DESIGN AND ANALYSIS OF URBAN CAR FOR SHELL ECO MARATHON ASIA

MUHAMMAD FIRDAUS BIN OTHMAN

BACHELOR OF ENGINEERING

UNIVERSITI MALAYSIA PAHANG

2012

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS

JUDUL: DESIGN AND ANALYSIS OF URBAN CAR FOR SHELL ECO MARATHON ASIA

SESI PENGAJIAN: 2011/2012

Saya **MUHAMMAD FIRDAUS BIN OTHMAN (890116-08-5235)**
(HURUF BESAR)
mengaku membenarkan tesis (Sarjana Muda / ~~Sarjana / Doktor Falsafah~~)* ini disimpan di perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis ini adalah hakmilik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

	SULIT	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
	TERHAD	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi / badan di mana penyelidikan dijalankan)
√	**TIDAK TERHAD**	

Disahkan oleh:

___________________________	___________________________
(TANDATANGAN PENULIS)	(TANDATANGAN PENYELIA)
Alamat Tetap:	Nama Penyelia:
NO 228B,Kongsi Baru **31700 Malim Nawar,** **Perak**	**MR. AZIM BIN MOHD ARSHAD**
Tarikh:	Tarikh:

CATATAN: * Potong yang tidak berkenaan.

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

◆ Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara Penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

EXAMINER'S APPROVAL DOCUMENT

UNIVERSITY MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled "*Reduction of Diesel Exhaust Emission by Using First Stage Seawater Spray System"* is written by *Muhammad Firdaus bin Othman*. I have examined the final copy of this project and in my opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Prof. Dr. Korada Viswanatha Sharma
Examiner Signature:

DESIGN AND ANALYSIS OF URBAN CAR FOR SHELL ECO MARATHONS ASIA

MUHAMMAD FIRDAUS BIN OTHMAN

Thesis submitted in partial fulfillment of the requirements
for the award of Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

Jun 2012

SUPERVISOR'S DECLARATION

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

Signature :

Supervisor's name : MR AZIM BIN MOHD ARSHAD

Position : LECTURER

Date :

STUDENT'S DECLARATION

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

Signature :
Name : MUHAMMAD FIRDAUS BIN OTHMAN
ID number : MH08036
Date :

ACKNOWLEDGEMENT

I am grateful and would like to express my sincere gratitude to my supervisor Mr Azim Arshad 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 bachelor 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 lab 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. Many special thanks go to member final year project group for their excellent co-operation, inspirations and supports during this study.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. 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. Special thanks should be given to my friends. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

Shell Eco Marathon Asia is a competition based on car race, which requires the participant to design and fabricate the most fuel saving cars. There are two categories being promoted, which are prototype and urban car type. The team representative in this category known as the SAE - UMP Chapter is taking the challenge and builds up the urban car for the 2010 events at the Sepang International circuit, Malaysia. For that year, the team has won the third place in urban category. Since then, various improvements have been carried out by the team to compete for the top 3 position and create the most efficient car ever build in Universiti Malaysia Pahang. This thesis focuses on improvement of existing model in terms of aerodynamics. Aerodynamic is an important area of study as it relates to parasitic load experienced by the engine. The objective of this project is to design fives new model with the improved drag coefficients compared to existing models and finally to select the best design in terms of the aerodynamic features. The CFD analysis was performed by using Flow Simulation in Solidworks with standard condition where air density equal to 1.184 kg/m^3 and at 1 atm environment pressure. The relative velocity of the analysis varies from 40 km/h to 90 km/h due to the minimum allowed velocity and maximum capable velocity of the car on track. The result of simulation shows that model 2 has the minimum drag coefficient which is equal to 0.281 and has improved the *Cd* by 37% compared to existing models. In this analysis, the most aerodynamic body is the one with minimum drag coefficient, minimum aerodynamic power and minimum relative pressure over the wake region. In selecting the model, the method of the Spider-web graph plot was used to visualize the widest area covered by the model in parameter axis. At the end of the analysis, model 2 shows the widest area covered in the graph plot which by quantitative measure, model 2 has the minimum drag coefficient, minimum aerodynamic power and minimum relative pressure over the wake region in the environment among the other 4 models.

ABSTRAK

Shell Eco marathon Asia ialah satu pertandingan berbentuk perlumbaan kereta yang memerlukan peserta untuk mereka dan membina kereta yang menjimatkan pengunaan bahan api. Terdapat dua kategori yang dipertandingkan iaitu kategori prototaip dan kategori urban. SAE-UMP Chapter telah menyahut cabaran tersebut dan bertanding untuk kategori urban dan telah memenangi tempat ketiga di Litar Antarabangsa Sepang, Kuala Lumpur. Sejak dari itu, pelbagai penambahbaikkan telah di lakukan bagi membolehkan kumpulan dari universiti pantai timur ini menduduki tempat ketiga teratas lalu membina kereta pertama di Universiti yang menjimatkan minyak. Tesis ini ditulis bertujuan menerangkan salah satu metod penambahbaikkan kereta dari segi aerodinamik. Objektif projek ini ialah membina lima buah model berdasarkan syarat pertandingan dan memperbaiki ciri aerodinamik kereta berbanding model kereta yang sedia ada terutama dari segi pekali geseran. Analisis CFD telah dilakukan dengan menggunakan Simulasi Aliran menggunakan perisian Solidworks dalam keadaan standard di mana ketumpatan udara sama kepada 1.184 kg/m^3 dan tekanan persekitaran pada 1 atm. Halaju relatif analisis berbeza dari 40 km/j hingga 80 km/j berdasarkan halaju minimum yang dibenarkan.Keputusan simulasi menunjukkan model 2 mempunyai pekali seretan yang minimum iaitu sama dengan 0.281 dan ia lebih rendah nilainya sebanyak 37% berbanding dengan model yang sedia ada. Dalam analisis ini, model yang paling aerodinamik ialah model dengan pekali seretan yang minimum, kuasa aerodinamik yang minimum dan tekanan relatif yang minimum . Dalam memilih model, kaedah plot Spider-web digunakan untuk menggambarkan kawasan yang paling luas yang diliputi oleh model dalam paksi parameter berdasarkan pemberat. Penghujung analisis menunjukkan, model 2 punyai kawasan yang paling luas yang diliputi dalam plot graf, dengan erti kata yang lain model 2 mempunyai pekali seretan minimum, kuasa aerodinamik yang minimum dan tekanan relatif yang minimum pada bahagian belakang model berbanding model-model kereta yang lain.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

A_f	Frontal area
a	Tunnel wide
b	Tunnel height
C_d	Drag coefficient
$C_{d(avg)}$	Average drag coefficient
C_l	Coefficient of lift force
d_h	Hydraulic diameter
F_d	Drag force
$F_{d(avg)}$	Average drag force
F_l	Lift force
L_e	Entrance length
ΔP	Relative pressure
P_w	Wake pressure
P_e	Environment pressure
P_a	Aerodynamic power
Re	Renault number
v_r	Relative velocity
v	Velocity
ρ	Fluid density
μ	Fluid dynamic viscosity

LIST OF ABBREVIATION

SEM Asia	Shell Eco Marathon Asia
SI	Standard International unit
CFD	Computational Fluid Dynamics
3D	Three dimensional
CAD	Computer Aided Design

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

This chapter basically discusses about the upper design of the urban car body proposed. Currently, urban areas suffer heavily from problems caused by the excessive use of the private car which also cause the congestion to air and noise pollution. Urban transport is not only a significant contributor to climate change, but also the main source of fine particulate matters. The pollutant may cause many cities in the world to exceed the thresholds given in the world air quality directive. One of the solutions that can be made is using an urban concept car. The urban concept car is a car designed to be used in city traffic. It is normally small in size and not design to run fast, but very fuel efficient. Urban cars are often hailed as the answer to the escalating levels of air pollution and traffic congestion that result from increases in numbers of larger personal vehicles. They are intended for use exclusively in or near cities and towns, but they are not suited to long journeys or fast travel on highways. They are very light, pollute little, take up a fraction of the space required by most vehicles, cost much less than most cars and trucks, and can be effectively recycled, Erdmenger and Fuhr, (2005).

This work intends to study the aerodynamic of the urban car`s. Moving object tend to be opposed by the existence of the air. This resistance varied with the ambient temperature and altitude from the sea level. The magnitude of the resistance is measured in term of drag force. The drag force of the moving car is supplied by the surface contact of the tire to the road, the drag force created on the car body surface and the pressure drag happened in the rear of the car. An object that was designed to develop minimum drag force is happened to be aerodynamic. When the drag force is reduced, the power from the engine to overcome this resistance is less and certainly used less fuel. Thus it is important to study what is the engineering approach that can be develop upon the car so that it will be aerodynamic and eventually create minimum drag force.

1.2 PROBLEM STATEMENT

Shell Eco Marathon Asia is a competition based on car race, which requires the participant to design and fabricate the most fuel saving cars. There are two categories being promoted, which are prototype and urban car type. The team representative in this category known as the SAE - UMP Chapter has took the challenge and builds up the urban car for the 2010 events at the Sepang International circuit, Malaysia. For that year, the team has won the third place in urban category. Since then, various improvements have been carried out by the team to compete for the top 3 position and create the most efficient car ever build in Universiti Malaysia Pahang. The existing model however do not obey the principal of streamline which significantly contributed to drag force. This thesis focuses on improvement of existing model in terms of aerodynamics. Aerodynamic is an important area of study as it relates to parasitic load experienced by the engine. The Shell Eco marathon is about to promote a car that runs with minimum fuel consumption.

1.3 OBJECTIVES

The main objective of this project is first to design fives model of urban car concept which is an improvement of existing model base on Shell Eco marathon Asia`s rules. The secondary objective is to analyze the aerodynamic features of the models and lastly to justify the best design based on required aerodynamic feature.

1.4 PROJECT SCOPES

This project consists of three main scopes that guide the project progress so that it get linear with the objectives. Firstly, the previous urban car design will be studied particularly in the shape of the body. The body of the car will be developed in Solidworks so that its drag coefficient can be obtained. After that, new five models will be developed in the same software and the drag coefficient is being analyzed. Among the fives design, there will be one design that will be chosen and that model is an improvement of the previous design in term of drag coefficient. The scope of the project can be summarized as follows;

i Focus on a development of 1-seated urban car body which consists of studying and designing of a 1-seated urban car

ii. Develop 3D models of the previous car body into Solidworks and perform CFD analysis using Flow Simulation to obtain drag coefficient.

iii. Configure the body shape of the car in term of size, dome angle, frontal area and apply those onto five new models to see the trend of aerodynamic drag.

iv. Perform computational fluid dynamic analysis to each design to obtain lowest drag coefficient.

v. The analysis of aerodynamic is focused on drag coefficient, drag force, aerodynamic power and relatives wake pressure and environment pressure

vi. Justify the best design based on aerodynamic feature.

CHAPTER 2

LITERATURE REVIEW

2.0 INTRODUCTION

This chapter briefly discusses about the study regarding analysis upon an upper body design for Shell Eco marathon competition in the urban car category. At first, the chapter discusses about the Shell Eco marathon Asia competition. This chapter then followed by a brief explanation regarding the urban car concept. Then, this chapter delivers the theory of aerodynamics of the ground vehicle. The section discusses about the official rules of the competition which is the most important part in considering the design configuration. As for the analysis, the review is about the significant of using Computational Fluid Dynamic analysis instead of a wind tunnel to determine the drag coefficient of the model.

2.1 INTRODUCTION TO SHELL ECO-MARATHON ASIA

Shell Eco-marathon challenges high school and college student teams from around the world to design, build and test energy efficient vehicles. With annual events in the Americas, Europe and Asia, the winner is the team that goes furthers distance using the least amount of energy. This event also affords an outstanding engagement opportunity for current and future leaders who are passionate about finding sustainable solutions to the world's energy challenge. There are two categories contested; urban category and prototype category. For this year, Malaysia hosts the stages for the third and the last time. This project reports however proposing the design of the urban car concept for the competition. The specific goal of the competition defines how the upper design of this concept car. On the other hand, the upper design is limited by rules and regulation as published by Shell in their official website. Basically the challenge is about to reduce the consumption of fuels as far as it goes. Understanding of how the upper car design helps to achieve this objective is the main issues in this report. As the car moves, the engine has to oppose the static air around the body. However, the body shape that is designed to streamline will reduce the power to overcome this static air and consequently allowing the flow of it through the body. This competition was an inspirational effort in order to promote automotive engineering and alternative fuel among the students. The lists of the winner from Asia level are shown in Table 2.1. It may not intrude into popular races of Formula One but with the competition leveling the technology achievements around the globe.

Table 2.1: Winners of Shell Eco Marathon Asia for 2010 and 2011 event`s

Place (Urban car)	**Fuel type**	**Events 2010**
1st	Hydrogen	NUS Urban Concept (Singapore). 612 km/l
2nd	Gasoline (Petrol)	MESIN ITS (Indonesia). 238km/l
3rd	Gasoline (Petrol)	Zamrud Khatulistiwa (Indonesia). 62km/l
4th	Gasoline (Petrol)	Yellow Makara (Indonesia). 54km/l
5th	Gasoline (Petrol)	UMP SAE Team (Malaysia). 39km/l.
Place (Urban car)	**Fuel type**	**Event 2011**
1st	FAME	MESIN ITS 4 (Indonesia). 150km/l
2nd	Gasoline (Petrol)	Cikal Nusantara (Indonesia). 117km/l
3rd	Gasoline (Petrol)	MESIN ITS 3 (Indonesia). 113km/l
4th	Gasoline (Petrol)	SEMART 2 (Indonesia). 71km/l
5th	Diesel	Team Up (Philippines). 60km/l

Source: http://www.shell.com/home/content/ecomarathon/results

2.2 CAR IN URBAN AREA

A city car (or urban car) is a small car intended for use primarily in an urban area. Mostly, this urban type car was invented using alternative fuel such as hybrid technology, fuel cell, electric powered system etc. This is on purpose to reduce the pollution in the urban area. According to Keiko Hirota in his article "Comparative Studies on Vehicle Related Policies for Air Pollution Reduction in Ten Asian Countries, (2010)", the contribution of carbon dioxide in Malaysia was 98 % discharged by automobiles. Where 42 % was discharged from passenger vehicles, 39.4 % are from van and lorries (truck) and 14.7 % was discharged from the motorcycle as recorded by the Department of Environment, (2004)

As the population increase it is important to keep the eco-system balance between the living thing and urbanization. Due to the approach of eco-friendly, the modern car was designed to be more efficient and fuel saving. Aligned with the concept of aerodynamics, the shape is likely to be rounded, small and lightweight. Technically, urban car is widely used in urban area.

