DEVELOP DRAG ESTIMATION ON HYBRID ELECTRIC VEHICLE (HEV) MODEL USING COMPUTATIONAL FLUID DYNAMICS (CFD)

REDZUAN BIN AHMAD

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS ◆

JUDUL: **DEVELOP DRAG ESTIMATION ON HYBRID ELECTRIC VEHICLE (HEV) MODEL USING COMPUTATIONAL FLUID DYNAMICS (CFD)**

SESI PENGAJIAN: 2008/2009

Saya **REDZUAN BIN AHMAD (860407-49-5683)**

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:
Kg. Buang Sayang,
89608, Papar, Sabah.

Nama Penyelia:
MR DEVARAJAN A/L RAMASAMY

Tarikh: **10 NOVEMBER 2008**

Tarikh: **10 NOVEMBER 2008**

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).

DEVELOP DRAG ESTIMATION ON HYBRID ELECTRIC VEHICLE (HEV) MODEL USING COMPUTATIONAL FLUID DYNAMICS (CFD)

REDZUAN BIN AHMAD

A report submitted in partial fulfillment of the
requirement for the award of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
Universiti Malaysia Pahang

NOVEMBER 2008

SUPERVISOR DECLARATION

We hereby declare that we have read this project report and in my opinion this project report is sufficient in terms of scope and quality for the award of Bachelor in Mechanical Engineering with Automotive.

Signature : ……..
Name of Supervisor : DEVARAJAN A/L RAMASAMY
Position : Lecturer
Date : ……..

Signature : ……..
Name of Panel : ……..
Position : Lecturer
Date : ……..

DECLARATION

I declare this thesis that was entitled "Develop Drag Estimation on Hybrid Electric Vehicle (HEV) Model using Computational Fluid Dynamics (CFD)" is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any degree.

Signature : ……

Name : REDZUAN BIN AHMAD

Date : ……

DEDICATION

First, I would like to show my expression of gratitude to Allah S.W.T whose guidance, help and grace was instrumental in making this analysis to be a valuable project. Beside, I want show my thankfulness to my beloved mother, Rabaiah Binti Mansor for her full support and not forgotten to my supervisor, lecturers, teachers, friends, brothers and sisters.

ACKNOWLEDGEMENTS

I would like express my gratefulness to my supervisor, Mr Devarajan A/L Ramasamy for his guidance and help during the whole course of completing my final year project. All his advices and ideas have eventually contributed to the success of this project.

I would like also to acknowledge with much of appreciation to my friends that also supervised by Mr. Devarajan, where is Johari Bin Ismail , Mohd Syazrin Bin Sopnan, Mohd Azrul Bin Mohd Othman , and Shasendran that whose have helped and give co-operation throughout the project and also Instructor Engineer, Mr Mohd Fazli B. Ismail. Then not forgotten to my friend, Hafnizar Bin Muhamad Shaharuddin for allowing me using his car to perform my test experiment.

Last but not least, I would also like to wish my gratitude to all my course mates and other friends for their encouragement, helps, and motivation that inspired me to do well my bachelor degree's final year project. Thank also to all lecturers, technician, and panels presentation that direct and indirectly contributed in completing and improving my quality of project.

ABSTRACT

Develop of drag estimation by using Computational Fluid Dynamics (CFD) on Hybrid Electric Vehicle (HEV) model was carried out on this project. The HEV model here means the Proton Iswara Hatchback body developed by researcher of Automotive Focus Group, Universiti Malaysia Pahang. To develop this HEV model, one of factor needs to consider and studied for giving better efficiency on the road is aerodynamics drag. Therefore, simulation of CFD and FEM have been the key features to aerodynamics drag studies in this project for the HEV model specifically is Proton Iswara Hatcback body. The objectives of this project are to estimate the drag of Proton Iswara Hatcback body at ranging speed between 40km/h to 110 km/h that designed by Computational Aided Design (CAD). The terminology to getting the drag estimation is by using input of CFD then export to FEM analysis to find the value of aerodynamics drag in terms of drag forces and drag coefficient. Besides that, CFD simulation results such as contour and trajectories plot also used to analyze the characteristics of streamlines flow or boundary layer that occurs on the body of this HEV model especially for the forebody, upperbody and rearbody. To achieve these objectives and rationalization made of project, aerodynamics studies, and study of CAD, CFD, and FEA engineering's software needed to optimize the development of aerodynamic design on HEV model and to estimate the drag using CFD and FEA software as an alternative after experimental process. In this project also, a simple experiment was done to validate the CFD simulation analysis. The experiment known as Pressure Experiment that gives valuable results to compare with the simulation results as a validation process to this project.

ABSTRAK

Menentukan aggaran daya rintangan udara terhadap Kenderaan Hibrid Elektrik (HEV) model menggunakan 'Computational Fluid Dynamics' (CFD) menjadi keutamaan dalam projek ini. Kenderaan Hibrid Elektrik model tersebut ialah Proton Iswara Hatchback yang telah di bangunkan oleh sekumpulan penyelidik automotif dikenali sebagai 'Automotive Focus Group' di Universiti Malaysia Pahang. Salah satu faktor yang dipertimbangkan dalam penyelidikan untuk melihat kecekapan HEV model tersebut ialah dari segi daya rintangan aerodinamik. Oleh itu, simulasi menggunakan CFD dan 'Finite Element Model' (FEM) menjadi kunci utama untuk menyelidik dan menganalisa daya rintangan aerodinamik terhadap HEV model dalam projek ini dan secara spesifiknya Proton Iswara Hatchback. Objektif melaksanakan projek ini ialah mendapatkkan nilai daya rintangan udara HEV model pada julat kelajuan, 40km/h hingga 110km/h pada setiap 10km/h. Terminologi mendapatkan nilai daya rintangan udara tersebut ialah menggunakan input daripada CFD seterusnya dihantar kepada FEM untuk dianalisis dan mendapatkan nilai dalam bentuk daya dan kualiti daya rintangan udara. Selain daripada itu, ciri-ciri aerodinamik yang berlaku di bahagian depan, atas dan belakang kenderaan tersebut di analisa menggunakan keratan dan garis aliran plot yang terdapat di dalam perisian CFD. Bagi mencapai objektif projek ini, perisian kejuruteraan seperti CFD, CAD dan FEA perlu di optimumkan untuk mendapatkan anggaran nilai daya dan kualiti daya rintangan udara seterusnya menjadikan CFD dan FEM sebagai alternatif lain selepas proses eksperimen. Dalam projek ini juga, satu eksperimen mudah telah dijalankan untuk mengesahkan keputusan proses simulasi. Eksperimen tersebut dikenali sebagai 'Eksperimen Tekanan' yang memberikan keputusan sangat berguna selepas dibandingkan dengan keputusan simulasi sebagai proses pengesahan terhadap projek ini.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF GRAPHS

LIST OF FIGURES

LIST OF APPENDICES

LIST OF SYMBOLS

D	Drag Force
D_f	Friction Drag Force
L	Lift Force
C_D	Drag Coefficient
Cp	Pressure Coefficient
C_L	Lift Coefficient
ρ	Air Density
A	Frontal Area
b	Length Normal to The Flow
h	Height of the Body
v	Speed of the Body
v_∞	Local Velocity
p_∞	Local Pressure
S_i	Mass-Distributed External Force per Unit Mass
E	Total Energy per Unit Mass
Q_H	Heat Source per Unit Volume
τ_{ik}	Viscous Shear Stress Tensor
q_i	Diffusive Heat Flux

LIST OF ABBREVIATION

HEV	Hybrid Electric Vehicle
CFD	Computational Fluid Dynamic
FEM	Finite Element Model
CAD	Computational Aided Design
CAE	Computational Aided Engineering
RANS	Reynolds-Averaged Navier-Stokes Equation
DNS	Direct Numerical Simulation
3D	Three Dimensional

CHAPTER 1

INTRODUCTION

1.1 Project Background

The importance of aerodynamics to a Hybrid Electric Vehicle (HEV) model needs a development of drag estimation to know how much the car performance on the road against air resistance beside to improve the stability, reducing noise and fuel consumption. In view of the fact that many of car makers like Toyota, Honda and Audi formulate a research and continue develop the HEV model focused on higher propulsion efficiency orderly integrate the energy saving by reduce the rolling resistance of wheel and reduce the drag by aerodynamically losses. At University Malaysia Pahang (UMP), Automotive Focus Group also was developed HEV model to achieve the aim of HEV's control strategy in term of efficiency. Proton Iswara Hatchback body was used by the group to modify the conventional power train to the hybrid power train. As an increasing of drag, the more power of car to do work than reducing the power train efficiency. Therefore, the body of passenger car (Proton Iswara Hatchback) needs to study in term of aerodynamics losses. In aerodynamic field there have two major studies need to be concerned where is study the airflow on the body and estimation of drag. To understand the aerodynamics on the HEV model, flow visualization is the best technique as usual does by wind tunnel. But, in this project Computational Fluid Dynamics (CFD) analysis will be used as the technology of computer simulation to estimate the drag of HEV model after conventional technique due to economical factor.

1.2 Problem Statement

In Universiti Malaysia Pahang, one group of automotive researcher known as Automotive Focus Group were studied the design and development on Proton Iswara Hatchback body as a hybrid electric car. Therefore, the main concerns in aerodynamics fields were focused to study to know the efficiency and performance of that HEV model. The aerodynamics fields consideration in that researches' is drag reduction which is be the most important factor of HEV model design.

Drag will cause many problems on the performance of HEV model like instability, noise and fuel consumption. Thus, in this project the CAD model of Proton Iswara's body was developed to analyze aerodynamics especially on the drag estimation. In addition, using CFD and FEM analysis as a possible procedure were develop the drag estimation and aerodynamics studies on the body due to no wind tunnel in UMP.

1.3 Objectives

1. To estimate the drag of HEV model (Proton Iswara's body)
2. Develop the drag estimation from Computational Fluid Dynamics (CFD) and Finite Element Model (FEM) analysis

1.4 Project Scopes

1. Study of aerodynamics on road vehicle
2. Study of Computer Aided Design (CAD) engineering software
3. Analyze the project with CFD and Finite Element Model (FEM) for various car speeds
4. Optimize the software engineering as a tool to develop aerodynamics design on passenger car.

CHAPTER 2

LITERATURE REVIEW

2.1 Theory of Aerodynamics

At this section, the fundamental of fluids mechanics and basics of aerodynamics were discussed to gain understanding in doing analysis of the project. The basics equation and terms in aerodynamics field or fundamental of fluid mechanics such as Bernoulli's Equation, pressure, lift and drag coefficient, boundary layer, separation flow, and shape dependence were studied.

2.1.1 Bernoulli's Equation

Aerodynamics play main role to defined road vehicle's characteristic like handling, noise, performance and fuel economy [1]. The improvement on the characteristic related through the drag force which is ruled by Bernoulli Equation.
Basic assumptions of Bernoulli's Equation for an air flows are;

1. Viscous effects are assumed negligible
2. The flow is assumed to be steady
3. The flow is assumed to be incompressible
4. The equation is applicable along streamline

$$p + \frac{1}{2}\rho v^2 = constant \tag{2.1}$$

From equation (2.1) shows the increasing of velocity will case the decrease in static pressure and vise versa. On the movement of road vehicle will produce a distribution velocity that's create the skin friction due to viscous boundary layer which act as tangential forces (shear stress) then contribute drag. Beside that, force due to pressure also created which acts perpendicular to the surface then contribute both lift and drag forces. The Bernoulli's Equation from equation (2.1) gives the important result which is [2], [4], [5];

Static pressure + Dynamic Pressure = Stagnation Pressure.

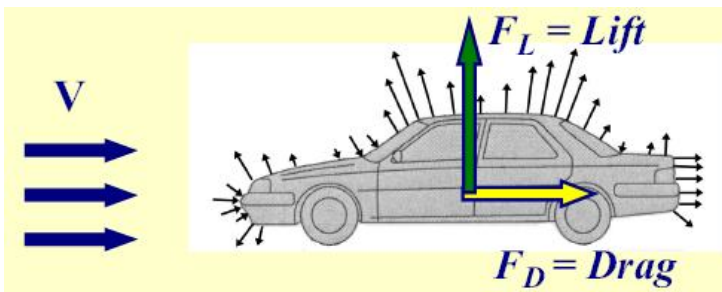


Figure 2.1 Drag and lift force due to pressure from velocity distribution [7]

2.1.2 Pressure, Lift and Drag Coefficient

Drag can generate by two main perspectives [1]:

1. From the vehicles (body)
2. From the moving fluid.

From the two perspectives, three major coefficients were produced from the two basic of aerodynamics forces. The first force is pressure distributions that normal (perpendicular) force to the body which is will produce pressure, drag and lift coefficient. The second force is shear force that tangential (parallel) to the surface of body's vehicle where is contribute drag coefficient only [2], [3].

2.1.2.1 Pressure Coefficient

The equation for coefficient of pressure (*Cp*) due to dynamic pressure can derive as [3],[4] ;

$$Cp = \frac{p - p_\infty}{\frac{1}{2}\rho v_\infty^{\ 2}} \tag{2.2}$$

The equation of dynamic pressure defined as [3],[4];

$$p_{tot} - p_\infty = \frac{\rho}{2} v_\infty^{\ 2} \tag{2.3}$$

In term of local velocity, the pressure coefficient (only valid for incompressible flow) can derive as [3],[4];

$$Cp = 1 - \frac{v^2}{v_\infty^{\ 2}} \tag{2.4}$$

The form of equation (2.4) is from the relation equation (2.2) and equation (2.5) as shown below [2],[3],[4];

$$p - p_\infty = \frac{1}{2}\rho(v_\infty^{\ 2} - v^2) \tag{2.5}$$

From the equation (2.4) where the local velocity on velocity is zero, the pressure coefficient is equal to 1.0 and when $v=v_\infty$, the pressure coefficient will be zero. While from equation (2.2) where $p=p_\infty$, *Cp* was become zero also. Pressure coefficient would become negative, since the local velocity is larger than the free stream velocity, v_∞ . Therefore, some typical value of pressure coefficient can summarize on table as shown in Table 2.1 below.

Table 2.1: Typical Values of Pressure Coefficient, *Cp* [3], [4].

Location	**Cp**	**Velocity, v**
Stagnation Point	1.0	0
On body's vehicle	0-1.0	$v < v_\infty$
On body's vehicle	Negative	$v > v_\infty$

2.1.2.2 Drag Coefficient

As was informed before the net drag is produced by both pressure and shear forces, thus the drag coefficient (C_D) for a vehicle body can define as [2], [3], [4];

$$C_D = \frac{D}{\frac{1}{2}\rho v_\infty{}^2 A} \tag{2.6}$$

Where D is the drag and A is the frontal area

Since, the C_D was defined as shown in equation (2.6). Thus, the drag force can derive as;

$$D = \frac{1}{2}\rho v_\infty{}^2 C_D.A \tag{2.7}$$

Besides that, the drag coefficient, C_{df} can derive from friction drag, Df, on a flat plate as [2];

$$C_{df} = \frac{D_f}{\frac{1}{2}\rho v^2 b.l} \tag{2.8}$$

Where D_f is friction drag, b and l are width and length of flat plate

2.1.2.3 Lift Coefficient

The lift force can be determined if the distribution of dynamic pressure and shear force on the entire body are known. Therefore the lift coefficient (C_L) can indicate as [3], [4];

$$C_L = \frac{L}{\frac{1}{2}\rho v_\infty{}^2 A} \tag{2.9}$$

Where L is lift force and A is the frontal area

Pressure and shear stress distribution is difficult to obtain along a surface for non geometry body either experimentally or theoretically but these to value can be obtained by Computational Fluid Dynamics (CFD) [1], [2].

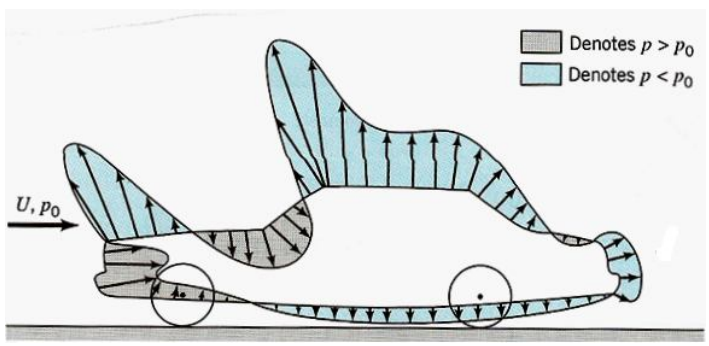


Figure 2.2 Pressure distributions on the surface of an automobile [2]

2.1.3 Boundary Layer

Boundary layer study in aerodynamics can be describe on a flat plate where is develop with two types flow which is laminar and turbulent flow. Due to fluid viscosity, a thin layer will exist when the velocity parallel to the static flat plate and then gradually

increase the outer velocity. The thickness of boundary layer also increases with the distance along the flat plate's surface [2], [3], [4].

Normally, the boundary layer is start from laminar flow and develops into turbulent flow. These two types of flow can determined with change of Reynolds number. Between the laminar and turbulent, form of transition region start occur when the change on laminar flow into turbulent flow [2], [3], [4]. The variation of boundary layer thickness can be seen in Figure 2.3;

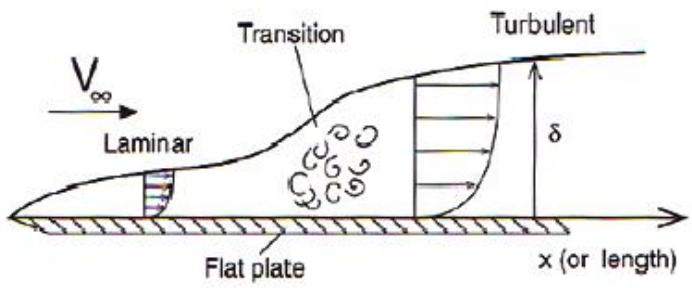


Figure 2.3 Variation of boundary layer thickness along flat plate [3]

2.1.4 Separation Flow

Separation flow can define as the fluid flow against the increasing pressure as far as it can; at point the boundary layer separates from the surface where the fluid within the boundary layer does not have such an energy supply [2].

