STUDY ON THE MODIFICATION OF DIFFUSER TO THE EFFECT ON AERODYNAMIC OF THE F1 IN SCHOOLS CAR

Submitted in accordance with the requirement of the Universiti Teknikal Malaysia Melaka (UTeM) for the Bachelor Degree of Manufacturing Engineering (Hons.)

by

LIM YIK SENG

B051710195

970829-14-5007

FACULTY OF MANUFACTURING ENGINEERING

Year 2020/2021

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

Tajuk: **STUDY ON THE MODIFICATION OF DIFFUSER TO THE EFFECT ON AERODYNAMIC OF THE F1 IN SCHOOL CAR**

Sesi Pengajian: **2020/2021 Semester 1**

Saya **LIM YIK SENG (970829-14-5007)**

mengaku membenarkan Laporan Projek Sarjana Muda (PSM) ini disimpan di Perpustakaan Universiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Laporan PSM adalah hak milik Universiti Teknikal Malaysia Melaka dan penulis.
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja dengan izin penulis.
3. Perpustakaan dibenarkan membuat salinan laporan PSM ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. *Sila tandakan (√)

☐ SULIT (Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysiasebagaimana yang termaktub dalam AKTA RAHSIA RASMI 1972)

☐ TERHAD (Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/ badan di mana penyelidikan dijalankan)

☑ TIDAK TERHAD

Disahkan oleh:

Alamat Tetap:

HD-34, Taman Hock Ann, Jalan Klang Lama, Bt 4 ¼, 58100, Kuala Lumpur.

Tarikh: 11/9/2021

Cop Rasmi:

Ir. Dr. Mohamad Ridzuan Bin Jamli
Senior Lecturer
Faculty of Manufacturing Engineering
Universiti Teknikal Malaysia Melaka

Tarikh: 11/09/2021

*Jika Laporan PSM ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh laporan PSM ini perlu dikelaskan sebagai SULIT atau TERHAD.

DECLARATION

I hereby, declared this report entitled "Study On The Modification Of Diffuser To The Effect On Aerodynamic Of The F1 In School Car" is the results of my own research except as cited in reference

Signature : ....

Author's Name : LIM YIK SENG

Date : 11/9/2021

APPROVAL

This report is submitted to the Faculty of Manufacturing Engineering of Universiti Teknikal Malaysia Melaka as a partial fulfillment of the requirement for the degree of Bachelor of Manufacturing Engineering (Hons.). The members of the supervisory committee are as follow:

Mohammad ridzuan Jamli
……………………………………………………

(Principle Supervisor) – Signature & Stamp

Ir. Dr. Mohamad Ridzuan Bin Jamli
Senior Lecturer
Faculty of Manufacturing Engineering
Universiti Teknikal Malaysia Melaka

ABSTRACT

F1 IN SCHOOLS is a global competition that promote exciting ways for students with the age range from 9 to 19 to study subject related with Science, Technology, Engineering and Mathematics (STEM). The goal in the competition is to finish the race on a 20 m track with the shortest time possible. There are two factors which affect the aerodynamic performance which are the drag and lift force. Drag force and extra downforce will slow down the speed of the car while extra lift force will cause imbalance of the car. Thus, study on the modification of underbody diffuser is important as it enables the control both drag and lift force of the car. Therefore, the objective of this study is to produce flow analysis on the F1 IN SCHOOLS Car. besides, this study aims to investigate the effect of diffuser angle on the drag and lift coefficient of the fast-moving car. Moreover, this study also aims to evaluate appropriate diffuser angle for maintaining suitable lift coefficient of the car. Tools that will be used in this study is by using Autodesk Fusion 360 for 3D modelling of the car and the CFD simulation will be run using Ansys Fluent. Before conducting CFD simulation for simplified 2D, 3D and complete 3D model analysis, mesh density analysis is done to obtain proper surface mesh element size that converges the result. Besides, comparison between own simulation results with previous wind tunnel test result is done to verify the fluent setting used. Ahmed body simulation result is then done to obtain the relationship between various angle of diffuser with drag and lift coefficient. The simulation results of simplified 2D, 3D and complete 3D model analysis shows different optimum diffuser angle for best aerodynamic performance. Besides, the result also shows that lift coefficient decreases as diffuser angle increases.