Apart from addressing the concern of society for environmental protection and energy conservation through upper body design, the power system of the car also being developed such as hybrid vehicles. According to J. Y. Wong, 2008; the term 'hybrid-drive' is used to denote a drive system consisting of two or more types of power source to propel the vehicle, such as a combination of the internal combustion engine and the battery powered electric motor, or a combination of the fuel cell and battery to power the electric drive. The example of hybrid car is shown in Fig. 2.1

Figure 2.1: The ‘Riversimple’, a hybrid car

Source: http://www.popsci.com/hybrid car

2.3 THEORY OF AERODYNAMIC

2.3.1 Parameters of aerodynamic drag

This study is concerned with the drag force that exerted upon a moving ground vehicle. There is a study known as Automotive Aerodynamics where it emphasized on the aerodynamics of road vehicles. The main concerns of automotive aerodynamics are reducing wind noise, minimizing noise emission, and preventing undesired lift forces and other causes of aerodynamic instability at high speeds. Basically, the drag force or resistance of the aerodynamic denoted as F_d, is governed by several parameters that value the level of efficiency of the car. In practice, the aerodynamic resistance is usually expressed in the following form (Theory of Ground Vehicle, J. Y. Wong, 2008);

$$F_d = \frac{1}{2} C_d \rho A_f v^2 \qquad (2.3.1)$$

where,

F_d : Drag force, N

ρ : Fluid density, kg/m^3

C_d : Drag coefficient

A_f : Frontal area, m^2

V_r : Relative velocity, m/s

Aerodynamic resistance is proportional to the squared of velocity. Thus the power used to overcome the aerodynamic drag increase with the cube of the velocity. Aerodynamic condition affected by the air density ρ, and hence aerodynamic resistance. An increase in ambient temperature from 0 ℃ to 38 ℃ will cause a 14 % reduction in aerodynamic resistance, and increase in altitude of 1219 m will to decrease to resistance drag by 17 %. However the significant effects of ambient conditions on aerodynamic resistance can be neglected in this study as we are not interested to manipulate the altitude. The moving vehicle will produce the distribution velocity that's created skin friction due to viscous boundary layer which acts as tangential forces (shear stress) and eventually contribute to drag.

The improvement in the characteristic related through the drag force is ruled by Bernoulli Equation. Basic assumptions of Bernoulli's Equation for an air flow are viscous effects that are assumed to be negligible, the flow is assumed to be steady, which assumed to be incompressible and the equation is applicable along streamlines.

$$P = \frac{1}{2}\rho\, v^2 \tag{2.3.2}$$

where,

P : Pressure, kPa

ρ : Fluid density, kg/m³

v : Velocity, m/s

In an incompressible fluid flow experiment, equation (2.3.2) used to relate the parameter of pressure and velocity of the fluid. From the equation above it shows that the increasing of velocity will cause the decrease in static pressure and vice versa. In relation to force distribution of a moving car, the force due to pressure differential along the body surface created a perpendicular resultant force that contribute both lift and drag forces.

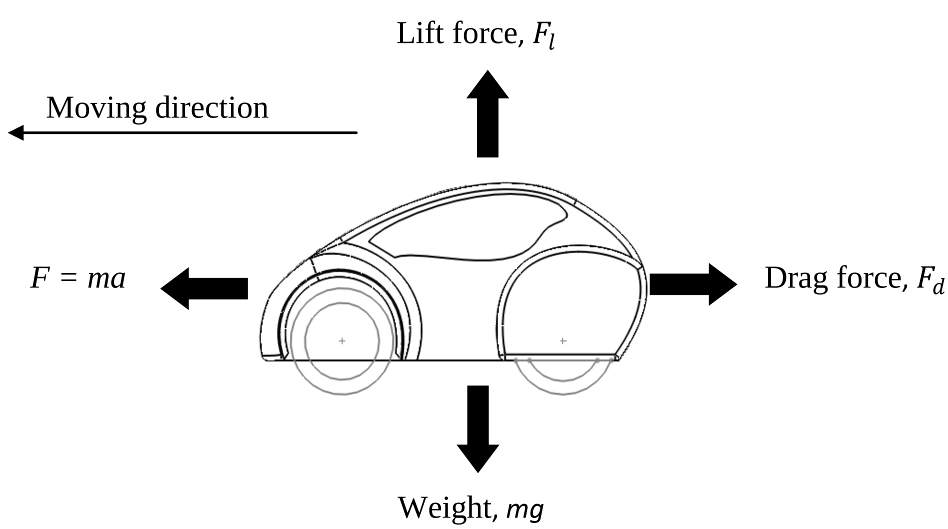


Figure 2.2: Component of force of a moving car.

However there are two components that contribute to total drag of a moving vehicle. The first one is the drag that was created along the body due to viscosity effect and is called drag force while the other one is the pressure drag which is due to the low pressure created at the rear of the body as shown in Fig. 2.3. The other component, directed vertically, is called the aerodynamic lift. It reduces the frictional forces between the tires and the road thus changing dramatically the handling characteristics of the vehicle. In addition to geometry, lift, F_l is a function of density, ρ and relative velocity, v_r. Lift is the net force (due to pressure and viscous forces) perpendicular to flow direction. The aerodynamic lift force is defined as follows.

$$F_l = \frac{1}{2} C_l \rho A_f \, v^2 \tag{2.3.3}$$

where,

F_l : Lift force, N

ρ : Fluid density, kg/m³

C_l : Lift coefficient

A_f : Frontal area, m²

v_r : Relative velocity, m/s

Aerodynamic lift and its proper front-and-rear-axle distribution is one of the key aspects in terms of on-road stability. As long as driving speed is low, below say 100 km/h, lift and pitching moment have only a small effect on the directional stability of a car, even in crosswinds. However, at higher speeds this is no longer true.

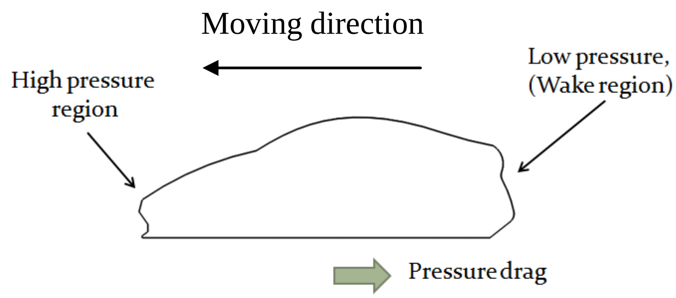


Figure 2.3: Pressure drag created in the wake region.

Some cars are designed with short tailed to comply with the compact mode. This can be seen clearly in urban car feature. The fact is the cut –off rear profile of the car create high changes of turbulent flow. As for the result, the air flow velocity is increased and consequently creates low pressure. This region is called wake region where the pressure is much lower compared to environmental pressure. The differential value of the pressure will trigger resultant force that opposed the motion of the car. The equation of resultant pressure expresses as follows;

$$\Delta P = P_e - P_w \tag{2.3.4}$$

where,

ΔP : Relative pressure, kPa

P_w : Wake pressure, kPa

P_e : Atmosphere pressure, kPa

Drag is considered as a parasitic load that reduces the performance of the engine. The aerodynamic power visualize the magnitude of power require to overcome the drag. The drag force affects the engine performance but the value does not correlate with the engine measurement. The power required to overcome the drag force is given by:

$$P_a = F_d \cdot v \tag{2.3.5}$$

where,

P_a : Aerodynamic power, kW

F_d : Drag force, N

v : Velocity, m/s

2.3.2 Implementation to improve aerodynamic feature

With growing emphasis on fuel economy and on the reduction of undesirable exhaust emissions, it has become increasingly important to optimize vehicle power requirements. To achieve this, it is necessary to reduce the aerodynamic resistance (drag), rolling resistance and inertia resistance. According to J. Y. Wong (2008), the aerodynamic resistance is generated by two sources: one is the air flow over the exterior of the vehicle body, and the other is the flow through the engine radiator system and the interior of the vehicle for purposes of cooling, heating, and ventilating.

The external air flow generates normal pressure and shear stress on the vehicle body. According to the aerodynamic nature, the external aerodynamic resistance comprises of two components, commonly known as pressure drag and skin friction. The pressure drag arises from the component of the normal pressure on the vehicle body acting against the motion of the vehicle, while the skin friction is due to the shear stress in the boundary layer adjacent to the external surface of the vehicle body. On the two components pressure drag is by far the larger, and constitutes more than 90% of the total external aerodynamic resistance of a passenger car with normal surface finishing (J. Y. Wong, 2008). Reducing aerodynamic drag is important for improved fuel consumption and higher top speed for a given power as shown in Fig. 2.4. The following is the regular way to approach aerodynamic drag reduction for land vehicle (Kalm, 2007).

i. The ground-up approach where the main body is shaped for low drag and then the non-aerodynamic element is designed within the body constraint.
ii. The improvement approach where the designer starts with the vehicle that already satisfies non aerodynamic constraint and finesses the detail to lower the drag as much as practical.

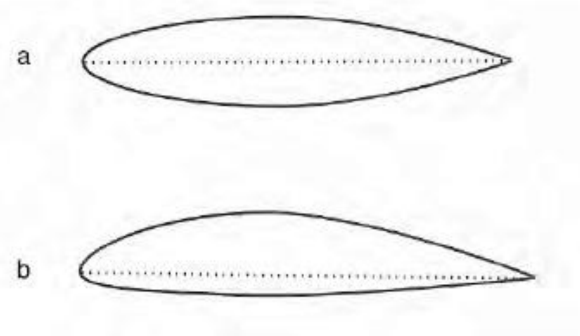


Figure 2.4: The shape of body (a) tear drop body, (b) aerofoil body

Source: Jaray and Klemperer (1920)

The first known streamlined land vehicle was developed by Jaray and Klemperer, (1920). They have discovered that, though an asymmetric teardrop body has the lowest drag in free air, as the body is brought closer to the ground, the drag force increases tremendously. For example, in the ground clearance found in automobiles, the drag force of the torpedo shape can be increased 50 %. If the ground clearance is zero, the drag force of the torpedo shape can be increase into almost 500 %. To preserve the robustness, Jaray and Klemperer invented the solution by chambering the body. To chamber the body, the belly can flattened or the top side of the body can be arched higher as in Fig. 2.4. b ,Kalm, (2007).

Around 1980`s, a professor of the Turin Technical University (Italy), Professor A. Morelli investigated whether it was possible for basis body near to the ground having an equivalent drag to the streamlined body in free air as shown in Fig. 2.5. The Morelli body achieved the minimum drag coefficient, based on frontal area under 0.05, matching that streamlined bodies are free air (Kalm, 2007)

Figure 2.5: Morelli streamlined car.

Source: Kalm, (2007)

The motion of the car on the ground introduces problems which differ greatly from those of an aircraft, primarily because of interference with the airflow between the car underside and the ground. That is because the car moves very closely to the ground this interference is one of the most important features of the airflow pattern around it. In many cars the underneath of the engine compartment is open to the ground to improve the cooling of the engine's crankcase. This cavity is formed full of structural or suspension members which may produce flow separation just behind the front bumper. Air inside this cavity is usually affected by the front grille and the high dynamic pressure region in front of the vehicle. This helps to create an additional aerodynamic lift force. Most of the car has a rough underside. The average roughness is ± 15 cm (or 6 inches) considered from a main surface level. According to A. J. Scibor-Rylski, (1984), airflow between the underside and the ground is however affected by the distance between the underside and the ground, the width, length and height ratio of the vehicle and the styling of the body shape, the roughness of the underside, and the lengthwise and crosswise curvature of the underside panel.

In a line at which the separation would be inevitable due to viscous friction, the shape is "bobtailed" as shown in Fig. 2.6. According to A. J. Scibor-Rylski, (1984), bobtailed the rear of the car reduces the size of wake region. The small size of the wake creates low changes of wake pressure to be developed. Eventually the pressure drag can be reduced.

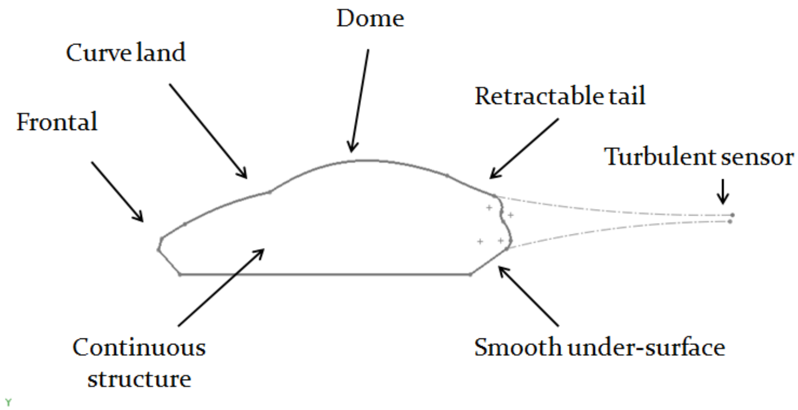


Figure 2.6: Drawing of the new airliner car.

