STUDY OF F1 CAR AERODYNAMICS FRONT WING USING COMPUTATIONAL FLUID DYNAMICS (CFD)

MOHD SYAZRUL SHAFIQ B SAAD

Report submitted in fulfillment 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

AWARD OF DEGREE

Bachelor Final Year Project Report

Report submitted in partial fulfilment of the requirements for the award of the degree of Bachelor of Mechanical Engineering with Automotives Engineering.

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project 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 SULAIMAN*

Position: LECTURER OF MECHANICAL ENGINEERING

Date: 6 DECEMBER 2010

STUDENT'S DECLARATION

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

Signature
Name: MOHD SYAZRUL SHAFIQ B SAAD
ID Number: MH 07043
Date: 6 DECEMBER 2010

ACKNOWLEDGEMENTS

I am grateful to Allah Almighty with His permission and blessings I can completed this thesis within the prescribed time. I also would like to express my sincere gratitude to my supervisor En Muhamad Zuhairi for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, and his belief that a 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 members 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.

Lastly I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life.

ABSTRACT

The aim of this thesis is to analyze a basic front wing of a typical Formula One car using Computational Fluid Dynamics (CFD). Commercial software Gambit and Fluent was used in this CFD analysis and SolidWorks as tool for designing the front wing. The front wing is crucial in term of its influence on the handling and dynamics of the car and efficiently directing the flow to other parts of the car. Design used for the front wing is based on NACA 23012, based on research from Freddie Mehta (2006). The validation study show that CFD capable of predicting the C_L with sufficient refine mesh and turbulence model assumptions. Turbulence model used in this analysis is Spalart-Allmaras turbulence model. C_L value for inverted NACA 23012 is -0.24, -0.49, -0.67, and -0.80 for angle of attack 2^0, 4^0, 6^0, and 8^0 respectively. For C_D is 0.06, 0.07, 0.09, and 0.12 for angle of attack 2^0, 4^0, 6^0, and 8^0 respectively. While for multiple wing C_L is -1.5 and C_D is 0.26. All the analysis was run around 90000 numbers of cells at interval size 5. The C_L values were approximately matched with the experimental result.

ABSTRAK

Tujuan kajian ini adalah untuk menganalisis sayap depan dasar kereta Formula Satu yang khas menggunakan Pengkomputeran Dinamika Fluida (CFD). Perisian komersial Gambit dan Fluent digunakan dalam analisis CFD dan SolidWorks sebagai alat untuk merancang sayap depan. Sayap depan sangat penting dalam hal pengaruhnya terhadap pengendalian dan dinamik dari kereta dan cekap mengarahkan aliran ke bahagian lain dari kereta. Rancangan yang digunakan untuk sayap depan didasarkan pada NACA 23012, berdasarkan kajian dari Freddie Mehta (2006). Kajian menunjukkan bahawa CFD mampu meramal C_L dengan mesh yang cukup memperbaiki dan andaian Model ombak. Ombak model yang digunakan dalam analisa ini adalah Spalart-Allmaras model ombak. C_L nilai untuk NACA 23012 terbalik adalah -0,24, -0,49, -0,67 dan -0,80 untuk sudut serangan 2^0, 4^0, 6^0, dan 8^0 masing-masing. Untuk C_D 0,06, 0,07, 0,09, dan 0,12 untuk sudut serangan 2^0, 4^0, 6^0, dan 8^0 masing-masing. Sedangkan untuk sayap C_L pelbagai ialah -1,5 dan C_D adalah 0,26. Semua analisis itu dijalankan sekitar 90,000 jumlah sel pada saiz selang 5. Nilai C_L kurang lebih sesuai dengan keputusan eksperimen.