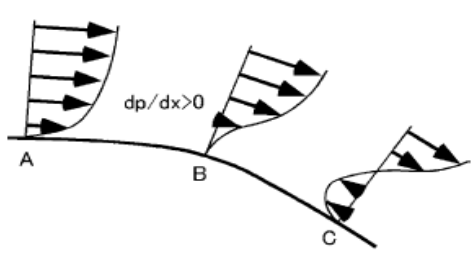


Figure 2.4 Schematic of velocity profile around a rear end [6]

In automobile shape, the rear end of vehicle becomes increasingly lower as the flow moves downstream then the extended airflow was formed there. Thus, it causes the downstream pressure increase while creates reverse force acting alongside the main flow and generates the reverse flow at downstream at point C as showed in Figure 2.4.

At Point A, no reverse occur because the momentum of the boundary layer is widespread over the pressure gradient. Between the Point A and Point C, the momentum of boundary layer and pressure gradient are balanced as stated at separation on Point B. The reverse force acting on separation point C is due to the viscosity of air (losses of momentum as it moves downstream) [6].

2.1.5 Shape Dependence

As discussed before the drag was depend on the shape of vehicle [1]. The Drag coefficient from equation (2.6) and equation (2.8) shown clearly the frontal area is give effect on the drag and lift coefficient means that increasing of frontal area or more blunt of body shaped will increase the both coefficient orderly increase the drag.

For the case, $0<Cd<1$ and $v< v_\infty$, the drag, $D=\frac{1}{2}\rho v_\infty^2 Cd.A$ that was defined from equation (2.7) [2]. Thus from the definition we can conclude that area of frontal area projected of composite body as [2], [8];

$$A=b.h \tag{2.10}$$

Where b is the length normal to the flow, and h is the height of the body

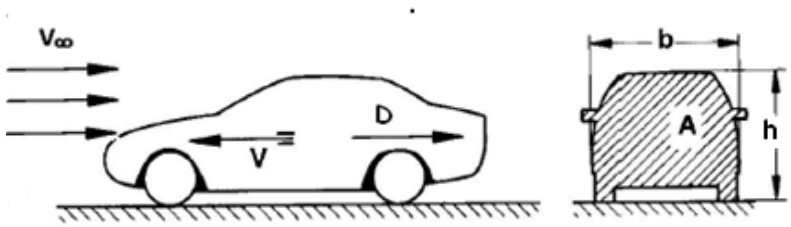


Figure 2.5 The relationship frontal area on vehicle body against the normal flow of velocity [8]

Shows above at Figure 2.5, the projected area of vehicle and the direction of local velocity, vehicle velocity and drag force. Beside body dependence, drag also depends on fluid viscosity, Reynolds number, compressibility effects, surface roughness and Froude number effects.

2.2 Road Vehicles Aerodynamics

Vehicles or cars defined as a bluff body where the boundary layer separates from their surface wide and generally unsteady wakes. Since all cars a bluff body but not all have same bluffness, the aerodynamics or drag of cars on the road are depend on the square of velocity, and shape or the frontal area of the cars. As a decreasing of frontal area, the drag coefficient also decrease to imply the decrease the aerodynamics drag. [7] The influences of various drag coefficients on velocity shown in Figure 2.6.

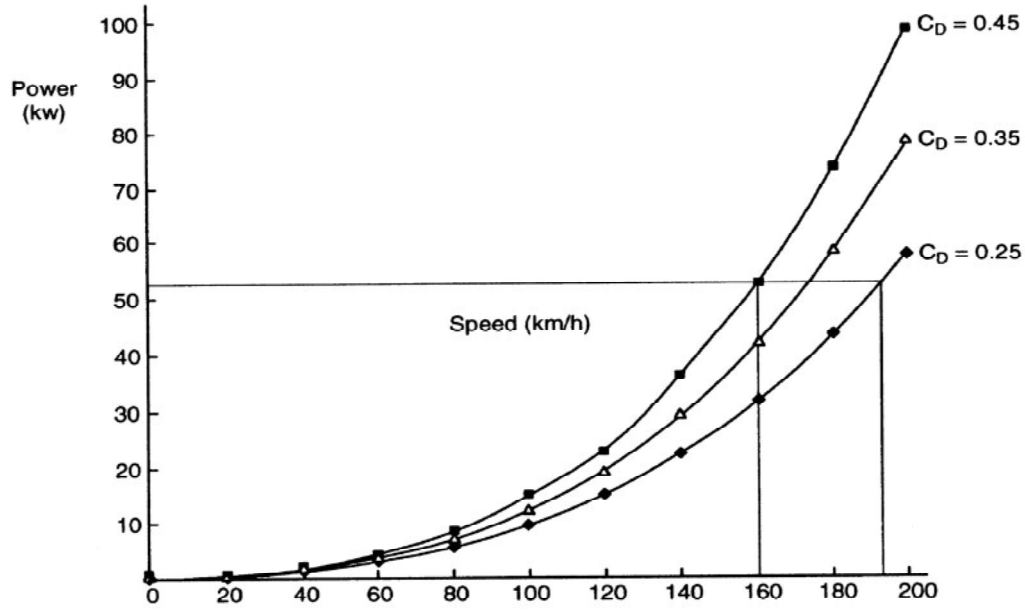


Figure 2.6 The influence of drag coefficients on velocity and Spent Power on road

2.2.1 History of Road Vehicles

Saving energy and fuel consumption reduction protect the global environment be the primary concern of automotive development, the evolution in improving fuel was started in 1970's when the world experience of oil crises [1].

Therefore, today one development on the road vehicle was developed in reduction of drag to improve the fuel consumption, better handling and as aesthetic values that contribute satisfaction to the customers [1], [8]. The requirement that influences concept of cars today can show at Figure 2.7.

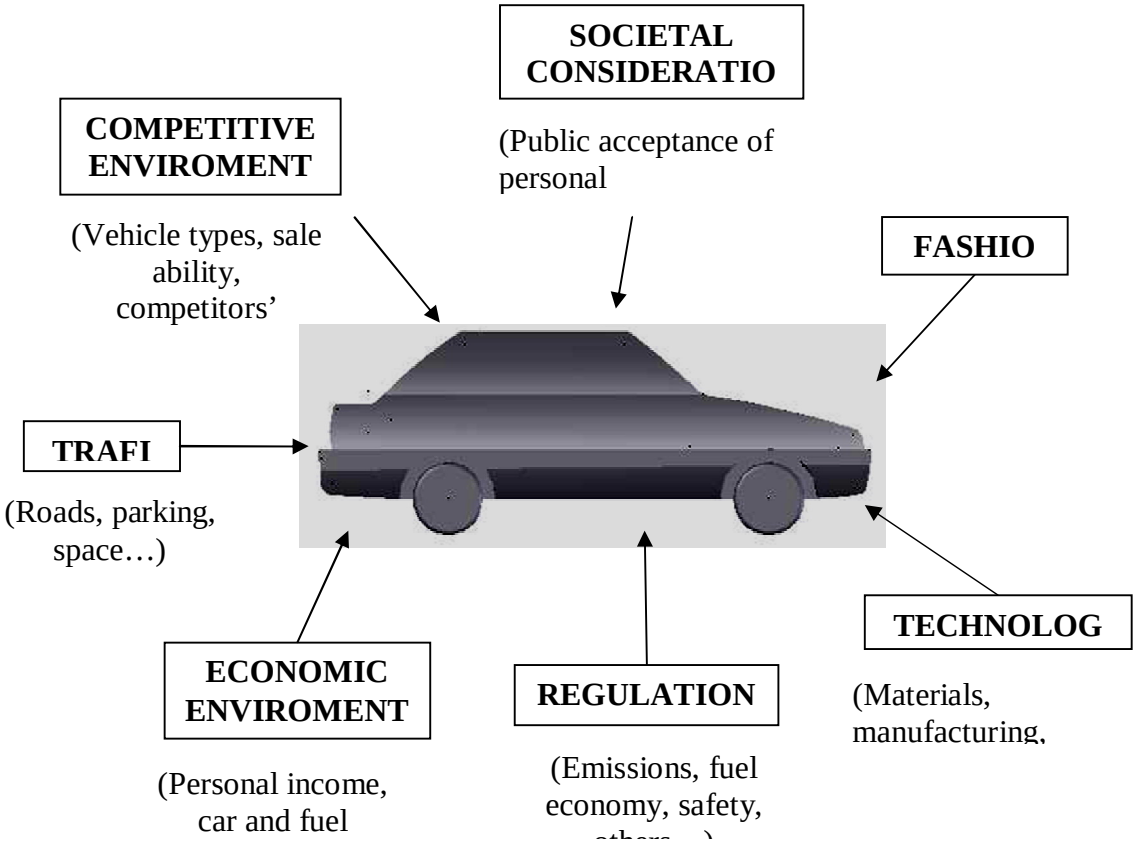


Figure 2.7 The concept of car is influenced by many requirements of very difference nature [1]

If looked at the history, the vehicle body's shape is made according to the aeronautical practice and from the naval architecture adapted shape that can seen at Figure 2.8 below [1], [8]. Since at, 1922, pattern of flow around half body of revolution in automobile shape was recognized by Klemperer where is when that the half body is brought close to the ground it was significantly change on the drag coefficient [1], [8].

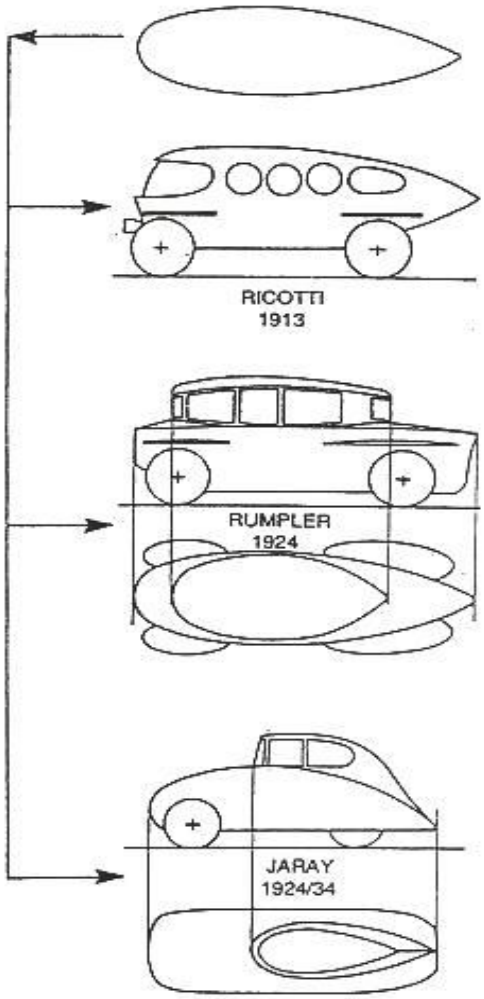


Figure 2.8 The early attempts to apply aerodynamic to road vehicle consisted of the direct transfer of shapes originating from aeronautical and marine practice [1].

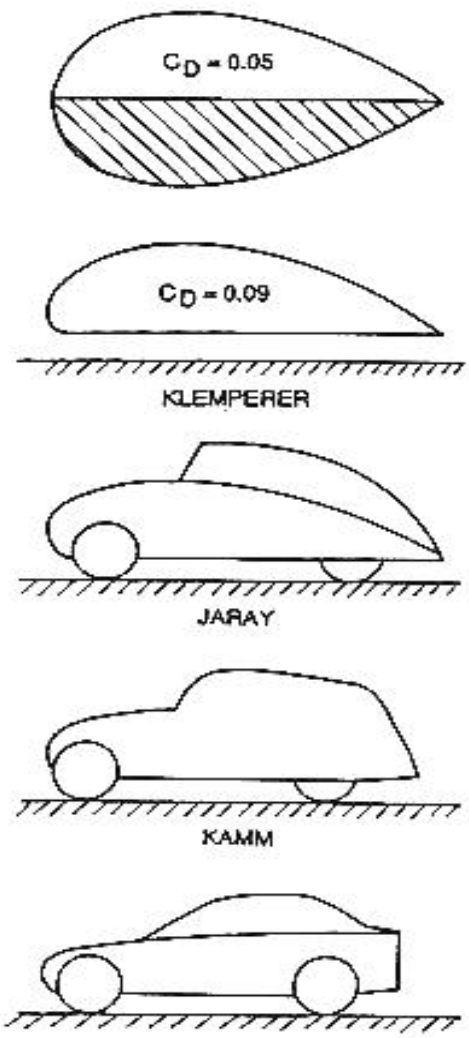


Figure 2.9 Klemperer recognized the flow over body revolution [1].

The drag value introduced by Klemperer in 1922 as shown in Figure 2.9 above demonstrate that drag coefficient as low as C_D=0.15 for a body with wheels [1], [7]. A brief overview of history on vehicle aerodynamic was summarized in Figure 2.10 below.

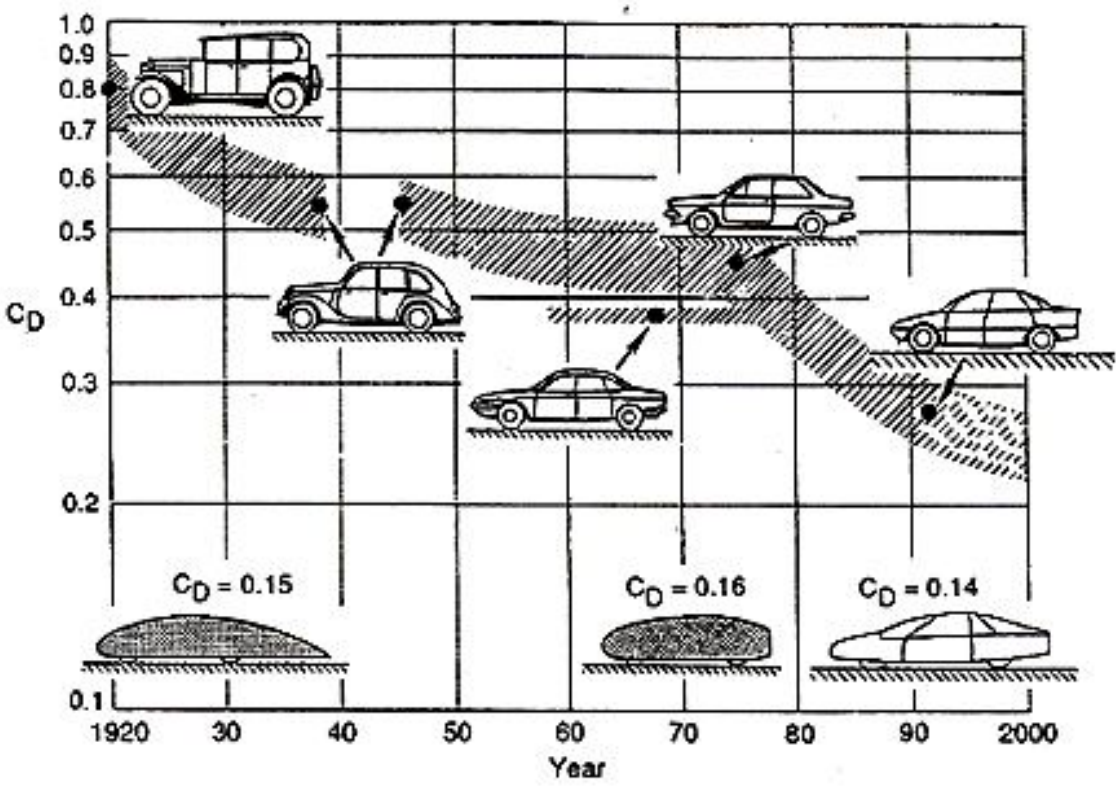


Figure 2.10 The drag history of cars using a logarithmic scale for drag emphasizes how difficult it is achieve very low drag values [1]

2.2.2 Relative of Aerodynamics on Passenger Car

The sub vision of drag is according to the regions on the body vehicle. But this consideration of type's classification is more difficult than an actual car [8]. This is because;

1. Pressure and shear stresses more are not known with the resolution needed.
2. The interference effects between the components.