Rear car rims technically create turbulent on the back of the car. Covering them reduces turbulence ordinarily created by the rotation of the wheels. Apart from that, much of the underside of the car is covered by flat sheets which streamline airflow beneath the car. Finally, there is the shape of the body, which mimics a falling teardrop. In real life application, a droplet of waterfalls, air and gravity mold it into the most aerodynamic shape, a teardrop. Honda`s engineers have created the car that mimics to the teardrop to reduce the drag as shown in Fig. 2.7

Figure 2.7: Honda Insight – covered rear wheel.

Source: www.km77.com

Aerodynamic factor has come to an important justification as it affect the power and performance. Drag coefficient technically represent the ratio of trust force to that drag created by the body. Table 2.2 show the type of object and rough drag coefficient obtained from experiment.

Table 2.2: The drag coefficient of object in moving fluid

Type of Object	Drag Coefficient
Old Car like a T-ford	0.7 - 0.9
Modern Car like Toyota Prius	0.26
Sports Car, sloping rear	0.2 - 0.3
Saloon Car, stepped rear	0.4 - 0.5
Convertible, open top	0.6 - 0.7
Common Car like Opel Vectra (class C)	0.29
Bus	0.6 - 0.8
Truck	0.8 - 1.0
Passenger Train	1.8
Motorcyvcle and rider	1.8

Source: : www.km77.com

2.4 SHELL ECO MARATHON ASIA OFFICAL RULES

The competition rules are important to determine the flow of the design. Without regulation, the design might be uncontrolled and the main objective of the competition as well cannot be achieved. In this competition, the organizer strict about the safety of the car, followed by other constrain mentioned in Shell Eco Marathon Official Rules. These rules changing every year as the team of competitor become more creative. One of the highlighted issues is that the car must not have any part that can be changed to help aerodynamic while running. It has to be by nature, the shape of the car itself to be aerodynamic.

The car must follow safety rules especially the roll bar and the bulkhead. Roll bar act like a bar extend 5 cm above the driver head to protect it if the car gets upside down. While bulkhead prevents any possibility of flame coming through from the engine compartment. Vehicle dimensions were briefly highlighted in the released rules. For examples, the vehicle height is in between 100 cm and 130 cm and the total vehicle width in range of 120 cm and 130 cm. The total vehicle length must be between 220 cm and 350 cm while the track width must be at least 100 cm for the front axle and 80 cm for the rear axle, measured between the midpoints where the tires touch the ground.

Besides that, the wheelbase must be at least 120 cm and the driver's compartment must have a minimum height of 88 cm and a minimum width of 70 cm at the driver's shoulders. Finally, the ground clearance must be at least 10 cm and the maximum vehicle weight (excluding the driver) must be 205 kg. The specification shown in Table 2.3

Table 2.3: The dimension of the urban car

Dimension specification in SEM 2012	**Value**
Height, cm	(100<h<130)
Width, cm	(120<w<130)
Length, cm	(220<l<350)
Wheel base distance, cm	(100<G<120)
Ground Clearance, cm	>10

2.5 COMPUTER AIDED DESIGN

Computer-aided-design (CAD) software allows the development of three dimensional (3D) designs from which conventional two-dimensional orthographic views with automatic dimensioning can be produced. The integration of computational fluid dynamics (CFD) methods in a wide range of engineering disciplines is rising sharply, mainly due to the positive trends in computational power and affordability, Joseph, (2008). The manufacturing tool path can be generated from the 3D model, and in some cases, part can be produced directly from a 3D database by using rapid prototyping and manufacturing methods (stereo lithography) -paperless manufacturing. Computer aided design bring new capabilities in designing a prototype. One of the branches of the analysis is a computational fluid dynamic analysis. Contrary to wind tunnel tests, the data can be viewed, investigated, and analyzed over and over, after the experiment ends. Furthermore, such virtual solutions can be created before a vehicle is built and can provide information on aerodynamic loads on various components, flow visualization, etc.

In summary, CFD became an important tool for studying the fluid flow over complex body profile such as a race car as shown in Fig. 2.8. It can be used as a preliminary design tool or to complement experimental methods. In providing fluid flow visualization information and details such as the aerodynamic load on an access door, expected pressure drop across a cooler, etc.

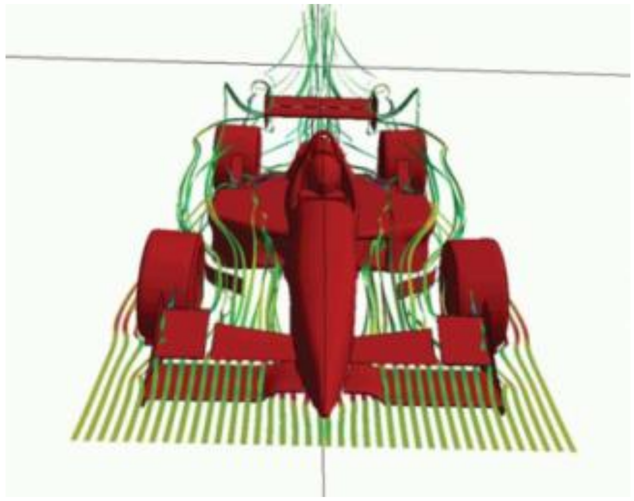


Figure 2.8: The streamlines release from the front wing cover until rear body.

Source: Aerodynamics of Race Cars, Joseph Katz (2008)

In this project, Solidworks are used to generate the model and to perform the analysis of fluid dynamic. There is a limitation of using CAD as the result is not reliable and restricted to approach the real condition. The main point is that the computational fluid dynamic approach helps a lot in preparing the preliminary results without having to spend expensive investment.

CHAPTER 3

METHODOLOGY

3.0 INTRODUCTION

The research methodology in the form of a flow chart is graphically shown in the Fig. 3.1, while the activity progress is shown in the Gant chart (Appendix). This planned activity organized the project into section by week. Basically, the research methodology of this project is related to the project scope as mentioned in Chapter 2, which are summarized as follows;

i. Identification of parameters
ii. Develop 3D models of the previous car body into Solidworks and perform CFD analysis using Flow Simulation to obtain the drag coefficient
iii. Implement aerodynamic feature onto new models.
iv. Perform computational fluid dynamic analysis to each design to obtain lowest drag coefficient.
v. Analyze the aerodynamic feature of the models in term of the drag coefficient, drag force, aerodynamic power and relatives wake pressure and environment pressure
vi. Justify the best design

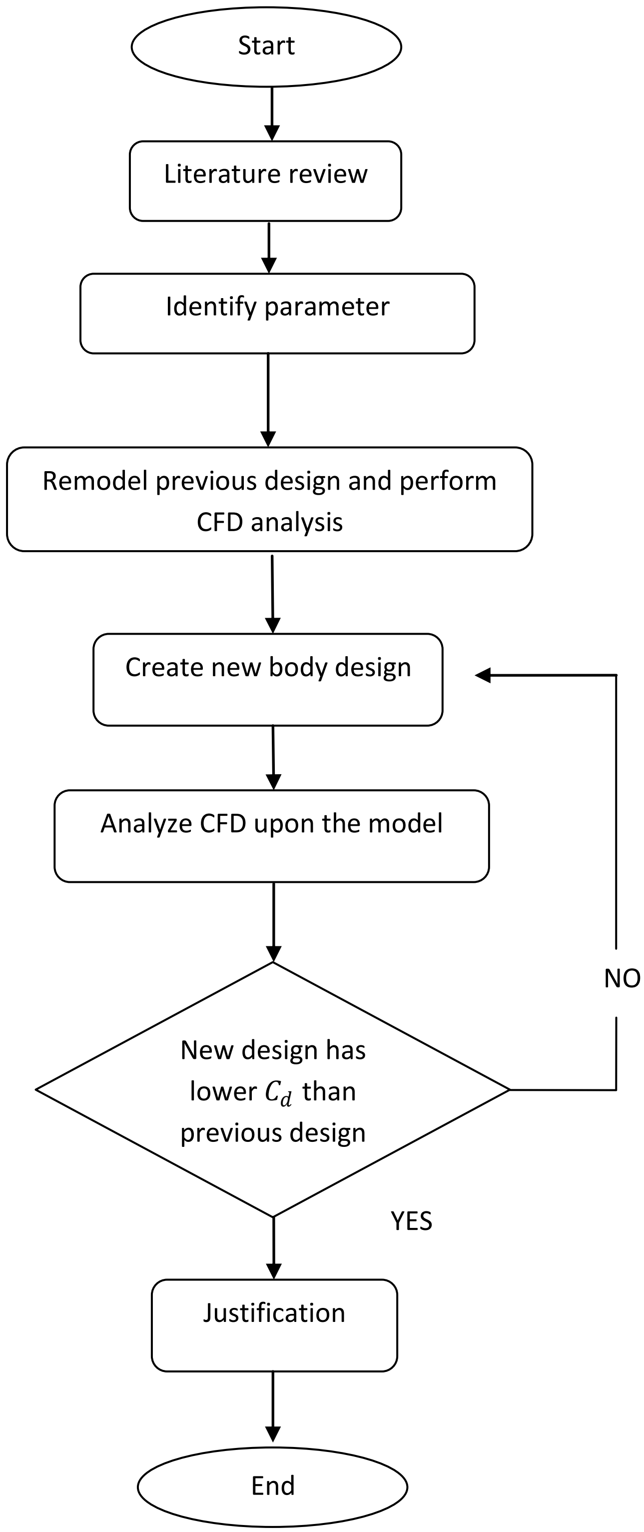


Figure 3.1: The flow chart of the project.

3.1 PARAMETER SELECTION

Basically the parameter involved in this analysis related to the scope of the project as mentioned in Chapter 1. In the process of selecting the best design, the parameter that is taken into concern is the drag coefficient, drag force, aerodynamic power and the relative pressure of wake region and environment pressure. However, there are several parameters that classified as constant variable and manipulated variable. The obvious variable being manipulated is the relative velocity. This is because the track on the circuit consists of 8 corners and 3 straight paths. It is difficult to maintain at constant speed. The parameters are summarized as in Table 3.1.

Table 3.1: List of parameters involved

Parameter	Symbol	Description
Drag force	F_d	Obtained from simulation.
Air density	ρ	Constant variable
Frontal area	A_f	Obtained from simulation (depend on car model)
Relative velocity	v_r	Manipulated variable
Drag coefficient	C_d	To be calculated from simulation data
Aerodynamic power	P_a	To be calculated from simulation data
Wake pressure	P_w	To be calculated from simulation data
Environment pressure	P_e	Constant variable

In Chapter 1, the best model selected will have a minimum value of the drag coefficient, aerodynamic power and relatives' wake and environment pressure. One model will be selected based on this feature and the method used will be shown in the next subtopic.

3.2 DRAG COEFFICIENT OF EXISTING MODEL

Existing model is an important part as to level the benchmark. The previous design of the Urban - Car by SAE UMP Chapter team is remodeled into Solidworks, Fig. 3.2. This model then analyzed using CFD to obtain the drag coefficient.

Figure 3.2: SAE UMP Chapter Urban car concept.

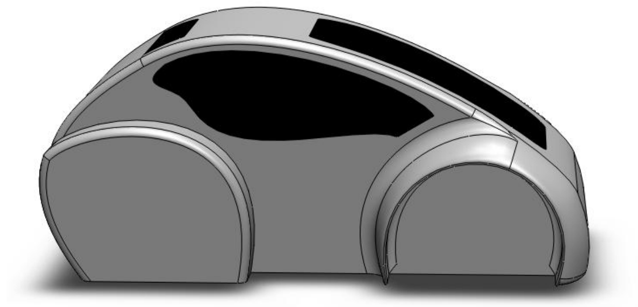


Figure 3.3: SAE UMP Chapter Urban 3D-model in Solidworks.

Once the existing model has been drawn into 3D model, the drag coefficient will be determined. Before performing the CFD analysis, it is important to decide the size of the computational domain. This computational domain represents the wind tunnel size as to fulfill the real condition as possible. The calculation of the dimension is shown in next subtopic.

3.2.1 Calculation of the entrance length of the tunnel

Although this analysis is run computationally, it is important to create a relation of real application and computer simulation. This calculation used to determine logical dimensions of the computational domain. In a real application of wind tunnel, the air flow does not develop constantly inside the chamber. This is due to friction between moving air and the internal wall of the tunnel. The viscosity of the fluid creates gradient velocity moving from the wall surface to the center of the duct. But at a certain distance, this flow will be fully developed as long as the shear stress being maintained all along the tunnel. This section intended to calculate the minimum length of the entrance length so that it will create fully developed flow to run over the car body.

The volume of the working chamber is taken as triple size of the model. The template model was used as the model size for the wind tunnel. Technically, the chamber was justified based on ergonomic factor. In real application it will be a lot easier to placing the car inside the room if the room is big enough. But if the size of the cross section is too big, more power (fan) will be needed to allow adequate air getting through the tunnel.

3.2.1 Parameter

There are three parameters that contribute to effective working chamber of the tunnel. The length of the tunnel must be long enough so that the distribution of the flow trajectory can be seen clearly. The parameters mentioned are as follows;

i. Velocity of the car, v (m/s)
ii. Cross sectional area of the working chamber tunnel, A (m^2)
iii. Renault number of air flow, Re

3.2.3 Entrance length, *El*

In a real application of air flow in a rectangular duct, the air flow velocity is not constant all along the tunnel. This is because the properties of boundary layer that gives friction upon the flow of the air. The shear between adjacent molecules of air creates a gradient velocity profile moving to the center of the duct. This shear can be maintained by using fans to power up the flow inside it. However the fully developed flow does not occur at the front of the tunnel. There is the minimum length of the chamber in order to achieve this laminar profile. This length known as entrance length and it is depending on the velocity of the air. The first rule of length selection determined by calculating the Reynolds Number of the air flow that entering the section. Then after the Reynolds Number was determined whether it is laminar or turbulent, the Reynolds Number then inserted into the Entrance Length equation to obtain the maximum required entrance length.

