Aerodynamics Analysis of UniMAP Automotive Racing Team 
Formula SAE Race Car spoiler via Simulation: Effect of Angle of Attack

A Wafi¹, M H Basha¹, M Tasyrif², M F Hamid²

¹Faculty of Mechanical Engineering Technology, Universiti Malaysia Perlis, Pauh Putra Campus, 02600 Arau, Perlis, Malaysia.
²School of Mechanical Engineering, Engineering Campus, Universiti Sains Malaysia, 14300, Nibong Tebal, Pulau Pinang, Malaysia.

Email: hafibasha@unimap.edu.my

Abstract. When a racing car moves at high velocity, aerodynamic factors will cause a lift and drag force, causing the speeding vehicle to lose stability, traction and speed. To solve this problem, racing cars use a rear spoiler as a medium to reduce drag and lift, increasing the car’s stability. This study explores the idea that focuses on the impact of the angle of attack on UniAR T SAE race car performance and the effect of angle of attack on drag force and downforce. ANSYS Fluent is used to generate the simulation. In the context of the lift force analysis, angle of attack equal to 12.5° produce the most outstanding results and generated the highest negative lift force value between three tested angles. The highest negative lift force generated is 35.658 N at the inlet velocity of 105 km/h. Furthermore, by referring to drag force analysis, it shows an increasing trend when the variable angle of attack parameter is changed to a higher value. Angle 25° produced the highest drag force data which is 60N and the inlet velocity condition is set equal to 105 km/h.

1. Introduction

Formula SAE is a design competition organized by the Society of Automotive Engineers (SAE) in which student engineers design, build, test and race an open-wheeled formula-style race car. Overall, all the automotive aspects are considered critically during team designs, builds and tests a prototype based on a series of rules which in purpose to ensure onsite event operations and promote clever problem solving [1]. Hence, aerodynamics is one of the crucial factors to be considered as automotive engineer thinking. Aerodynamics is the concept that focuses on the interaction between the motion of airflow and bodies moving through air [2]. The main concerns of automotive aerodynamics are reducing drag and wind noise, improving vehicle stability, and preventing undesired lift forces at high speeds. Spoiler is one of the devices that is invented based on many research with the purpose of utilising the aerodynamics concept. In this project, the factor to be focused on are the drag force and downforce due to the different angles of the spoiler on the rear side of the car. In other research, data gain conclude that the lift coefficient of a fixed-wing aircraft is affected by the angle of attack [3]. On the other hand, an inverted airfoil that is spoiler design will affect the flow of the process in an inverse way, where the downforce is associated with the increase of the angle of attack. Hence, the angle of
attack is a critical point to be considered during the design and manufacturing process of a spoiler. Furthermore, this research objective is to study the effect of angle of attack towards spoiler performance in the context of drag force and downforce analysis.

2. Methodology

2.1 Vehicle generic models and dimensions
Some guidelines are highlighted by the SAE organization in determining the wheelbase parameters, which is the car body design, must have a wheelbase of at least 1525mm (60 inches). CATIA software is a powerful tool that is used to sketch and design the car generic model. The wheelbase for these body components is 1581.07mm, the height average is 500mm, and the width is 1218.75 mm. Hence, whole body specifications follow the guidelines that have been assigned by the SAE organization.

2.2 Spoiler generic models and dimensions
Spoiler size has been set equal to 181.74 cm for width parameter and 1005 cm for height parameter. The spoiler is designed with a contour shape to induce a new turning of the airflow creating the downforce [4]. The dimension of the spoiler is compatible for installation on the rear part of the car. The use of CATIA software to design and sketch the whole model is important in accelerating the entire research process.

![Figure 1: Car model assembled with spoiler](image)

2.3 Determination of car speed in surrounding channel
The power of 600cc engine and the maximum of car speed produced on the road are identified through speed testing process held at the UniMAP circuit. Maximum car speed is determined to be equal to 85 km/h based on data analysis. For the purpose of further research, the variable of speed is manipulated into a low-speed prediction of 65km/h and a high-speed prediction of 105km/h. By focusing to recreate airflow velocity based on real situations, the testing and prediction speed result have been set as inlet velocity on ANSYS fluent software.