TABLE OF CONTENTS

		Page
SUPERVISOR'S DECLARATION		ii
STUDENT'S DECLARATION		iii
ACKNOWLEDGEMENTS		v
ABSTRACT		vi
ABSTRAK		vii
TABLE OF CONTENTS		viii
LIST OF TABLES		xi
LIST OF FIGURES		xii
LIST OF SYMBOLS		xiv
LIST OF ABBREVIATIONS		xv
CHAPTER 1	**INTRODUCTION**	
1.0	Introduction	1
1.1	Objective	2
1.2	Problem Statement	2
1.3	Scope	3
CHAPTER 2	**LITERATURE REVIEW**	
2.1	Introduction	4
2.2	Formula One	4
2.3	The Concept and Usage of CFD	5
2.4	Usage of CFD in Formula 1	6
2.5	Formula One Aerodynamics	6
2.6	The Formula One Front Wing	7
2.7	Lift Force and Drag Force	8
2.8	Static Pressure Around The Aerofoil	10
2.9	Turbulent Model	10

CHAPTER 3 METHODOLOGY

3.1 Introduction 12
3.2 Modeling NACA 23012 in 3D 12
3.2.1 The Design of Aerofoil 12
3.2.2 Design a Straight Aerofoil 13
3.3 FIA Regulations 14
3.4 Front Wing Design 16
3.5 Modeling and Meshing NACA 23012 in 2D 16
3.6 Meshing Processes 17
3.7 Named Selection 19
3.8 Setup on Fluent 19
3.8.1 Grid Modification 21
3.8.2 Definition of Solution Parameters 22
3.8.3 Solution 24
3.9 Modeling Assumption 24
3.10 Spalart-Allmaras Turbulent Model 25
3.11 Simulation Configuration 26
3.12 List of Parameters 26
3.13 Converge Criteria 27
3.14 Material Properties 27
3.14.1 Air Properties 27
3.15 Flow Chart 28

CHAPTER 4 RESULTS AND DISCUSSIONS

4.1 Introduction 29
4.2 Grid Dependency Test 29
4.3 Variation of Angle of Attack Effect 30
4.4 C_L, C_D and C_P 32
4.5 Effect on C_L C_D and C_P on Multiple Wing 33
4.6 Summary of Result 34

CHAPTER 5 CONCLUSION

5.1 Conclusions 35
5.2 Further work 36

REFERENCES 37

APPENDICES

1. Appendix A 38
2. Appendix B 40
3. Appendix C 42

LIST OF TABLES

Table No.	Title	Page
3.1	FIA regulations 2010	15
3.2	Front wing dimension design	16
3.3	Mesh size value for various angle of attack (^{0})	26
3.4	List of parameter for 2D analysis	26
3.5	Air properties	27
4.1	Grid Dependency Test	29
4.2	Variation of angle of attack that effect C_L and C_D – inverted NACA 23012	31
4.3	Variation of Value C_L C_D and C_P	32
4.4	C_L C_D and C_P on multiple wing	33
4.5	Summary of result	36

LIST OF FIGURES

Figure No.	Title	Page
2.1	Aerodynamics flow around Formula One car	7
2.2	Winglets of Formula One car	8
2.3	Lift and drag force direction	9
2.4	Example of modern F1 car wing	10
3.1	NACA 23012 coordinates	13
3.2	Select the front plane when creating the sketch	14
3.3	FIA front bodywork	15
3.4	Isometric view	16
3.5	Mesh around NACA 23012	17
3.6	Skewed element mesh	18
3.7	Highly skew element mesh	18
3.8	Fluent software	20
4.1	Value of C_L vs number of cells	30
4.2	Coefficient of lift experimental and analytical vs angle of attack	31
4.3	Coefficient of drag experimental and Analytical vs angle of attack	32
4.4	C_L C_D and C_P vs Angle of Attack	33
4.5	Pressure distribution around front wing	35
A1	Streamline velocity on multiple wings	34
A2	Pressure contour on multiple wings	34
A3	Velocity contour on multiple wings	35
B1	Overall Width F1 Car	40
B2	Overall Height F1 car – Front Bodywork	40
B3	Overall Length F1 car	41

LIST OF SYMBOLS

ρ	Fluid Density
A	Area
C_D	Coefficient of Drag
C_L	Coefficient of Lift
C_L/C_D	Lift Drag Ratio
C_P	Coefficient of Pressure
F_D	Drag Force
F_L	Lift Force
S_i	Mass Distribution
V	Velocity
u	Fluid Velocity

LIST OF ABBREVIATIONS

2D	Two Dimensional
3D	Three Dimensional
CAD	Computer-aided Drafting
CFD	Computational Fluid Dynamics
F1	Formula 1
FIA	Fédération Internationale de l'Automobile
NACA	National Advisory Committee for Aeronautics

CHAPTER 1

INTRODUCTION

1.0 INTRODUCTION

Formula One, also known as Formula One or F1, and currently officially referred to as the FIA Formula One World Championship, is the highest class of auto racing organized by the Fédération Internationale de l'Automobile (FIA). The "formula" in the name refers to a set of rules to which all participants and cars must followed.

