# STUDY OF F1 CAR AERODYNAMIC REAR WING USING COMPUTATIONAL FLUID DYNAMIC (CFD)

MOHD SHAHMAL BIN MOHD SHAHID

Report submitted in fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

DECEMBER 2010

**UNIVERSITI MALAYSIA PAHANG**

**FACULTY OF MECHANICAL ENGINEERING**

We certify that the project entitled "*Study Of F1 Car Aerodynamic Rear Wing Using Computational Fluid Dynamic (CFD)* "is written by *Mohd Shahmal Bin Mohd Shahid*. We have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

MUHAMMAD AMMAR BIN NIK MU'TASIM

Examiner Signature

## SUPERVISOR'S DECLARATION

I hereby declare that I have checked this report and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

| | |
|---|---|
| Signature | : |
| Name of Supervisor | : MUHAMAD ZUHAIRI BIN SULAIMAN |
| Position | : LECTURER |
| Date | : 6 DECEMBER 2010 |

**STUDENT’S DECLARATION**

I hereby declare that the work in this report is my own except for quotations and summaries which have been duly acknowledge. The report has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature :

Name : MOHD SHAHMAL BIN MOHD SHAHID

ID Number : MH07042

Date : 6 DECEMBER 2010

## ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Mr. Muhamad Zuhairi Bin Sulaiman for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He always impressed me with his outstanding professional conduct, his strong conviction for science, and his belief that a Bachelor's Degree program is only a start of a life- long learning experience. I appreciate his consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naïve mistakes, and his commitment to my future career. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

My sincere thanks go to all my research's mates and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable.

Special thanks should be given to my committee members. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. I would like to acknowledge their comments abd suggestions, which was crucial for the successful completion of this study.

**ABSTRACT**

This thesis deals with the study on the rear wing of the F1 car. This research is about the analysis of drag coefficient, $C_d$ and lift coefficient , $C_L$ using the Ansys software. The analysis consisit of two cases. First case by 2D analysis and other case by using 3D analysis. The design the rear wing of formula one car, based on the FIA 2009 regulation by using Solidwork software. NACA of NACA 2312, NACA 2308 and NACA 2104 were used. The $Re$ = 1.2 $\times$ $10^6$ and $Ma$ = 0.147 are use in the research. 2D result obtained from the research of the NACA 2312, NACA 2308 and NACA 2104 for the drag coefficient, $C_d$ are in range 0.01697 until 0.04340 and for lift coefficient, $C_L$ are in range -0.26427 until -0.35808. 3D result for NACA 2312, NACA 2308 and NACA 2104 on drag coefficient, $C_d$ are in range 0.02979 until 0.04046 and lift coefficient, $C_L$ in range -0.00386 until -0.00434.

# ABSTRAK

Tesis ini adalah berkenaan mengkaji semula sayap belakang kereta formula satu. Kajian ini mengenai analisis pekali rintangan dan juga pekali angkat.mengguanakan perisisan Ansys. Kajian ini juga terbahagi kepada dua kes. Kes pertama mengenai 2D analisis dan kes berikutnya mengenai 3D analisis. Untuk mereke bentuk sayap belakang kereta formula satu, peraturan 2009 FIA digunakan sebagai rujukan dan direka menggunakan perisian Solidwork. Profile NACA 2312, NACA 2308 dan NACA 2104 digunakan dalam kajian ini. $1.2 \times 10^6$ nombor Reynolds dan 0.147 nombor Mach digunakan sebagai parameter dalam kajiana ini. Hasil 2D analisis untuk profil NACA 2312, NACA 2308 dan NACA 2104, pekali rintangan adalah diantara 0.01697 hingga 0.04340. Manakala untuk pekali angkat pula antara -0.26427 hingga -0.35808. Untuk hasil 3D analisis pula,pekali rintangan untuk ketiga-tiga profil NACA itu adalah diantara 0.02979 hingga 0.04046 dan untuk pekali angkat pula diantara -0.00386 hingga -0.00434.

