DESIGN AND ANALYSIS FRONT HOOD FAIRING FOR CARS

MAHASAUFEE BIN MAHAZIS

BACHELOR OF MECHANICAL ENGINEERING WITH AUTOMOTIVE
FACULTY OF MECHANICAL ENGINEERING
UNIVERSITI MALAYSIA PAHANG

2009

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

We certify that the project entitled *Design and analysis front hood fairing for cars* is written by *Mahasaufee Bin Mahazis*. We have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfilments of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

AGUK ZUHDI MUHAMMAD : ………………………….

Examiner Signature

Date : ………………………....

DESIGN AND ANALYSIS FRONT HOOD FAIRING FOR CARS

MAHASUFEE BIN MAHAZIS

A report submitted in fulfillment of the requirements

for the award of the Bachelor of

Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering

UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

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

Signature	:	....
Name of Supervisor	:	Devarajan Ramasamy.
Date	:	20 November 2009

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

I Mahasufee Bin Mahazis declare that this report entitled " *Design and Analysis Front Hood Fairing for Cars (Mechanical)* " is the result of my own research except as cited in the references. The report has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : ....

Name : Mahasaufee Bin Mahazis

Date : ....

Dedicated to my beloved abah and emak

ACKNOWLEDGEMENTS

Indeed, the emotional moments have come.

Alhamdulillah, thanks to him for granting me such precious breathing spaces to finally complete my wonderful years of study in UMP. I am truly thankful to you for giving me those huge tests during to complete this research which I believed they will be my greatest teachers ever in order to survive for upcoming moments. Alhamdulillah.

My infinite gratitude goes to both my parents who in my opinion are the best parents in the world and also my family members for their enduring patience, moral and financial supports. Your sacrifices are truly invaluable. My sincere appreciation also extends to all my colleagues and others who have provided assistance at various occasions. Their views and tips are useful indeed.

Then, I would like to express my sincere appreciation to my wise supervisor, Mr. Devarajan a/l Ramasamy, who generously shared his insights and suggestions, for his critics, trust, encouragement, and attention. Without his continued support and interest, this project report would not have been the same as presented here. From the bottom of my heart, I want said thank you. Thank you for everything. May God bless all of you.

ABSTRACT

This report is an outcome of the work carried out in doing and completing my final year project, Design and analysis front hood fairing for cars. The paper presents a model of the front hood fairing for cars that can reduce the drag coefficient of the car, model that been used is proton Iswara. The overall duty is modeling and analyzes to get the best model that can reduce the coefficient of drag of the proton Iswara because of the high drag force at front windscreen. It happen when the airflow at the windscreen is disturbed by car wiper during high speed. It is often that the wiper system generates unwanted noise and vibration during the high speed. The report starts off with an introduction in aerodynamics vehicle and design. Then proceed to investigation about the problem at the front windscreen to get information. After gathering all the relevant information, the project undergoes design process. In this steps, from the knowledge gathered before is use to make a design refers to case data that suitable for the project. Several sketches have been made and only a few have been selected based on the suitability of the front hood and dimension of the windscreen. Based on the sketches, all the sketches will be draw into CAD software and simulate in the CFD software. At the end, the fairing that can reduce more coefficient of drag will be selected. The result shown by adding fairing at the front hood will decrease the pressure and wind noise also can be reduced. Then, the airflow will smooth and the performance of the car greater than before.

ABSTRAK

Laporan ini hasil daripada kajian dalam projek sarjana muda yang bertajuk rekaan dan analisis struktur bantu di bonet hadapan untuk kereta. Kertas kerja ini memapaparkan dan menerangkan model struktur bantu di bonet hadapan yang dapat mengurangkan pekali heretan kereta. Model kereta yang digunakan ialah Proton Iswara. Keseluruhan tugas adalah untuk mereka dan menganalisis model-model terus memilih yang terbaik yang dapat mengurangkan daya heretan di kaca hadapan kereta. Ia berlaku akibat aliran udara di bahagian kaca hadapan kereta dihalang oleh wiper semasa pemanduan di halaju yang tinggi . Hal ini sering bahawa sistem wiper menghasilkan hingar dan getaran yang tidak diingini pada kelajuan tinggi. Laporan bermula dengan pengenalan dalam aerodinamik kenderaan dan rekaan. Seterusnya penyiasatan terhadap masalah di bahagian kaca hadapan untuk mendapatkan maklumat lanjut. Apabila semua dokumen berkaitan selesai dikumpul,projek ini diteruskan dengan fasa rekabentuk model. Dalam fasa ini, rekabentuk dilakukan berdasarkan dokumen yang dikumpul serta bersesuaian dengan projek. Beberapa lakaran model dihasilkan dan dipilih mengikut kesesuaian dengan bonet hadapan serta kaca hadapan kereta. Lakaran ini kemudiannya akan di lukis didalam perisian CAD dan di analisis didalam perisian CFD. Model yang dapat mengurangkan pekali seretan paling banyak akan dipilih sebagai struktur bantu yang terbaik. Akhir sekali, struktur bantu yang dapat mengurangkan pekali seretan yang terbaik akan dipilih. Keputusan projek menunjukkan dengan penambahan struktur bantu dapat mengurangkan tekanan dan hingar angin dapat dikurangkan. Kemudian, aliran udara akan lancar dan prestasi kereta yang lebih baik daripada sebelumnya.

TABLE OF CONTENT

CHAPTER 2 LITERATURE REVIEW

CHAPTER 3 METHODOLOGY

CHAPTER 4 RESULT AND DISCUSSION

APPENDICES

LIST OF TABLE

LIST OF FIGURES

LIST OF SYMBOLS

ac	Acceleration
p	Pressure
U	Velocity
ρ	Density
Cp	Coefficient of pressure
v_∞	Stream velocity
D	Drag
D_f	Friction Drag
A	Area
L	Lift
C_L	Coefficient of lift
u	Fluid velocity
S_i	Mass-distributed
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 ABBREVIATIONS

3-D	Three Dimensional
CAE	*Computer-aided engineering*
CFD	Computational Fluid Dynamics
CAD	Computational Aided Design
RANS	Reynolds-averaged Navier-Stokes equation
DNS	Direct numerical simulation
HEV	Hybrid electric vehicle

CHAPTER 1

INTRODUCTION

1.1 Project Background

The performance, handling, safety and comfort of an automobile are significantly affected by its aerodynamics properties. Defecting drag was the first major focus of automotive aerodynamics, beginning in 1960's. Low drag is important for fuel economy and low emissions. Other aspects of vehicle aerodynamics are no less important for quality of automobiles such as directional stability, wind noise, cooling of engine, ventilating and air conditional these all depend on flow field around and through vehicle.

Nowadays, automotive designer rely on aerodynamics principle to create improvement in the power and handling of vehicle at high speeds. Passenger cars have become more shapely over the years as manufacturer discovered how streamlining can increase fuel efficiency, allowing a car to travel at the same speed using less horsepower. These designs reduce air resistance, or aerodynamics drag. Low drag coefficient make the vehicle enable to move easily through the surrounding viscous air with minimum of resistance. As an increasing of drag, the more power of car to do work than reducing the power train efficiency.

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

A windscreen wiper is a device used to wipe rain and dirt from a windscreen. Almost all motor vehicle, including train, aircraft and watercraft, are equipped with windscreen wiper, which are usually a legal requirement. It is often that the wiper system generates unwanted noise and vibration during the high speed. The high speed flow through the exposed structures of the wiper can cause high wind noise levels [8]. It is because the wiper blocks the airflow through the car body.

Reverse flow also exist when the flow is disturbed by the wiper during the high speed level. The reverse flow will increase the number of drag force. Aerodynamics drag is the force of air along the length of traveling car, opposing the car's force. As the car cuts a path through the air, some air molecules collide with the front windscreen and producing resistance. A passenger car driving on the highway spends an estimated 60 percent of its energy overcoming air drag, a far greater percentage than tire friction and the energy needs of the drive train itself.

Hence, often the only solution available is to hide the wiper by tucking them behind the rear edge of hood or putting them behind some sort of flow deflector. While this does not address the wind noise generated while the wiper are being used, this is less of concern, as the increase in wind noise due to the wipers during a rain storm is usually masked by higher levels of tire noise due to the wet roads and by raindrop impact noise.

