Research Progress in Mechanical and Manufacturing Engineering Vol. 2 No. 2 (2021) 1003-1008

RPMME

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

A Study on Aerodynamic Effect of a Generic Car Spoiler Using Computational Fluid Dynamics (CFD)

Muhammad Lokman Hakim Shafee[1], Azwan Sapit[2]*

[1]Faculty of Mechanical and Manufacturing Engineering,
University Tun Hussein Onn Malaysia, Batu Pahat, 86400, MALAYSIA

[2]Combustion Research Group, Faculty of Mechanical and Manufacturing Engineering,
University Tun Hussein Onn Malaysia, Batu Pahat, 86400, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.02.109
Received 30 July 2021; Accepted 30 Nov. 2021; Available online 25 December 2021

Abstract: Most of the time, the car normally breaks through the air barrier when a human drives the vehicle. In an undesired condition where the handling and stability of the car can deteriorate as a result of tires losing the traction to the road because of lift, it can be encountered by installing the spoiler. In this research work, the properties of air flow have been investigated around the spoiler where the value of the drag and lift forces and its coefficient has been computed. The research is to distinguish the forces acting on three types of spoiler with a varied angle of attack and subjected to different velocity. Ansys Fluent is utilised to analyse and simulate a computational fluid dynamic (CFD) technique. It can be seen the angle of attack and velocity affected the magnitude of lift and drag force and their coefficient. At velocity = 45 m/s, all spoiler 1, 2 and 3 experienced downforce with lift coefficient at -0.02632, - 0.03705 and -0.08887, respectively. For this kind of spoiler, the adjustment of angle of attack onspoiler 2 at 10° AOA could be ideal in terms of lift and drag coefficient optimization based on the Lift over Drag Ratio recorded at 4.56. The best spoiler design should have the least drag coefficient and considerably higher negative lift coefficient.

Keywords: Ansys Fluent, Spoiler, Lift & Drag Coefficient

1. Introduction

There are many types of aerodynamic devices that can be integrated to the car body as additional parts such as spoiler, side skirt, diffuser, winglets and vortex generator. Most commonly used is the rear spoiler placed on the trunk of the car. Lack of spoiler could lead to a disastrous situation of deteriorating handling and stability especially when the car is in motion with high speed.

Technically, by collecting more air refills in the low pressure area, a spoiler controls the air flow around the rear end so that more high pressure region is generated for better traction and the car stick still to the road [1]. The objectives of this study are to simulate the air flow around the spoiler for different air speed and to compare the air flow analysis in terms of CD and CL of three different degree angle of attack of car spoiler.

1.1 Lift concept

Lift is a force that opposes the weight of the object and holds the object in the air. In this case which is cars, the pressure difference of the frontal end which is high pressure to the lower pressure at the back end generates lift. This is correlated to the simple physics explained in Bernoullis' principle. According to David Bernoulli, the pressure is directly proportional to the local velocity of the air. This varies whenever velocity changes, the pressure also changes. Lift is always perpendicular to the airflow over the cars. This can be described by the following equation of aerodynamics:

$$F_L = 1/2\ \rho v^2 C_L A$$

1.2 Drag concept

In general, drag is referred to as the resistive force experienced by an object or body while in motion with respect to the fluid surrounding it. Drag forces depends to velocity of the object and is defined by the formula:

$$F_D = 1/2\ \rho v^2 C_D A$$

1.3 Bernoulli's equation

A principle emphasized called Bernoulli's equation which held for fluids in ideal state, where speed at a point in a fluid and pressure are inversely proportional to each other. In a simpler meaning, high moving fluids exert less pressure than slower fluid speed [2].

$$P + \frac{1}{2}\rho v^2 + \rho g h = constant$$

1.4 Relation between drag and lift coefficients

The lift/drag ratio is calculated by dividing the lift coefficient by the drag coefficient, CL/CD, to represent the relationship between lift and drag. The L/D ratio reflects the efficiency of an airfoil. Higher L/D ratio aircraft are more efficient than lower L/D ratio aircraft.

2. Materials and Methods

Figure 3-5 shows the dimension of the spoiler where recorded a length span of 1000 mm and height of 146 mm. The chord length of the spoiler is measured at 153 mm.