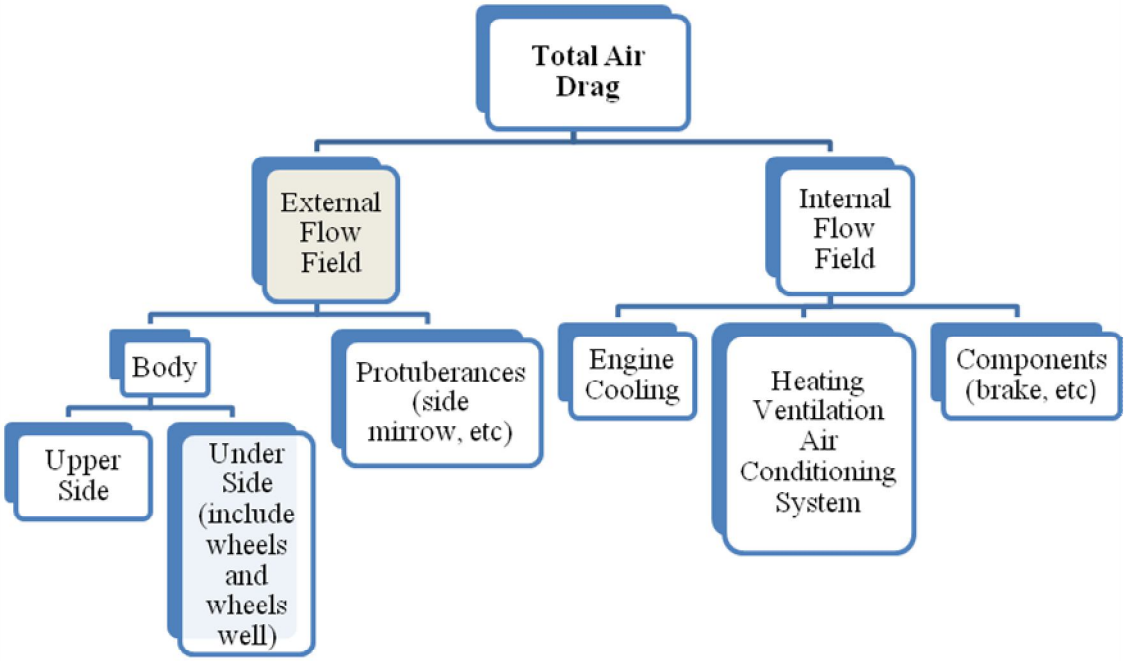
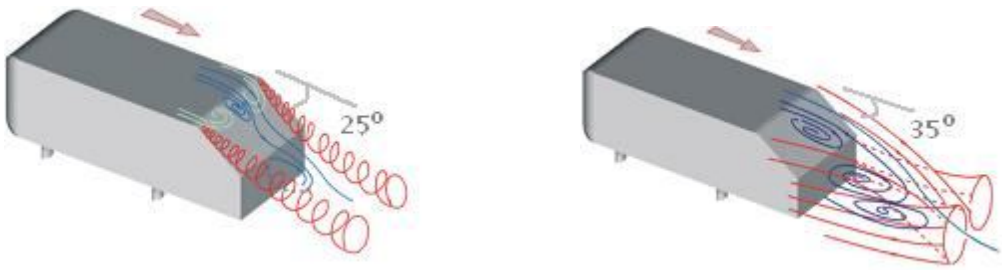


Figure 2.11 Breakdown of drag according to the locations of generation [8]

Figure 2.11 shows that the breakdown that divided into two general division where is external and internal flow field. Base on the Ahmed body, for a generic car it can be divide into local drag contributions. Four geometric zones was distinguished for smooth body without attachment, there are [8];

1. Front end
2. The rear slant
3. The base (i.e. vertical plane at the rear)
4. The side panels, roof and under body

Figure 2.12 Ahmed body view (a) 25° rear slant; (b) 35° rear slant [9]



(a) (b)

Figure 2.13 Development of the flow for the (a) 25° and (b) 35 ° slant angle. [9]

For basic understanding in estimation of drag study on vehicle body, the external flow at the upper side of body is the best way as consideration for study in aerodynamic field. The main concern for upper body is from the front end bonnet (hood) until the rear end (boot) or base area. Streamlines as imaginary line of flow can visualized as a pattern of air movement on the body vehicle. The Bernoulli equation theory or Venturi effect can be the basic of explanation the streamline of air velocity and the pressure over the car's body [10].

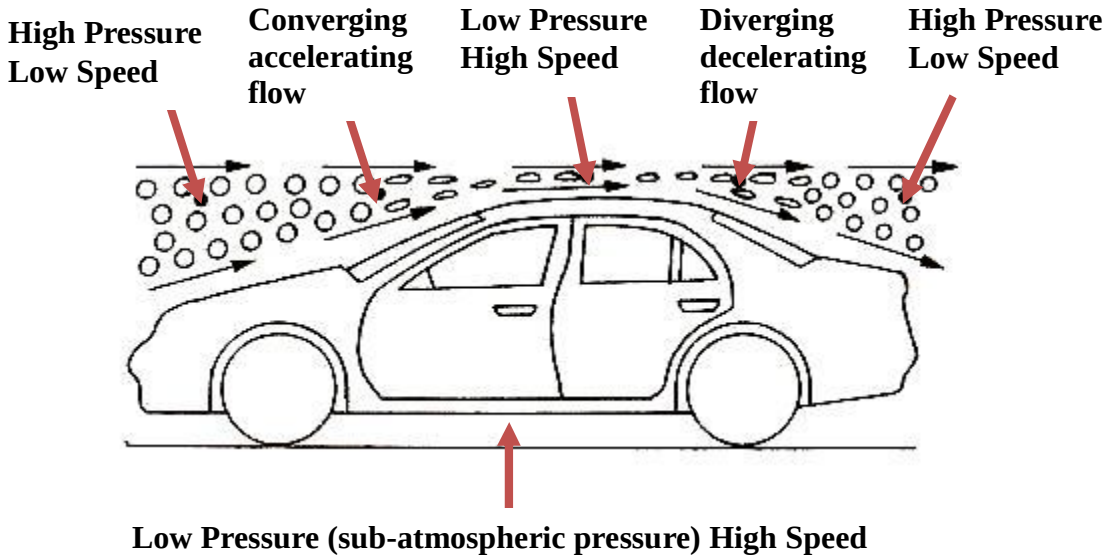


Figure 2.14 The relative velocity of air and pressure condition over the upper profile of a moving car [10]

From the Figure 2.14 above, as air flow horizontally over the vehicle body from front end bonnet (hood) and windscreen profile and the back end screen and boot (trunk) profile will produce the diverging and converging wedge flow respectively. When air moving into the converging wedge it will accelerate and reduce the air pressure. Over the roof the Venturi was narrowest so the movement of air will increase and in the mean time reduction in air pressure. As the air move to the rear through the diverging wedge region it decelerates to manage the enlarged flow space [10].

2.2.3 Detailed Surface Flow on Car's Body

The detailed surface flow on car's body have a many reason to investigated, where is to know the proper location of opening inlets and outlets, to determine the forces on particular body parts and sources of wind noise and to find the means controlling water flows are placed in region of high pressure [1].

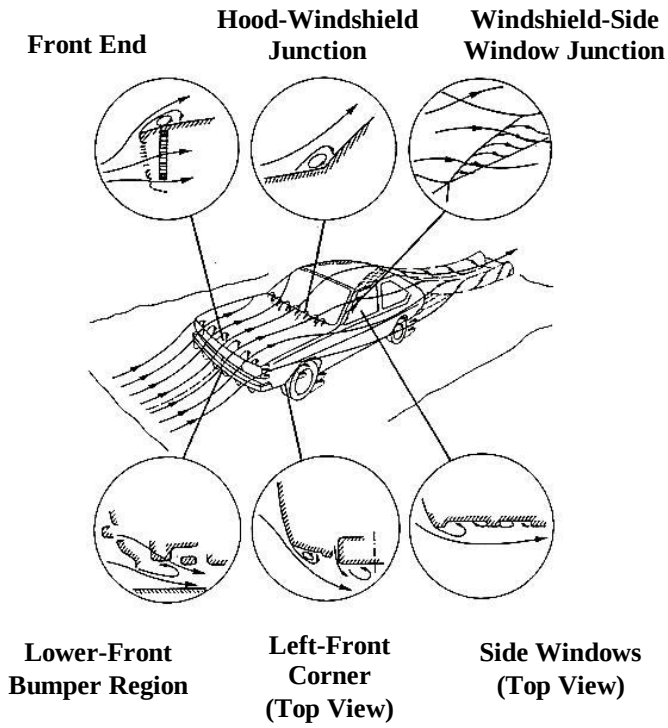


Figure 2.15 Flow around a car, and major of locations of flow separation [1]

The Figure 2.15 show the separation flow around the upper body vehicle, the major location of separation flow are at the front end, hood-windshield junction, windshield-side window junction, lower front-bumper region, left-front corner, and side windows. Beside that, the separation also occurs at a shortened rear surface, leading to a wake which includes a zone of recirculation usually called dead water [8].

2.2.4 Pressure Distribution for Car's Body

As a mentioned before the flow around surface body of car need to investigate to know the best location of inlets and outlets for example the cooling and ventilation system, to know the side mirror location and others [8]. The fundamental equation of inviscid flow was simplified on the vehicle body as shown in Figure 2.16 below as a two-dimensional flow, this simplification actually from three-dimensional flow around a car's body shaped [3], [8].

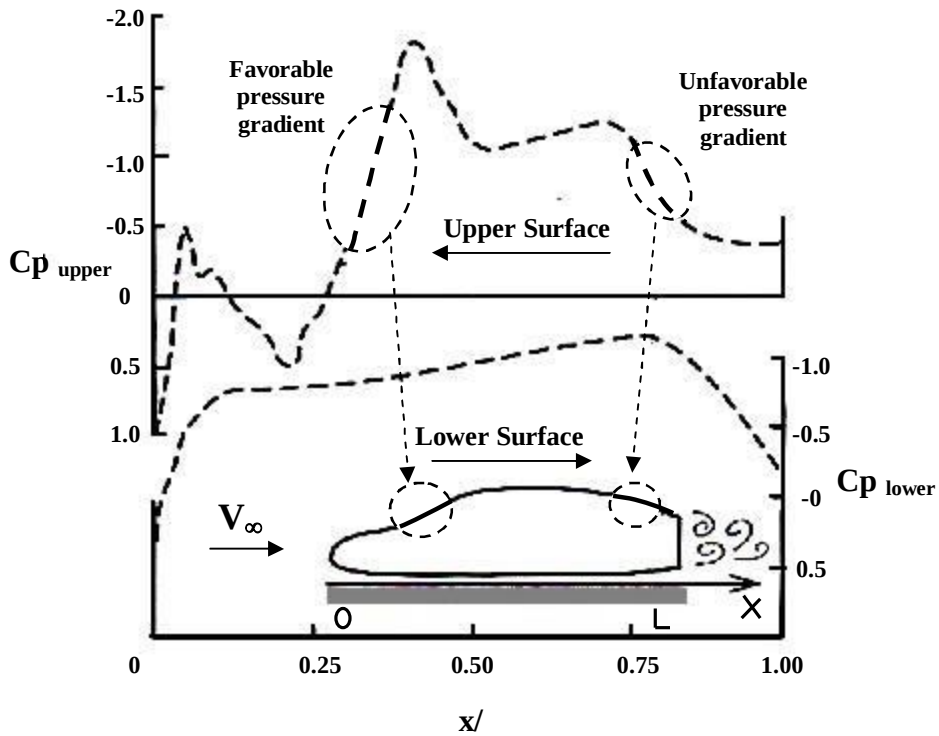


Figure 2.16 Pressure Coefficient Distribution over an automobile shape [3].

The favourable gradient (favourable pressure distribution area) in the Figure 2.16 above shown the flow stays close longer and the boundary layer undisturbed free stream will stay laminar for longer distance along the body surface that will form less of drag and less of friction. While, the unfavourable pressure gradients set off flow separations and transition to turbulent boundary layer [3]. From the types of pressure gradients, computational tools can predict the drag, boundary layer characteristics like transition to turbulent and flow separation if the local speed and the slope of pressure distribution known [3].

2.3 Introduction of Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) as a branch of Fluid Mechanics that uses numerical methods and algorithm to solve and analyze the airflows for a various field in engineering such as submarines, aeronautical, aerospace, automobile and wind turbines. CFD is computers tools that have fundamental basis of equation which is Navier-Stokes Equations to solve many types airflow. Nowadays, CFD widely uses in automobile industry nowadays to reduce expensive experimental test and saving time that required for aerodynamics studies.

2.3.1 CFD as a Tool for Aerodynamics Simulation

The force due to the pressure from velocity distribution is difficult and close impossible to obtain experimentally for complexity shape [1]. Therefore the direct evaluation for detailed surface of road vehicle not practical and on the other hand CFD is the grown tool in CAE tool that be a popular for analysis of many airflow situation, including road [3]. CFD is the validated computational codes create by programmer as a branch of Fluid Mechanic that have complex mathematical basis [3]. There is having four equations as a solver followed in application to road vehicle that the documented recently by Ahmed 1992, Kobayashi and Kitoh 1992 [1]. There are based on:

1. Laplace’s equation,
2. Reynolds-averaged Navier-Stokes equation (RANS),
3. Instantaneous Navier-Stokes equation, called direct numerical simulation (DNS)
4. Zonal models (hybrid)

Aerodynamic characteristic on road vehicle can visualize and investigate with CFD analysis. The graphic or streamline of velocity and pressure distribution can be measure. Beside that, CFD also offer the image processing of visualized flow fields, computed tomography and many more [3].

The results obtained with various types of flow visualization are useful to understand the flow field that obtained from wind tunnel. On this project, details on the surface body as a consideration of study and also the aerodynamic characteristics which is base on the CFD analysis to estimate the drag force.

2.3.2 Equation Solved by CFD

The aim of CFD is to resolve the equations that drive theoretically every kind of flow [12]:

1. The continuity equation
2. The momentum equations
3. The energy equation

All the three equation above were generally used to solve the fluid motion and also known as Navier Stokes equation of mass, momentum and energy. The conservation form can derive as [13]:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k}(\rho u_k) = 0 \tag{2.11}$$

$$\frac{\partial \rho u_i}{\partial y} + \frac{\partial}{\partial x_k}(\rho u_i u_k - \tau_{ik}) + \frac{\partial P}{\partial x_i} = S_i \tag{2.12}$$

$$\frac{\partial(\rho E)}{\partial y} + \frac{\partial}{\partial x_k}((\rho E + P)u_k + q_k - \tau_{ik} u_i) = S_k u_k + Q_H \tag{2.13}$$

where u is the fluid velocity, ρ is the fluid density, S_i is a mass-distributed external force per unit mass, E is the total energy per unit mass, Q_H is a heat source per unit volume, τ_{ik} is the viscous shear stress tensor and q_i is the diffusive heat flux.

2.3.3 Basic Steps of CFD Computation

Generally the steps of computation in CFD analysis are essentially the same irrespectively of the method. Since the method was decided, the steps basic was following in termed are [8], [11]:

1. Preprocessing- discretization of body surface or the computational domain
2. Solving- aerodynamic computation
3. Postprocessing- a suitable graphic representation of the numerical results

The preprocessing means discretization is done by using Preprocessors Codes where is dealt with the grid generation. While the solving or solution refers to the solver that programmed in computer from the numerical equation for an example the Navier-Stokes equation. The third step is postprocessing, means the importance for user defines the results by monitoring or display, manipulation and analysis of the vast amount of data contained in CFD solution [8] for an example results plot (contours, vector, isolines) and process results (XY plots, goals, report, etc) .

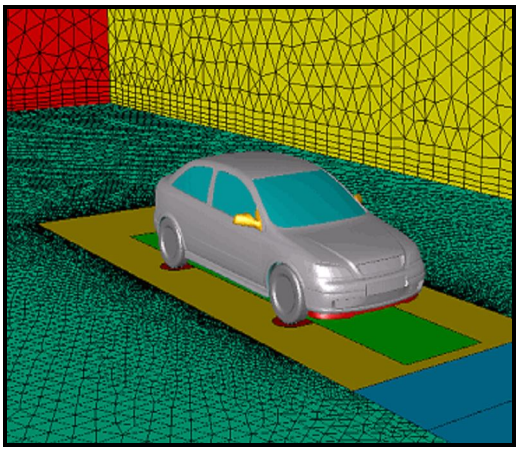


Figure 2.17 The 3D-hybrid grid [11]

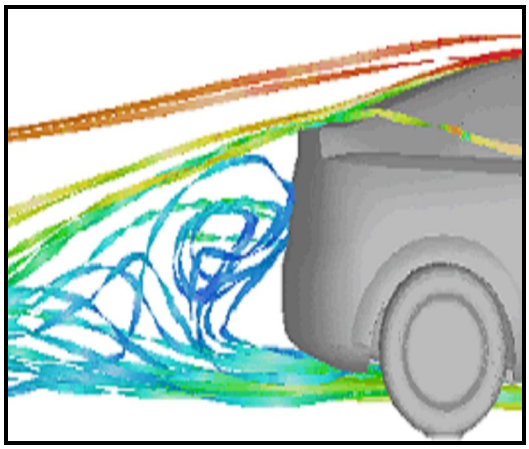


Figure 2.18 Flow analysis after a design study (pathlines) [11]

2.3.4 Surface Mesh Generation in CFD

To perform successful CFD-calculations, construction of a proper surface grid is very important. The mesh of CFD was produce triangular geometry that modeled on the surface of the body's model. The mesh was later used as input to the volume grid generator and was to comply with set of constraints and quality measures dictated by the grid generator and the CFD software. These measures can divide in two categories which is geometric quality and mesh quality. The usually meshing step used in CFD analysis is generating the initial mesh directly from basic mesh of the model. To enhance the quality of calculation and analysis, the smoothing, decimation, refinement and mesh optimization process continued as iterative process after initial mesh [12].

2.3.4.1 Refinement of thin areas

Refinement of model is the further mesh construction, in COSMOSFloworks software refinement can divide into four main types of refinements. Each refinement has its criterion and level. The refinement criterion denotes which cell cans have to be split, and the refinement level denotes the smallest size to which the cell can be split. The smaller cell size is very important for resulting the computational mesh of the basic mesh. The main types of refinement are [13]:

1. Small Solid Feature Refinement
2. Curvature Refinement
3. Narrow Channel Refinement
4. Square Difference Refinement

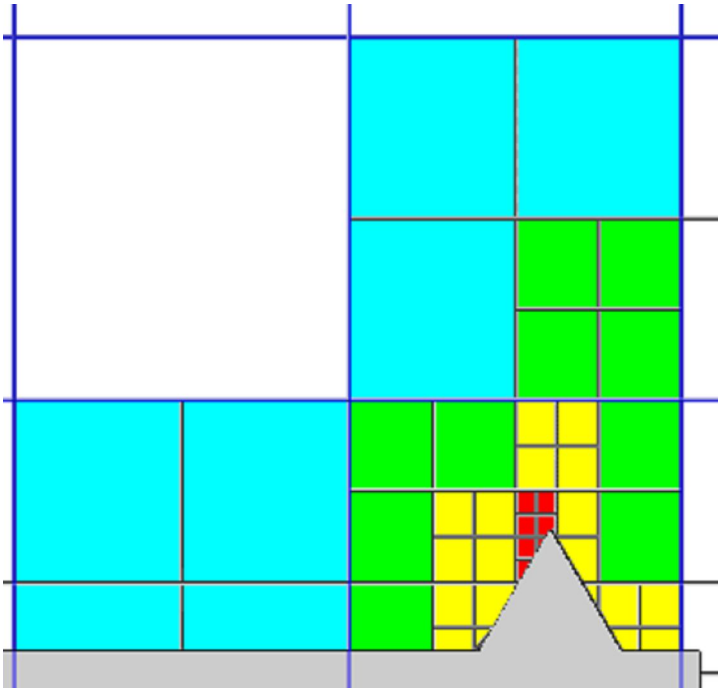


Figure 2.19 Fluid cell refinements due to the Cell Mating rule [13]

The mesh at this stage is called the primary mesh. The primary mesh implies the complete basic mesh with the resolution of the solid/fluid (and solid/insulator) interface by the small solid features refinements and the curvature refinement also taking into account the local mesh settings [13].