In this PSM project, the element that important to be study is aerodynamics. Part that be focused is at the front hood and the windshield of the car. Proton Iswara is car that be chosen as in this project because it is widely used by customer in Malaysia. As reducing the drag and the wind noise problem, the airflow will smooth and the performance of the car greater than before.

1.3 Objectives

1. To design the fairing hood car that make the airflow over the car body is not disturbed by the car wiper.

2. Reducing the drag by addition of fairing at front hood of car.

1.4 Project Scopes

1. Study of Proton Iswara body structure.
2. Modeling software in CAD.
3. Analyze the project with CFD.

CHAPTER 2

LITERATURE REVIEW

2.1 Theory of Aerodynamics

In this section, the fundamental of fluids mechanics and basics of aerodynamics were discussed to improve the understanding in doing analysis of the project. Many studies had been done in flow over bodies, there are many factor that effect the performance of car. In terms of aerodynamics field or fundamental of fluid mechanics the factor that will affect the aerodynamics of car 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]. All of these characteristics are influenced by drag force which is ruled by Bernoulli Equation.

$$p + \frac{1}{2}\rho_a U^2 = \text{a constant} \tag{2.1}$$

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

From equation (2.1) shows the increasing of velocity will case the decrease in static pressure and vice versa. It is because these two elements are proportional inversely with each other. 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. Besides 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.

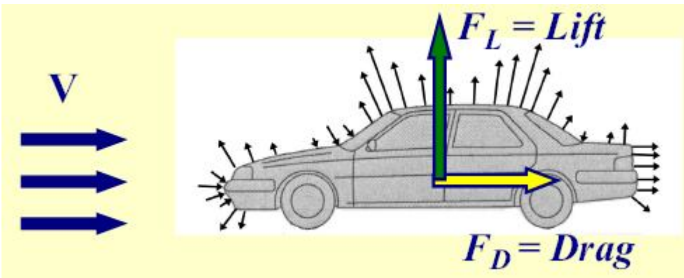


Figure 2.1 Drag and lift force due to pressure from velocity distribution [7]

2.1.2 Pressure, Lift and Drag Coefficient

Drag can generate by two main perspectives [1], firstly from the vehicle body and from the moving fluid that attached to the surface of the vehicle body. From these 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*

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

Source: Luca Iaccarino and John D. Anderson Jr. [3], [4].

2.1.2.2 Drag Coefficient

Drag coefficient is a measure of effectiveness of streamline aerodynamic body shape in reducing the air resistance to the forward motion of a vehicle. Low drag coefficient make the vehicle enable to move easily through the surrounding viscous air with minimum of resistance [10]. Drag is primarily due to the pressure distribution on the body, with only a small contribution to the drag due to skin friction. Hence, the difference between the pressure acting on the front of the vehicle and on the back increases. The skin friction drag is due to the viscous shear forces acting on the surfaces of the vehicle [6].

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, D_f, on a flat plate as

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

Where D_f is friction drag, b and l are width and length of flat plate [2]

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

The aerodynamics lift coefficient (C_L) is a measure of the different in pressure created above and below a vehicle's body as it moves through the surrounding viscous air. A resultant upthrust or downthrust may be produced which mainly depend upon the body shape [10].

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].

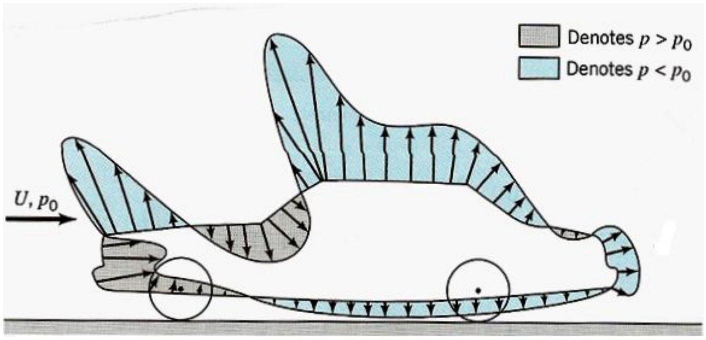


Figure 2.2 Pressure distributions on the surface of an automobile [2]

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

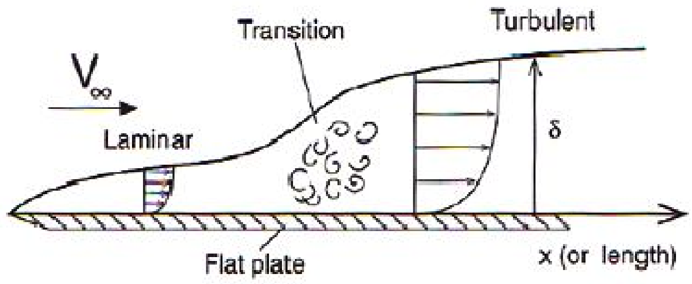


Figure 2.3 Variation of boundary layer thickness along flat plate [3]

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].

The air velocity and pressure therefore reaches its highest and lowest values, repectively, at the top of the front windscreen ; towards the rear of the roof and when the screen tilts downwards there will be a reduction in air speed and rise in pressure. In figure 2.4, the pressure rise will be large so that the mixing rate of mainstream air with the boundary layer cannot keep the inner layer moving, consequently the slowed down boundary layers thicken. Under these condition the mainstream air flow breaks away from the contour surface of the body, this being known as flow separation. An example of the flow separation followed by reattachment can be visualized with air flowing over the bonnet and front windscreen in figure 2.4 [10].

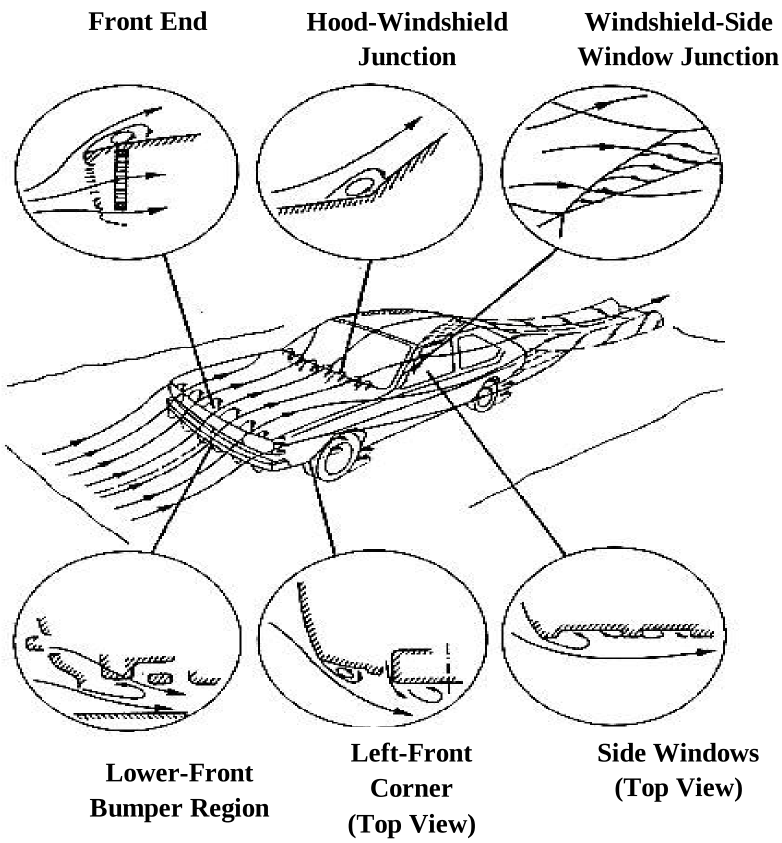


Figure 2.4 Flow around a car, and major of locations of flow separation [1]

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. Besides 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 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.