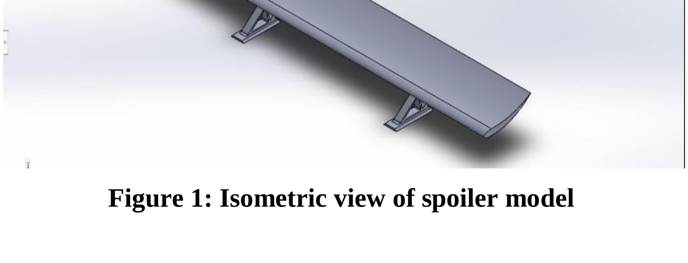

Figure 1: Isometric view of spoiler model

2.1 Computational domain

In this case study, researcher use external aerodynamics approach as it suits the air flow analysis over the spoiler surface. In external aerodynamics, CFD simulates the flow around the geometrical model, therefore the computational domain is a volume of reasonable dimensions around the geometrical model. The computational domain resembles a rectangular box. This case study involves the presents of physical boundaries act like the wall of wind tunnel.

2.2 Meshing

The purpose of this process is to discretise the flow domain into cells and elements. In order to assist the meshing process, ANSYS Meshing is utilized. Initially for mesh generation, the ANSYS Meshing will generate the meshes regarding the default settings. Due to the curvatures around the car body detected by the ANSYS Meshing, improvement of mesh needs to be taken such as modifying the mesh sizing parameter [3].

2.3 Setup parameter

Table 1: Setup

Item	Parameter Name	Variable Value	Unit or Dimension
1	Velocity	15, 30, 45	m/s
2	Angle of attack	0, 10, 20	°
3	Number of iterations	500	Dimensionless
4	Type of air flow	k-epsilon	Dimensionless
5	Density of air	1.225	kg/m^3

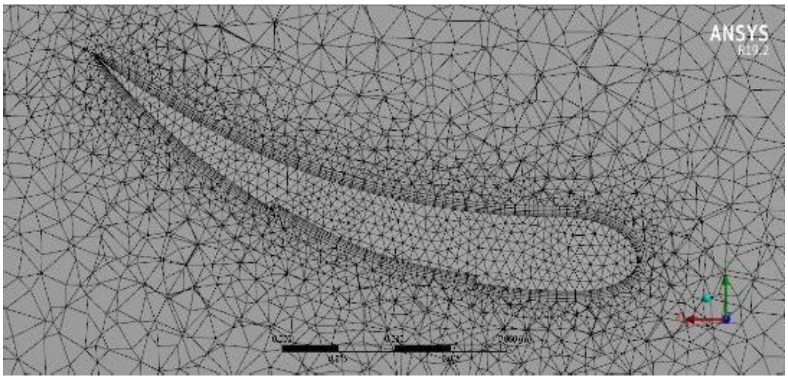

Figure 1: Sample of mesh generation with alteration

3. Results and Discussion

The findings to be compared in terms of lift and drag force and coefficient of lift and drag, C_D & C_l. Furthermore, the analysis to be assisted by the means of velocity streamline, velocity contour and pressure contour. The findings will be shown and presented through table, graph and contour display. The aerodynamic effect of car spoiler has been analysed to study the fluid flow characteristics from 0 degree, 10 degree to 20 degree angle of attack.

3.1 Results

Table 2: The data comparison of lift coefficient of three spoiler

Velocity (m/s)	Lift coefficient 1	Lift coefficient 2	Lift coefficient 3
15	-0.02531	-0.03640	-0.08708
30	-0.02605	-0.03682	-0.08830
45	-0.02632	-0.03705	-0.08887

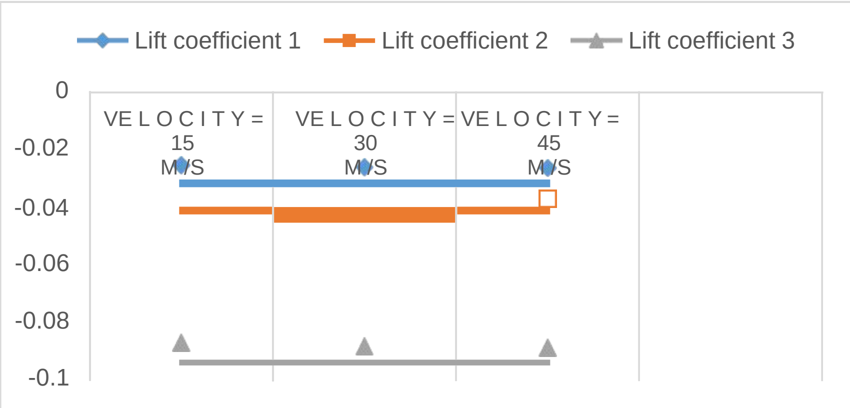

Figure 2: The graph of lift coefficient against velocity (m/s).

All types of spoiler show decreasing trend of lift coeffient as the velocity increases. All spoiler experienced downforce as lift coefficient experience slight changes due to velocity.