# TABLE OF CONTENTS

## CHAPTER 3 METHODOLOGY



## CHAPTER 4 RESULTS AND DISCUSSION

**CHAPTER 5 CONCLUSION**



**APPENDICES**

# LIST OF TABLE

## LIST OF FIGURE

# LIST OF SYMBOLS

| | |
|---|---|
| $C_L$ | Lift Coefficient |
| $C_d$ | Drag Coefficient |
| $F_d$ | Drag force |
| $F_L$ | Lift force |
| $\rho$ | Density |
| $F$ | Force |
| $\mu$ | Viscosity |
| $Re$ | Reynold number |
| $Ma$ | Mach number |
| $a$ | Speed of Sound |
| $mm$ | Millimetre |
| $m$ | Metre |
| $kg$ | Kilogram |
| $m/s$ | Metre Per Second |
| $m^3$ | Metre Cubic |
| $V$ | Free Stream Velocity |
| $c$ | Chord length |
| $t$ | Thickness |
| $y$ | Y axis |
| $x$ | X axis |
| $A$ | Reference surface area |
| $Pa$ | Pascal |

## LIST OF ABBREVIATIONS

| | |
|---|---|
| CFD | Computational Fluid Dynamic |
| F1 | Formula one |
| NACA | National Advisory Committe for Aeronautics |
| IGES | Initial Graphics Exchange Specification |
| S.I | International System of Units |

# CHAPTER 1

# INTRODUCTION

## 1.1 PROJECT BACKGROUND

Formula one is the one of event of motorsport currently referred to as the FIA Formula One World Championship. In typical of journalist's language, F1 racing in the early 1980's, it's witnessed revolutionary changes because of the introduction of electronic driver aids, and active suspension for the sports cars. These two features were introduced in the Lotus Esprit and Lotus 91 Formula One racing in 1982. In the 1990s traction control and semi-automatic gearboxes were new additions to the sports car models. Ever increasing competition, changes in regulations, the ever increasing need to cut costs by reduced track testing and, maybe in the future, a freeze on design changes collectively contribute to the need of dependence on computer simulations for various aspects of performance enhancement. (FIA, 2007)

Aerodynamics has become key to success in the Formula One sport and spends of millions of dollars on research and development in the field each year. The aerodynamic design has two primary concerns. First, the creation of downforce to help push the car's tires onto the track and improve the cornering force. Secondly, to minimizing the drag that caused by turbulence and act to slow the car down. To create the downforce to the car, the wing operate with air flow at different speeds over the two sides of the wing by having to travel different distances over its contour and form this creates a difference in pressure. This pressure can make the wing tries to move in the direction of the low pressure. One car capable developing cornering force by three and a half times its own weight produces from aerodynamic down force. That means, in theoretical, at high speed the car could drive upside down. (F1, 2010)

The third of the car's total downforce can come from the rear wing. The rear wings of the car create the most drag the rear aerodynamic load. As air flow over the wing, it's disturbed by the shape causing a drag force. Although this force is usually less than lift or downforce, it can seriously limit the top speed and cause the engine to use more fuel to get the car through the air. To assist development of aerodynamic in Formula One cars now days, almost all the team use some software package of CFD. The main advantage is that the results are obtained without construction of the required prototype. The major concern over a software simulation is the validity of its results. The accuracy of the obtained results cannot be guaranteed for a given study. Hence, before analyzing the results obtained from the CFD simulations, a validation study has to be carried out in order to know the specific parameters and conditions under which the software yields the most accurate results when compared to a set of established data. ( Henrik D, 2008)

## 1.2 OBJECTIVES

The objectives of this project are:

1. Design of aerodynamic rear wing of formula one's car based on 2009 regulation of FIA.
2. Analyse the drag coefficient, $C_d$.and lift coefficient, $C_L$ for the designed aerofoil by using CFD.

## 1.3 SCOPES

The scopes of this project covered the design of NACA profile based on FIA 2009 regulation using Solidwork and analysis of design using commercial software, Ansys. Turbulent model for the analysis were using in the Ansys. Drag coefficient, $C_d$ and lift coefficient, $C_L$ will be studied. 2D and 3D analyse are consideration in this research. This research using 51.11 m/s as the velocity design, $Re$ = 1.2 $\times$ $10^6$ and $Ma$ = 0.147.