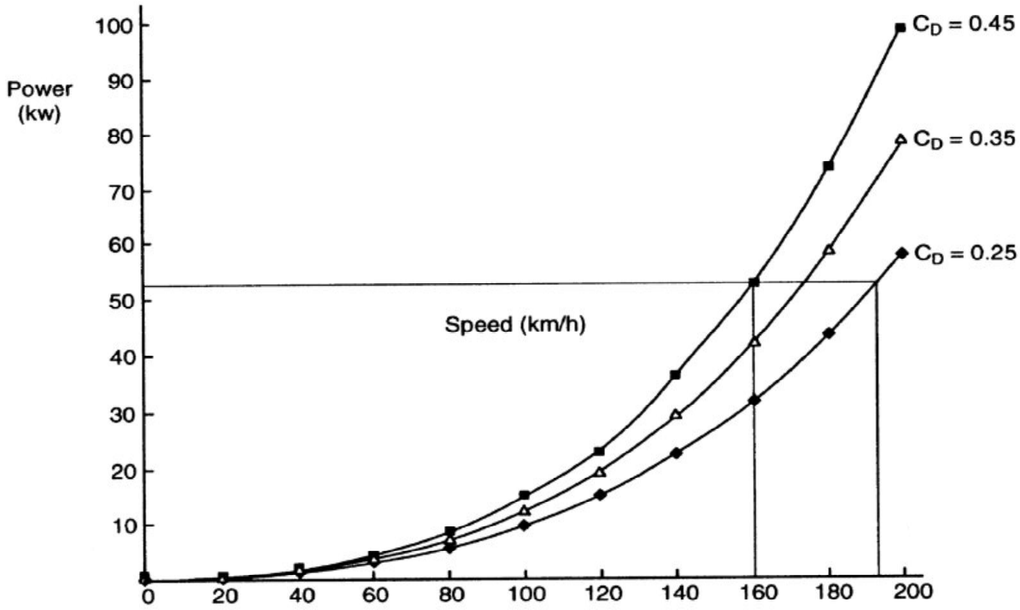


Figure 2.5 the influence of drag coefficients on velocity and spent power on road [7]

2.2.1 Relative air speed and pressure

The air gap between the horizontal air streamlines and front end bonnet (hood) and windscreen profile and the back end screen and boot (trunk) profile produces a diverging and converging air wedge, respectively. As the air moves into the diverging wedge it has to accelerate to maintain the rate of volumetric displacement.

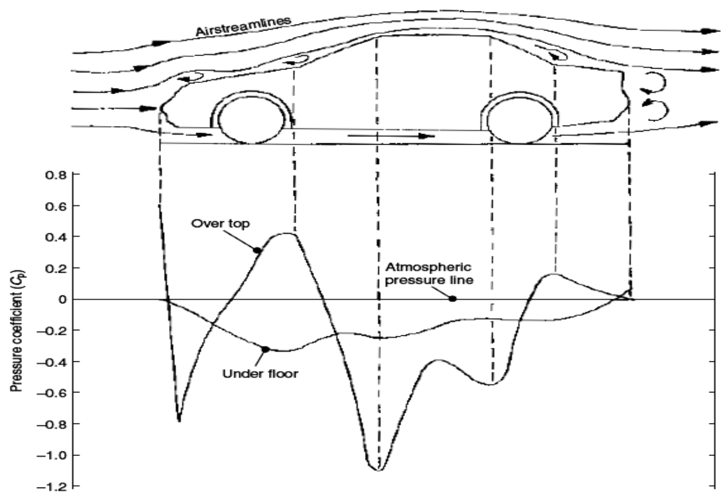


Figure 2.6 the pressure coefficient arounf the car body

2.2.2 Drag reduction

2.2.2.1 Fairing

A further step in improving truck aerodynamics was the invention of the cab spoiler. Road tests carried out by A.W. Sherwood [15] at the University of Maryland in 1953 indicated that a drag reduction was possible by using a roof fairing to guide the air from the roof of the cab to the roof of the container without flow separation.

Existing fairing have achieved good reduction in fuel consumption. An example of this is the “Nose Cone”. Its manufacturer claims a reduction in fuel consumption of 5% under ideal circumstances [14]. Similar type of devices, such as the “Air Shield“, makes the claims of the same order of saving in fuel consumption.

2.3 Wind noise

As an automobile travel down the road, the air in front of it is displaced and flows around the car, causing aerodynamics forces such as drag and lift. In addition, this airflow interacts with the surface of the car body to generate aerodynamics noise, or, less formally knowing as wind noise. Wind noise is generally a problem only at higher speeds. Wind noise can add to driver fatigue on a long highway trip [1]. The layman may expect that car with low aerodynamics drag will have low levels of winds noise. This is not found to be true in practice [8].

One explanation for this lack of correlation is that the aerodynamics drag depends largely on the exterior airflow over the rear of the car where the wake separates from the vehicle. Conversely, interior wind noise depends largely on the details of the exterior airflow around the A-pillar and windshield [1].

2.3.1 Windshield wiper

In recent times, windshield angle have increased for both styling and aerodynamics reasons. Hence, most cars do not have a region of separated flow at the base of the windshield for the wiper to hide in, and the wipers may be exposed directly to the high-speed flow over the windshield. The high speed flow through the exposed structure of the wipers can cause the high wind noise level. Hence, often the only solution available is to hide the wipers by tucking them behind the rear edge of the hood or putting them behind the some sort of flow deflector [8].

It is often that the wiper system generates unwanted noise and vibration. Noise and vibration in the wiper system can be classified into three groups, namely, squeal noise, chattering and reversal noise. Squeal noise, sometimes called squeaky noise, is a high-frequency vibration of about 1000 Hz. Chattering or beep noise, is a low frequency vibration of 100Hz or less. Reversal noise is an impact sound with a frequency of 500 Hz or less produced when the wiper reverses. These types of noise and vibration phenomenon lead to visual and audible annoyance for the driver and passengers [9].

2.4 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.

2.4.1 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. Besides 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.

2.4.2 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 [12], [13]:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k}\left(\rho u_k\right) = 0$$

(2.11)

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

(2.12)

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

(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 [12], [13].

2.4.3 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) .

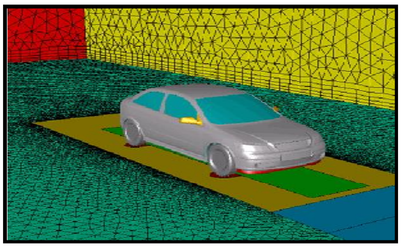


Figure 2.7 The 3D-hybrid grid [11]

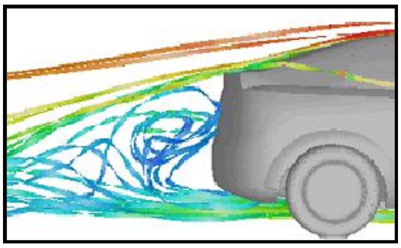


Figure 2.8 Flow analysis after a design study (pathlines) [11]

2.4.4 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].

2.4.4.1 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

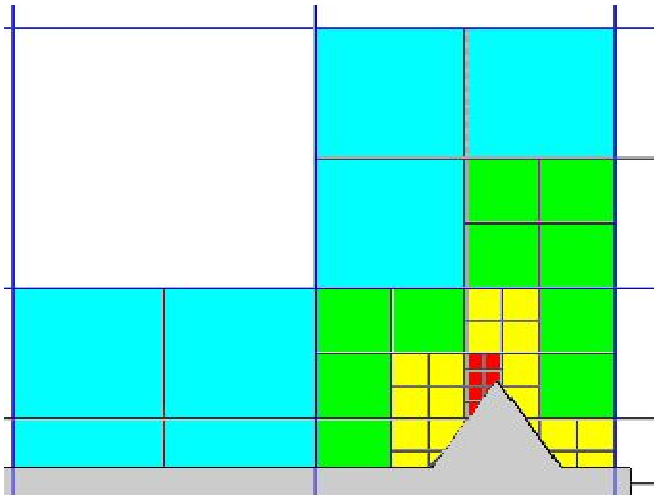


Figure 2.9 Fluid cell refinements due to the Cell Mating rule

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 reducing of drag for Proton Iswara body by adding a fairing at the front hood is taking with 4 stages, and it complete with through the flow chart in step by steps. The stage that include in this project is;

- Literature review
- Measuring
- Modeling with CAD
- Analysis with CFD

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 a terminology of works and planning show in the flow chart at Figure 3.1.