The red cell indicates the fourth level which is after resolving the cog then it split up to third level as a yellow cell. Then, it actuates the subsequent refinement producing the second level green cells and the first level blue cells. The white zero level cell (basic mesh cell) remains unsplit since it borders with first level cells only, thus satisfying the rule. The rule is strict and has higher priority than the other cell operations. This is especially important for the definition of the local initial mesh settings [13].

CHAPTER 3

METHODOLOGY

3.1 Introduction

For the methodology in estimation of drag for Proton Iswara body, the information from literature reviews are important to referring, plan, and analyze during the simulation and calculation for this project. The information such as aerodynamics theory, coefficient drag equation, simulation CFD and FEM of the model very important to defines the problems, to collect the data, to calculate drag coefficient and frontal projected area and interpretation of data.

3.2 Methodology of Flow Chart

To achieve the objectives of project, a methodology were constructs base on the scopes of projects as a guiding principle to formulate this project successfully. The important of this project is to simulating and calculating. Therefore to achieve the objectives of this project where is estimation of drag using CFD and FEM analysis, a terminology of works and planning show in the flow chart at Figure 3.1.

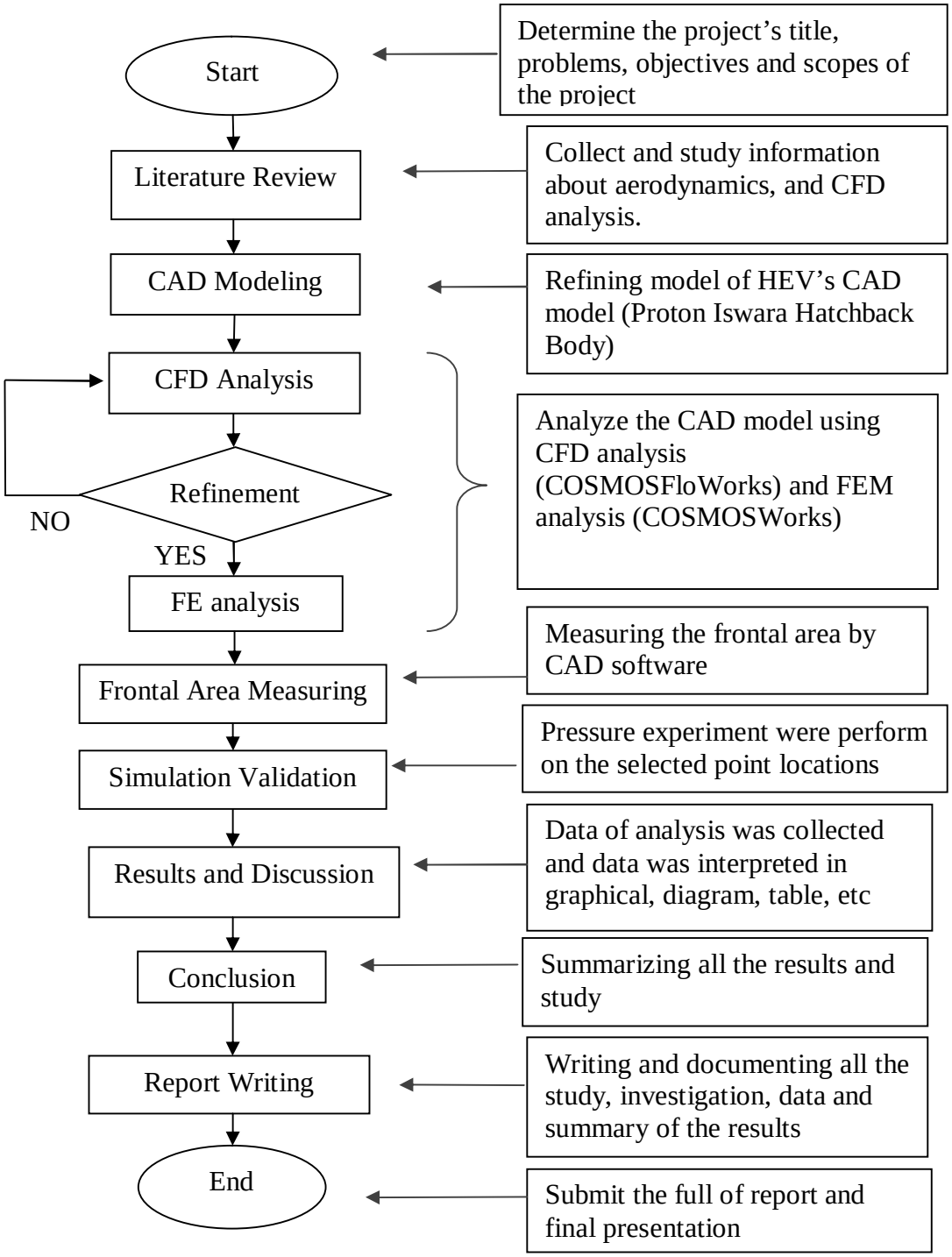


Figure 3.1 Flowchart of the Overall Methodology

3.3 Data Collecting

The data of dimension for Proton Iswara body was collected from the internet and measured then modeling the body by SolidWorks software. Data collecting of dimension as accurate as possible is very important for Proton Iswara's body to simulating the model in CFD and FEM analysis to get the appropriate or expected value of drag coefficient. The dimension of Proton Iswara Hatchback show in Figure 3.2 below:

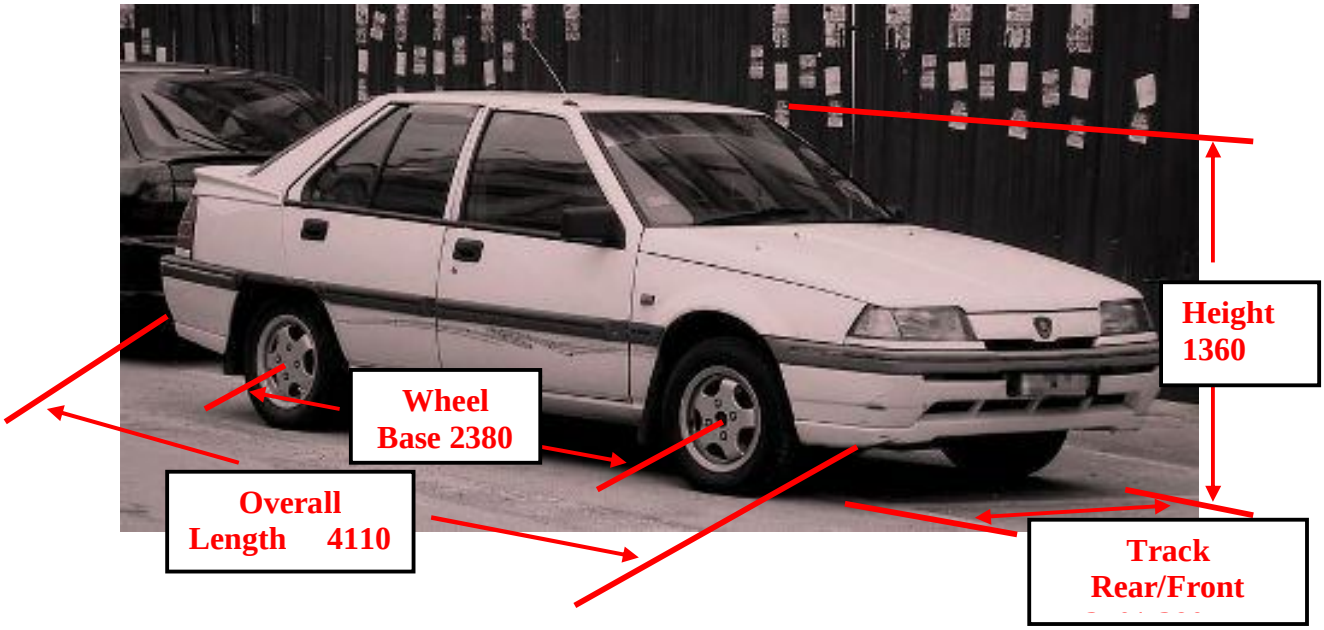


Figure 3.2 The dimension of Proton Iswara Hatchback

3.3.1 CAD Modeling

After collecting the dimension of the car body, the car bodies from the previous CAD model were modeling in SolidWorks software followed by refining the model as an improvement for the previous model design.

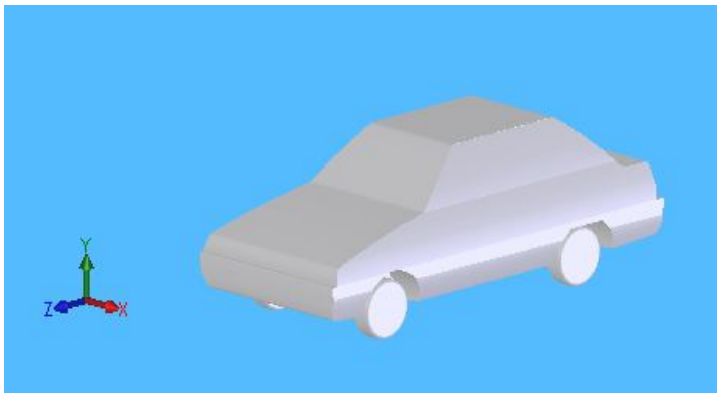


Figure 3.3 CAD model of Proton Iswara Hatchback's body in dimetric view

3.4 CFD Analysis

Since the car body was modeled in CAD and refined it, the next stage is importing the CAD model into CFD software. CFD analysis in this project will be run inside the COSMOSFloWorks software. At this stage, the various car speeds will analyze at ranging 40 km/h to 110 km/h for every 10 km/h. The boundary condition for this analysis is external flow with adiabatic wall. Beside that, the types of flow considered are laminar and turbulent flow for CFD model analysis. Figure 3.4 below show the boundary study of CFD analysis.

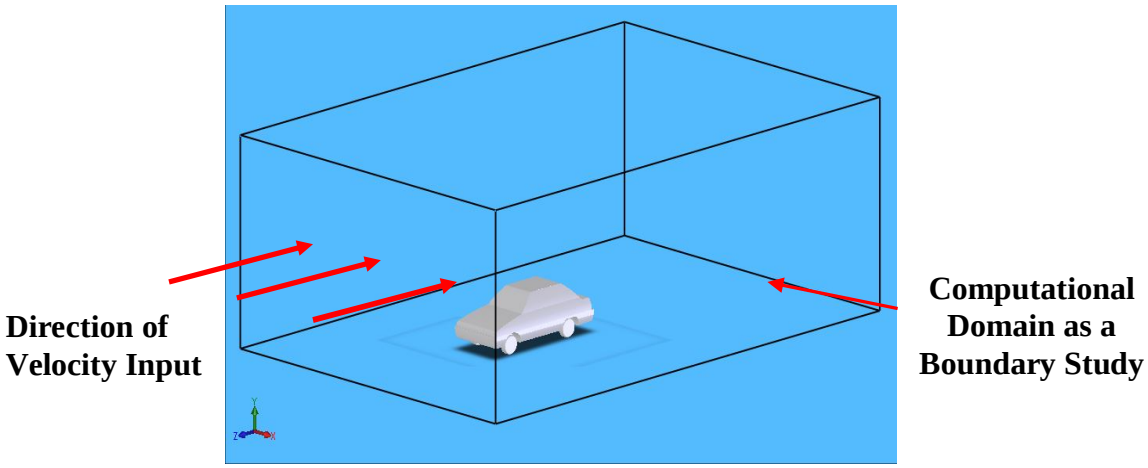


Figure 3.4 Boundary condition of CFD analysis

3.4.1 Refinement

During analyzing the car model, the basic mesh were refined by strategy refinement that able in the COSMOSFloWorks. The tabular refinement form was used to analysis the model where is the initial or basic mesh will be re-mesh during calculation. Before applying the strategy refinement, make sure the level of refinement is possible for the boundary condition. The mathematical solution of the solver assuming be done successfully and vise versa if the calculation were stop.

3.5 FEM Analysis

To find the drag force, FEM analysis will be assumed as a rigid body by way of a conception wind tunnel. Entire wheel of vehicle were constraint that it cannot move upward (vertically to Y-axis) and sideward (laterally to X-axis). In addition, the constraint of fixed point at the rear applying as weld joint so the model will supporting the pressure force by wind in opposite direction of Z-axis as shown in Figure 3.5. All the constraints are similar for the car tied in wind tunnel. But, in the meantime the wheels can move linearly either backward or forward (horizontally to Z-axis).

After constraining and deciding the boundary condition of FEM, the pressure forces from the CFD results simulation exporting to the COSMOSWorks to run the FE analysis. Subsequently, read the maximum reaction force at rear support fixed point on the model. The maximum reaction force obtaining that contributes from the fluid flow from the rear support will be used as a drag force and used to find the drag coefficient.

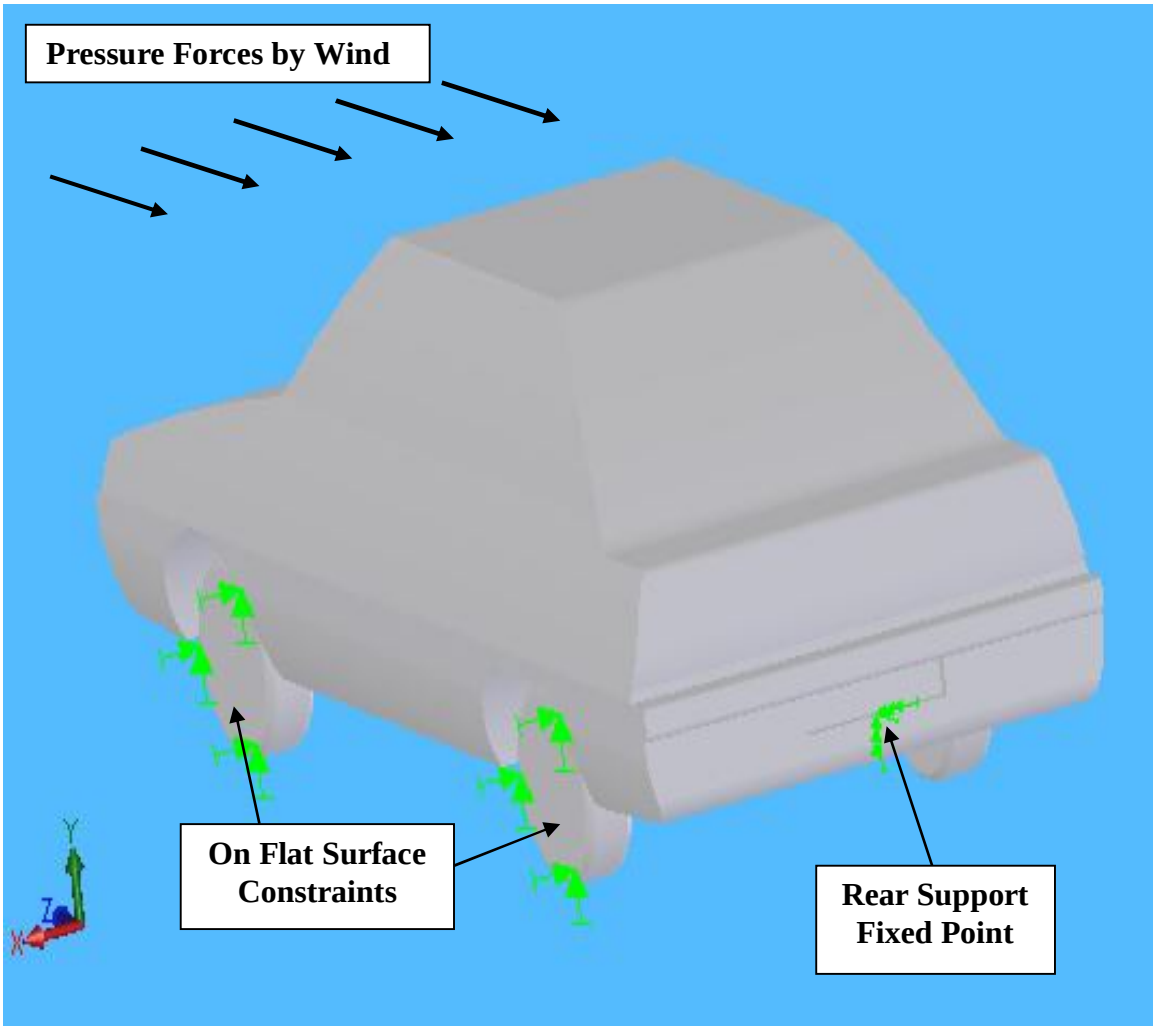


Figure 3.5 All constraints of FEM analysis as in a wind tunnel

3.6 Frontal Area Measuring

Before calculating the drag coefficient, the frontal area projected were constructing from the SolidWorks software. The frontal area were sketching then extruding it perpendicularly to the front plane of sketching area. Thus, measuring the area by measure tool while select the surface of frontal area extruded to get the value of area.

