all degrees of freedom. Insert the results of the lift coefficient and drag coefficient analyses at the end of the solution. Solve the simulation then and wait for the solution. Finally, repeat procedure with velocity of 140 km/h, 160 km/h, 180 km/h and 200 km/h including Model 2, Model 3 and Model 4.

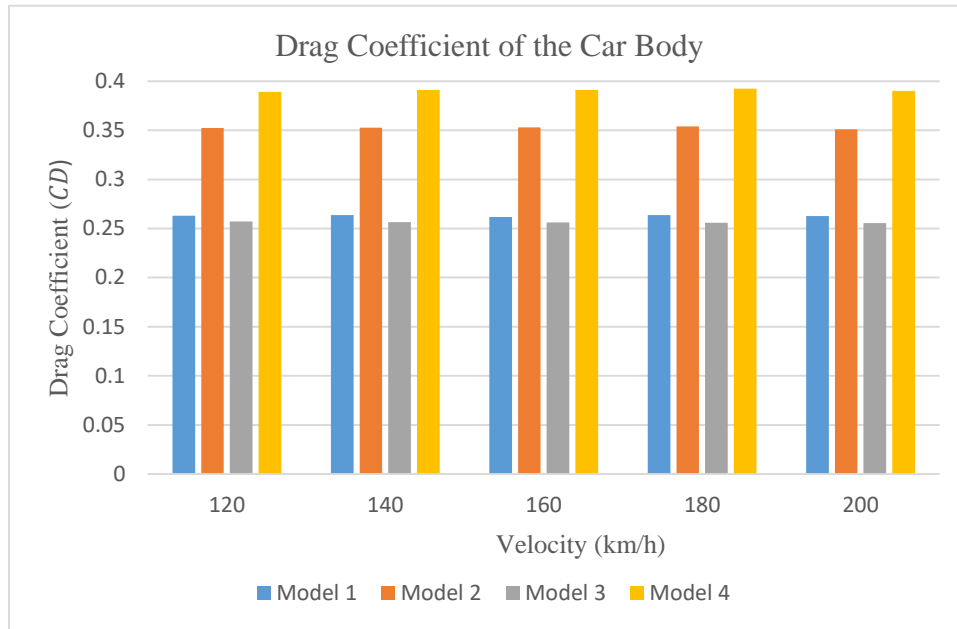
## 3.0 Results and Discussion

### 3.1 Drag Coefficient

Table 3.1 and Figure 3.1 shows the lift coefficient of the car body with different velocity with different spoiler.

**Table 3.1 - Data drag coefficient of the car body with different velocity and different model**

Velocity (km/h)	Drag Coefficient ( $C_D$ )			
	Model 1	Model 2	Model 3	Model 4
120	0.26318797	0.35240334	0.25708303	0.38914806
140	0.26379228	0.35271138	0.25636202	0.39107987
160	0.26176608	0.35298547	0.25631449	0.39109123
180	0.26362112	0.35402429	0.25589284	0.39236024
200	0.26273471	0.35110497	0.25554880	0.39013293



**Fig. 3.1 - Comparison drag coefficient of the car body with different velocity and different model**

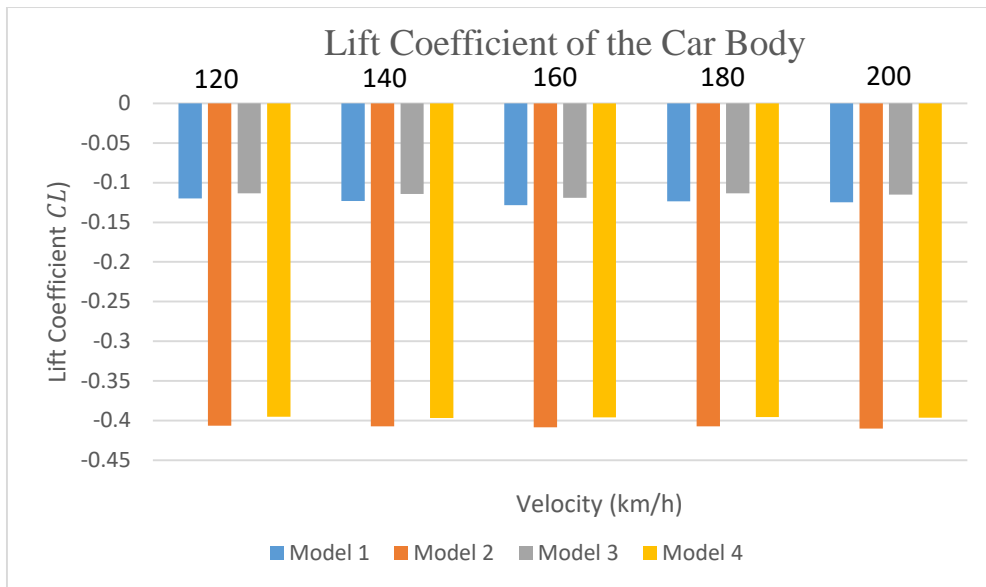
Based on Table 3.1 and Figure 3.1, the lowest drag coefficient of the car body is Model 3 at velocity 200 km/h. The highest drag coefficient of the car body is Model 4 at velocity 180 km/h. Designing spoilers vehicle requires an understanding of aerodynamics to create better flow and reduce drag. The better a design for aerodynamics, the better fuel efficiency a model will have. It reduces the amount of turbulence the vehicle generates as it moves. Thus, it helps to spoil or reduce the turbulent flow and air drag. According to the simulation results, spoilers have a significant impact on determining which vehicles have the best and worst drag coefficients. Following that, this result indicates that Model 3 has the best spoiler compare to Model 1, Model 2 and Model 4. Besides, Model 4 is the worst spoiler model and it might be dangerous on the road if driven at high speeds.