The Renault number was obtained by assigning the standard condition. In this calculation the standard condition is to follow the room condition where the parameter was assumed to be constant. The assumptions are as follow;

i. The study is conducted at $T = 25^{o}C$

ii. Relative velocity, v = 11.11 m/s (minimum velocity allowed for the urban car)

iii. The pressure is 1 bar in normal room condition.

iv. Air density, ρ = 1.184 $\mathrm{kg/m^3}$

v. Air dynamic viscosity, μ = 0.01827 kg/m·s

Reynolds Number;

$$Re = \frac{\rho v d_h}{\mu} \tag{3.1}$$

where,

Re : Reynolds Number (to be determined)

ρ : Air density, kg/m^3

v : Velocity, m/s

d_h : Hydraulic diameter rectangular duct, m

μ : Air dynamic viscosity, kg/m·s

From previous consideration the cross section of the duct is twice as the size of the car-front-cross sectional. Thus, area, A of the rectangular duct

$$A = (1.3 \text{ x } 1.3) \text{ m}^2.$$

Hydraulic diameter, d_h;

$$d_h = \frac{h \text{ x } (a \text{ x } b)}{(a \text{ x } b)^{0.25}} \tag{3.2}$$

where;

$$a = (1.3 \text{ x } 2) + 1.3$$
$$= 4\text{m}$$
$$b = (1.3 \text{ x } 2) + 1.3$$
$$= 4\text{m}$$

Thus, hydraulic diameter is;

$$d_h = \frac{1.3 \text{ x } (4 \text{ x } 4)}{(4 \text{ x } 4)^{0.25}}$$

$$d_h = 4.373 \text{ m}$$

So Renault number is then;

$$Re = \frac{(1.1845)(11.11)(4.373)}{(0.01827)}$$

$$Re = 3149.85$$

For internal flow of fluid in closed chamber, the range of Reynold number is laminar for Re number equal to 2300 while the flow is categorized as turbulent for Re greater than 3000. Reynold number in between this range is considered as transition flow. In this calculation the Reynold number will be considered as turbulent in order to cover the worst cases.

For turbulent value of Reynolds Number, The entrance length formula is;

$$El \text{ turbulent} = 4.4\, Re^{\frac{1}{6}} \tag{3.3}$$

Thus, *El* turbulent,

$$El = (4.4)(3149.85)^{1/6}$$

$$El = 16.846 \text{ m}$$

Then the entrance length value is inserted in Length of Entrance formula.

where;

$$L_e = \frac{E_l}{d_h} \tag{3.4}$$

Thus,

$$L_e = \frac{16.846}{4.733}$$

$$L_e = 3.852 \text{ m}$$

Therefore, the minimum length of the full airflow developed is 3.582 m. Technically the velocity of the air inside the duct is at 40 km/h. Thus this length is adequate to create fully developed flow so that the air that coming over the body is constantly at 40 km/h. The structure of the working chamber is shown in Fig. 3.4

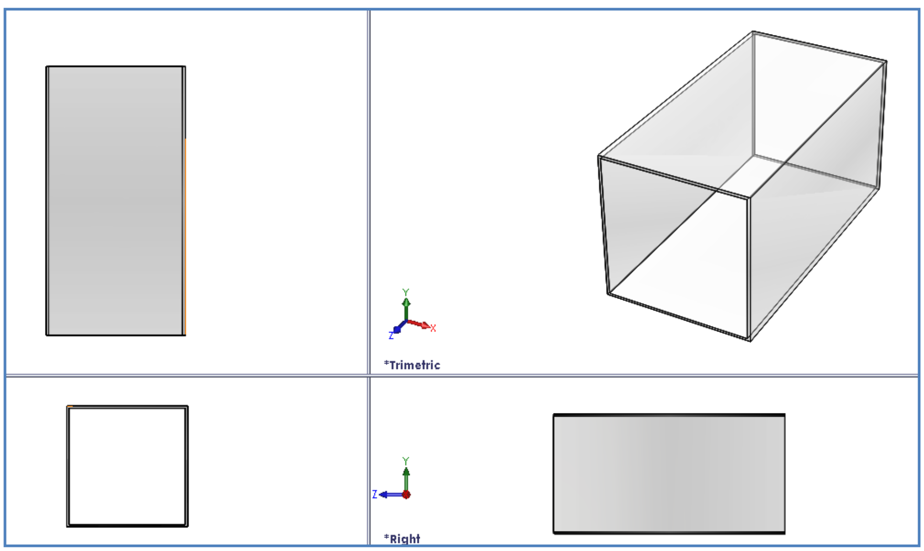


Figure 3.4: The wind tunnel designed in Solidworks.

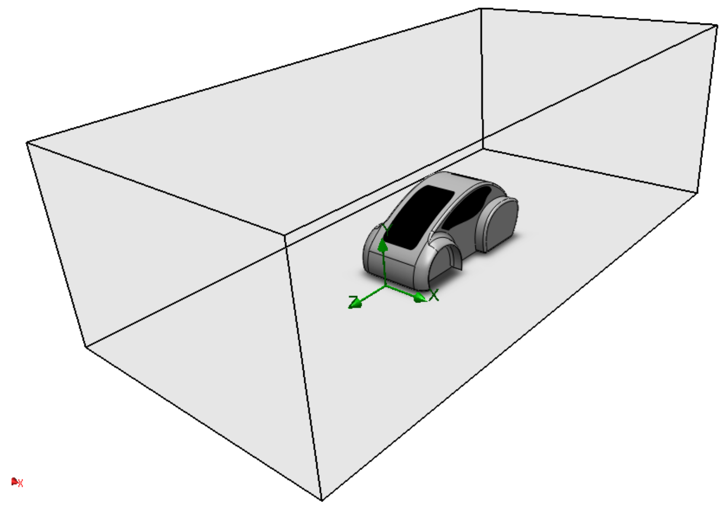


Figure 3.5: 3D model of SAE UMP Chapter Urban-car in computational domain setup.

3.2 .4 Drag coefficient of existing model

In this analysis, the speed of the car was set to be 40km/h while the air pressure is at 1 atm, the temperature of environment is 25 °C and the air density approximately equal to 1.184 kg/m^3. The drag force is marked as goal plot at the end of the simulation. Following table shows the value of drag force computed.

Table 3.2: Drag force of the model

Name	Unit	Value
Drag force	N	94.005

Calculation of drag coefficient was calculated using eq. 2.3.1 as follow;

$$C_d = \frac{2F_d}{\rho v^2 A_f}$$

$$C_d = \frac{(2)(\ 94.005)}{(1.184)(11.11)^2(2.8863)}$$

$$C_d = 0.45$$

Previous model of Urban car designed by SAE UMP Chapter was analyzed using the CFD method and the drag coefficient of the model obtained is 0.45. This value technically quite high for a car designed to reduce the fuel consumption. Thus, this value will be the benchmark for the next model so that it will be lower than 0.45. From Table 2.2, the drag coefficient that of 0.40 is similar to saloon car profile where the drag coefficient is in between of 0.4 to 0.6. This value can be decrease. The room of improvement was then identified from the model as shown in Fig. 3.6

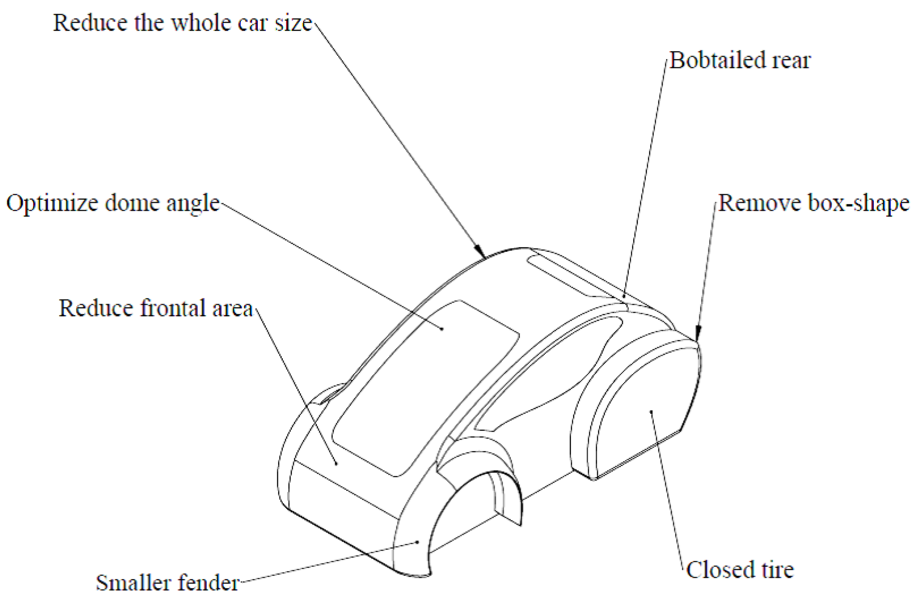


Figure 3.6: Identified component to be improved

3.3 DESIGNING THE NEW MODEL

Initially, there are various aspect needs to be considered to fulfill the design requirements. The existing body was designed according to the Shell Eco-marathon rules and regulation which include the maximum width, maximum height and maximum length of the car. There are few parameters need to be considered to analyze the body design. They are frontal area, drag force of the car, relative velocity and the air density. All of these parameters were applied to the different velocity value from 40 km/h to 80 km/h. The organizer has provided the dimension to standardize the size and the look of the car. Basically the rules emphasize on ergonomic feature especially for the safety of crews and the driver. The designed car must pass the inspection session before the competition. Otherwise, the team will be rejected before being able to send the car into the track. The dimension specification is as below.

Table 3.3: The dimension specification.

Dimension specification in SEM 2012	**Value**
Height, cm	$(100<h<130)$
Width, cm	$(120<w<130)$
Length, cm	$(220<l<350)$
Wheel base distance, cm	$(100<G<120)$
Ground Clearance, cm	>10
Door opening area, cm^2	$>(50x80)$
Luggage area, cm^2	(50x40x20)
Wheel allowed diameter, cm	$(33.02< d <43.18)$

As to make the design process easier, the dimension of the car is proposed and sketched into Solidworks as the template. This will help the process of customizing for which part and component should be placed onto the new design. The template designs are as shown in Fig.3.6

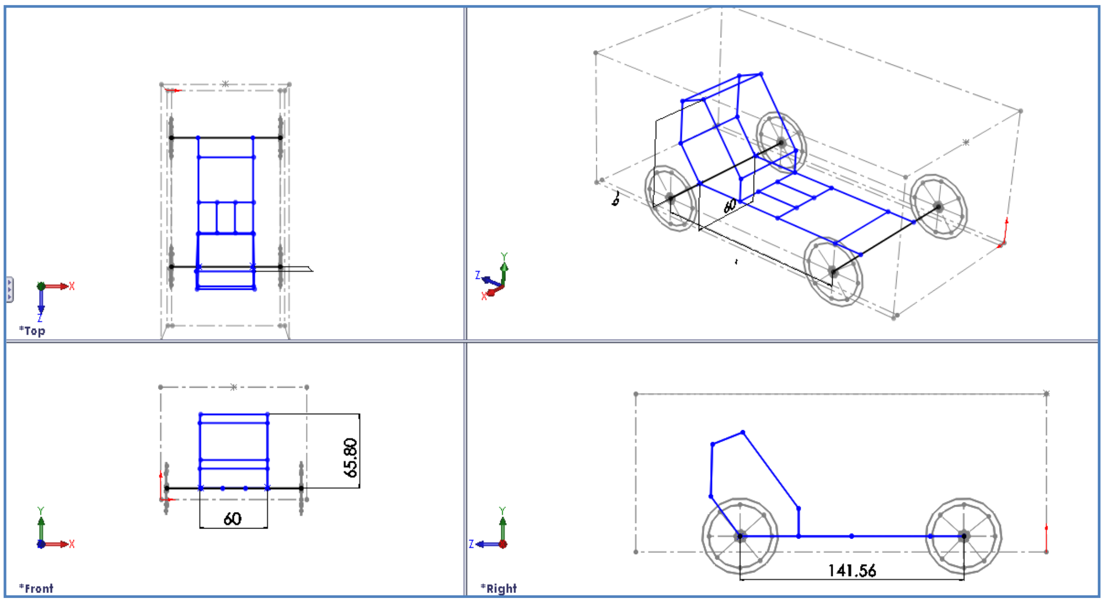


Figure 3.7: The template design for upper body profile.

3.4 DESIGN IMPLEMENTATION

The new design will have several improvement based on literature review. The improved feature is implemented on separated model as it cannot be applied into one single model. The problem and solution was summarized as in Table 3.4 while the aerodynamic feature that are installed into the model is as proposed in Table 3.5

Table 3.4: Source of aerodynamic drag and the solution.

Problem	Parameter	Solution
Flow separation of the bluff body create turbulence flow, causing high drag force on the body	C_d	Create streamlined body, allowing smooth flow of the air upon the car body.
Skin friction	Total surface area, A	Reduce total surface area of the car, create small but ergonomic for driver.
Boundary layer pressure loss		Create thin bodies instead of ticker, because thin bodies tend to reduce the drag due to boundary layer pressure loss.
Drag to the body increase with lift force.	C_L	Minimum drag occurs when the body has zero lift, and so the angle attack of the car should be adjusted to zero lift.
Pressure drag due to wake region	F_d	Create bobtail feature at the rear of the body to reduce premature flow separation.

Problem	Parameter	Solution
Static pressure accumulated at the orthographic projection of the car normal to the direction of moving.	Drag area, A_f	Reduce the frontal area such as create a lean feature started from the front to the top side of the car.
Turbulence flow on the rear wheel	C_d	Cover the rear wheel to avoid turbulence that create pressure drag.