ABSTRAK

F1 IN SCHOOLS adalah sebuah pertandingan global yang mempromosikan pembelajaran yang menarik bagi pelajar yang berumur antara 9 hingga 19 tahun untuk mempelajari subjek yang berkait dengan Sains, Teknologi, Kejuruteraan dan Matematik (STEM). Matlamat dalam pertandingan ini adalah untuk menamatkan perlumbaan di lintasan sepanjang 20 m dengan masa sesingkat mungkin. Terdapat dua faktor yang mempengaruhi prestasi aerodinamik iaitu daya tarikan dan daya angkat. Daya tarik dan daya turun tambahan akan melambatkan kelajuan *F1 IN SHOCCLS Car* sementara daya angkat tambahan akan menyebabkan ketidakseimbangan *F1 IN SHOCCLS Car*. Oleh itu, objektif kajian ini adalah untuk menghasilkan analisis aliran pada *F1 IN SCHOOLS Car*. Selain itu, kajian ini bertujuan untuk mengkaji pengaruh sudut diffuser pada pekali tarik dan angkat pada *F1 IN SCHOOL Car* yang bergerak pantas. Di samping itu, kajian ini juga bertujuan untuk megkaji sudut *diffuser* yang sesuai untuk mengekalkan pekali angkat yang sesuai. Alat yang akan digunakan dalam kajian ini adalah dengan menggunakan *Autodesk Fusion 360* untuk pemodelan *3D F1 IN SHOCCLS Car* dan simulasi *CFD* akan dijalankan menggunakan *Ansys Fluent.* Sebelum melulakan simulasi CFD untuk analisis model 2D, 3D, dan 3D lengkap analisis, analisis kepadatan jala dilakukan untuk mendapatkan ukuran elemen permukaan permukaan yang tepat. Selain itu, perbandingan antara hasil simulasi dengan hasil ujian *wind tunnel test* dilakukan untuk mengesahkan *fluent setting* yang digunakan. Hasil simulasi *Ahmed body* kemudian dilakukan untuk mendapatkan hubungan antara pelbagai sudut *diffuser* dengan pekali seret dan angkat. Hasil simulasi analisis model 2D, 3D dan 3D lengkap menunjukkan sudut *diffuser* optimum yang berbeza untuk prestasi aerodinamik terbaik. Selain itu, hasilnya juga menunjukkan bahawa pekali angkat menurun ketika sudut *diffuser* meningkat.

DEDICATION

Only

my beloved father, Lim Boon Beng

my beloved mother, Lo Chin Liang

for giving me support mentally

my respected supervisor, Dr. Mohamad Ridzuan Bin Jamli

for giving me suggestion and informative ideas

Thank you so much

ACKNOWLEDGEMENT

I would like to thank all the person involved in assisting me to complete my final year project.

I would like the express my gratitude to my supervisor, Dr. Ridzuan Bin Jamli, who never hesitates to offer suggestion and ideas on improvement in helping me to complete the study.

Besides, I would like to appreciate my family for giving me support, money and encouragement in completing the study.

Lastly, special thanks to my friend Muhammad Nadzirul Mubin bin Abdul hamid who always share and discuss ideas regarding the content of the study.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

STEM	-	Science, Technology, Engineering and Maths
CAD	-	Computational-Aided Design
CAE	-	Computational-Aided Engineering
F1	-	Formula One
CO2	-	Carbon Dioxide
CFD	-	Computational Fluid Dymamics
RANS	-	Reynolds-averaged Navier–Stokes
SIMPLE	-	Semi-Implicit Method for Pressure Linked Equations
FVM	-	Finite Volume Method
COP	-	Coefficient of Pressure
RMS	-	Root Mean Square