# CHAPTER 2

# LITERATURE REVIEW

## 2.1 INTRODUCTION

This chapter deals with the concept of aerodynamic on the rear wing F1 car. This chapter continues with the application of aerodynamic in the rear wing F1 car of study. Then, discussion continues with the design and analyze the rear wing F1 car using Computational Fluid Dynamics and make the comparison with the present design was created.

## 2.2 AERODYNAMIC

Aerodynamics is a branch of dynamics concerned with studying the motion of air, particularly when it interacts with a moving object. Aerodynamics is a subfield of fluid dynamics and gas dynamics, with much theory shared between them. Aerodynamics is often used synonymously with gas dynamics, with the difference being that gas dynamics applies to all gases. Understanding the motion of air (often called a flow field) around an object enables the calculation of forces and moments acting on the object. Typical properties calculated for a flow field include velocity, pressure, density and temperature as a function of position and time. By defining a control volume around the flow field, equations for the conservation of mass, momentum, and energy can be defined and used to solve for the properties. The use of aerodynamics through mathematical analysis, empirical approximation and wind tunnel experimentation form the scientific basis for heavier than air flight. (Von K, Theodore, 2004).

Aerodynamic problems can be identified in a number of ways. The flow environment defines the first classification criterion. External aerodynamics is the study of flow around solid objects of various shapes. Evaluating the lift and drag on an airplane, the shock waves that form in front of the nose of a rocket or the flow of air over a hard drive head are examples of external aerodynamics. Internal aerodynamics is the study of flow through passages in solid objects. For instance, internal aerodynamics encompasses the study of the airflow through a jet engine or through an air conditioning pipe. (Von K, Theodore, 2004).

In the Formula One racing, it more study to the objects through the air. It is closely related to fluid dynamics as air is considered a compressible fluid. From aerodynamic criteria can produced downforce to the car. This downforce can be likened to a virtual increase in weight, pressing the car down onto the road and increasing the available frictional force between the car and the road, therefore enabling higher cornering speeds. (F1, 2010)

F1 can be considered to be canard configurations in the sense that the front and back wings are on opposite sides of the centre of gravity and both are lifting strongly in the same direction, in this case creating downforce. Other than that, the car's body also can be optimized for the required downforce at the minimum of drag. Practically, every component has its influence on the behaviour of the car and cannot be regarded as an individual component. (F1, 2010)

### 2.2.1 Lift

Lift is the components of the pressure and wall shear force in the direction normal to the flow tend to move the body in that direction. It can prevent the object from fly to the air when use the negative lift coefficient. The pressure difference between the top and bottom surface of the wing generate an upward force that tends the wing to lift. For the slender bodies such as wings, the shear force acts nearly parallel to the flow direction, thus its contribution to the lift is small. The lift force depend on the density, $\rho$, of the fluid, the upstream velocity $V$, the size, shape, and orientation of the body, among other things, and it is not practical to list these force for a variety of situations. Instead, it is found convenient to work with appropriate dimensionless numbers that present the drag and lift characteristics of the body. These numbers are the lift coefficient, $C_L$. It is defined as

$$F_L = \frac{C_L}{2\rho V^2 A} \tag{2.1}$$

Where $A$ is ordinarily the frontal area (the area projected on a plane normal to the direction of flow) of the body. ½ $\rho V^2$ is the dynamic pressure and $F_L$ is lift force. (Anderson, John D, 2004)