Aerodynamics is very important in Formula One race. The aerodynamic designer has two primary concerns: the creation of down force, to help push the car's tires onto the track and improve cornering forces; and minimizing the drag that gets caused by turbulence and acts to slow the car down.

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. The main advantages of using CFD software is that the results are obtained without the construction of the required prototype and this is very important because it can reduces the cost in constructing the F1 cars. The validity of the results is the most important things that we need to concern about while using the software simulation. Therefore the specific parameters and conditions while analyzing the data need to be valid.

There are two types of benefits that we can get when using CFD analysis: -

1. Technical advantages
   i. Faster evaluation of new ideas, products and processes
   ii. New insights into our process and performance
   iii. Maximise effectiveness of our manufacturing resources
   iv. Save time and cost, and get better results

2. Business advantages
   i. Increase customer confidence
   ii. Increase credibility with customers
   iii. Win more business

The most crucial component of the entire F1 cars is the front wing. This is primarily because front wing is the first part of the car to come in contact with the oncoming air hence influences the air flow over the rest of the car.

1.1 OBJECTIVES

The objectives of this study are:

i. Design front wing of formula one car for NACA 23012 based on 2010 FIA regulations.
ii. Analyze the drag coefficient and lift coefficient for the design aerofoil using CFD

1.2 PROBLEM STATEMENT

One of the problems in formula one’s car is the aerodynamic of front wing part. In front wing formula one’s car, engineers need to maximize the down force of the formula one’s car to push it to the track so the tire does not slip. Also the down force can improve the cornering force of the formula one’s car. The design of front wing is important because the down force of the formula one’s car can be improve. This project considered the previous design of front wing of formula one’s car team to be analyzed.

This study will use CFD analysis because it can save time and cost and can get better result. CFD also can explain the behavior of the flow characteristic. It also can maximize effectiveness of manufacturing resources.

1.3 SCOPE

This project is focusing on F1 car front wing. This focus area is done based on the following few aspects. Firstly, 2D analysis will be considered. This is because 2D analysis should be used wherever possible. Isothermal flow will be used so that type of material for the front wing will not affect the result. The front wing design will be based on NACA 23012. The analysis will be done in turbulent flow since for external flow the $Re > 8.21 \times 10^5$.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTIONS

This chapter explains about the concept of aerodynamics on front wing F1 car. This continues with the application of CFD in Formula One. Then, discussion about the design and analyze the front wing F1 car using CFD.

2.2 FORMULA ONE

In Formula One every milliseconds is very significant. It could make a significant difference in winning or lose the race. Therefore, aerodynamics is one of the predominant means of enhancing the performance of the car. The car can traverse a corner faster than the competitor if the car can generate greater down force. The different is noticeable over a matter of laps. It also would make a very significant difference during 60 odd laps in each race. Moreover, less drag would translate to a better top speed along straight laps.(Mehta, 2006)

Engineers in Formula One doing their best effort to maximize the levels of down force while keeping the aerodynamics drag to a minimum level. Not only the front and rear wing generated down force, but also by the under-body shape which create a pressure different, hence literally sucking the car towards the ground.(Mehta, 2006)

2.3 THE CONCEPT AND USAGE OF CFD

CFD is a computational technology that enables researcher to study the dynamics of things that flow. Using CFD, a computational model that represents a system or device can be build. Then the fluid flow physics and chemistry can be applied to this virtual prototype and the software will output a prediction of the fluid dynamics and related physical phenomena. Therefore, CFD is a sophisticated computationally-based design and analysis technique. Besides that, CFD software can give the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling.(Slater, 2008)

Using CFD, the product will get to the market faster. This is because CFD can save time that used for building prototype. CFD is commonly used for aerodynamics of air craft and vehicles (lift and drag), hydrodynamics of ships, power plant combustion (I.C engines and gas turbines), marine engineering (loads on off shore structures), meteorology (weather prediction) and etc.(Malalasekera, 1995)