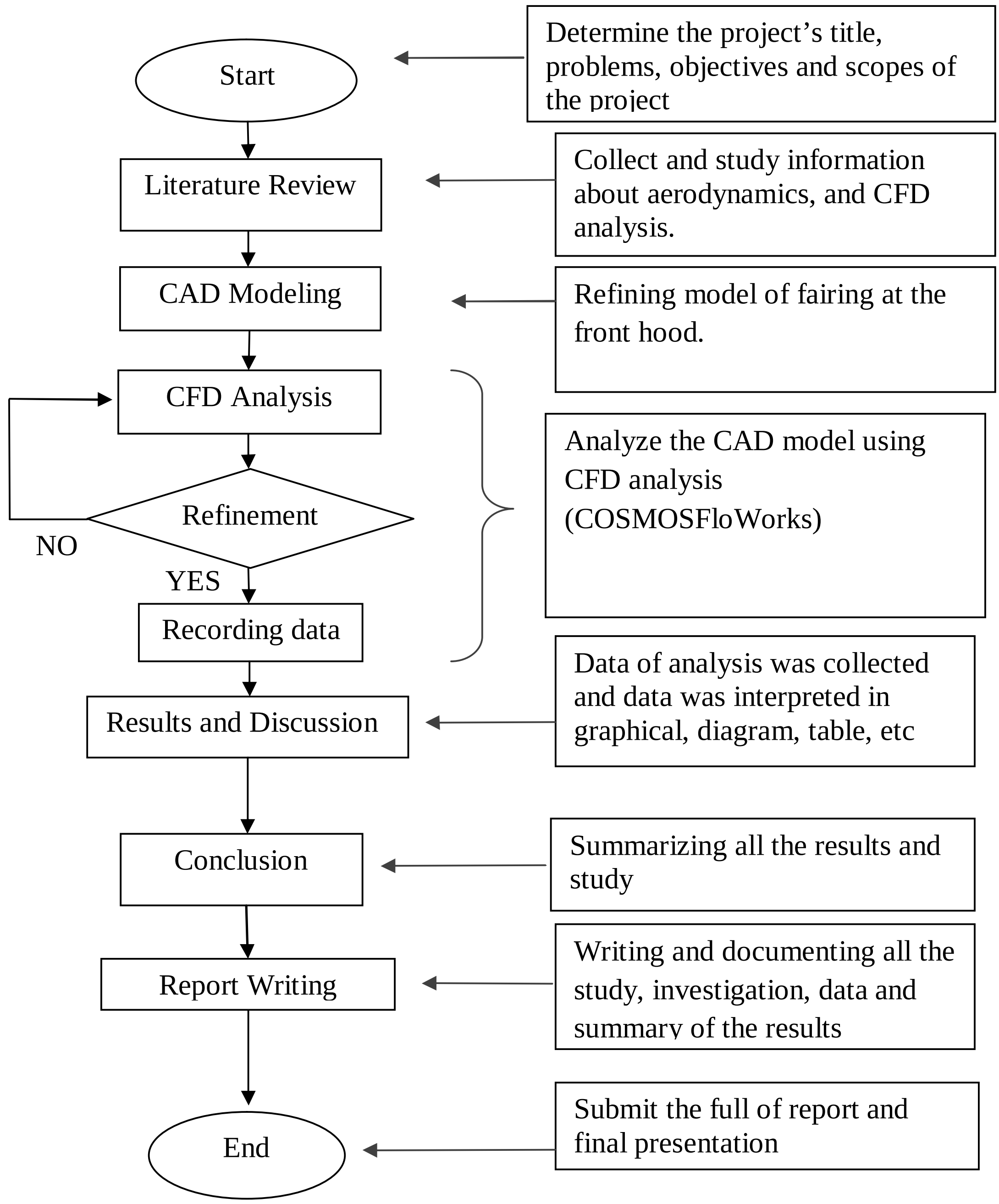


Figure 3.1 Flowchart of the Overall Methodology

3.2.1 Literature review

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 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.

All the information is collected with many source, such as;

- Article from internet
- Journal
- Discussion with supervisor

3.2.2 Measuring

The data of dimension for Proton Iswara body was collecting from the internet and measuring 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 analysis to get the appropriate or expected value of drag coefficient. The dimension of Proton Iswara Hatchback show in Figure 3.2 below:

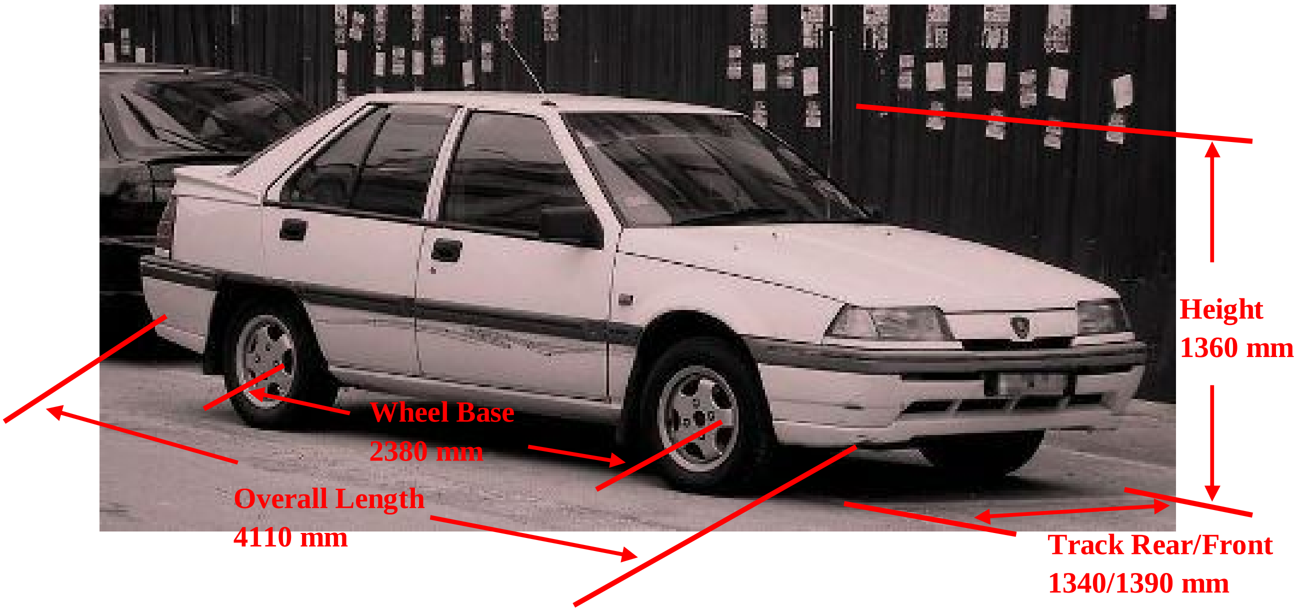


Figure 3.2 The dimension of Proton Iswara Hatchback

3.2.3 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.

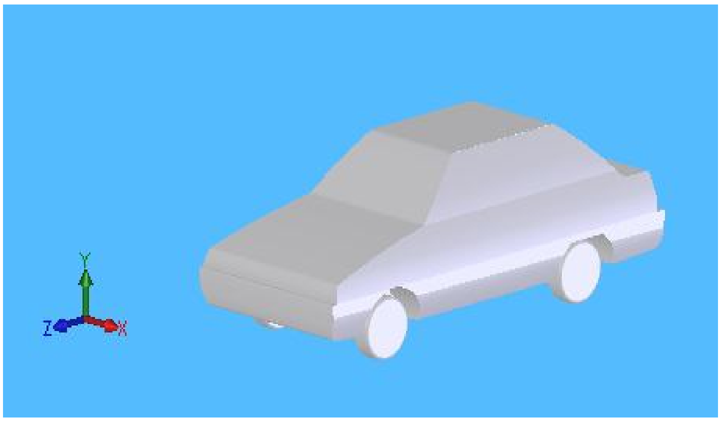


Figure 3.3 CAD model of Proton Iswara Hatchback's Body in Dimetric View

In this project, 4 model fairing is designed according to the dimension of Proton Iswara front hood. This 4 model have different of characteristics of the shape. So, it can affect the numbers of drag force at the fairing.