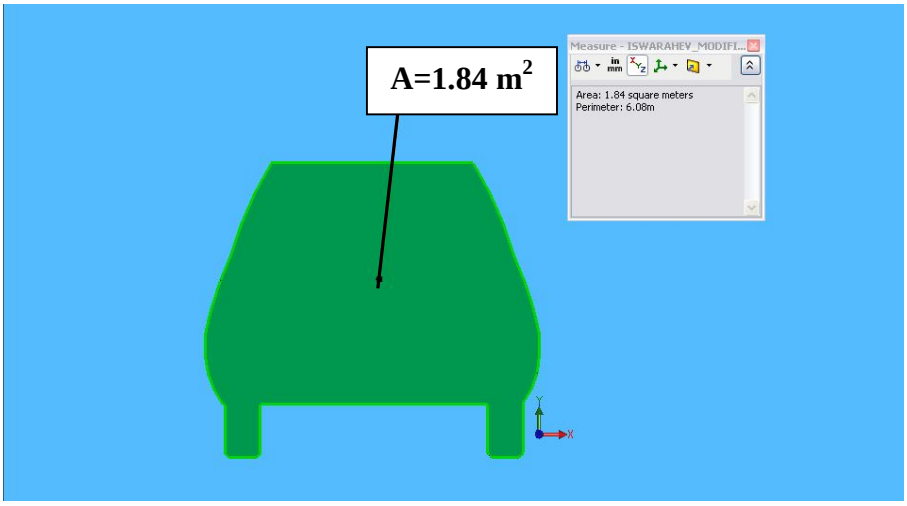


Figure 3.6 The frontal area projected of CAD model

Since the value of frontal area projected were getting, the drag coefficient easily to determine by using the equation (2.6) that defines in Chapter 2.

3.7 Simulation Analysis Validation

After run the simulation on CFD and FEM analysis, the experiment was constructing to validate the simulation analysis results. The validation scope is to get the value of total pressure at the certain point on the body of Proton Iswara.

The tools of experiment:

1. Set of Laptop (WINDAQ Acquisition Software)
2. DATAQ Instruments Hardware Manager Device
3. Pressure Transducer
4. Pressure Pad
5. Connecting Box

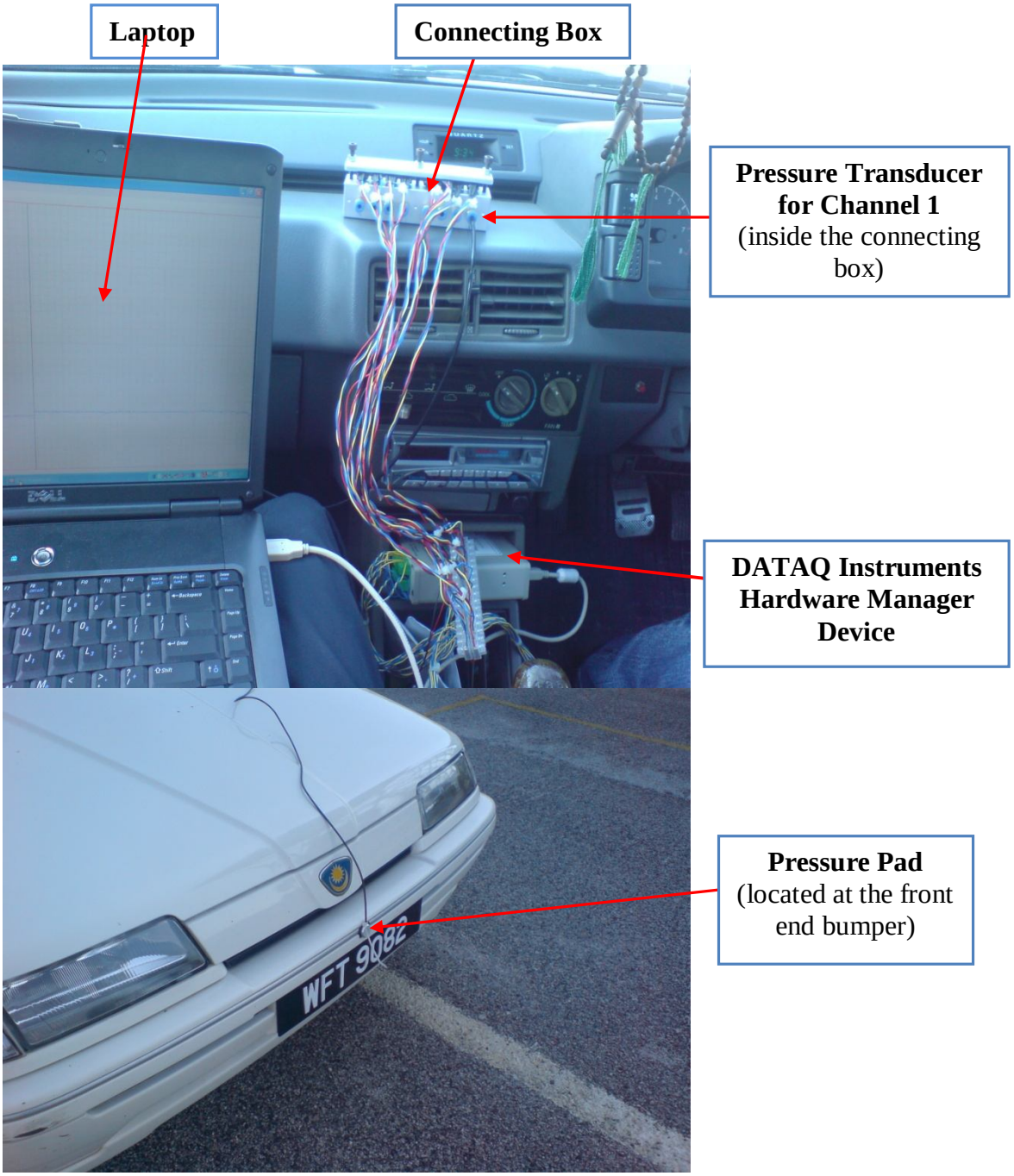


Figure 3.7 Figure of the experiment tools setup

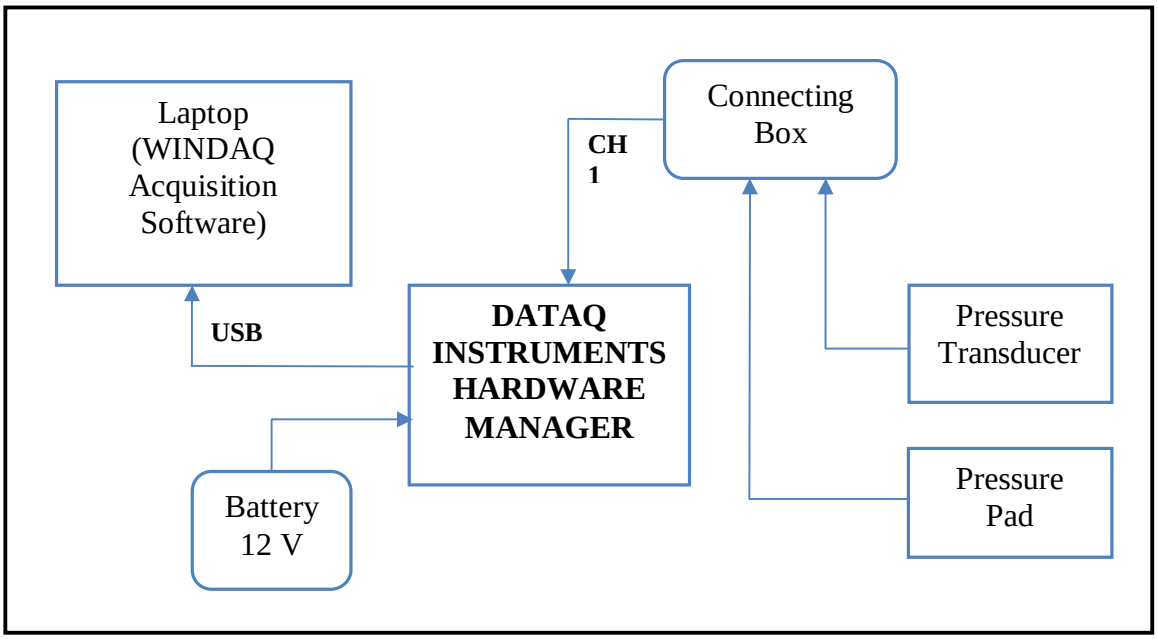


Figure 3.8 Diagram of pressure experiment setup tools

The Figure 3.7 and Figure 3.8 above shows the figure of experiment tools setup and its diagram and how the tools were connected each other. The main device for pressure experiment is DATAQ instruments hardware manager device. The functions of every tool were discussed in Table 3.1 below.

Table 3: Table of experiment tools and the function

Tool	Function
DATAQ Instruments Hardware Manager	Allows to effectively manage and run multiple instruments installed and connected to PC or network or even over the internet
WINDAQ Acquisition Software	It was accessed through the DATAQ Instruments Hardware Manager. All available devices will automatically appear in the list box in PC system when running this software.
Connecting Box	It used as port of pressure transducer and have about 6 channels while connect to DATAQ Instrument Hardware device.
Pressure Transducer	It used to sense the pressure of airflow from the pressure pad. After that, transmit the input as a pressure gage to the DATAQ Instruments Hardware Manager and display by the WINDAQ Acquisition Software.
Pressure Pad	Function as a device that receive the airflow then pass on it by the tube to pressure transducer.

The experiment was performing along the straight road outside the laboratory of Universiti Malaysia Pahang. The speed test of experiment is 40km/h and 50km/h only, then the gage pressure were measured by the experiment tools. The pressure pad was located at 4 differences point location where is:

1. Front End Bumper
2. Bottom of Windshield
3. Front End Roof
4. Rear End Roof

Before the experiment, the ambient temperature was taken to define the air density and ambient pressure. After the experiment tools was setup completely, the data will start recorded at about speed of car 30km/h and let the speed of car remains constant for a long distance and several seconds at the test speed. The test was run in two times for every point location then the average value of pressure was taken and analyze for 40km/h and 50km/h of speed test.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Data Collections

4.1.1 Reference Point of Flow Analysis

The reference point of flow analysis for simulation was defined before run the calculation of CFD. The reference points of flow analysis are detailed below:

(a) Fluid type: Air

(b) Ambient parameter conditions

i. Ambient Pressure: 101325 Pa

ii. Ambient Temperature: 300.15 K

iii. Density of Air: 1.225 kg m^{-3}

(c) Analysis type: External Flow

(d) Wall Condition: Adiabatic (Incompressible Flow)

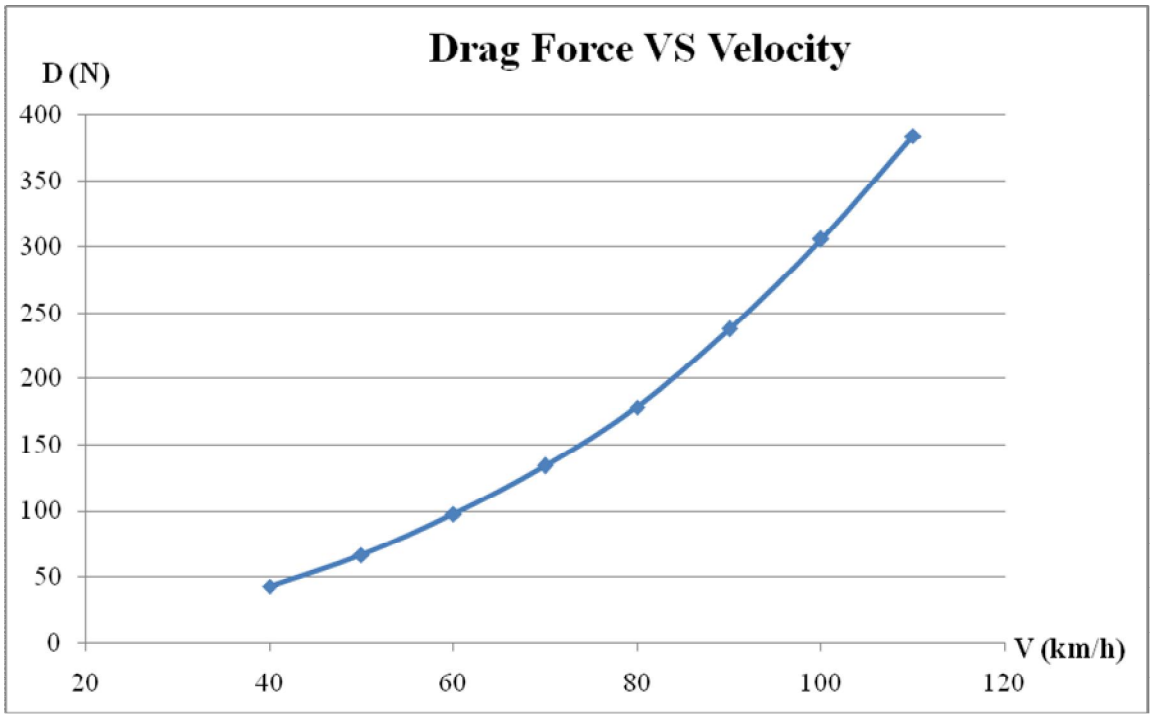
4.1.2 Data of Various Velocities and Drag Forces

The drag forces data collections that were getting started from CFD and FEM analysis for various velocities ranging from 40 km/h to 110km/h was listed in the Table 4.1 and the graph from the data has been plotted in Graph 4.1.

Table 4.1: Table of various velocities and drag forces

Velocity, (km/h)	**Drag Force, (N)**
40	42.36
50	66.84
60	97.52
70	134.00
80	178.00
90	237.70
100	305.50
110	383.60

The data from Table 4.1 was plotted into the graph of drag forces, F_D against the velocity, V. Then, the graph series of D VS V was shown in Graph 4.1 below.



Graph 4.1 Graph drag forces, *D* against velocity, *V*

From the Graph 4.1 above shown the curve of positive quadratic form graph or exponential graph. It shows clearly the drag forces, F_D was function to the square of velocity from 40 km/h up to 110km/h for every 10km/h. Thus, it means the increasing of velocity would affected to the increasing of drag forces.

4.1.3 Value of Projected Area

The value of projected area was getting from the CAD software. The projected area means as frontal area of the Proton Iswara's body that has overall height and width is 1.36 m and 1.655 m. While, the value of projected area were measured is about 1.84 m^2 directly from the CAD's Software.

So, $A = 1.84\ m^2$

4.2 Data Analysis

4.2.1 Calculation of Drag Coefficient

From the all data collections of Table 4.1, and Graph 4.1**,** reference point measurement, and projected area of the car. The drag coefficient, C_D was calculated base on the equation (2.6). The calculation were including for every speed of analysis where is between 40 km/h to 110km/h for every 10km/h then plotted into a graph.

4.2.2.1 Sample Calculation for Drag Coefficient

Therefore, drag coefficient for every velocity of car were calculated from the equation (2.6). Beside, the value of percentages C_D rise between the two velocities was considered too for analysis.

1. Drag Coefficient for velocity of car is 40km/h;

$$C_D = \frac{42.36}{\frac{1}{2}(1.225)(11.11)^2(1.84)}$$

$$= 0.304512$$

2. Drag Coefficient for velocity of car is 50km/h;

$$C_D = \frac{66.84}{\frac{1}{2}(1.225)(13.89)^2(1.84)}$$

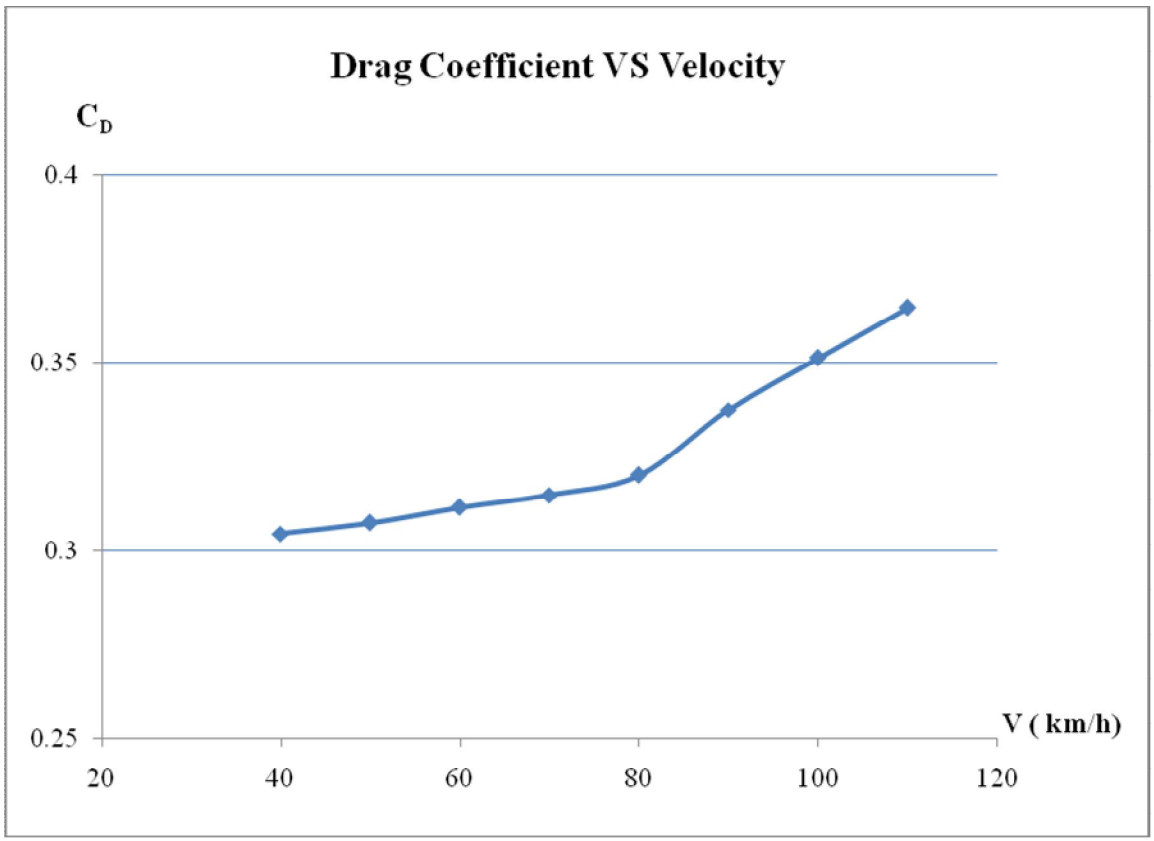
$$= 0.307403$$

4.2.3 Data of Drag Coeffiecient for Various of Velocity

Since the coefficient has been calculated for every velocity, the drag coefficient and velocity was listed and plotted in Table 4.2 and Graph 4.2.