### 3.2 Lift Coefficient

Table 3.2 and Figure 3.2 shows the lift coefficient of the car body with different velocity with different spoiler.

**Table 3.2 - Data lift coefficient of the car body with different velocity and different model**

Velocity (km/h)	Lift Coefficient ( $C_L$ )			
	Model 1	Model 2	Model 3	Model 4
120	-0.11975095	-0.40658292	-0.11327268	-0.39500180
140	-0.12313474	-0.40713093	-0.11441892	-0.39665228
160	-0.12825942	-0.40843251	-0.11922401	-0.39581904
180	-0.12353367	-0.40715051	-0.11336596	-0.39542696
200	-0.12464764	-0.41022152	-0.11525322	-0.39628556



**Fig. 3.2 - Comparison lift coefficient of the car body with different velocity and different model**

Based on Table 3.2 and Figure 3.2, the highest lift coefficient of the car body is Model 3 at velocity 120 km/h. The lowest lift coefficient of the car body is Model 2 at velocity 200 km/h. The spoilers act as means to create better airflow around and over the car which results in better grip on the road. This way, cars moving faster are easier to control without needing extra weight to be added. This spoiler also reduces the effect of the turbulence created by the airflow. It does so by spoiling the streamlined flow of air and providing a cushion like boundary layer. However, different types of airflow may need spoilers to work differently. Hence, it is clear from this data that spoiler on Model 3 is the best at preventing undesired lift forces. Furthermore, the spoiler on the Model 2 is the worst and might be unsafe when driving at high speeds. It can be concluded from these data and graph that spoilers have a significant impact on lift coefficient.

### 3.3 Pressure Contour

When the flow come and hit the body, the high pressure is acting on the front bumper of the car because the flow gets hit directly and then the pressure become decrease when it flows to the car hood because the flow separation at right and left. The pressure at the windshield is quite high because the wind gets hit directly and when it flows to the roof the pressure is low because of the flow separation at the roof and high again when it moves to spoiler. The lowest acting on vehicle surface is at the back bumper because there is no wind hit it. The highest pressure contour is Model 4 at 200 km/h which is 1938.647 Pa. The high pressure shifted to windward and front of the car. At the windshield area, there is a little pressure acting and the pressure is low at the leeward because of the flow separation that happens to the leeward area. Then, when the velocity increased to 120 km/h, 140 km/h, 160 km/h, 180 km/h and 200 km/h the same thing happens where the high pressure slowly shifted more to the windward area and low at the leeward area and the top of the car because of the flow separation depend on the model of spoilers. Thus, it can be said that spoilers play a significant influence in reducing the pressure contour.

### 3.4 Velocity Streamline

The velocity streamline were acquired on a plane placed horizontally across the car which originates from the middle of the car. The highest velocity streamline is Model 2 at 200 km/h which is 102.516 ms<sup>(-1)</sup>. There is high pressure in front of the car and low pressure behind the car. The velocity streamlines directly hit the front of the bumper and its flows on the car. There is a slight vortex formed at the back of the car due to the flow separation and reattachment. When the velocity increased from 120 km/h to 200 km/h, the velocity streamline also increase depend on the model of spoilers. Thus, it can be concluded that spoilers play an important role to reduce the velocity streamline.

### 4.0 Conclusion

In conclusion, the objective of this simulation is to select the parameter of the car body at different velocity with different spoiler and to stimulate and analyse the drag coefficient and lift coefficient of the car body at different velocity with different spoiler. The material that been assigned to the car body is alumina and carbon steel for the spoiler. The simulation was conducted on different velocity which is 120 km/h, 140 km/h, 160 km/h, 180 km/h and 200 km/h. After performing analysis using ANSYS Workbench on four different model which is Model 1, Model 2, Model 3 and Model

4 in aspect of drag coefficient and lift coefficient analysis, the results show the Model 3 has been dominated the drag coefficient and lift coefficient analysis. So, from the analysis that have been made, the conclusion can be made where the spoiler of Model 3 is the best in aspect of aerodynamics and make it better in lift and lowest the drag on the road.

### **Acknowledgement**

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

### **References**

- [1] C. M. M and Y. Priyanka, "Aerodynamic Analysis on a Car To Reduce Drag Force Using Vertex Generator," *Int. Res. J. Eng. Technol.*, vol. 6, no. 11, pp. 1833–1843, 2019
- [2] R. P. Verma, N. Kumar Chaudhary, and S. Avikal, "Effect of direction of lip spoiler on the aerodynamic performance of a small passenger vehicle," *Mater. Today Proc.*, no. xxxx, 2021, doi: 10.1016/j.matpr.2020.12.448
- [3] P. Gowdhaman, K. Antonyraj, and V. Annamalai, "An effective approach on physical and dielectric properties of PZT- PVDF composites," *Int. J. Adv. Sci. Res.*, vol. 1, no. 08, pp. 322–328, 2015, doi: 10.7439/ijasar
- [4] P. A. Nigal Ashik, P. Suseendhar, N. Manoj, and S. Wasim Feroze, "Reduction of drag in box-type and half streamlined automobile vehicles," *Mater. Today Proc.*, no. xxxx, 2020, doi: 10.1016/j.matpr.2020.11.109.