Model 1

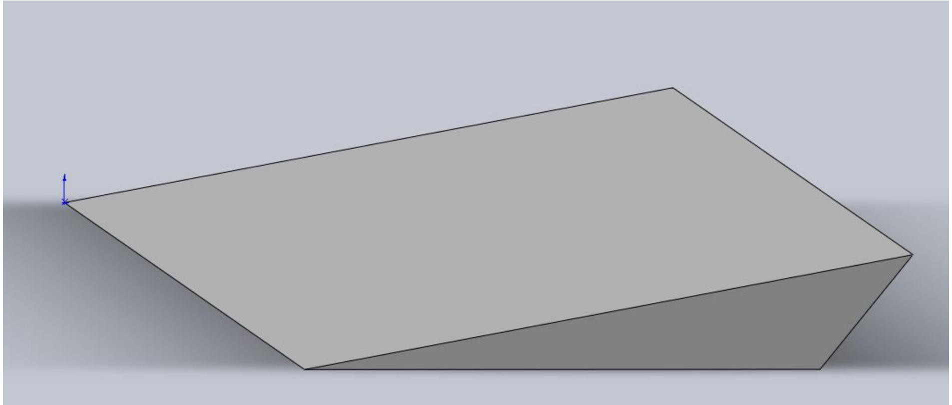


Figure 3.4 model of fairing model 1

These models have 5 straight surfaces with all sharp edge and have various angle of the top surface.

Model 2

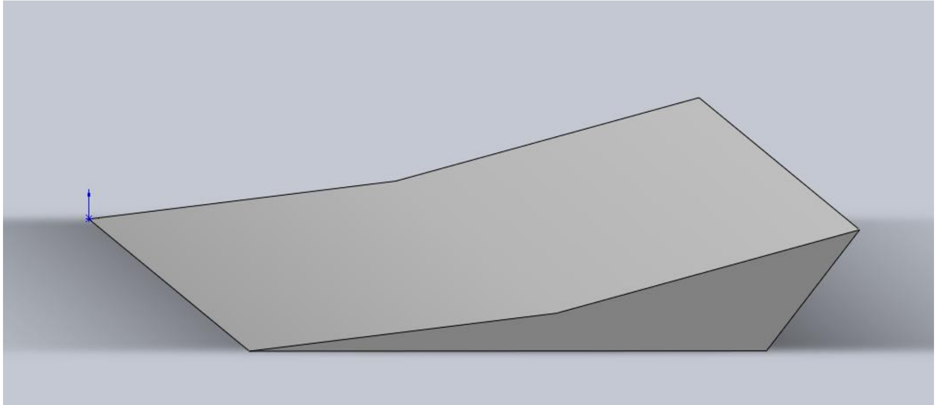


Figure 3.5 model of fairing model 2

These models have 4 straight surfaces and 1 curve surface. All the edge is sharp and has various angle of the top surface.

Model 3

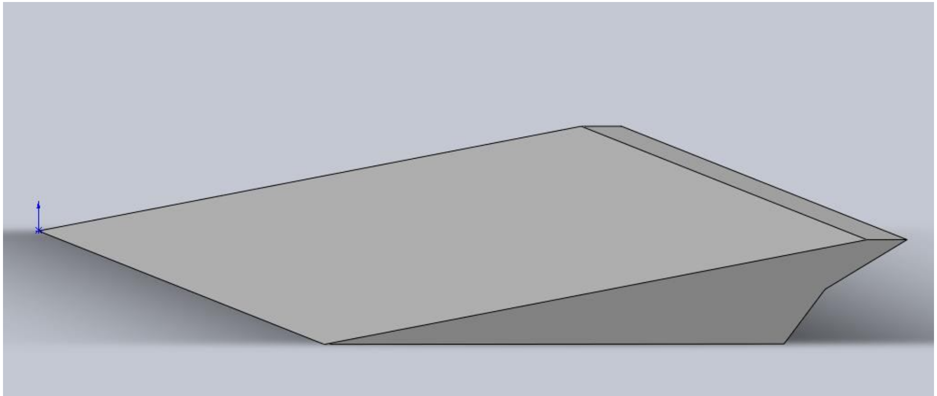


Figure 3.6 model of fairing model 3

These models have 6 straight surfaces with all sharp edge and have various angle of the top surface. At the top of the fairing, it has flat surface that parallel to the bottom of the fairing

Model 4

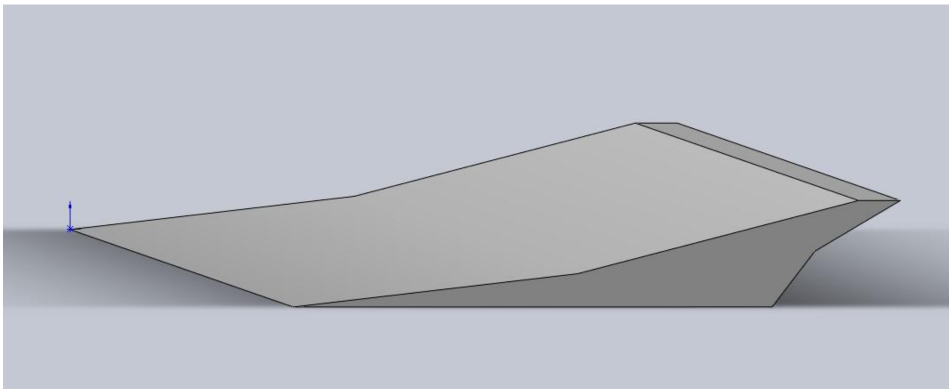


Figure 3.7 model of fairing model 4

These models have 5 straight surfaces and 1 curve surface. All the edge is sharp and has various angle of the top surface. At the top of the fairing, it has flat surface that parallel to the bottom of the fairing.

3.3 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 car speeds will analyze at 110 km/h for every simulation according to the maximum speed at the highway. The boundary condition for this analysis is external flow with adiabatic wall. Besides that, the types of flow considering are laminar and turbulent flow for CFD model analysis. Figure 3.4 below show the boundary study of CFD analysis.

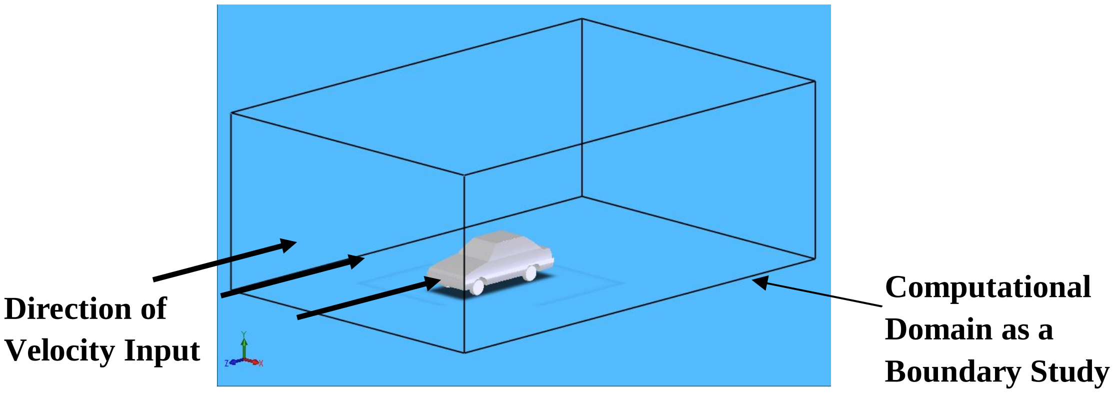


Figure 3.8 Boundary Condition of CFD analysis

3.3.1 Refinement

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

3.3.2 Local initial mesh

In this project fairing and windscreen area is been focused in the calculation. So before running the simulation, local initial mesh is been setting to focus the mesh refinement at the fairing and windscreen area. Then, the number of meshing will increase at that area and the calculation of the simulation will more precise.