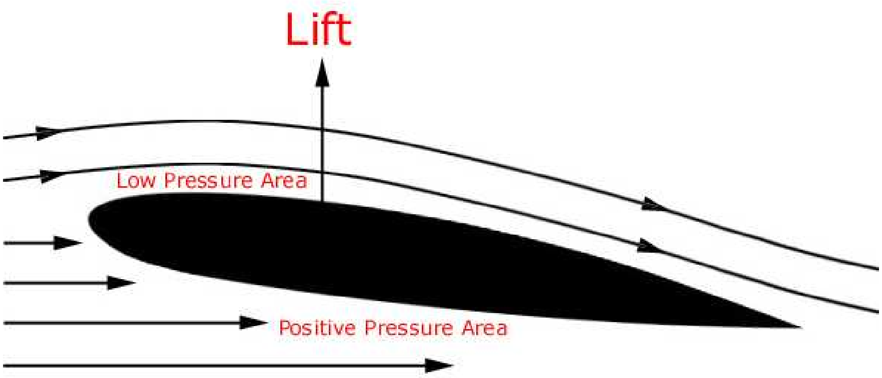


**Figure 2.1**: Lift direction

Source: Anderson, John D. (2004),

### 2.2.2 Drag

Drag is the aerodynamic force that is opposite to the velocity of an object moving through air or any other fluid. Its size is proportional to the speed differential between air and the solid object. Drag comes in various forms, one of them being friction drag which is the result of the friction of the solid molecules against air molecules in their boundary layer. Friction and its drag depend on the fluid and the solid properties. A smooth surface of the solid for example produces less skin friction compared to a rough one. For the fluid, the friction varies along with its viscosity and the relative magnitude of the viscous forces to the motion of the flow, expressed as the Reynolds number. Along the solid surface, a boundary layer of low energy flow is generated and the magnitude of the skin friction depends on conditions in the boundary layer. (Steven.D.G. 2009)

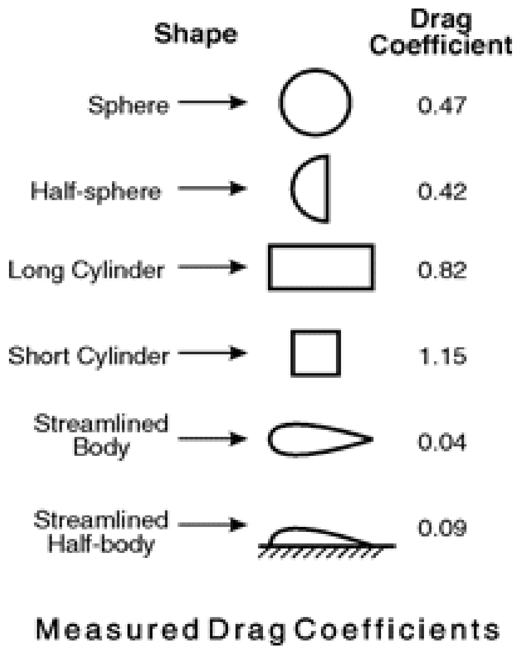


**Figure 2.2**: Type of drag coefficient, $C_d$

Source: Steven.D.G. 2009

Additionally, drag is a form of resistance from the air against the solid moving object. This form of drag is dependent on the particular shape of a wing. As air flows around a body, the local velocity and pressure are changed effectively creating a force. Interference drag or induced drag on the other hand is the result of vortices that are generated behind the solid object. Due to the change of direction of air around the wing, a vortex is created where the airflow meets unchanged, straight flow. The size of the vortex, its drag strength increases with an increasing angle of attack of the aerofoil. As a primary source of possible drag reduction, Formula One teams try to counteract this drag by adding end plates to wings or with fillets at the suspension arms. (Steven.D.G. 2009)

The amount of drag that a certain object generates in airflow is quantified in a drag coefficient. This coefficient expresses the ratio of the drag force to the force produced by the dynamic pressure times the area. Therefore, a $C_d$ of 1 denotes that all air flowing onto the object will be stopped, while a theoretical 0 is a perfectly clean air stream. At relatively high speeds of high Reynolds number $(R_e > 1000)$, the aerodynamic drag force can be calculated by this formula:

$$F_d = \frac{C_d}{2\rho V^2 A} \qquad (2.2)$$

Where

| | |
|---|---|
| $F_d$ | = Force of drag |
| $\rho$ | = Density of the air |
| $V^2$ | = Speed of the object relative to the fluid (m/s) |
| $A$ | = Reference surface area |
| $C_d$ | = Drag of coefficient |

Note that, minus sign which indicate that the resulting drag force is opposite to the movement of the object. (Steven.D.G. 2009)

Other sources of drag include wave drag and ram drag. For the wave drag, it is unimportant for normal race cars as it occurs when the moving object speeds up to the speed of sound. Ram drag on the other hand is the result of slowing down the free airstream, as in an air inlet. (Steven.D.G. 2009)