The aim of CFD is to resolve the equation that drives theoretically every kind of flow:

i. The continuity equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k}(\rho u_k) = 0 \tag{2.1}$$

ii. The momentum equation

$$\frac{\partial \rho u_i}{\partial y} + \frac{\partial}{\partial x_k}(\rho u_i u_k - \tau_{ik}) + \frac{\partial P}{\partial x_i} = S_i \tag{2.2}$$

Where u is the fluid velocity, ρ is the fluid density, S_i is a mass-distributed external force per unit mass.(Slater, 2008)

2.4 USAGE OF CFD IN FORMULA ONE

In CFD simulation, it is easier to optimize and change certain features in the design rather than changing the shape of prototype in the wing tunnel tests. This is a significant advantage as it leads to a reduction in time.

In Formula One not only case of aerodynamics can be used in CFD. CFD also can be used to monitor the heat transfer from the brakes. Formula One brake comprise of carbon fiber discs with carbon fiber brake pads. During heavy braking, the driver experiences a force of nearly 5.5 Gs, the brakes may reach temperatures as high as 1000 degrees centigrade. Hence, cooling them down is extremely crucial for their consistent performance and reliability.(Mehta, 2006)

It is vital to ensure that the optimum air flow is achieved for engine induction in the engine intake opening. This can be done by using CFD. Hence, the air flow can be optimized to increase the power of the engine.(Mehta, 2006)

Another advantage of CFD is that the level of detail of the data that is obtained from the solution is wide. This data can be viewed from different perspectives, which is a big advantage.

2.5 FORMULA ONE AERODYNAMICS

Aerodynamics is the science that studies objects moving through air. It is closely related to fluid dynamics as air is considered a compressible fluid. Nowadays, aerodynamics becomes the most important factor in Formula One car performance.

In aerodynamics, creating down force is important because it pressing the car down onto the road and increasing the available frictional force between the car and the road, therefore enabling higher cornering speeds.(Groote, 2006)

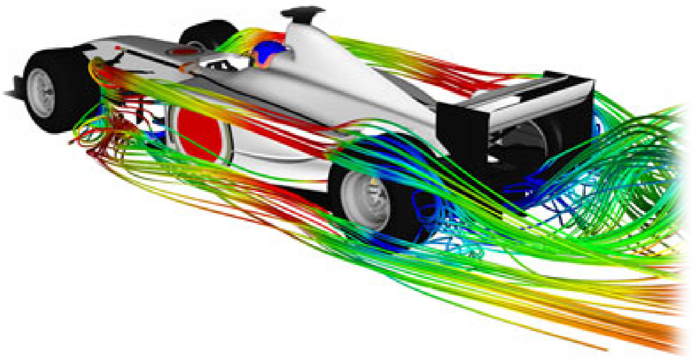

Figure 2.1: Fluid flow around Formula One car.

Source: www.f1technical.net

2.6 THE FORMULA ONE FRONT WING

The front wing of the Formula One car is the single most crucial aerodynamics component. This is because it directly influences the way in which the airflow takes place over the rest of the car since the front wing is the first part of the car to come in contact with the air. It also influences the flow of air into the brake ducts, radiator, diffuser, and also to the main engine intake.

It is usually made into a readily modifiable component where the flaps can be adjusted to provide different levels of front down force distribution as per the steering and handling requirement of the driver during the race.

The main factor influencing the down force generated by the front wing is the ground effect where more down force is generated with an aerofoil moving close to the ground. A typical lift (negative): drag ratio for a front wing is usually 7 to 9.(Mehta, 2006)

The wing also incorporates many small winglets which refine the flow into the radiator, underbody, and the rest of the bodywork. Hence it ensures that the flow is as laminar and attached as possible in order to attain good aerodynamics results. Flow over the bodywork is particularly important since the bodywork itself develops around 35% of the total cars down force. (www.f1technical.net)

Figure 2.2: Winglets of Formula One car.