Table 3.5: Aerodynamic feature and models

Component feature	Model				
	1	2	3	4	5
Curved shape	✓		✓		✓
Covered wheel	✓	✓	✓	✓	✓
Fender	✓		✓	✓	✓
Bobtailed rear	✓	✓		✓	

3.5 COMPUTATIONAL FLUID DYNAMIC ANALYSIS

From the car model designed, CFD analysis is set to be run at constant condition for all models. For example, the air density and computational domain is constant at varied velocity for each models. From the simulation, the value of drag force can be obtained. However the aerodynamic factor is not only relying on drag force. It is important to consider the drag coefficient, the aerodynamic power and the wake pressure as to select the best aerodynamic body. From Chapter 2, we know that the efficiency of the car depends on the drag force. As if the drag force is small, then the cars require less power to overcome the drag. It is common to compare the aerodynamic feature in term of drag coefficient. At the end of simulations, the drag coefficient for all models will be calculated to select the best model in term of drag coefficient. Apart from that since the other component of the drag force is coming from the pressure drag, the wake pressure of the car is taken into consideration as to determine the best body in term of pressure drag. Best model should have small relative pressure between wake pressure and environment pressure. Lastly is the aerodynamic power. This parameter is calculated at the end of the simulation in order to show the strong relation between drag force and the engine performance. Usually the engine will have to overcome the parasitic load before it can move the car. But in terms of aerodynamics, the parasitic load is identified from the drag force. Power is equal to force times the velocity. The higher value of drag force at high speed, the greater power will be required to overcome the drag. This will increase fuel consumption and reduce the engine efficiency. The best model selected must have the minimum aerodynamic power among the others. The method of model selection will be discussed in the next subtopic.

3.6 MODEL SELECTION METHOD

Model selection is based on the criteria mentioned in Chapter 2. Basically, the design selected must fulfill the aerodynamic feature which has the minimum average drag force, minimum drag coefficient, minimum aerodynamic power and minimum relative pressure of the wake region of environment pressure. The technique used is by assigning the weighting of the value into the range of selection number. The result of the calculation of all parameters is filled in the table and the highest weighted will show the best model. This weightage then will be imported into Spider-web graph to determine the widest area filled by the model. The high value of weightage should fill up wider area in Spider-web graph. Thus the model with widest area will be the best design.

CHAPTER 4

RESULT AND DISCUSSION

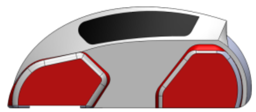
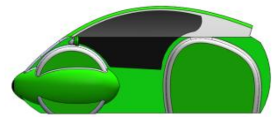
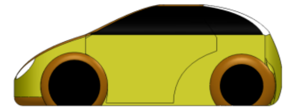
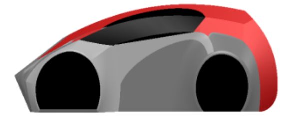
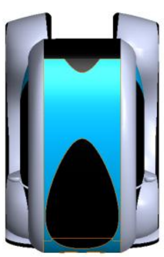
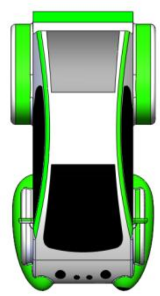
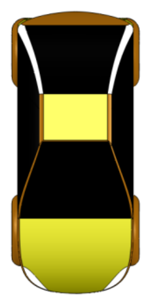
4.0 INTRODUCTION

This section discusses the result obtained from the simulation. At first, the new designed model is presented in section 4.1. The layout of the model in this section is intended to view the obvious different of the upper body shape created. After that, data analysis of the simulation will be presented in section 4.2. Along with the result, the parameter for drag coefficient, aerodynamic power and relatives wake pressure to environment pressure will be calculated and presented in table and graph. The relation of parameter can be seen clearly in data presentation sub-chapter. Detail discussion on graph trend and model selection is presented in section 4.3 and 4.4.

4.1 THE NEW MODEL

According to Chapter 3, the new design will have several improvements from existing models and design implementation of literature review. However, the customization is applied onto fives different models. The following models are created within the range of giving dimension in Shell Eco marathon Asia`s rules. From Fig. 4.1, the first model is created in wide curvature; the shape is likely to be rounded from the front to the rear. The rear is bobtailed while the fender is created biggest among the other. Model 2 is designed so that it has a smaller frontal area compared to the first model. The shape is almost streamlined but the fender is designed in a boxy shape. The rear part is not bobtailed and the total area is smaller compared to the first model. Model 3 has is the combination of model 1 and model 2. This makes the total area of the model is the widest among the other models. The frontal area is designed to be smaller compared to model 1 and model 2. Model 4 is still the combination of model 1 and model 2 but the fender has been removed. The rear part is bobtailed more compared to the first model. The front angle is created to be leaner compared to other previous three models. Model 5 is a recreation of model 4 but the total area has been reduced. This makes the model become the smallest among the others.

Table 4.1: Projection of top, front and right view of the five new models that being tested in Flow simulation Solidworks.

	Model 1	**Model 2**	**Model 3**	**Model 4**	**Model 5**
Right view					
Front					
Top view					

4.2 DATA PRESENTATION

All data simulations are recorded and analyzed in this section. Once the simulation in Flow Simulation is complete, the drag coefficient can be determined. Apart from that, the aerodynamic power also been calculated. Aerodynamic power is the power that required by the engine to overcome the drag.

4.2.1 The result of car models drag force.

Table 4.2.1: Drag force in function of velocity with different model.

Velocity , km/h	**Drag Force, F_d (N)**				
	Model 1	**Model 2**	**Model 3**	**Model 4**	**Model 5**
40	56.13	57.16	72.52	69.34	52.32
50	92.52	91.32	113.12	114.64	82.62
60	132.87	132.86	178.46	167.98	119.46
70	181.23	182.34	253.54	231.17	162.90
80	243.51	286.98	354.98	387.65	215.14

4.2.2 Result of drag coefficient of car models

Table 4.2.2: Drag coefficient in function of velocity with different models.

Velocity, km/h	**Drag coefficient**, C_d				
	Model 1	**Model 2**	**Model 3**	**Model 4**	**Model 5**
40	0.310	0.263	0.302	0.273	0.289
50	0.325	0.268	0.301	0.289	0.292
60	0.324	0.271	0.330	0.294	0.293
70	0.327	0.273	0.345	0.297	0.294
80	0.336	0.329	0.369	0.382	0.297

Table 4.2.3: The averages drag coefficient of the models.

Model	**Average drag coefficient,** $C_{d(avg)}$
Model 1	0.324
Model 2	0.281
Model 3	0.329
Model 4	0.307
Model 5	0.293

4.2.3 Aerodynamic power of the models at different velocity.

Power is the rate of changes of work. In case of aerodynamic analysis, the aerodynamic power refers to the power required to overcome the drag force at specific velocity. From equation of drag force;

$$F_d = \frac{1}{2} C_d \rho A_f v^2 \tag{2.3.1}$$

And power is known as;

$$P_a = F_d \cdot v \tag{2.3.5}$$

Thus, model 1, at a velocity of 40 km/h, with drag force obtained from simulation is 56.13 N gives aerodynamic power as follow.

$$P_a = F_d \cdot v$$

$$P_a = (\, 56.13).(11.11)$$

= 6.9072 kW

Table 4.2.4: Aerodynamic power of all models

Velocity, km/h	Model 1		Model 2		Model 3		Model 4		Model 5	
	Drag force, N	Aerodynamic power, kW	Drag force, N	Aerodynamic power, kW	Drag force, N	Aerodynamic power, kW	Drag force, N	Aerodynamic power, kW	Drag force, N	Aerodynamic power, kW
40	56.00	6.9072	57.16	7.0502	72.52	8.9451	69.34	8.5525	52.32	6.4533
50	92.00	17.7497	91.32	17.6186	113.12	21.8245	114.64	22.1177	82.62	15.9401
60	132.00	36.6813	132.86	36.9203	178.46	49.5921	167.98	46.6798	119.46	33.1966
70	181.00	68.4024	182.34	68.9088	253.54	95.8162	231.17	87.3623	162.90	61.5623
80	243.00	119.9760	286.98	141.6902	354.98	175.2637	387.65	191.3938	215.14	106.2207

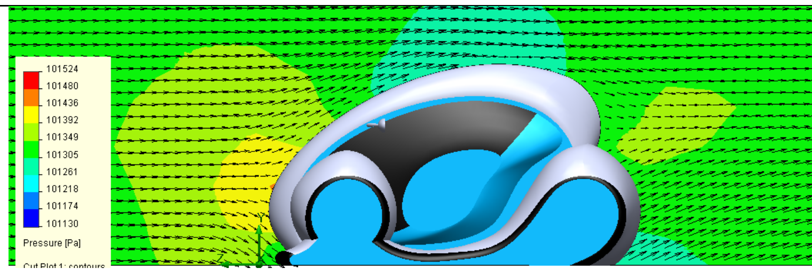
Table 4.2.5: Mean aerodynamic power of all models

Model	Mean aerodynamic power, kW
1	9.374
2	10.109
3	65.411
4	65.778
5	42.006

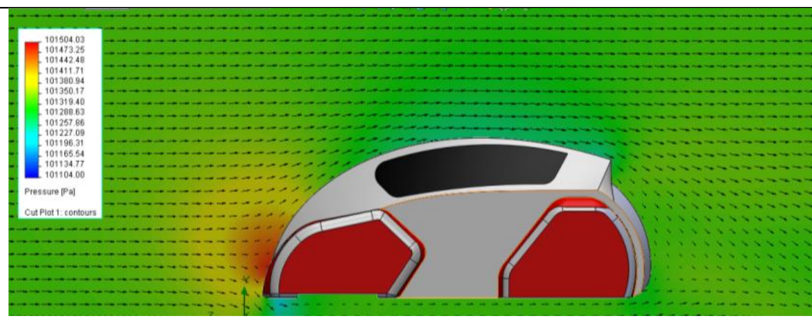
4.2.4 Relative pressure of wake region to environment

From the simulation, the pressure distribution around the body can be determined. Aerodynamic drag in external flow analysis consists of two components which is drag force and pressure drag. The pressure drag is the drag due to wake region located in the cutoff body feature. The relative pressure between wake region and atmospheric pressure cause the car to move in opposite directions. The following figure shows the cut-plot of the pressure distribution.

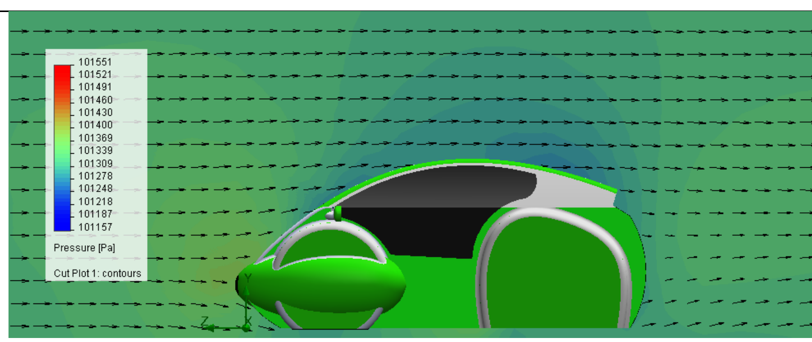
Model 1



Model 2



Model 3



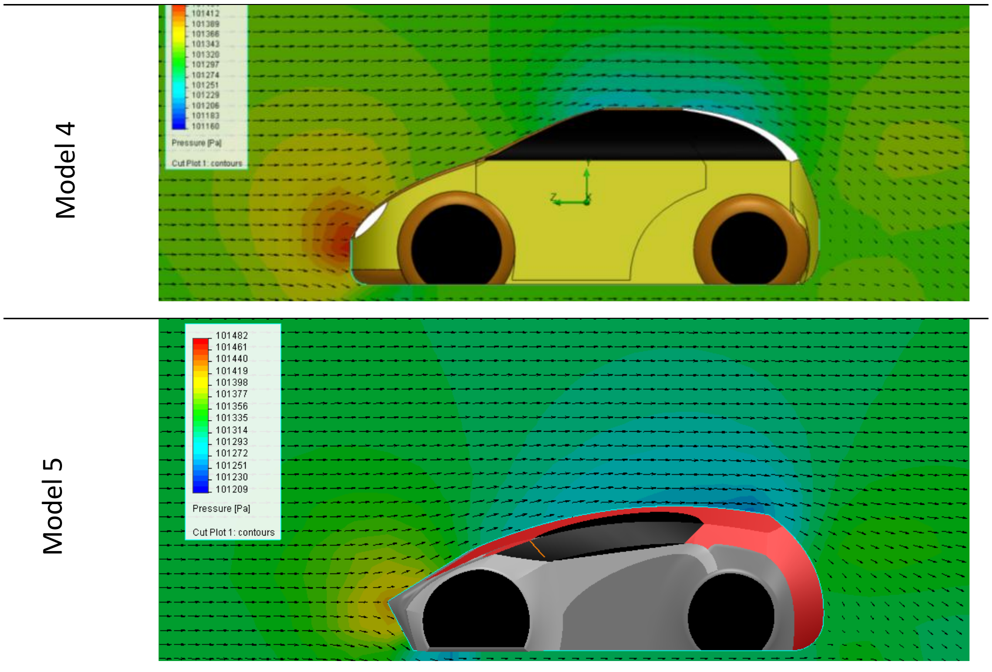


Figure 4.2: The cut-plot of pressure distribution of the car models.

Fig. 4.2 located the low pressure and high pressure area around the body. From the cut-plot of the right side view, the wake region is clearly shown in dark color indicating the high velocity area. This analysis is intended to identify the model that creates minimum relative of wake pressure to atmospheric pressure. The value of relative pressure between wakes region and atmospheric at mean velocity are as shown in following table.

Table 4.2.6: Relative pressure between wake pressure and atmospheric pressure.