LIST OF SYMBOLS

F_D	-	Drag Force
F_L	-	Lift Force
F_S	-	Side Force
T_{RM}	-	Rolling Moment
T_{PM}	-	Pitching Moment
T_{YM}	-	Yawing Moment
L	-	Length
W	-	Width
H	-	Height
α, α_d, β	-	Diffuser Angle
G_c	-	Ground Clearance
C_D	-	Draw Coefficient
C_L	-	Lift Coefficient
C_P	-	Pressure Coefficient
l_d	-	Diffuser Length
N	-	Newton

CHAPTER 1

INRODUCTION

1.1 Background of Study

F1 IN SCHOOLS is a global competition held once a year, and participants' ages range from 9 to 19. This competition offers exciting ways to learn subjects related to Science, Technology, Engineering and Maths (STEM). F1 IN SCHOOLS competition's main concern is to achieve the highest speed possible for the car to complete a race on a 20-meter track in the shortest time. To accomplish that, knowledge of fluid dynamics, aerodynamics and physic is indispensable (Mansor, 2017). There are a few parameters affect the speed of the car, which are drag force and downforce.

Drag force is resistance or forces that act on a fast-moving object with the direction opposite to the direction of motion. In FI IN SCHOOLS car, drag force that occurs can be sub-divided into two categories, which are skin friction drag and pressure drag. Skin friction drag occurs between the moving car surface and the fluid (Paturrahman et al., 2018). In other words, the surface of the F1 IN SCHOOLS Car needs to be polished to reduce the skin friction drag with the air to achieve its optimum moving speed. Pressure drag happens to be always in the opposite direction with a moving object. This happens where the pressure accumulated at the front part of the F1 IN SCHOOLS car is too much compared to the car's rear part when at high speed (Mosiężny et al., 2020). This is because air molecules collide with the front of the car and create stagnation pressure, which creates pressure that pushes the car from moving forward, resulting in slowing down the car. On top of that, there is also a pulling force generated at the rear of the car. This situation happens due to flow separation. Flow separation plays a major role in racing cars in real life and in F1 IN SCHOOLS car. It is responsible for the major portion of the aerodynamic drag of racing cars (Hassan et al., 2014). When fluid flows through a change in geometry at high speed which the fluid inertial forces higher than the viscous forces trying to keep the fluid flow attached to the surface, flow separation occurs (Ryan T. Kell). The flow separation zone generated on the rear of the moving car creates a low-pressure zone which causes pressure drag. The rear part's pressure is much lower than the front part of the car because the air molecules cannot fill in the zone. This creates a pulling force behind the car which decreases the moving speed.

Downforce is also an essential factor that affects the speed of a car and its stability in real life. Stability is an important issue because a fast-paced car would lose its balance and the tyre would lose contact with the ground. This situation applies to F1 IN SCHOOLS car because the car is accelerated using a CO_2 canister which generates high thrust force at the rear part of the car. The lift force and downforce must be balanced because the guiding thread which redirects the car direction applies pulling force at the front of the car. Imbalanced lift force of the car with the guiding thread's pulling force may create a resultant force that interrupts the acceleration. Ground effect is also another method of controlling the lift force of the car. It was discovered that large amounts of downforce could be generated from the airflow between the underbody of the car and the ground plane (SEAS, 2020). In real life, F1 car uses ground plane almost like the floor of venturi duct to generate low pressure underneath the car. The shape of the venturi duct is inverted wing profile and the airflow tunnel underneath the car is narrowing down able to accelerate the airflow under the car. This creates low pressure underneath the car in accordance with Bernoulli's principle (Gordon McCabe, 2008). In F1 IN SCHOOLS car, the ground effect needs to be examined to control the lift force on the car so that the car is always in a balanced position while moving at high speed and to minimize the friction in between the wheels and track.

1.2 Problem Statement

The main factors that affect the aerodynamic performance of F1 in SCHOOLS car are drag force and lift force.

Drag force happens when the car moving at high speed which always has the opposite direction to the motion of a car. The difference between the front part and the rear part of the car causes pressure drag while skin drag happens between the car's surface with the fluid. Flow separated zone on the rear part of the car also creates drag force to the fast-moving car. To obtain better aerodynamic performance on the car, the drag coefficient must be as small as possible while the lift coefficient must be just right. The main factors that affect the aerodynamic performance of F1 IN SCHOOLS car are drag force and lift force.