Source: www.f1technical.net

2.7 LIFT FORCE AND DRAG FORCE

Lift, or down force is the force generated perpendicular to the direction of travel for an object moving through a fluid (gas or liquid). The same effect occurs when a fluid moves over a stationary object, such as an airfoil in a wind tunnel. Airfoils are the most efficient shapes found so far that can generate lift while at the same time minimizing drag.(Mehta, 2006)

Drag is an unavoidable consequence of an object moving through a fluid. Drag is the force generated parallel and in opposition to the direction of travel for an object moving through a fluid. Drag can be broken down into the following two components:

i. Form drag (or pressure drag) - dependent on the shape of an object moving through a fluid
ii. Skin friction - dependent on the viscous friction between a moving surface and a fluid, derived from the wall shear stress

The lift and drag force depend on the density ρ of the fluid, the upstream velocity *V*, the size, shape and orientation of the body. (Cimbala, 2006) Lift and drag coefficient can be defined as:

$$\text{Lift coefficient: } C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A} \tag{2.3}$$

$$\text{Drag coefficient: } C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A} \tag{2.4}$$

Where:

F_L = Lift force

F_D = Drag force

ρ = Fluid density (sea level air is 1.204 kg/m^3)

V = Velocity

A = Frontal area

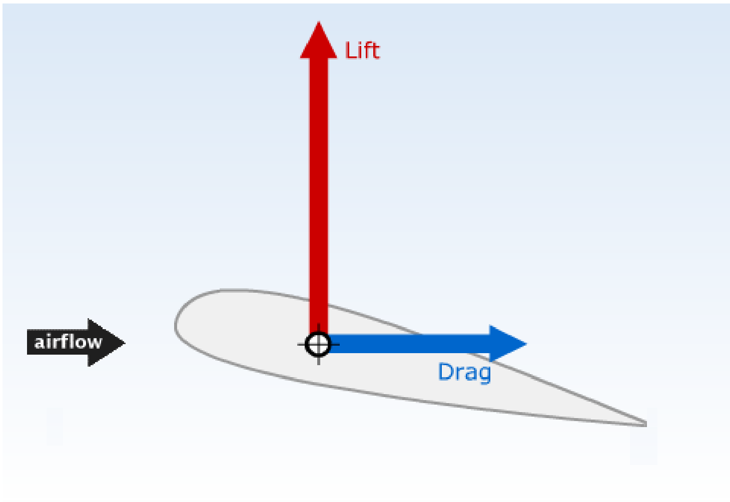

FIGURE 2.3: Lift and drag force direction

Source: Groote, 2006

2.8 STATIC PRESSURE AROUND THE AEROFOIL

Aerofoil shape can produce different pressure on upper and lower side. This can be explained in term of velocity and pressure. As explained by Bernoulli’s Principle, an increase in velocity occurs simultaneously with decrease in pressure. This principle is a simplification of Bernoulli’s equation which states that the sum of all forms of energy in a fluid flowing along an enclosed path (a streamline) is the same at any points in that path. (Mehta, 2006) When air flows over the longer edge on the underside of the aerofoil, it experiences an increase in velocity and it makes the area experience a drop in pressure as compared to the shorter side of the aerofoil.

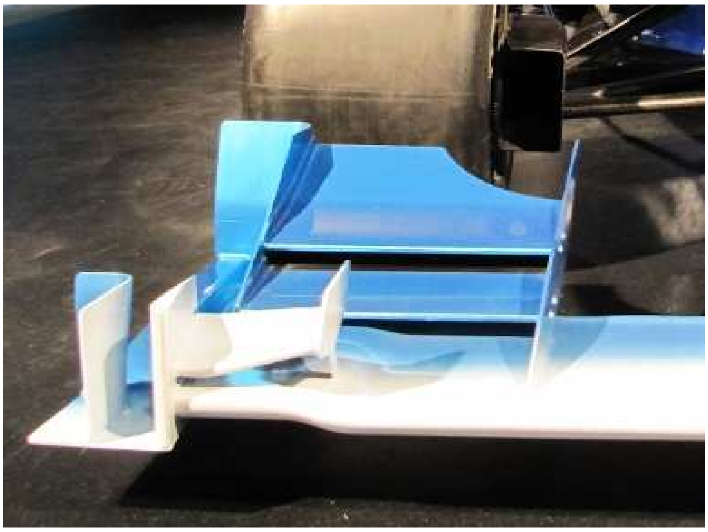

Figure 2.4: Example of modern F1 car wing

Source: www.paultan.org

2.9 TURBULENCE MODEL

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. This model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. It is also gaining popularity for turbo machinery applications.(Javaherchi, 2010)