Model	Wake region pressure, kPa	Atmospheric pressure, kPa	Relative pressure, kPa
1	101.305	101.325	0.020
2	101.319	101.325	0.006
3	101.278	101.325	0.047
4	101.274	101.325	0.051
5	101.314	101.325	0.011

4.3 RESULT AND DISCUSSION.

In this section, data obtained from the simulation was transferred into the graph. The trend of the drag of the fives model against velocity was shown in Fig. 4.3. While the profile of the drag coefficient of the fives model against velocity shown in Fig. 4.4. Aerodynamic power is an important aspect needed to be considered as to reduce the fuel consumption to overcome the drag. The aerodynamic power for all models is shown clearly in Fig. 4.6.

4.3.1 Drag force of the models at different velocity.

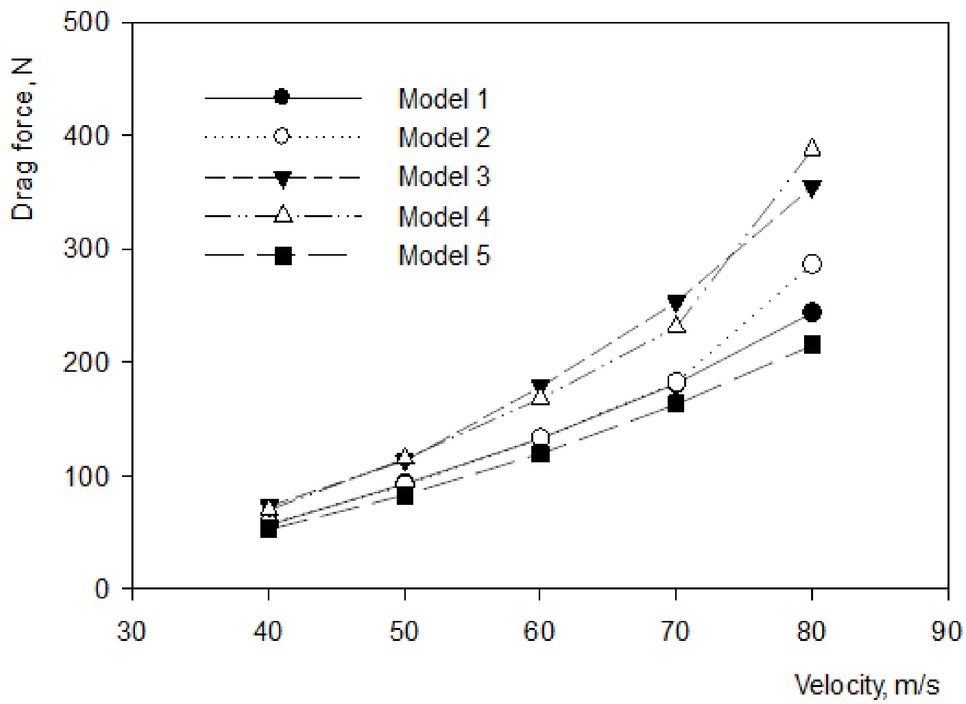


Figure 4.3: Drag force of the models at different velocity.

The simulation result shows the different trend of the drag force created by each model at different velocities. The velocity of the air flow was set to be 40 km/h, 50 km/h, 60 km/h, 70 km/h and 80 km/h. From the graph of drag force against velocity shown in Fig.4.4, the drag force is proportional to the velocity of the air flow. From the graph, the value of drag force that changes from minimum to maximum velocity was determined.

According to the figure, model 4 shows biggest increase of drag force which is equal to 82%, followed by model 2, which is approximately equal to 80% and model 1, and increased as much as 76%. Model 5 and model 3 shown the lowest drag force changed which is equal to 79% and 75% respectively. For the drag force that coming from the body, it is mostly manipulated by parameter of velocity, fluid density and frontal area. Model 4 has the biggest drag force changes because of the frontal area of the model is the widest among the other which is equal to 3.42 m^2. While the frontal area for model 1, model 2, model 3 and model 5 is 2.47 m, 2.98 m, 3.28 m and 2.47 m respectively. According to Fig. 4.3, model 3 shows the lowest drag force changes. Model 3 has potential to be selected because theoretically it will save fuel consumption.

4.3.2 Drag coefficient of the models at different velocity.

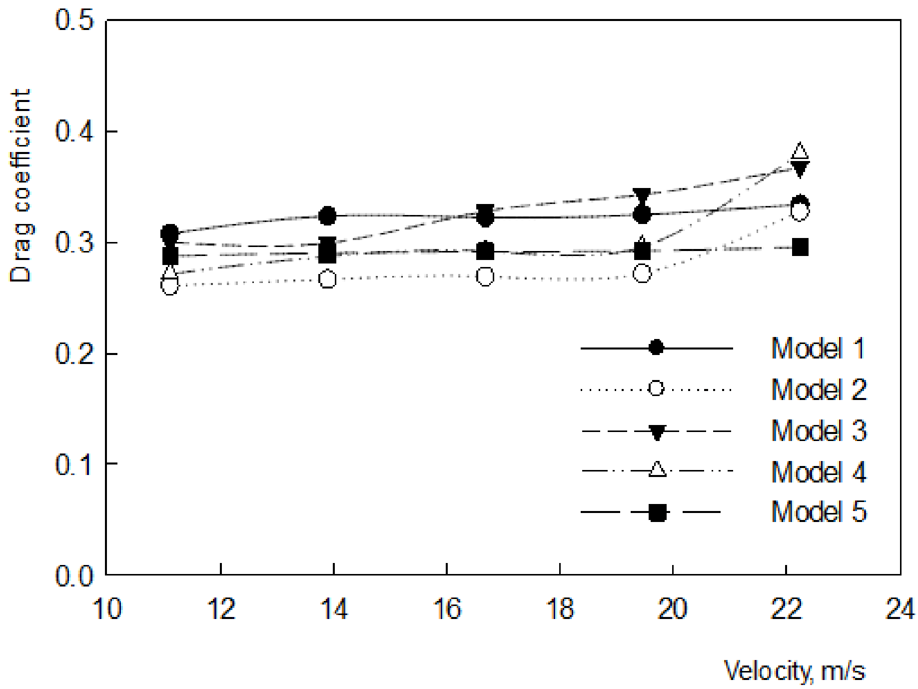


Figure 4.4: Drag coefficients of the models at different velocities.

The drag coefficient for all models was taken as an average value. This is because, the drag coefficient is not constant at different velocities. However the trend of the drag coefficient changed with velocity was clearly shown in Fig. 4.4. From the figure, model 3 has the highest drag coefficient at all velocities compared to the others. From table 4.3, mean drag coefficient for model 3 is 0.329, while the average drag coefficient for model 1, model 2, model 4 and model 5 is 0.324, 0.281, 0.307 and 0.293. From Fig.4.4, the order of increasing mean drag coefficient of the model is model 2, model 5, model 4, model 1 and model 3. According to mean drag coefficient value shown in table 4.4, model 2 has the potential to be selected because of the minimum mean drag coefficient among the others. From the equation of drag force, drag coefficient changed proportional to the drag force. The small values of drag coefficient create less drag force. This will help the engine to reduce the power to overcome the drag force that opposes the car moving direction. In terms of drag coefficient, model 2 is highly recommended to be the selected model.

4.3.2 Aerodynamic power of the models at different velocity.

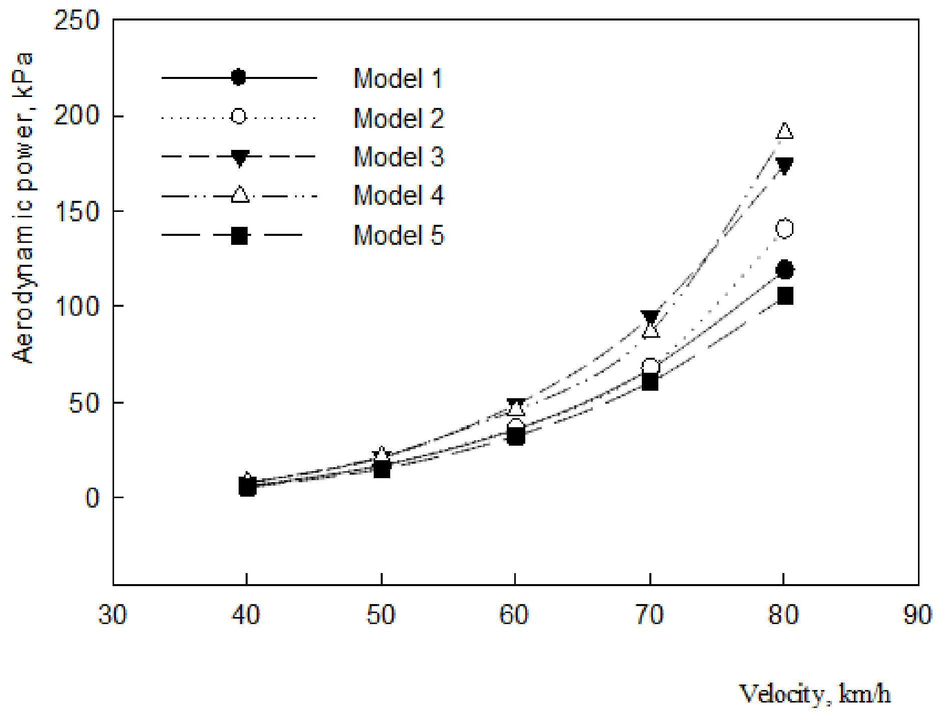


Figure 4.5**:** Aerodynamic power of all models at different velocity.

In general, the car performance was measured from its engine power. However, the engine technically has to overcome several parasitic loads that reduce the efficiency. From Fig. 4.5, model 1 show that the aerodynamic power changes are the minimum among the other which is in the average value it is approximately equal to 9.374 kPa. The parasitic load in increasing magnitude is followed by model 2 which is approximately equal to 10.109 kPa, model 4 by 42.006 kPa, model 3 by 65.411 kPa and model 5 which is equal to 65.778 kPa. According to the Fig 4.5, the best model was represented by model 1 which have the minimum aerodynamic power or parasitic load and this show that it will save fuel consumption when bring into fabrication.

4.3.3 Relative pressure of the models at different velocity.

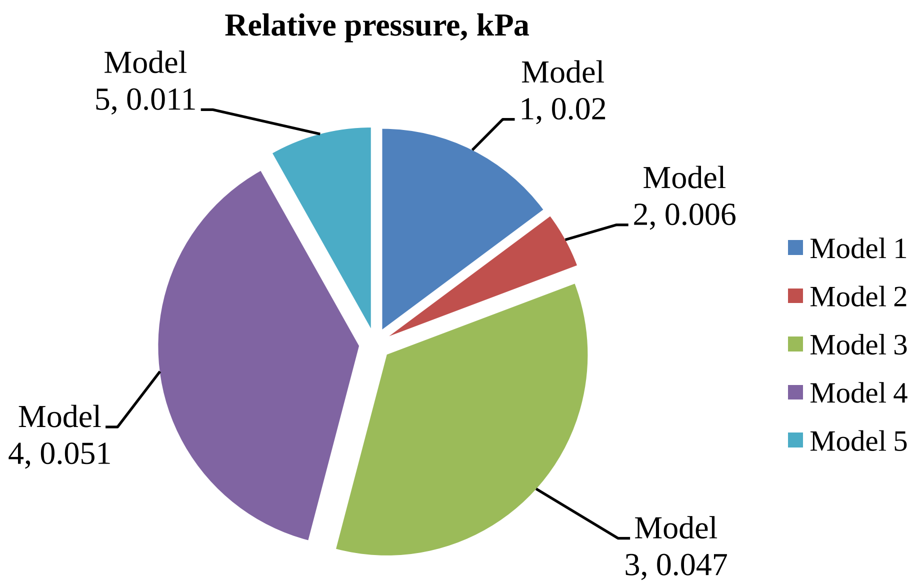


Figure 4.6: The relative pressure (kPa) of wake region to atmospheric pressure.

From Fig. 4.6, the lowest relative pressure of the wake region to that atmospheric pressure is represented by model 2. This showed that the rear part of model 2 has created small wake regions that induce pressure drag. In other words, model 2 has a minimum pressure drag. The order of increasing pressure drag is model 5, model 1, model 3 and model 5. The significance of quantitative analysis of relative pressure is to determine the total external drag exerted on the body. The analysis shown that model 2 has a high possibility to be selected as the best model. Based on justification mentioned in the previous graph, one model will be selected. The method of selection is shown in the next subtopic.