Other than that, the velocity of the car only depends on the CO2 canister that generates high thrust force at the beginning of the race and the wheels of the car are not motorised. Thus, the downforce generated on the car body must be just right. This is because excessive downforce generated will increase the friction between the car's wheels and the racing track. The imbalanced force generated on the body due to the pulling force of guiding thread and extra lift force will also interrupt the car acceleration.

To obtain better aerodynamic performance on the car, the drag force must be as small as possible while the lift force and downforce must be balanced.

1.3 Objectives

There are three objectives in this study:

- To produce airflow analysis on the F1 IN SCHOOLS Car.
- To investigate the effect of different diffuser designs on the drag and lift coefficient of fast-moving F1 IN SCHOOLS Car.
- To evaluate appropriate diffuser design for maintaining suitable lift coefficient of F1 IN SCHOOLS Car.

1.4 Scope

The main focus of this study is to improve the aerodynamic performances which is the drag and lift of the F1 IN SCHOOL Car. The software used to complete this study is 3D Computer-Aided Design (CAD) modelling software and Computer-Aided Engineering (CAE) software. The CAD software used is Autodesk Fusion 360 (educational licensed) while the CAE software used is Ansys Fluent. The solver of Ansys Fluent is based on laminar flow due to the Reynold Number of model smaller than 500 for the open channel flow. The model used for simulation is k-epsilon model.

The study is also focused on examining the effects of the ground effect produced under the fast-moving F1 IN SCHOOLS Car body. Effects on the lift and drag coefficient of the fast-moving car are also focused by using various diffuser angle. Besides that, minor study on sidepot also conducted to evaluate the effect of removing sidepot on the aerodynamic performance of car body.

1.5 Significant/Important of Study

This study provides an understanding of the behaviour of airflow on the car body. As the behaviour of airflow is different when it passes through different geometry such as an edge, plane surface, or curved surface. Besides, visualization of turbulent form at which geometry can also be done easily by examining the airflow behaviour. Turbulent will form when the air flow through a change of geometry on the body usually an edge as air particles try to stick with the surface due to its viscosity characteristics. The required part for modification to smoothen the airflow can also be determined easily by examining the airflow behaviour. After that, airflow simulation can be rerun to check the airflow behaviour after modification.

The study is also important for participants in the competition to understand and visualise the airflow on the body. After the simulation is done, they can determine the drag and lift force experienced on the fast-moving car body. Visualization pressure changes and velocity of airflow under the car body also helps them to examine the ground effect experienced under the car body. With this, adjustment can be made on the height of ground clearance for a more balanced lift force on the car body.

1.6 Organization of The Report/Thesis

The organization of the report is started by introduction, followed by literature review and lastly methodology. In chapter 1 introduction, the topic discussed are the background of the research, problem statement, objectives, scope of the research, importance of the study, and the topics' summary. Chapter 2 literature review discussed the reviews of previous research related to the importance of aerodynamic in automotive, the background of Computational Fluid Dynamics (CFD), studies related to drag and downforce on fast-moving body and studies of various modification on diffuser to the aerodynamic performance. The research objective, strategies, and findings were emphasized in the review of the diffuser related research. Chapter 3 methodology describes the planning and method used to complete this research. The modelling of the 3D model, steps done on simplification of model, analysis, mesh refinement, and results validation is discussed in the topic.

1.7 Summary

In the chapter introduction, the background of F1 IN SCHOOLS competition is stated while the two factors that affect the aerodynamic performance of the F1 IN SCHOOLS Car were discussed. The first factor is drag force which can be categorized into skin friction drag and pressure drag. The mechanism on how skin friction drags and pressure drag affect the aerodynamic performance is discussed. The second factor is the lift force its effect on the car while at high speed. Then, problems that interrupt aerodynamic performance are further discussed regarding the car's drag and lift force during a race. The objectives and scopes of this study are also stated in detail after the problem statement. Next, the importance of this study is discussed by explaining where the outcome will be benefited. Lastly, the organization of the report is discussed.