Table 4.2: Table of various velocities and drag coefficients

Velocity, (km/h)	**Drag Coefficient, C_D**
40	0.305
50	0.307
60	0.311
70	0.315
80	0.320
90	0.337
100	0.351
110	0.364
Average of C_D	**0.326**



Graph 4.2 Graph of Drag Coefficient, C_D against Velocity, V

The Graph 4.2 shown that the drag coefficient is proportionally increase to the increasing of velocity. For low velocity ranging between 40 km/h to 80 km/h from the graph, illustrating the drag coefficients were small slightly change of increasing velocity. But, at the high velocity of 90km/h the drag coefficient is 0.337 were significantly from the drag coefficient of 80 km/h to 90 km/h about 5% changes, while the drag coefficients were gradually increase up to 100km/h and 110km/h which is about 0.351 and 0.364.

Clearly this graph show of increasing of drag coefficient as increasing of velocity and might be related to the increasing of drag force by square of velocity that were discussed before for the Graph 4.1. Since the value of the drag coeffients were calculated, the average of drag coeffient for this analysis of Proton Iswara's body that ranging 40 km/h to 110km/h is 0.326.

4.2.4 Calculation for Percentages of C_D Rises

4.2.4.1 Sample Calculation for Percentages of C_D Rises

Calculation of percentages C_D Rise between 40km/h and 50 km/h:

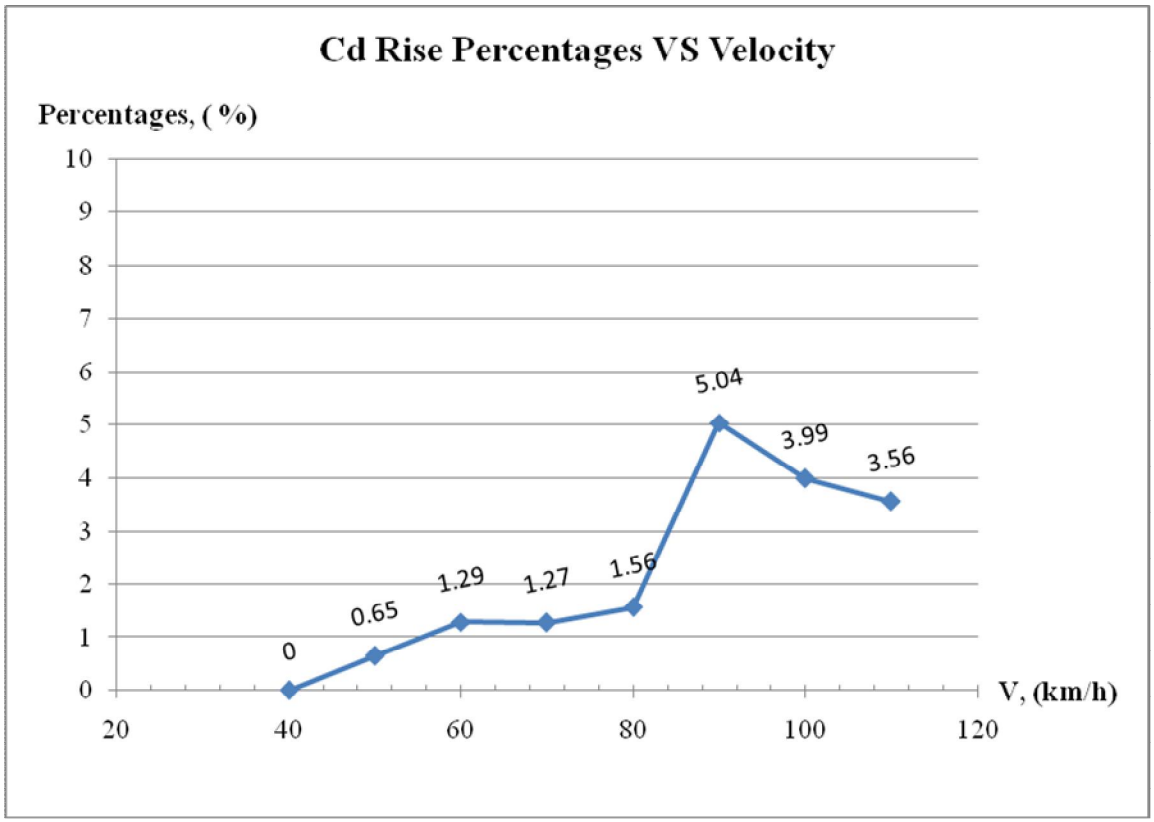
$$Percentages\ C_D\ Rise, \%$$

$$= \frac{0.307 - 0.305}{0.307} x100\%$$

$$= 0.65\ \%$$

Table 4.3 Table of various velocities and percentages of C_D rise by velocity

Velocity, (km/h)	**Percentages of C_D Rise by Velocity (%)**
40	0
50	0.65
60	1.29
70	1.27
80	1.56
90	5.04
100	3.99
110	3.56



Graph 4.3 Graph of C_D Rises Percentages against Velocity

The percentages rises of aerodynamics drag in term of drag coefficient for ranging of velocity's analysis between 40 km/h to 110km/h was be able to see in Graph 4.3 above. The graph shown, from the 40 km/h to 80 km/h the average percentages of aerodynamics drag rises is about 1.2% where is no significant change of aerodynamics drag at that range but its can say that the aerodynamics drag already build up at the ranging of velocity.

Despite the fact that, the percentage of aerodynamics drag was change drastically at velocity 90 km/h about 5.04% then it gradually decrease at ranging 90 km/h to 110 km/h where is 3.99% and 3.56%. Since the value of drag coefficient of 100km/h and 110km/h is higher but it does not means the percentages rises of drag coefficient was increased. This is because, it might be at the high speed of 100km/h and 110km/h the aerodynamics streamline more better than the velocity of 90 km/h.

4.2.5 Countour Plot of Velocity and Pressure

The contour plot of velocity and pressure were figured in Figure 4.1 and Figure 4.2. The contour plot of velocity and pressure were shows to analyzed the characteristic of wake region and the pressure's boundary layer around the Proton Iswara's body.

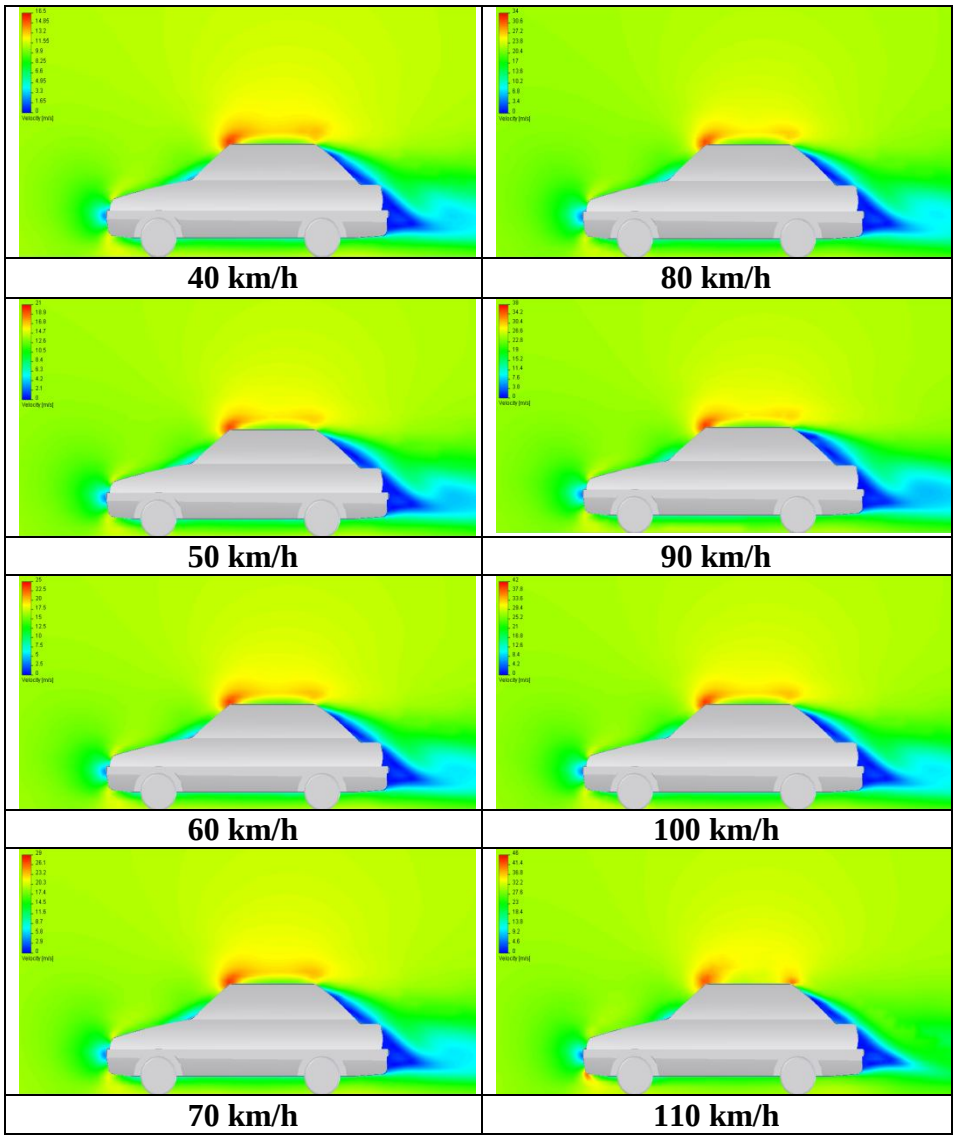


Figure 4.1 Figure of table of velocity plot from the center body for various velocities

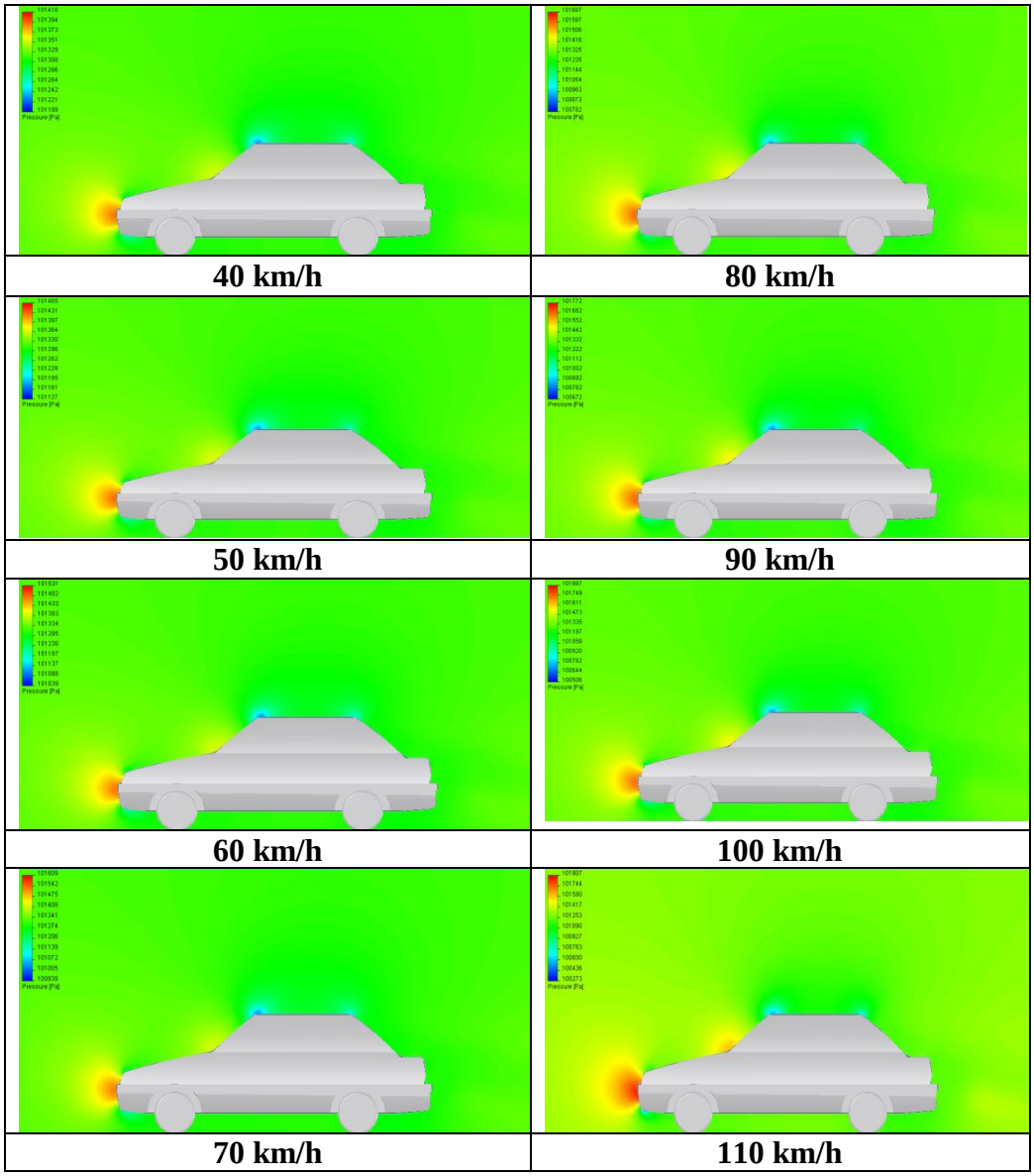


Figure 4.2 Figure of table of pressure plot from the center body for various velocities

Figure 4.1 shows the contour plot of velocity for the ranging velocity's analysis. From the figure above was demonstrating the patent of the contour plot of velocity quite same for every velocity of analysis. The red and blue color from the contour plot of velocity shows high and low velocity.

If looked detail, the big slight differences contour plot of velocity at the high velocity of 110km/h from others. The clear differences can look at the blue region behind the body called as a wake region where the turbulent flows built up after laminar flow from the separation point located at about of above rear windshield or at the rear end edge of roof. The wake region or blue color is small than other, show that better of separation flow from the streamline of forebody.

At 90 km/h, the wake region or separation region also clearly different where the intensity of blue color is more than the others in addition has a big of region. It shows the base pressure drag is high and affected to the increasing of drag force. At the same time as, shows rationalization of analysis before from Graph 4.3.

For the Figure 4.2, the contour plot of pressure needs to consider also after contour plot of velocity consideration to know the form of pressure drag. The red colors develop at the end of front bumper indicated the high dynamics pressure known as stagnation point as velocity is low or zero.

While, the blue color at the two end point of the roof's body shows the negative or low of pressure distribution. This is because, the high of streamline velocity during convergence and divergence wedge of air flow on the top of body that can see from the Figure 4.1.

The local pressure depicts clearly difference for the velocity of 110km/h where apparently the yellow color than the green color for others velocity's analysis. The yellow color means the high pressure drag or high of drag coefficient. Beside that, the big red color of stagnation point means as high dynamics pressure also give significant value for the aerodynamics drag.

4.2.6 Trajectories Velocity Flow Analysis

The detail analysis of the flow on the body by trajectories velocity flow analysis diagram were figured in the Figure 4.3. The figure was shown as a general flow on the Proton Iswara body and the velocity of 40km/h was chosen to help in determining the pressure field on the surface. The pathlines means the path taken by an air particle that's start out at a given point of value in the flow.