4.4 MODEL SELECTION

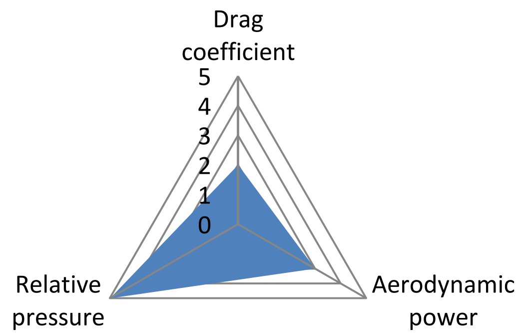
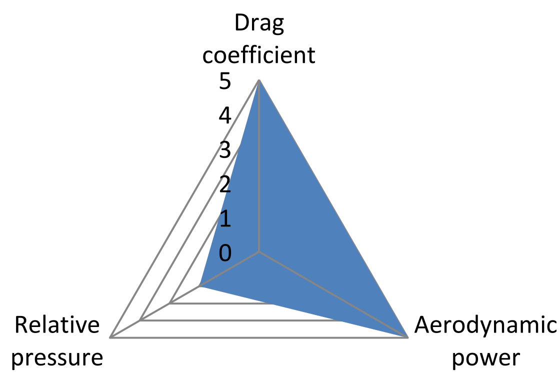
This section shows the method of justifying the best model. The plot of spider-web graph is used as to show the area covered by the model in required weightage. The magnitude of the result is assigned to the weightage ranged from 1 to 5 as shown in Table 4.4.1

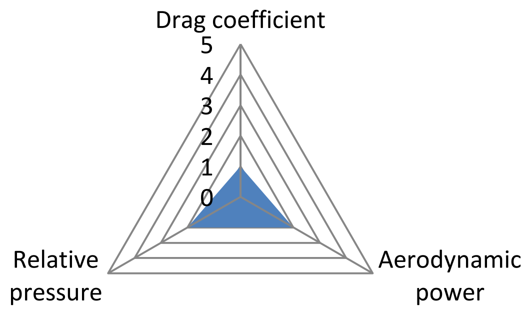
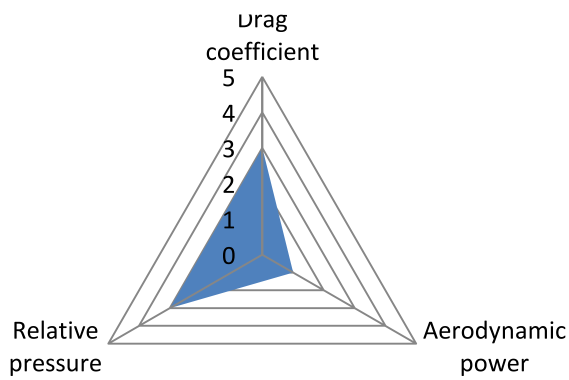
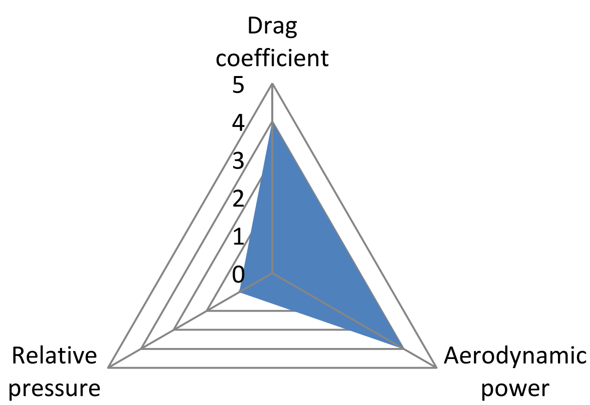
Table 4.4.1: The parameter weightage.

Parameters	**Weightage**				
	5	**4**	**3**	**2**	**1**
Drag coefficient, $C_{d(avg)}$	0.281	0.293	0.307	0.324	0.329
Relative pressure, *ΔP,* (kPa)	0.006	0.011	0.020	0.047	0.051
Aerodynamic power, P_a, (kW)	9.374	10.109	42.006	65.411	65.778

According to the table, magnitude of weightage is inversely proportional to the magnitude of parameters. The area cover by each model in Spider-web plot is shown in Table 4.4.2.

Table 4.4.2: Area covered by the models in the Spider-web graph.

Model	**Spider-web plot**	**Area computed, *A***
1		1050 cm²
2		2000 cm²

Model	Spider-web plot	Area computed, *A*
3		300 cm²
4		300 cm²
5		1000 cm²

From Table 4.6, it is clearly seen that model 2 have the highest weightage in all parameters and a computed area of 2000 cm^2 in Spider-web plot. This indicated that model 2 has the minimum drag coefficient, minimum aerodynamic power and minimum relative pressure of the wake region of environment pressure. Thus, model 2 is selected as the best model among the other fives. Model 2 was designed so that it has a smaller frontal area compared to the first model. The shape is almost streamlined but the fender is designed in a boxy shape. The rear part is not bobtailed and the total area is smaller compared to the first model. This analysis shows that the combination of small total surface area and streamlines create low pressure drag at an average velocity of 60 km/h. Small total surface creates less friction drag between the air and the body surface while the streamline body keeps the air flow remain on the body surface, preventing development of wake region.

CHAPTER 5

CONCLUSION

5.1 CONCLUSION

The main objective of this project is to design new fives model of urban car based on Shell Eco marathon Asia rules while improving the existing design in term of drag coefficient. The drag coefficient of existing model made by SAE UMP Team in Shell Eco marathon Asia 2010 competition`s is 0.45 while mean drag coefficient for the new model is 0.324 for model 1, 0.281 for model 2, 0.329 for model 3, 0.307 for model 4 and 0.293 for model 5. Maximum improvement is achieved at model 2 where the drag coefficient has decreased by 37%. From the simulation of CFD, the aerodynamic profile of the model was analyzed. The best model was selected among the models based on aerodynamic feature. In this project, the selected model is model 2 because it shows the minimum drag force changes over velocity, minimum average drag coefficient, minimum aerodynamic power and minimum relative wake pressure of the wake region to the environment. Model 2 is considered as the most streamline body among the others.

5.2 RECOMMENDATIONS

In this analysis, the result of the simulation can be improved by taking consideration of real condition to simulate the real situation such as surface roughness, temperature distribution etc. It is recommended to use the equation of Renault number to obtain the drag coefficient because the airflow properties varied at different velocity. Besides that, in order to validate the result, it is recommended to fabricate small scale model and test the CFD analysis in the wind tunnel to get the drag coefficient. The expected result of the experiment will deviate with the simulation. This percentage of deviation has to be considered as a simulation error in obtaining the reliable result.

REFERENCES

A. Morelli - “Low Drag Bodies Moving in Proximity of the Ground” - ASME Aerodynamics of Transportation Niagara Falls, June, 1979.

Alexander B. Hoerl, Derek K. Larson, Matthew S. Stenson, Kendall D. Dennis and Yildirim B. Suzen –“Aerodynamic Design Improvement of NDSU Solar Car through Computational Fluid Dynamics”-SAE International, Januari 2008

Antonello Cogotti –“A Parametric Study on the Ground Effect of a Simplified Car Model”-SAE International,February 1998.

A. Morelli, L. Fioravanti, A. Cogotti -”The Body Shape of Minimum Drag” SAE Congress, publ. n. 760186 Detroit, February 23-27, 1976.

J.Y.Wong-2008, “Theory of Aerodynamic”- (pp221-pp222), Carlinton University, Ottawa,Canada

Hucho, W. (Ed).(1998). Aerodynamic of road vehicles: From fluids mechanics to vehicle engineering. (pp. 13-26 and 49-51) Warrendale, PA: Society of Automotive Engineers, Inc.

Una McGeough, Doug Newman, Jay Wrobel “Model for Sustainable Urban Design With Expanded Sections on Distributed Energy Resources”- Oak Ridge National Laboratory-February 2004

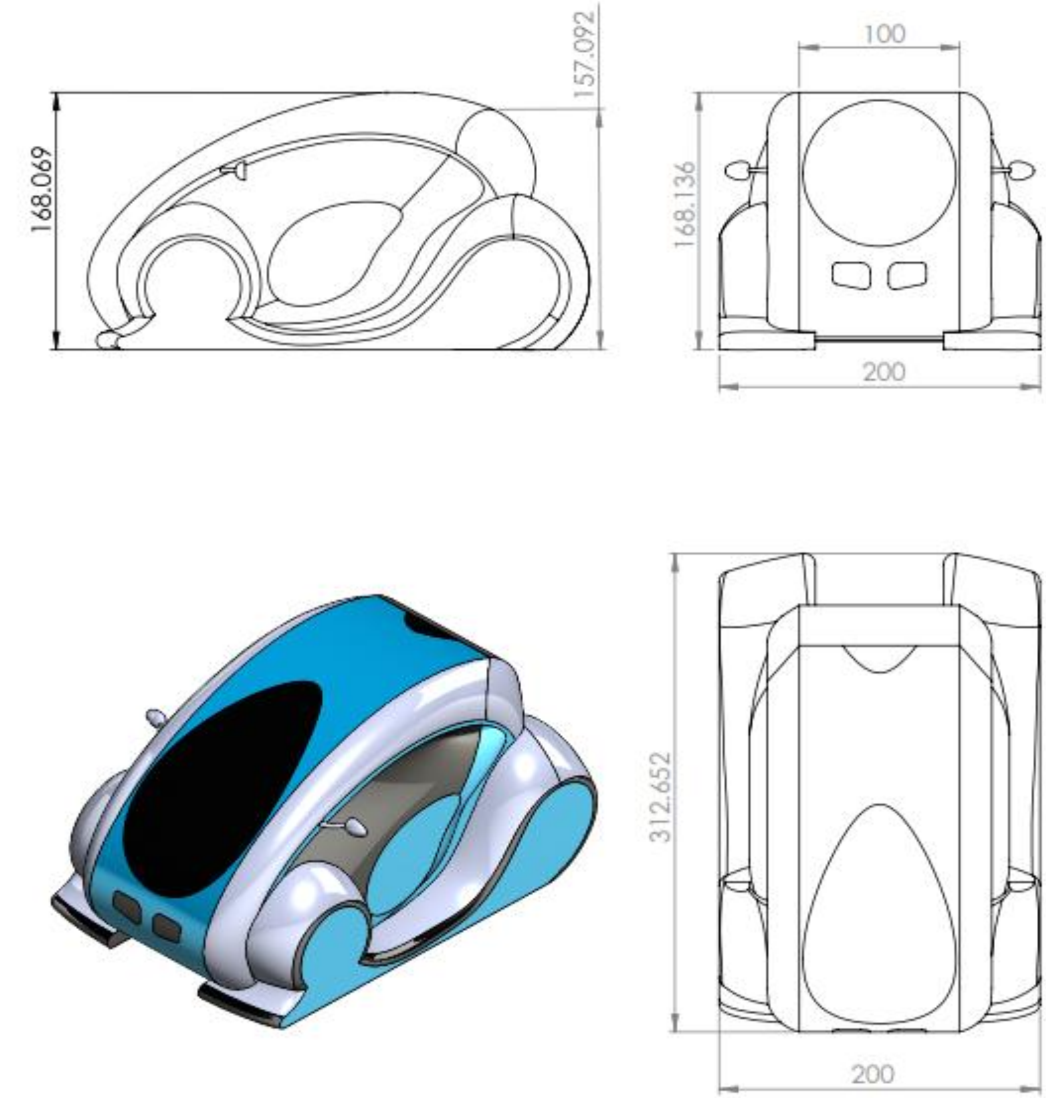
PROJECT ACTIVITIES	**W1**	**W2**	**W3**	**W4**	**W5**	**W6**	**W7**	**W8**	**W9**	**W10**	**W11**	**W12**	**W13**	**W14**
Confirm title with supervisor														
Verification of the title with FYP coordinator														
Collecting information from previous team														
Collecting information about Shell eco Marathon Asia														
Research of aerodynamic (Literature review)														
Develop model of previous car														
Perform CFD analysis of previous car														
Verification of analysis with SV														
Design another 3 new model in Solidworks														
Perform CFD analysis of the new model														
Prepare preliminary result														
Presentation														

Figure 3.1 Show the Gant chart of the project. This Gant chart proposed the activity that was carried out throughout the Final Year Project 1.

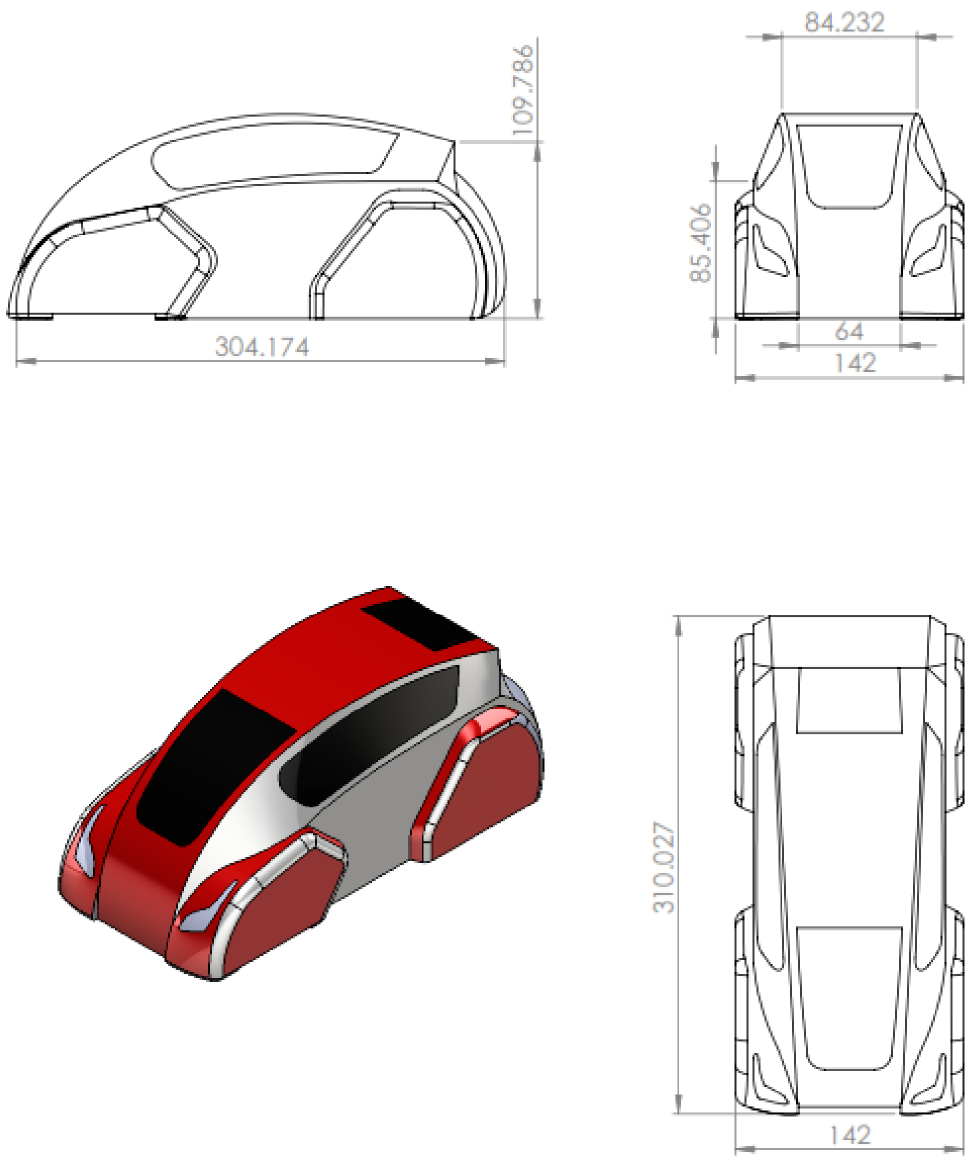
APPENDIX (FYP 1, GANT CHART FOR FYP 2)