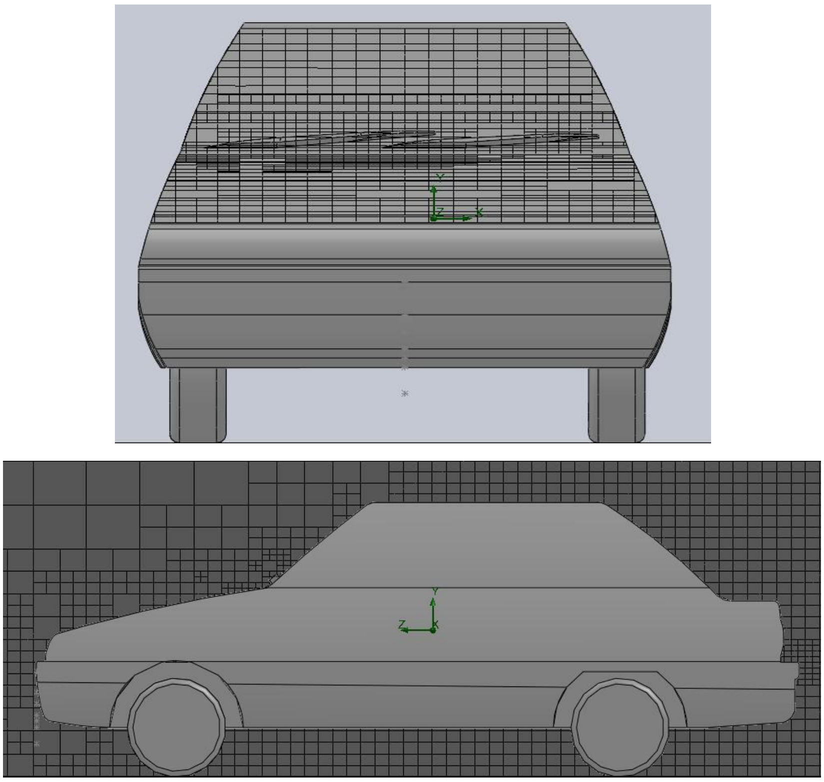


Figure 3.9 critical area of meshing at front windscreen

3.4 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.

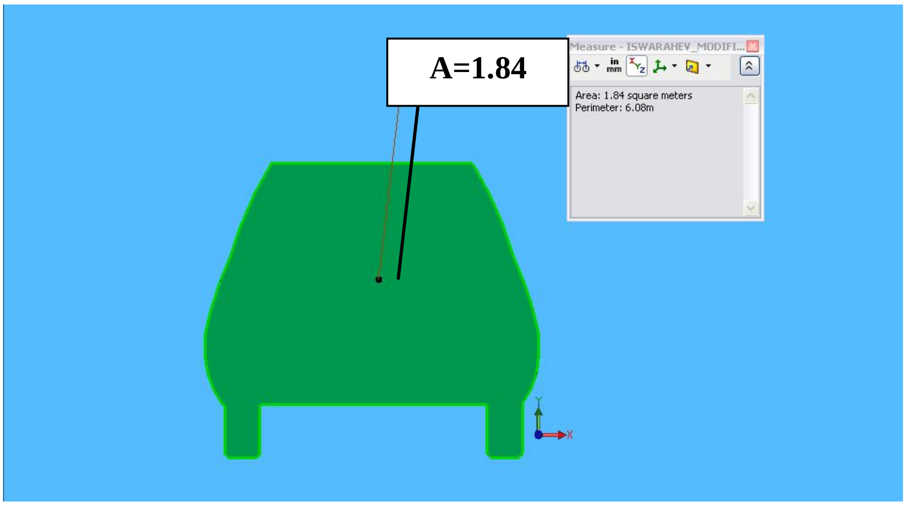


Figure 3.10 the frontal area projected of CAD model

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.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Data Collections

4.1.1 Reference Point of Flow Analysis

(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)

(e) Speed: 110km/h

4.1.2 Data of drag force with and without fairing

The drag forces data collections that were getting started from CFD analysis for proton Iswara model with and without fairing was listed in the Table 4.1 and the graph from the data has been plotted in Figure 4.1.

Table 4.1: Table of analysis for proton Iswara model with and without fairing

	Unfairing	Model 1	Model 2	Model 3	Model 4
Drag force (N)	394.054	388.477	390.783	388.540	390.850

The data from table 4.1 is an average of the 3 times of running of simulation in CFD. The entire model is in 10.5^{o} of angle taken randomly. To get the good result average value is collected. All the data from Table 4.1 was plotted into the graph of drag forces, *D* against the type of fairing. Then, the graph series was shown in Figure 4.1 below.

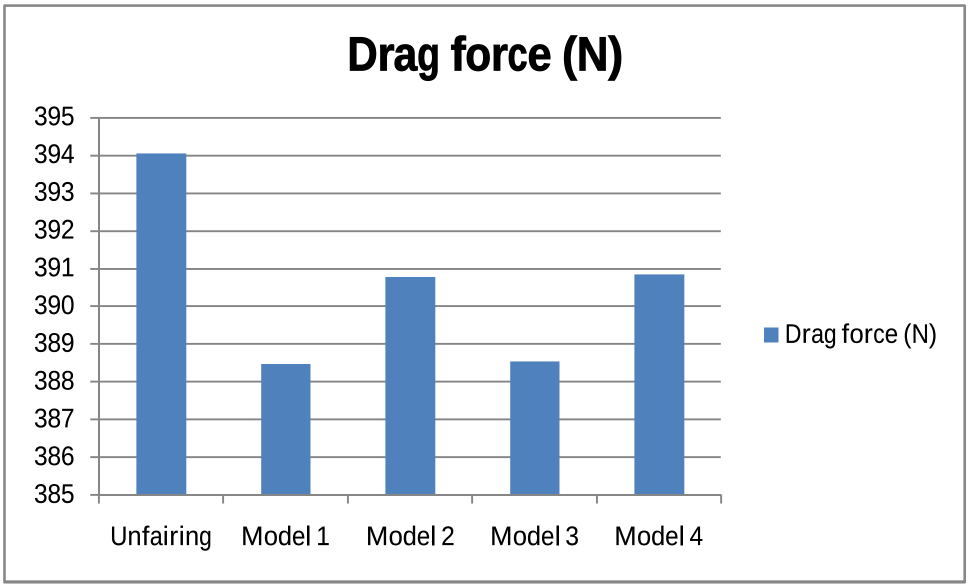


Figure 4.1 Graph drag forces, *D* against type of fairing

From the Figure 4.1 above shown the proton iswara without fairing have higher of drag force with 394.054 when 110 km/h air speed is selected in the simulation. It is because the airflow over the car body is disturbed by the wiper that exposed to the airflow at the front windscreen. In the first simulation, all model is randomly taken with angle of 10.5 as shown in table 4.1. Fairing model 1 and model 3 is the best model that can reduce drag force from 394.054 to 388.477 by model 1 and 388.540 by model 3. So that, this two models is been selected to proceed in the next step of simulation.

4.1.3 Value of the drag force in the different angle

The drag forces data collections that were getting from CFD analysis for proton Iswara model with fairing model 1 and model 3 with different angle of model was listed in the Table 4.2. All the value is average of 3 times of running simulation for one model.

Table 4.2: Table of various angle and drag forces

Angle	Model 1	Model 3
8.5°	388.627	388.725
9.0°	383.860	383.860
9.5°	383.724	383.757
10.0°	383.628	383.633
10.5°	388.477	388.540
11.0°	388.684	388.730
11.5°	388.693	388.763

After past the first step of model selection simulation, the model that been selected is been running again in the CFD to be analyze with different angle of the model. In this steps fairing model 1 and model 3 is been chosen to analyze with 7 angle of model. From table 4.2, fairing model 1 is the best model that can reduce more drag force than the fairing model 3.

4.1.4 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 4.2**,** 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 angle of analysis between 8.5° to 11.5° then plotted into a graph.

4.2.1.1 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 fairing model 1 angle 8.5° ;

$$Cd = \frac{388.627}{(0.5)(1.225)(30.56)^2(1.84)}$$

$$= 0.369$$

2. Drag Coefficient for fairing model 2 angle 10.5°;

$$Cd = \frac{383.628}{(0.5)(1.225)(30.56)^2(1.84)}$$

$$=0.364$$

4.2.2 Data of Drag Coeffiecient for Various angle of model 1

Since the coefficient has been calculated for every angle, the drag coefficient and angle was listed and plotted in Table 4.3 and Figure 4.2.