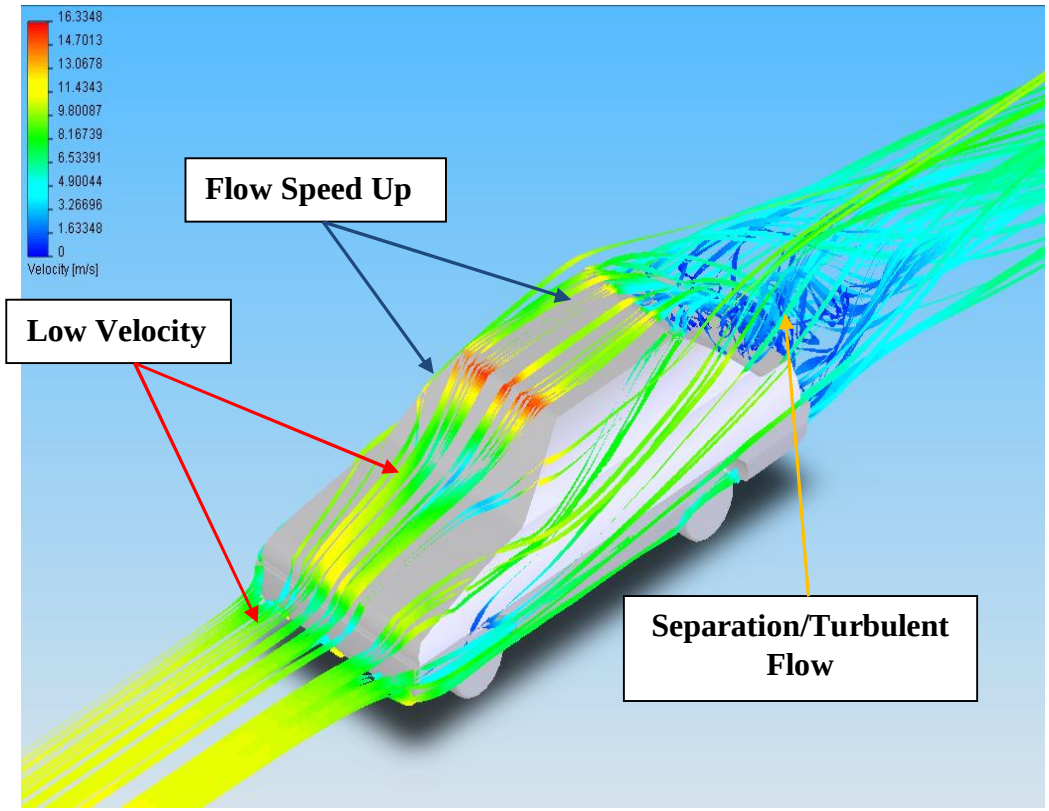


Figure 4.3 The isometric view of trajectories velocity flow of 40 km/h

Shown at the Figure 4.3 above, the area labeled with the usual phenomena in aerodynamics of car. As an air move onto the car, the air flow would be wedged at front end bumper. This area or point was known as stagnation point, where the flow of air would be stop or reducing the velocity of air after hit the bumper then give the high pressure forces to the car.

After that, the air moving and attached on the front hood of car then hit again at the hood-windshield junction before diverged into diverging wedge. Since the air hit the area of hood-windshield junction, the area would produce the positive pressure or high pressure distributions.

After air flow across the hood-windshield junction the air was accelerated at the front end roof of car and then attached over until the rear end roof of car. The high of velocity were occurring at the two end points, front and rear end roof of car. Despite the fact that the reduction in pressure.

Since the air reach the end rear point of car roof, the air was decelerates through the rear windshield or diverging wedge region then the air flows creates an enlarged flow space known as wake region. If looked detailed, the wake region is due to the streamline flow at the trailing edge of side and roof of the car. Therefore, its mean the wake region or turbulent flow can be controlled by the good streamline at the fore body and upper body of car.

4.3 Validation of Simulation Analysis

4.3.1 The results of experiment for the validation of simulation analysis

The experiment details and data was collected during the test was discussed to the table form and graphical. Show below the table of experiment details in Table 4.4.

Table 4.3: Table of experiment details

1. Name of Experiment:	Pressure Experiment
2. Model of Car used:	Proton Iswara
3. Date of Experiment:	10 October 2008
4. Time of Experiment:	9.00 a.m
5. Location of Experiment:	Outside the laboratory of UMP
6. Temperature:	31.5 ^{o}C
7. Density of Air:	1.159 kg.m^{-3}
8. Ambient Pressure:	101301 Pa

4.3.2 Location of Selected Point

The location point of measurement was spotted base on the pressure plot of simulation where is:

1. Front End Bumper
2. Bottom of Windshield
3. Front End Roof
4. Rear End Roof

The figure below shows the four locations of pressure test base on the pressure plot of simulation.

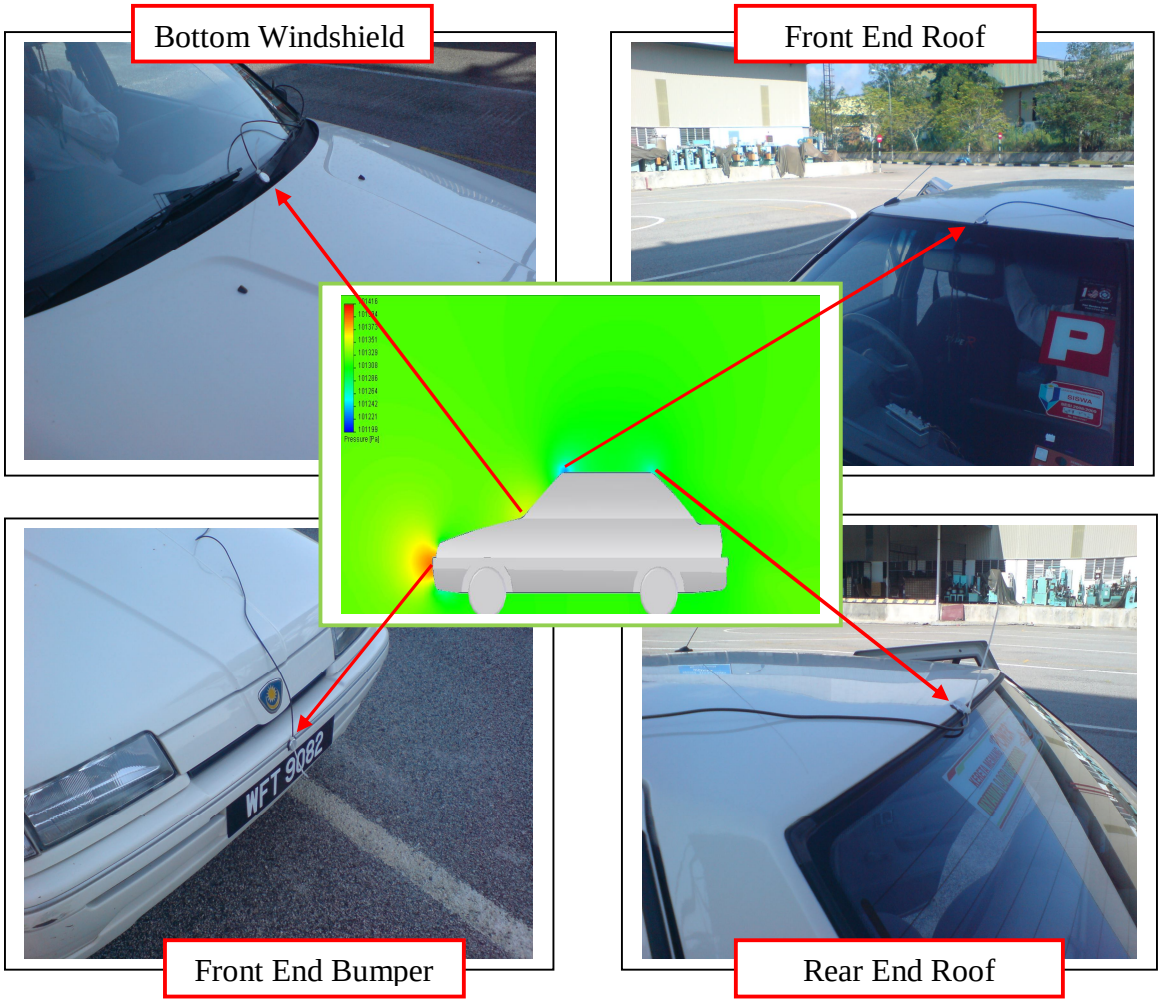


Figure 4.4 The location point of high and lower pressure distribution

4.3.3 Pressure Reading of Experiment and Simulation

Base on the experiment and simulation results, the absolute value of pressure were tabled into Table 4.5 and Table 4.6 then plotted into Graph 4.4 and Graph 4.5. The differences between the experiment and simulation results also calculated to define the validation between the experiment and simulation results. Shows below the Table 4.5, Table 4.6, Graph 4.4 and Graph 4.5 for speed 40 km/h and 50 km/h of analysis.

1. Reading for Speed 40km/h

Table 4.5: Table of Pressure Reading of Experiment and Simulation and the differences for 50 km/h

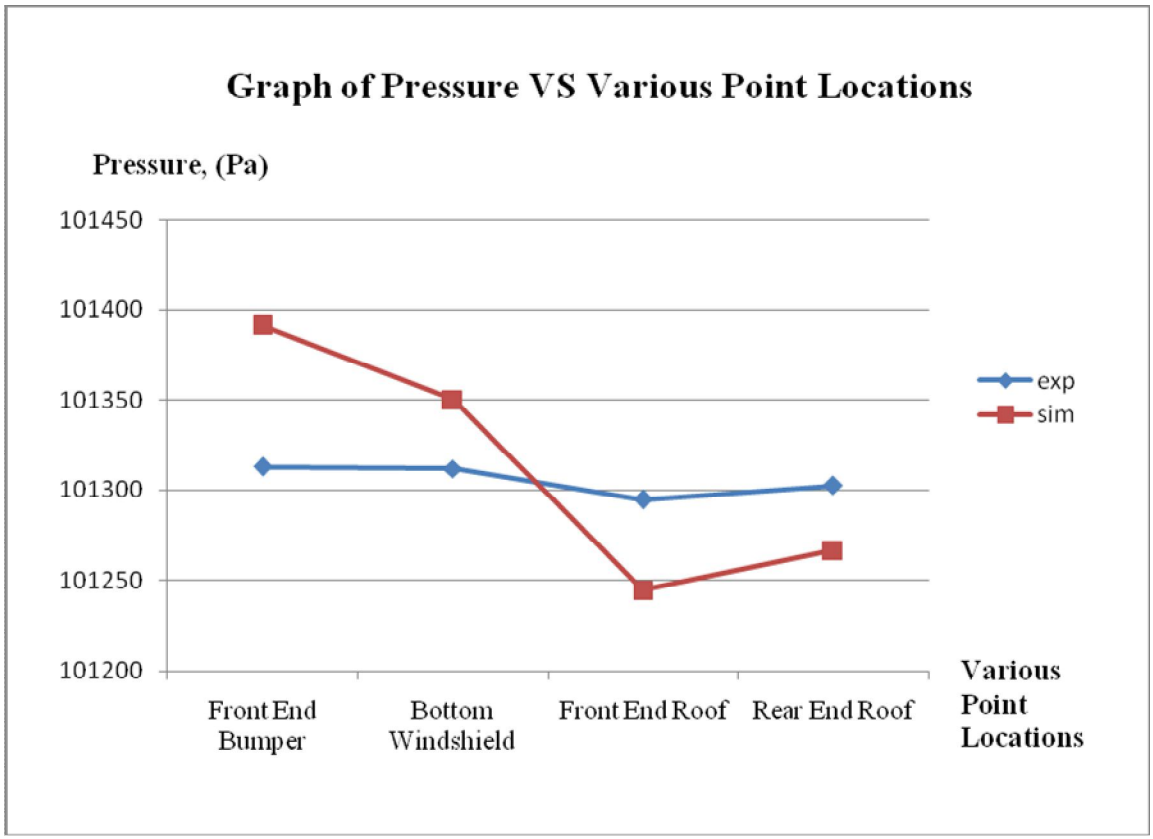
Reading	**Front End Bumper**	**Bottom of Windshield**	**Front End Roof**	**Rear End Roof**
Experiment, (Pa)	101337.19	101336.04	101318.78	101326.62
Simulation, (Pa)	101391.18	101350.60	101244.72	101266.66
Difference (%)	0.05	0.01	0.07	0.06

2. Reading for Speed 50km/h

Table 4.6: Table of Pressure Reading of Experiment and Simulation and the differences for 50 km/h

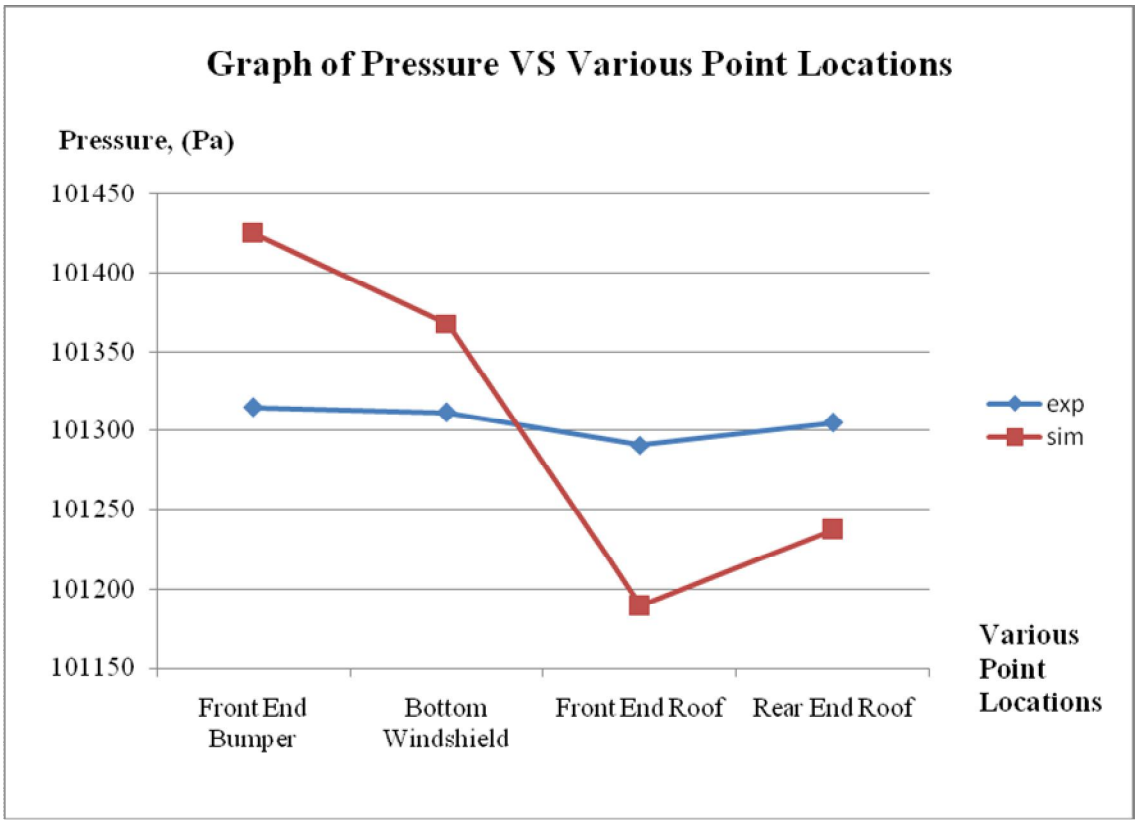
Reading	**Front End Bumper**	**Bottom of Windshield**	**Front End Roof**	**Rear End Roof**
Experiment , (Pa)	101338.72	101335.48	101314.99	101329.03
Simulation, (Pa)	101425.00	101367.78	101189.40	101236.90
Difference (%)	0.09	0.03	0.12	0.09

4.3.4 Discussion of Graph Validation of Pressure



Graph 4.4 Graph of Pressure Reading of Experiment and Simulation against Various Point Location s for speed 40km/h

From the graph 4.4 show that the data of pressure reading from the experiment and simulation for various point locations of 40 km/h. Both of data was represent the differences for front end bumper 0.05%, , the bottom windshield is 0.01% and the front end roof is 0.07% and the rear end roof is 0.06%. The highest of differences between the experiment and simulation data analysis were expected from the inaccuracy of CAD modeling at certain body and lack of meshing in CFD simulation analysis.



Graph 4.5 Graph of Pressure Reading of Experiment and Simulation against Various Point Locations for speed 50km/h

Graph 4.5 above show the same trend with the Graph 4.4 before, where is the difference for the front end bumper and rear end roof is about 0.09%, the bottom windshield is 0.03%, and the front end roof is 0.12%. If make a comparison between Graph 4.4 and Graph 4.5, the accuracy at bottom windshield point of the comparison between experiment and simulation is very high with low of difference percentages. The accuracy at that point was expected due to the high performance of pressure transducer at that area during the experiment and good of re-meshing that define from the refinement of mesh during CFD calculation.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

As a conclusion from this project, learning process base on the estimation of drag achieved using CFD and FEM analysis specifically for upper body as well as the study of aerodynamics on the vehicle especially passenger car and hybrid electric vehicle. Aerodynamics drag for Proton Iswara's body is 0.326 of drag coefficient at ranging velocity between 40km/h and 110km/h of every 10 km/h analysis. The analysis show aerodynamics drag in term of drag forces or drag coefficient were proportionally increase to the square of velocity for Proton Iswara's body that has frontal area is 1.84 m^2.

Since the aerodynamics drag were proportionally increase to the square of velocity, the max rise of aerodynamics drag was occur between velocity of 80km/h and 90km/h which is about of 5.04% increases of drag coefficient. While, at velocity of 100km/h and 110km/h is higher of drag coefficient but it can said that the point of velocities, aerodynamics streamline were better that can rationalize by the decreasing of percentages C_D rise and other than that the better of streamline shows by velocity plot.

The contour plot of velocity and pressure were shown the rationalization of aerodynamics drag graph analysis as a visualization analysis. The patent of visualization for every velocity depict quite same either for velocity contour plot or pressure contour plot. However at velocity of 90 km/h and 110km/h, the plot of velocities obviously depicts variation. For the plot of pressure, the difference clearly shows at 110km/h.

Moreover, the trajectories flow was also give the significant value of analysis to the Proton Iswara's body. The streamline or flow of velocity can looked next to the whole body of Proton Iswara. The turbulent flow that called wake region shows visibly at the rear that comes from the fore body and side body of vehicle. Besides, the point of the streamline accelerates and decelerates be able to determine for further analysis.

In addition, a simple experiment named as Pressure Experiment was successfully done. The experiments were give the valuable results to validate simulation analysis results of high pressure distribution on the four location at body of Proton Iswara which is front end bumper, bottom windshield, front and rear end roof. The validation show about less than 1% differences between simulation and experiment results for analysis 40 km/h and 50km/h of test speed.