Table 3: The data comparison of drag coefficient of three spoiler

Velocity (m/s)	Drag coefficient 1	Drag coefficient 2	Drag coefficient 3
15	0.00765	0.00808	0.02234
30	0.00749	0.00805	0.02228
45	0.00744	0.00805	0.02227

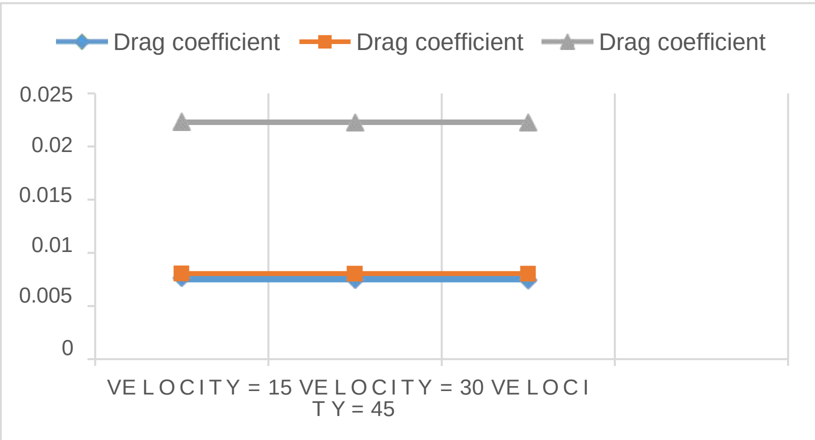

Figure 3: The graph of drag coefficient against velocity (m/s).

In terms of drag, all three types of spoiler recorded an decrease coefficient trend once the velocity increase. Drag coefficient for spoiler 3 recorded the highest drag coefficient compared to another two spoiler.

3.2 Discussions

In order to select the appropriate spoiler which satisfy the requirement of greater downforce and smaller drag, implementation of the Lift to Drag Ratio (L/D) has been used. The average lift coefficient was divided by the average drag coefficient to obtain the L/D ratio for all types of spoiler. The result will indicate the ideal efficiency of the spoiler in terms of lift and drag relationship. The results of the ratio are as follows:

Table 4: L/D ratio

Types	L/D Ratio
Spoiler 1 (0° AOA)	3.44
Spoiler 2 (10° AOA)	4.56
Spoiler 3 (20° AOA)	3.95

Based on the results of L/D ratio, we can assume that Spoiler 2 with 10° angle of attack possess the highest L/D ratio which, in conclusion, indicating the spoiler experienced the greater lift at reduced drag compared to spoiler 1 or 3.

4. Conclusion

The objectives of this study were achieved where to simulate the air flow around the spoiler by using computational fluid dynamic software such as Ansys Fluent. The study also enable the researcher to compare the air flow analysis in terms of lift force, drag force and their coefficient acting on three types of spoiler which subjected to dissimilar angle of attack at separate velocity. Moreover, the velocity streamline, velocity contour and pressure contour were generated during the analysis.

It can be understood that the greater the velocity magnitude, the greater the drag force generated and while the spoiler is to be a concern in terms of downforce, it may only works well in an ideal angle of attack. As for this study, all spoiler which adjusted at 0, 10 and 20 degree were able to generated sufficient downforce despite varies in drag. Hence, to make it worth discussion, the lift over drag ratio result may indicated spoiler 2 (10 degree) as a better option where it recorded highest ratio at 4.56. This makes it an indicator of having greater lift at reduced drag. It is known that there are various advantages of having downforce such as improves braking and cornering performance other than improve better traction to the vehicle.

The selection of finer angle of attack could lead to a better ratio between angle 10 to 20 degree for this kind of spoiler. Moreover, as the development of technology in automotive industry growing rapidly, it could be beneficial for the future researchers to study the engagement of hydraulic system in spoiler design where the angle and height of the spoiler might be varies according to the velocity. The implementation of software that automatically control the spoiler height or angle of attack could make the research exciting to study.

Acknowledgement

The authors wish to thank to the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia that has supported on the accomplishment of research activity.

References

[1] Ipilakyaa, T. D., Tuleun, L. T., & Kekung, M. O. (2018). Computational fluid dynamics modelling of an aerodynamic rear spoiler on cars. Nigerian Journal of Technology, 37(4), 975. https://doi.org/10.4314/njt.v37i4.17

[2] Bajpai, P. (2018). Hydraulics. In *Biermann's Handbook of Pulp and Paper* (pp. 455–482). Elsevier. https://doi.org/10.1016/B978-0-12-814238-7.00023-4

[3] Cakir, M. (2012). Scholar Commons CFD study on aerodynamic effects of a rear wing/ spoiler on a passenger vehicle. http://scholarcommons.scu.edu/mech_mstr