2.4 Simulation process in ANSYS workbench.
Car model and spoiler design assembled in CAD software have been save in igs format to make it compatible to be transferred to ANSYS workbench. ANSYS fluent software has been used for simulation process that focuses on the analysis of fluid flow in this research. To create a situation that is similar to the actual situation, the speed of the car in this experiment was adjusted in conjunction with the speed resulting from the actual speed of the car on the track during the time trial was perform. During the meshing analysis process, meshing rate for model geometry surfaces has been set to maximum point in order to generate an accurate result. The fluent condition for the car simulation set up by making the viscous model equal to k epsilon (2eq) and the material condition is air for fluid
flow and aluminium for the car body. Furthermore, operation pressure at outlet pressure of surrounding channel will be set to zero and inlet velocity are divided into three variables, which are 65km/h, 85 km/h and 105 km/h, flow. Other than that, turbulent kinetic energy & turbulent dissipation rate as Second-Order Upwind and all other parameter are used as default to ensure that all process run smoothly. Lastly, the result of the simulation will be displayed as the setup settings.

![Simulation arrangement](image)

**Figure 2**: Simulation arrangement

### 2.5 Angle of attack of spoiler
Angle of attack of the spoiler refers to the edge between a reference line on a spoiler (regularly the chord line of an airfoil) and the vector representing the relative movement between the body and the air particle through which it is moving. Hence, in this research, several angles have been decided to be tested on the spoiler, which is 0°, 12.5° and 25°.
3.0 Result and discussion

3.1 The comparison of the lift force results generated from three different angle of attack of the spoiler.

Figure 4 shows the effect of angle of attack of the spoiler on the lift force (downforce) for different velocity. Based on simulations, the downforce value has shown the highest reading when tested with 12.5° angle of attack which is -35.6858 N at inlet velocity equal to 105 km/h. The working principle that occurs at 12.5° is the result of an ideal fraction of air particles when they collide with geometry spoiler. The streamline velocity at the top spoiler is slower than at the bottom, hence based on inversely proportional relation between speed and the pressure at the spoiler increases that in turns press the spoiler geometry down [5]. At 0° angle of attack, it generated an amount of downforce cause division of air through the spoiler geometry. Air particles that are divided on top of the geometry spoiler and the bottom are approximately equal on this condition, hence resulting in low downforce on the geometry surface are exert. By integrating all data, at speed inlet velocity equal to 105 km/h, it’s relatively shown highest value through all the experiment. At angle 25° indicated as the limitation of spoiler angle before its reach the division of the boundary layer that unexpectedly loses downforce at a high angle to the streamline flow. For higher angles of attack, however, the dependence relays in a quite complex scope. As a tested object through the air-surrounding channel, air molecules collide and stick to the geometry surface of the object. This develops a thin layer of air near the surface called a boundary layer that, in effect, changes the geometry of the object. The flow turning reacts to the edge of the boundary layer just as it would to the physical surface of the object. The boundary layer properties may lift off or "separate" from the body and create an effective shape much different from the physical shape [6]. Furthermore, the angle of attack increases the region of the negative pressure tends toward the leading edge and the pressure difference is decreased which is directly proportional with downforce ability [7].
3.2 The comparison of the drag force generated from three different angle of attack of the spoiler
The working principle behind the natural phenomena of drag force is related to the force in the opposite direction of motion that acts on a body moving through the air particle [8]. The skin friction plays a major role in affecting the drag properties based on the area that is attached to air particles and resist it [6]. Furthermore, in spoiler aerodynamics context and study point can be related to frontal pressure effect with the consequence of the air trying to float across the front of the spoiler. Air particles approach the front geometry of the spoiler components, they begin to compress, simultaneously enhance the pressure in the front of the spoiler Through all angles, when inlet velocity is 105 km/h, the highest drag force value was recorded. Reduction in the value of drag force is clearly shown when the inlet velocity parameter was reduced to 85 km/h and 65 km/h. Figure 5 shows the effect of angle of attack of the spoiler on the drag force for different velocity. Drag force shows an increasing trend when the angle of attack parameter increased. At angle of 25°, the highest drag force of 60N was obtained when the inlet velocity condition is set to 105 km/h. 25° angle of attack resulted in the highest drag force compared to other angle of attack tested. Vigorously flow separation at the rear part contribute to the formation of turbulent flow, which acts as a factor that increases the value of drag force in this condition [9]. The extraction data process obtained from the research point shown that at angle 12.5°, it shows second highest value of drag force between three-angle parameter. This angle parameter highest drag force value equal to 40N at inlet velocity equal to 105 km/h. Production of minimum drag force value between three angles of attack have been produced when the angle of attack was changed to a value of 0°. It is obvious so much the greater the angle, the more turbulence exerted at car spoiler or the farther the turbulence of the whirl interior and the spoiler geometry [10]. This implies as a wider range leads to a large stretch out. At 0°angle, the concentration of air particles that acts to the spoiler area point is lower, hence contributed to high streamline velocity and low pressure exerted on the top surface of the spoiler.