Table 4.3: Table of various angle and drag coefficients

Angle of fairing model 1	Coefficient of drag
8.5°	0.369
11.5°	0.369
9.0°	0.365
9.5°	0.365
10.0°	0.364
10.5°	0.369
11.0°	0.369

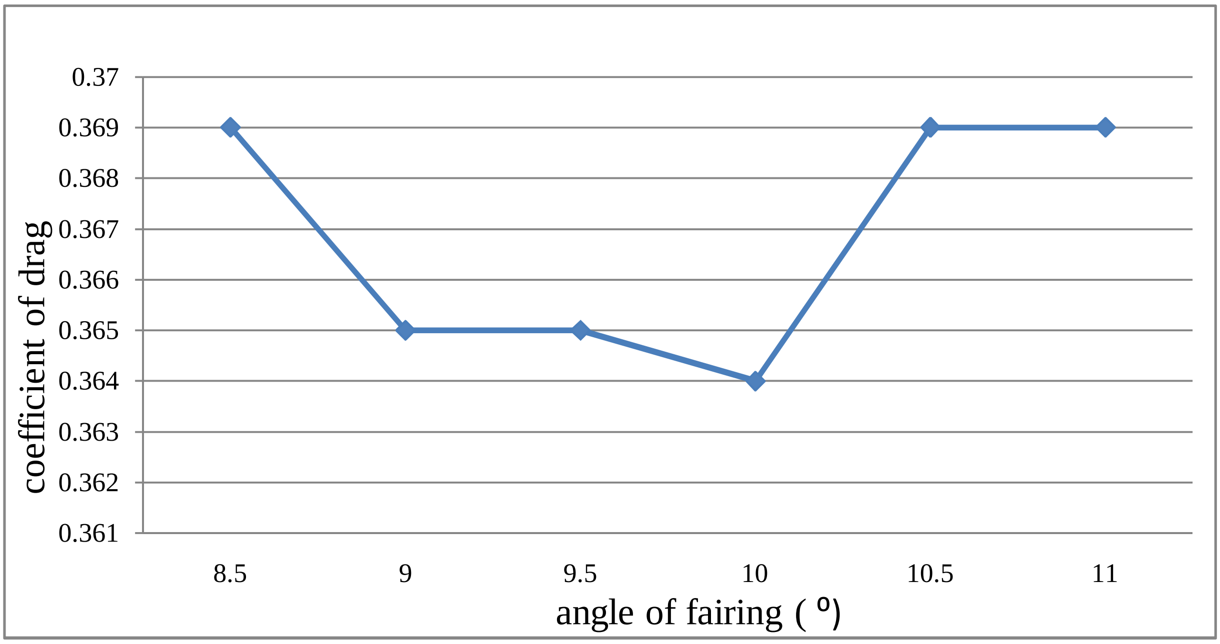


Figure 4.2 Graph of Drag Coefficient, C_D against angle of model

The Figure 4.2 shown that the value of drag coefficient is decreasing when the angle of model is increased from 0.369 to 0.364 but at the value of 10.5⁰, the value of drag coefficient will increasing to 0.369 back. It is because at this angle, the fairing did not diverge the airflow but it block the airflow. So the simulation is stop at this angle.

The best fairing angle that can reduce effectively of drag coefficient is the fairing angle 10.0⁰ according to the graph 4.2.

4.2.3 Comparision betweenmodel proton Iswara unfairing and fairing model 1

In the CFD result, surface of of the windscreen is been analyze due to the colour of the pressure as shown in Figure 4.1 below. This figure shown that the different of proton Iswara without fairing and when adding fairing model 1 with angle 10.0⁰.

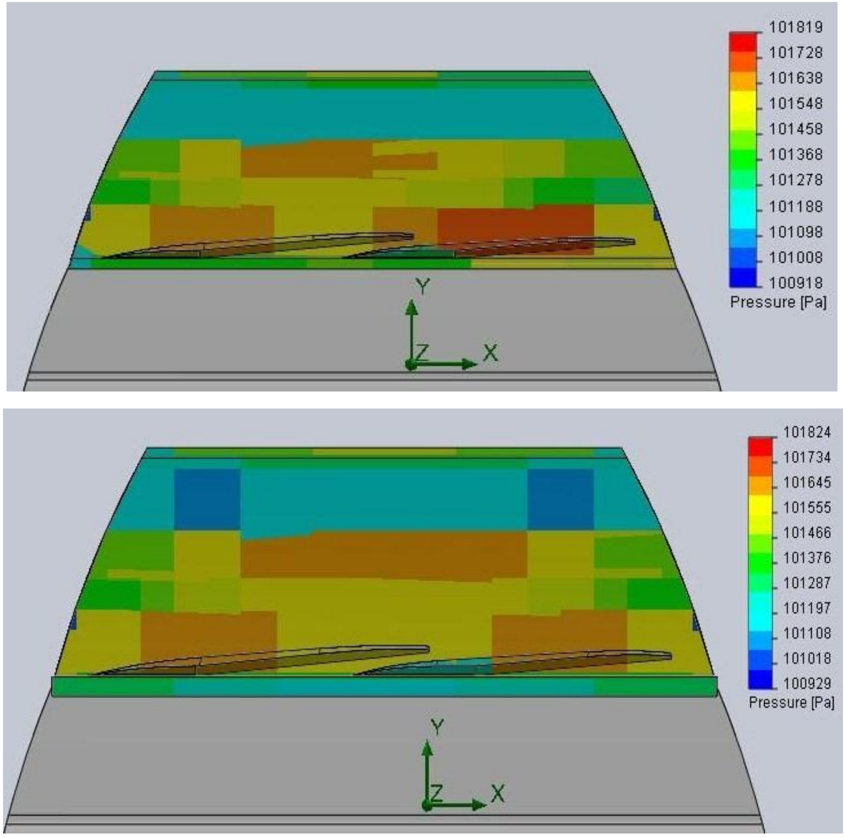


Figure 4.3 Different surface plot of pressure

From the Figure 4.1, surface of proton Iswara without fairing is high of presure with more red and orange colour. In the result shows that, when the colour goes to red colours, the pressure also increase due to the colour range. So, when the proton Iswara is adding fairing model 1 with angle 10.0^{o} the pressure surface plot colour is decreasing and more light colour appear at the front windscreen of proton Iswara. That mean, the pressure at the front windscreen is decreasing when fairing model 1 with angle 10.0^{o} is adding at the front hood of the car.

4.2.4 Comparison of Data of Drag Coeffiecient

Model 1 with angle 10.0^{o} is the best model that been choosen from the reducing of drag coefficient. Table 4.4 shows that the comparison of the drag coefficient between model proton iswara without fairing and with the best model from the simulation.

Table 4.4 comparison of the drag coefficient

Model	Unfairing	Model 1 (10.0°)
Drag coefficient	0.374	0.364

4.2.5 percentage of reducing

$$\text{percentage of drag reducing} = \frac{0.374-0.364}{0.374} \times 100$$

$$= 2.67\ \%$$

From the simulation, value of drag coefficient that can be reducing is 2.67% with adding fairing model 1 with angle of 10.0° to model proton Iswara at front hood.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

As a conclusion from this project, learning process base on the reducing of drag achieved using CFD analysis specifically at front windscreen as well as the study of aerodynamics on the vehicle especially passenger car and HEV model. Aerodynamics drag for Proton Iswara's body is 0.374 of drag coefficient at velocity 110km/h analysis. The analysis show aerodynamics drags in term of drag coefficient were proportionally increase to the drag force for Proton Iswara's body that has frontal area is 1.84 m^2.

Since the aerodynamics drag were proportionally increase to the drag force, so the area that produce drag force need to be develop for reduce the pressure that occur by reverse flow or turbulent flow. At the front windscreen the airflow is disturbed by the exposed structures such as wiper. By adding the fairing at the front hood will improve the aerodynamic drag and make airflow smooth over the car body. For passenger car proton Iswara fairing model 1 with angle of 10.0^{o} is the best model from 4 models that have been simulate in CFD. Reducing of 0.01 of drag coefficient with 2.67% percentage of reducing drag from unfairing model will improve the aerodynamics of car.