Table 3.0 Show the planned Gant chart for the next FYP 2 programmer

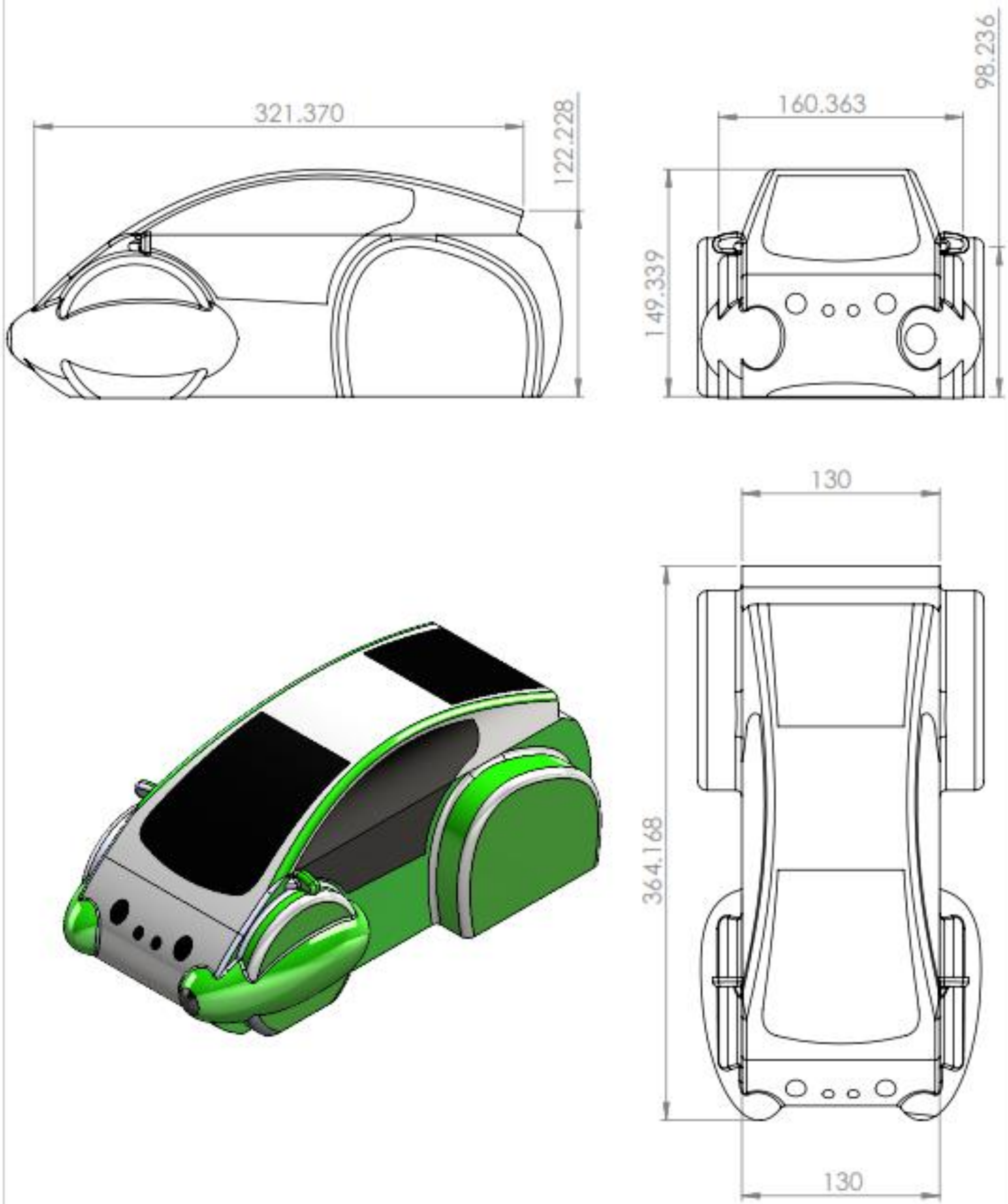
PROJECT ACTIVITIES	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14
Design and analysis														
Result validation from Supervisor														
Design Improvement														
Result and discussion support/correlation with other findings														
Validation of conclusion with Supervisor														
Preparing final report														
Presentation														



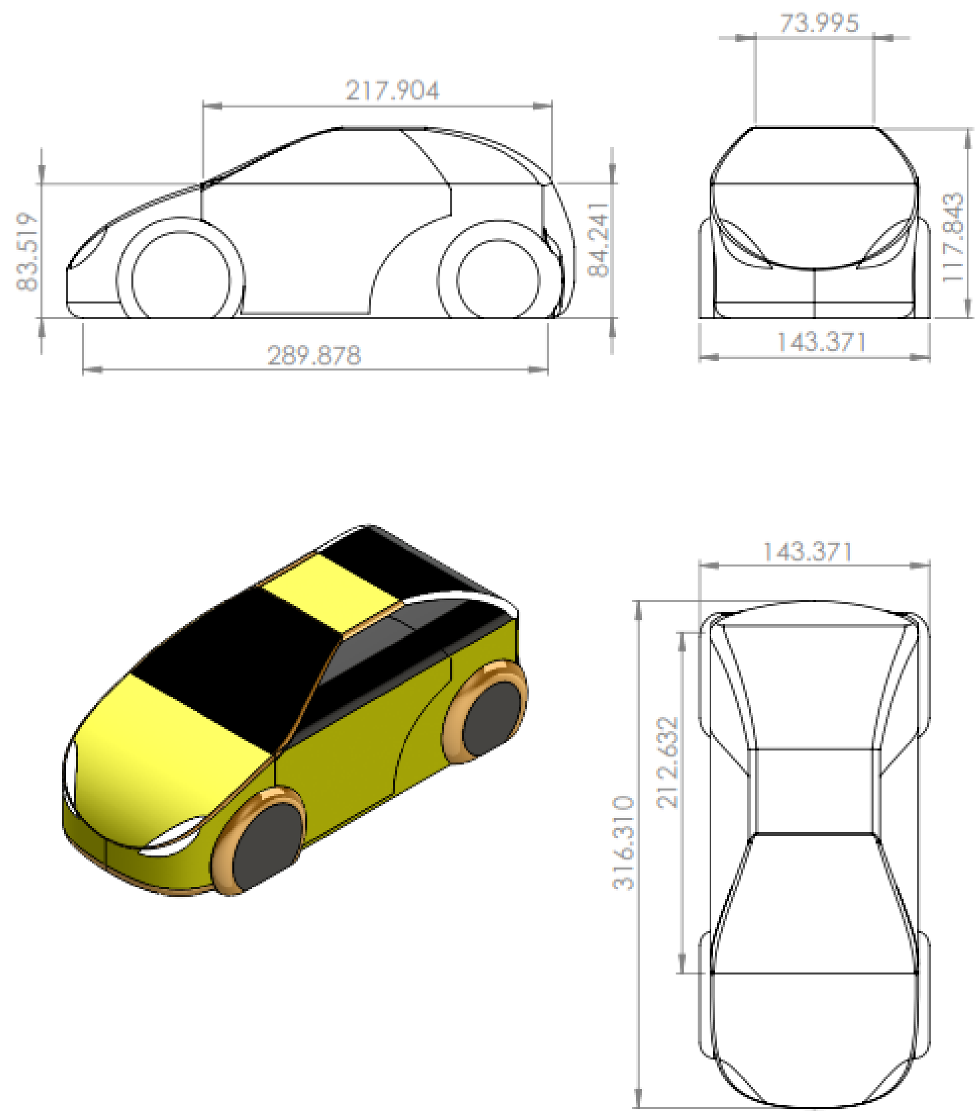
Title	Urban-car Model 1	Scale	1:35
Name	Muhammad Firdaus Othman	Dimension	Milimeter
Matric	MH08036	Item	APPENDIX C



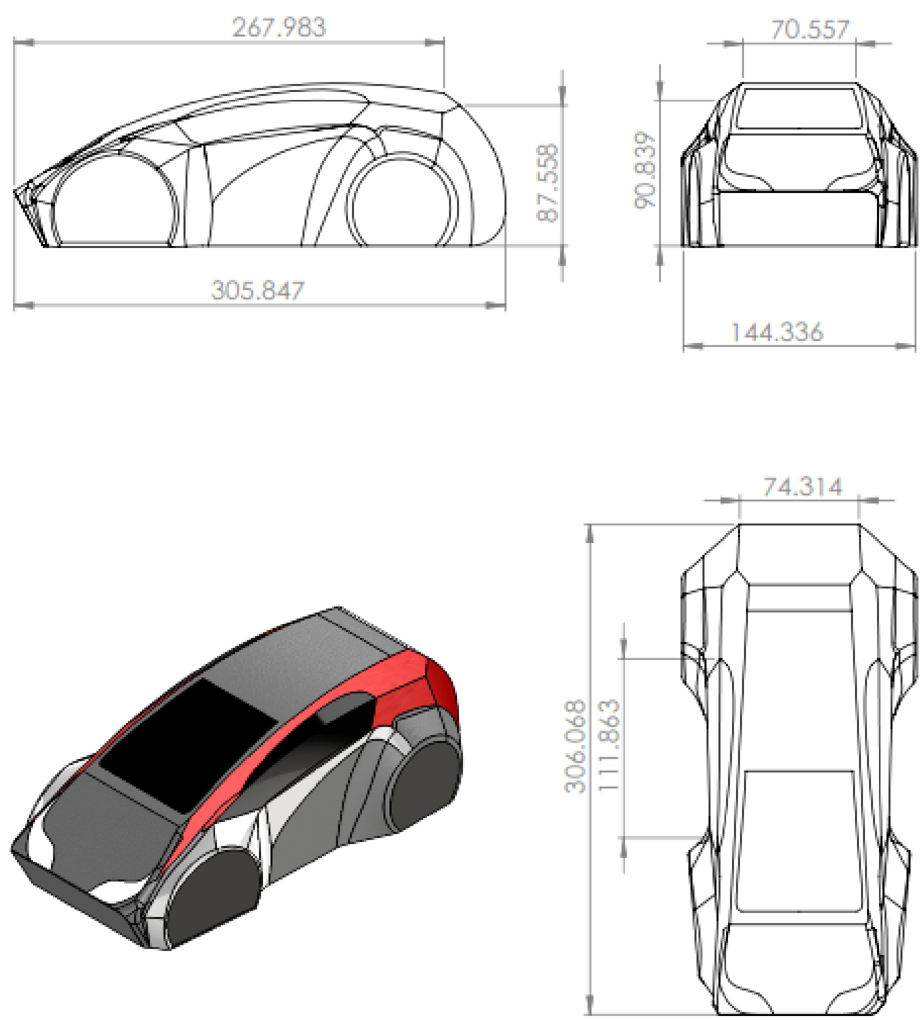
Title	Urban-car Model 2	Scale	1:35
Name	Muhammad Firdaus Othman	Dimension	Milimeter
Matric	MH08036	Item	APPENDIX D



Title	Urban-car Model 3	Scale	1:35
Name	Muhammad Firdaus Othman	Dimension	Milimeter
Matric	MH08036	Item	APPENDIX E



Title	Urban-car Model 4	Scale	1:35
Name	Muhammad Firdaus Othman	Dimension	Milimeter
Matric	MH08036	Item	APPENDIX F



Title	Urban-car Model 5	Scale	1:35
Name	Muhammad Firdaus Othman	Dimension	Milimeter
Matric	MH08036	Item	APPENDIX G

APPENDIX H

Calculation of drag coefficient;

i. Model 1, Velocity = 11. 11 m/s, Drag force = 56. 13 N, $A_f = 2.4758$ m²

$$C_d = \frac{2F_d}{\rho v^2 A_f}$$

$$C_d = \frac{(2)(56.13)}{(1.184)(11.11)^2(2.4758)}$$

$$C_d = 0.310$$

ii. Model 2, Velocity = 11. 11 m/s, drag force = 57. 16 N, $A_f = 2.9808$ m²

$$C_d = \frac{2F_d}{\rho v^2 A_f}$$

$$C_d = \frac{(2)(57.16)}{(1.184)(11.11)^2(2.9808)}$$

$$C_d = 0.263$$

iii. Model 3, Velocity = 11. 11 m/s, drag force = 72. 52 N, $A_f = 3.2886$ m²

$$C_d = \frac{2F_d}{\rho v^2 A_f}$$

$$C_d = \frac{(2)(72.52)}{(1.184)(11.11)^2(3.2886)}$$

$$C_d = 0.302$$

iv. Model 4, Velocity = 11.11 m/s, drag force = 69.34 N, $A_f = 3.4743$ m²

$$C_d = \frac{2F_d}{\rho v^2 A_f}$$

$$C_d = \frac{(2)(69.34)}{(1.184)(11.11)^2(3.4743)}$$

$$C_d = 0.273$$

v. Model 5, Velocity = 11.11 m/s, drag force = 52.32 N, $A_f = 2.4758$ m²

$$C_d = \frac{2F_d}{\rho v^2 A_f}$$

$$C_d = \frac{(2)(52.32)}{(1.184)(11.11)^2(2.4758)}$$

$$C_d = 0.289$$