4. Conclusion
In the scope of angle of attack, variation of angle in this sub experiment has shown a concrete data to clarify effectiveness of the downforce is directly proportional with the specific parameter of spoiler geometry angle. Angle 25° indicated as the limitation of spoiler angle before it reaches the separation of the boundary layer, which explains why spoiler will abruptly lose downforce at high angles. Furthermore, at variable 0° angle of attack; it still generated the small amount of downforce due to the division of air through the spoiler geometry. Through all angle variable which is 0°, 12.5° and 25°, when inlet velocity equal to 105 km/h, they produce relatively high highest drag force data at that
condition. Hence, proven from the theoretical formula and experiments data, angle of attack has a directly proportional relationship to the effectiveness of the drag force

Figure 5: Drag force result comparison between angle 0°, 12.5° and 25°

Acknowledgement
The authors would like to thank Universiti Malaysia Perlis for the technical support and facilities provided for this work.

Reference
[1] A. Kourta, P. Gilliéron “Impact of the Automotive Aerodynamic Control on the Economic Issues”, Journal of Applied Fluid Mechanics, Vol. 2, No. 2, pp. 69-75, 2009, ISSN 1735-3645
[2] Sumantran, V., & Sovran, G. (1996). Vehicle aerodynamics. Warrendale, PA: Society of Automotive Engineers.
[3] Wolf-Heinrich Hucho: Aerodynamic of Road Vehicle; Fourth Edition; Society of Automotive Engineers, Inc. 1998
[4] Racecar engineering, 2006. F1: Adjustable rear wings how they work. [Online] Available at: http://www.racecar-engineering.com/articles/f1/f1-adjustable-rear-wings-how-they-work
[5] Anderson, John D. (2004), Introduction to Flight (5th ed.), McGraw-Hill, pp. 257–261
[6] Sadraey, M., 2013. Wing Design. [Online] Available at: <http://faculty.dwc.edu/sadraey/chapter 5. Wing Design.pdf>[Accessed 10 June 2014].
[7] Mashud, M., Ferdous, M., & Omee, S. H. (2012). Effect of spoiler position on aerodynamic characteristics of an airfoil. International Journal of Mechanical and Mechatronics Engineering, Bangladesh, 12(6).
[8] Katz, J. (1995). Race car aerodynamics: Designing for speed. Cambridge, MA, USA: R. Bentley
[9] Wang, Y., Xin, Y., Gu, Z., Wang, S., Deng, Y. and Yang, X. (2014) Numerical and Experimental Investigations on the Aerodynamic Characteristic of Three Typical Passenger Vehicles. International Journal of Engineering Technology and Advanced Engineering, 7 (4).
[10] Murugan, C. K. B., Ashik, P. N., & Raju, P. CFD Analysis and Optimization of a Car Spoiler. Journal of Aerodynamics 48.7 (2013)