### 2.2.3 Downforce

Aerofoil in motorsports are called wings where can generate high downforce by having a high angle of attack, thus increasing the drag of the aerofoil. The evolution of aerofoil now is mainly thanks to the genius and research of a few well known scientists. In 1686, Sir Isaac Newton presented his three laws of motion, one of them being the conservation of energy. He stated that energy is constant in a closed system, although it can be converted from one type to another. Out of that theory, Daniel Bernoulli deducted a formula proving that the total energy in a steadily flowing fluid system is a constant along the flow path. An increase in the fluid's speed must be matched by a decrease in its pressure. Adding up the pressure variation times the area around the entire body determines the aerodynamic force on the body. (Steven.D.G. 2009)

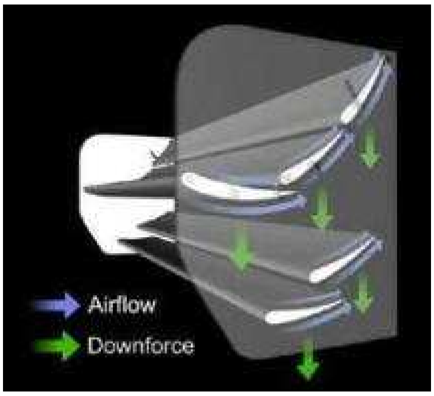


**Figure 2.3**: Downforce direction

Source: Steven.D.G. 2009

An aerofoil's operation can be easily explained when consider a wing in a steady laminar flow of air. As air is a gas, its molecules are free to move around and may have a different speed at different locations in the airstream. As downforce generating aerofoils are mostly designed with more thickness on the lower side, the lower airstream is slightly reduced in surface, hence increasing the flow speed and decreasing the pressure. On top of the wing, the airspeed is lower, and thus the pressure difference will generate a downward force on the wing. Additionally, and in line with Newton's third law of motion, downforce wings are never straight and induce a new turning of the airflow. More specifically, the shape of the wing will turn air upwards and change its velocity. Such speed creates a net force on the body. ( Steven.D.G. 2009)

$$F = m_a = m\frac{(V_1 - V_0)}{(t_1 - t_0)} = M\frac{dV}{dt} \qquad (2.3)$$

This shows that a force, causes a change in velocity, $V$, or also a change in velocity generates a force. Note that a velocity is a vectorial unit, having a speed and a direction component. So, to change of either of these components, force must be imposing. And if either the speed or the direction of a flow is changed, a force is generated. (Steven.D.G. 2009)

Downforce is often explained by the "equal transit time" or "longer path" theory, stating that particles that split ahead of the aerofoil will join together behind it. In reality however, the air on the longer side of the wing will flow much faster, further increasing the downforce effect. While these simplified versions are the basics of lift and downforce generation, the reality can hardly be simplified and is a complex study, requiring high power computer systems. For a gas, we have to simultaneously conserve the mass, momentum, and energy in the flow. Hence, a change in the velocity of a gas in one direction results in a change in the velocity of the gas in a direction perpendicular to the original change. The simultaneous conservation of mass, momentum, and energy of a fluid (while neglecting the effects of air viscosity) are called the Euler Equations after Leonard Euler. Several computer algorithms are based on these equations to make an approximation of the real situation. (Steven.D.G. 2009)

Because of the complexity, today's formula one cars are designed with CFD (computational fluid dynamics) and CAD (computer aided design) that allows engineers to design a car, and immediately simulate the airflow around it, incorporating environmental parameters like traction, wind speed and direction, and much more. From commercial CFD software, the drag coefficient, $C_d$ and lift coefficient, $C_L$ can be known to create downforce for the formula one car. (Steven.D.G. 2009)

## 2.3 THE CONCEPT AND USAGE OF CFD

The development of modern computational fluid dynamics began with the advent of the digital computer in early 1950s. It uses finite difference methods and finite element method as the basic tools used in the solution of partial differential equations in general and computational fluid dynamics. The fundamental basic of almost all computational fluid dynamics problems are the Navier-Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler Equation and removing terms describing vorticity