5.2 Further Study Recommendation

In view of the fact that, this project represents the first known insight of flow over the Proton Iswara's body and still not have previous work of data either experimentally or simulation. Therefore, some of recommendations were list below to improve this analysis of project to give a better performance on the HEV model especially in developing of aerodynamics performance. The recommendations are:

1. Construct the experimental analysis by wind tunnel to validate the simulation analysis.
2. Refine the model geometry by 3D scanner to get the accuracy during CFD and FEM analysis on the body of Proton Iswara
3. Use the better software such as FLUENT software to perform the simulation analysis of aerodynamics drag.

REFERENCES

[1] Dr. V. Sumantran and Dr. Gino Sovran. *Vehicle Aerodynamics*. Society of Automotive Engineers, Inc. 1996.

[2] Bruce R. Munsan. Donald F. Young and Theodore H. Okiishi. *Fundamental of Fluid Mechanics*. Fifth Edition. John Wiley & Sons (Asia), Inc. 2006.

[3] Luca Iaccarino. *Cranfield University Formula 1 Team: An Aerodynamics Study of the Cockpit*. School of Engineering. Cranfield University. August 2003.

[4] John D. Anderson Jr. *Fundamental of Aerodynamics.* Fourth Edition. Mc Graw Hill. 2005.

[5] A.C. Kermode. *Mechanics of Flight*. 11th Edition. Pitman Books Ltd. 2006.

[6] Masaru Koike, Tsunehisa Nagayoshi and Naoki Hamamoto. *Research on Aerodynamic Drag Reduction by Vortex Generators*. Mitsubishi Motors. 2004.

[7] Guido Buresti. *The Influence of Aerodynamics on the Design of High-Performance Road Vehicles*. Department of Aerospace Engineering University of Pisa, Italy. 19 March 2004.

[8] Wolf-Heinrich Hucho. *Aerodynamic of Road Vehicle.* Fourth Edition. Society of Automotive Engineers, Inc. 1998.

[9] Emmanuel Guilmineau. *Computational Study of Flow around a Simplified Car Body*. Elsevier Ltd. 0167-6105. 2007.

[10] Heinz Heisler. *Advanced Vehicle Technology.* Second Edition. Elsevier Butterworth Heinemann. 2002.

[11] Ing. Andreas Kleber. *Simulation of Air Flow Around an OPEL ASTRA Vehicle with FLUENT*. JA132. 2001.

[12] Marit Kleven* and Morten C. Melaaen. *Fluid Flow And Particle Deposition Simulations In The Human Nose.* Telemark University College (HiT-TF) and Telemark R&D Centre (Tel-Tek).

[13] *FundamentalsGuide*. COSMOSFloWorks 2003. COSMOS. 2003

APPENDIX A

PROJECT GANTT CHART

Appendix A1: Gantt Chart for FYP 1

Activities/ Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Project's title briefing																
Verify the project title, objectives and scopes																
Write down the background, abstract, objective and scope																
Literature Study																
Study of theory aerodynamics																
Study of aerodynamics																
Study on CFD and FEA																
Determine best method of methodology																
Concept selection (CFD and FEM analysis)																
Detailed Methodology																
Submit proposal and draft of report																
Presentation of Proposal																

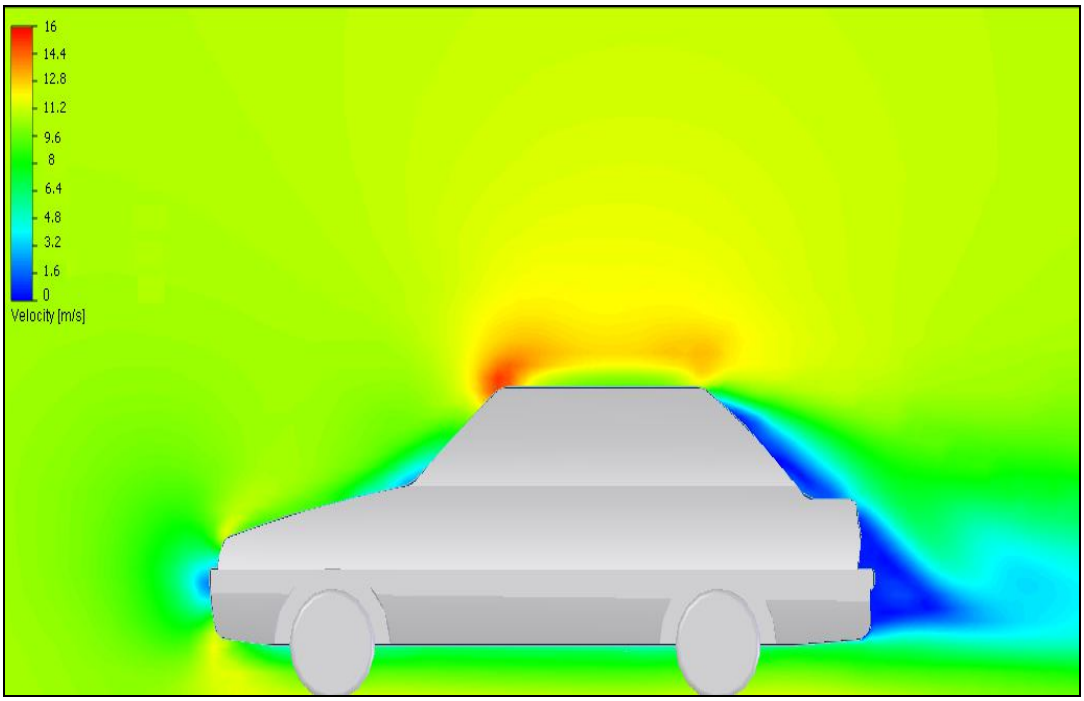
Appendix A2: Gantt Chart for FYP 2

Activities/ Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Literature Study																
Analysis project																
Collect the data																
Analysis of data and results																
Simulation validation																
Interprets data																
Conclusion of the project																
Final Presentation																
Preparing and submit report																

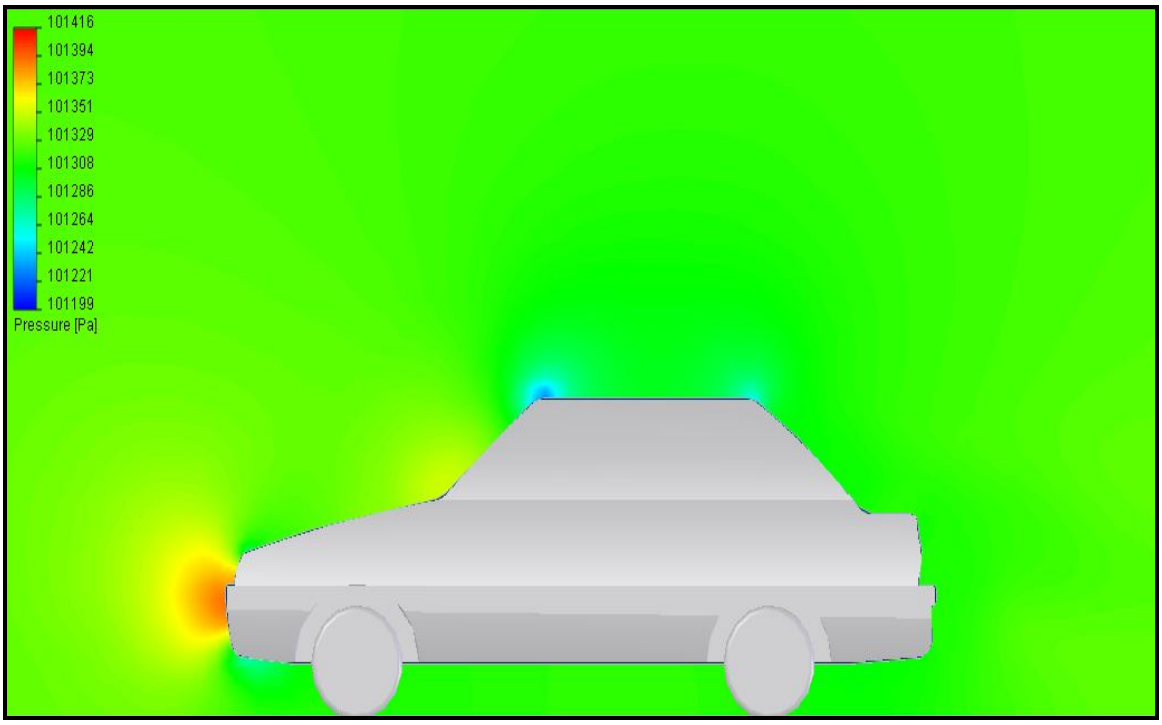
APPENDIX B

ANALYSIS PLOT OF
COMPUTATIONAL FLUID DYNAMICS (CFD)

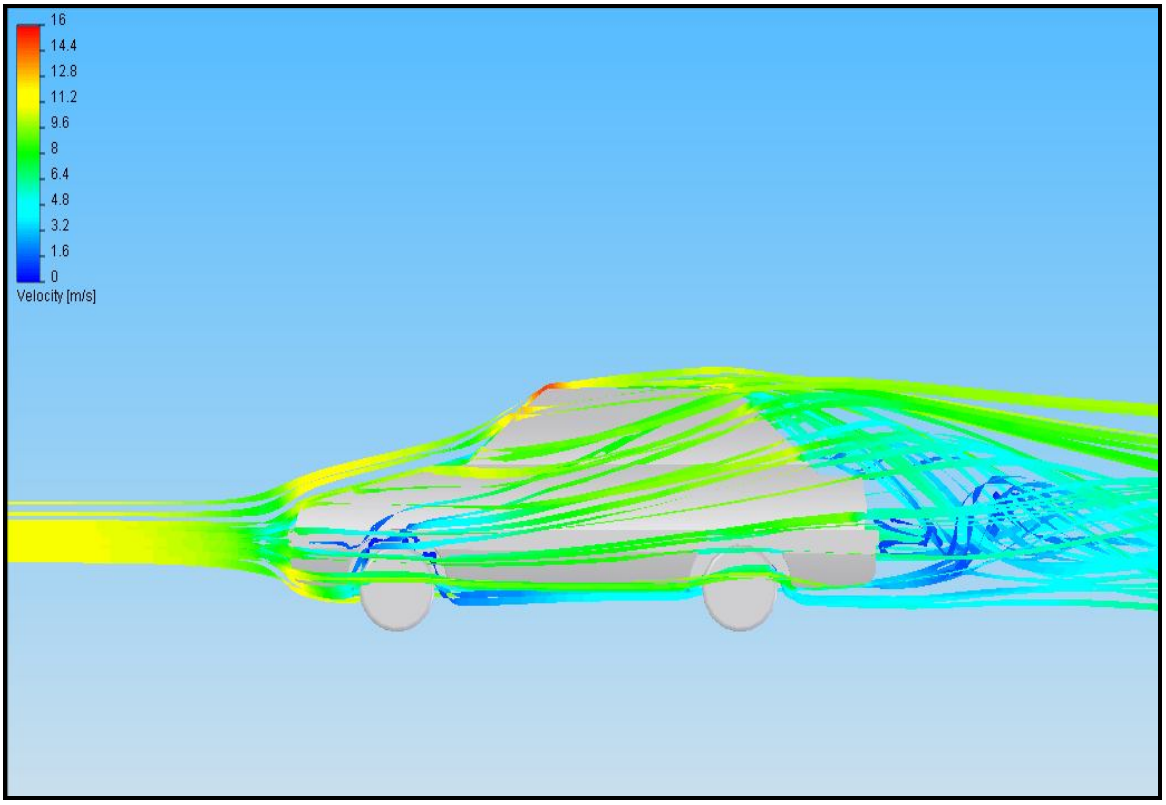
Appendix B1: Simulation of HEV Model with Boundary Layer of Velocity



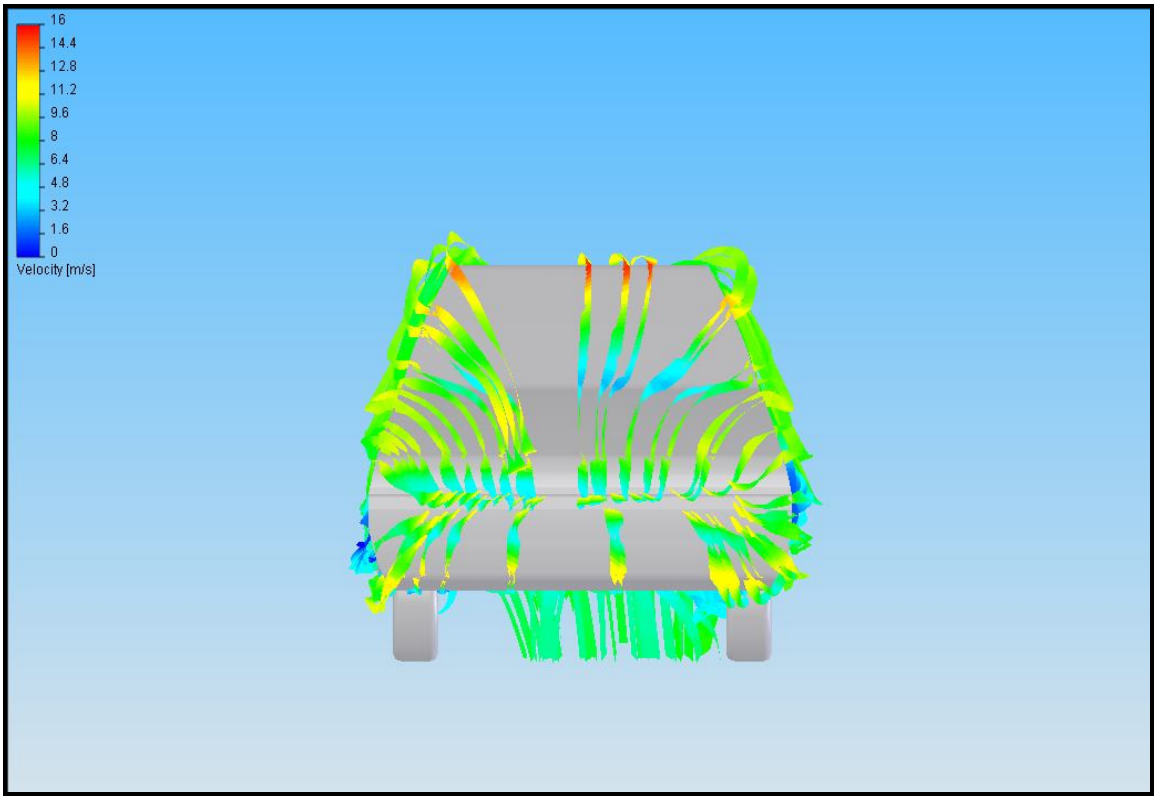
Appendix B2: Simulation of HEV Model with Boundary Layer of Pressure



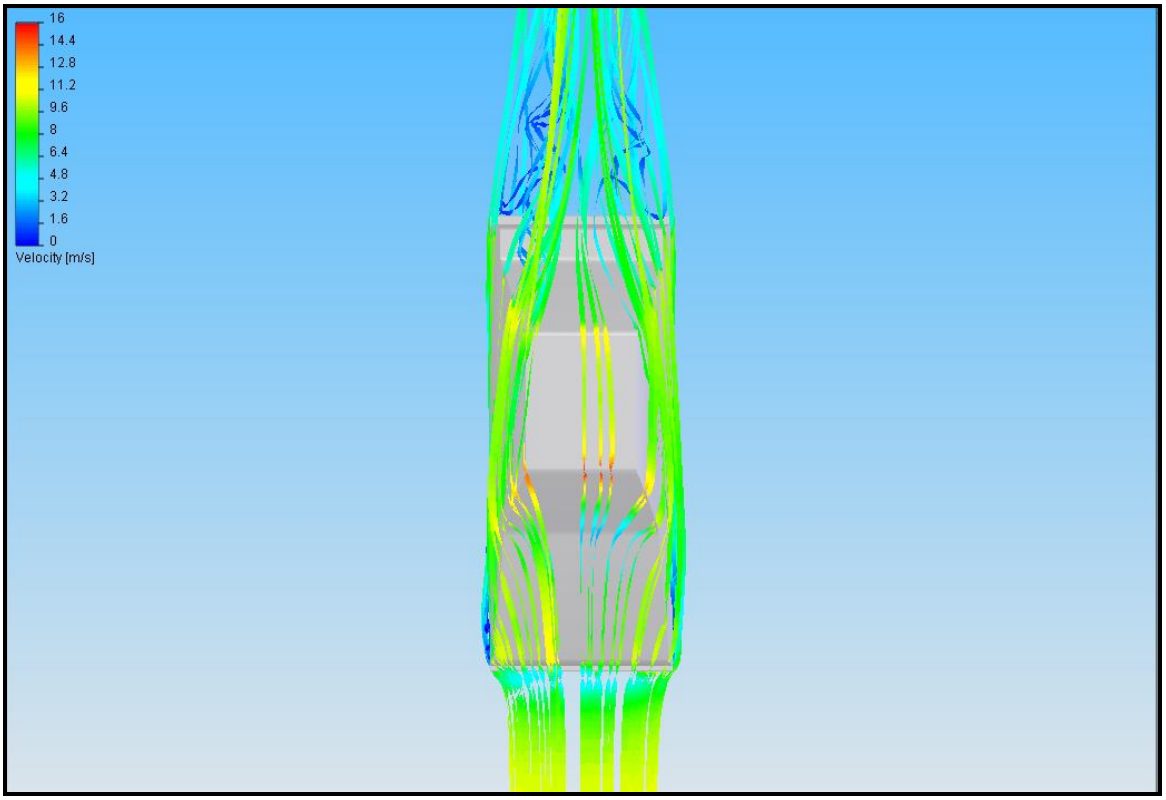
Appendix B3: Simulation of HEV Model with Velocity Trajectories Plot in Right View



Appendix B4: Simulation of HEV Model with Velocity Trajectories Plot in Front View



Appendix B5: Simulation of HEV Model with Velocity Trajectories Plot in Top View



Appendix B6: Simulation of HEV Model with Velocity Trajectories Plot in Isometric View