The surface plots of pressure were shown the rationalization of aerodynamics drag analysis as a visualization analysis. The result shown by adding fairing at the front hood will decrease the pressure with change the color at the front windscreen. By reducing the pressure at the windscreen will make the wind noise also can be reduced. Then, the airflow will smooth and the performance of the car greater than before.

5.2 Further Study Recommendation

In view of the fact that, this project represents the first known insight of modeling fairing hood for cars 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 fairing model especially in developing of aerodynamics performance of cars. 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] P.W. Bearman, Review-bluff body flow applicable to vehicle aerodynamics, Trans. ASME J. Fluids Eng. 102 (1980) 265 274.

[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] I.M.Awang, C.Y.Leong, A.R.Abu-Bakar, Abd-Ghani,R.Abd-Rahman and M.Z. Md- Zain.*Modelling and Simulation of automotive wiper noise and vibrations using Finite Element Method,* University Teknologi Malaysia report.2008.

[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.

[14] Van Setters,R., Service manager, Intercontinental Truck Body Inc., B.C., Canada,1992.

[15] Shewood, A.W., ”Wind Tunnel Test of Trailmobile Trailers”, University of Maryland Wind Tunnel Report Nr. 35, 1953

APPENDIX A

Project Gantt chart 1

Activities/ Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Title of project briefing by supervisor	■															
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						■	■	■	■	■	■	■				
Determine best method of methodology									■	■						
Modeling the fairing model										■	■					
Analysis for preliminary result											■	■	■			
Submit proposal and draft of report														■		
Presentation of Proposal															■	■

APPENDIX B

Project Gantt chart 2

Activities/ Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Title of project briefing by supervisor																
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																
Determine best method of methodology																
Modeling the fairing model																
Analysis for preliminary result																
Submit proposal and draft of report																
Presentation of Proposal																

DESIGN AND ANALYSIS FRONT HOOD FAIRING FOR CARS

M.Mahazis

Faculty of Mechanical Engineering, Universiti Malaysia Pahang,
26300 UMP, Kuantan, Pahang, Malaysia, Phone: +609-5492223 Fax: +609-5492244.
Email: mahasaufeezis@gmail.com

ABSTRACT

The performance, handling, safety and comfort of an automobile are significantly affected by its aerodynamics properties. By improving the aerodynamics will increase the performance of the cars. The paper presents a model of the front hood fairing for cars that can reduce the drag coefficient of the car, model that been used is proton Iswara. The overall duty is modeling and analyzes to get the best model that can reduce the coefficient of drag of the proton Iswara because of the high drag force at front windscreen. It happen when the airflow at the windscreen is disturbed by car wiper during high speed. CAD software is used to draw all the model of fairing include proton Iswara and simulate in the CFD software. The result shown by adding fairing at the front hood will decrease the coefficient of drag, pressure and wind noise also can be reduced. Then, the airflow will smooth and the performance of the car greater than before.

Keywords: Coefficient of drag, CFD, drags reducing, automobile aerodynamics

INTRODUCTION

Defecting drag was the first major focus of automotive aerodynamics, beginning in 1960’s. Low drag is important for fuel economy and low emissions. Other aspects of vehicle aerodynamics are no less important for quality of automobiles such as directional stability, wind noise, cooling of engine, ventilating and air conditional these all depend on flow field around and through vehicle.

Nowadays, automotive designer rely on aerodynamics principle to create improvement in the power and handling of vehicle at high speeds. Passenger cars have become more shapely over the years as manufacturer discovered how streamlining can increase fuel efficiency, allowing a car to travel at the same speed using less horsepower. These designs reduce air resistance, or aerodynamics drag. Low drag coefficient make the vehicle enable to move easily through the surrounding viscous air with minimum of resistance. As an increasing of drag, the more power of car to do work than reducing the power train efficiency.

.

LITERATURE REVIEW

A further step in improving truck aerodynamics was the invention of the cab spoiler. Road tests carried out by A.W. Sherwood [5] at the University of Maryland in 1953 indicated that a drag reduction was possible by using a roof fairing to guide the air from the roof of the cab to the roof of the container without flow separation.

Existing fairing have achieved good reduction in fuel consumption. An example of this is the “Nose Cone”. Its manufacturer claims a reduction in fuel consumption of 5% under ideal circumstances [4]. Similar type of devices, such as the “Air Shield“, makes the claims of the same order of saving in fuel consumption.

As an automobile travel down the road, the air in front of it is displaced and flows around the car, causing aerodynamics forces such as drag and lift. In addition, this airflow interacts with the surface of the car body to generate aerodynamics noise, or, less formally knowing as wind noise. Wind noise is generally a problem only at higher speeds. Wind noise can add to driver fatigue on a long highway trip [1]. The layman may expect that car with low aerodynamics drag will have low levels of winds noise. This is not found to be true in practice [2].

The high speed flow through the exposed structure of the wipers can cause the high wind noise level. Hence, often the only solution available is to hide the wipers by tucking them behind the rear edge of the hood or putting them behind the some sort of flow deflector [2]. It is often that the wiper system generates unwanted noise and vibration. These types of noise and vibration phenomenon lead to visual and audible annoyance for the driver and passengers [3].

METHDOLOGY

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.

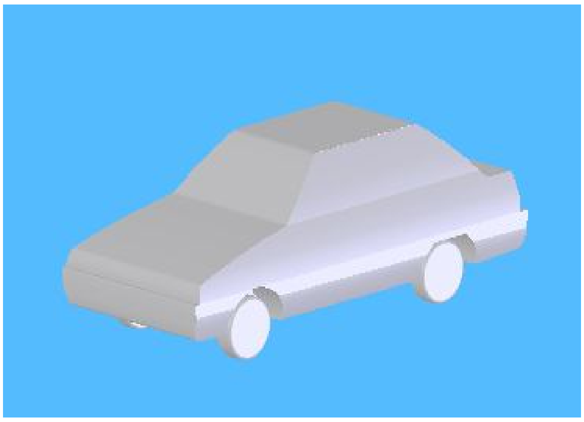
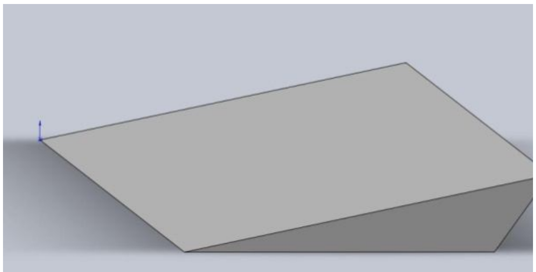
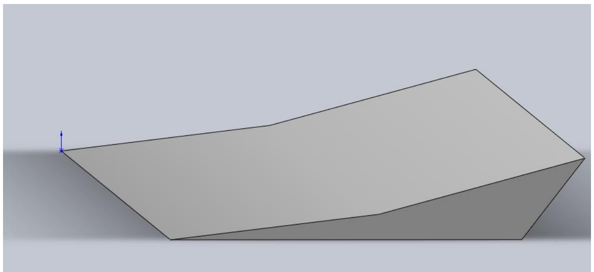


Figure 3.3 CAD model of Proton Iswara Hatchback's Body in Dimetric View

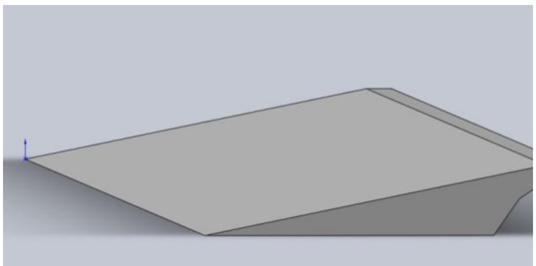
Then , 4 model fairing is designed according to the dimension of Proton Iswara front hood. This 4 model have different of characteristics of the shape. So, it can affect the numbers of drag force at the fairing. All these fairing are assembled at the front hood near to the wiper at the windscreen.



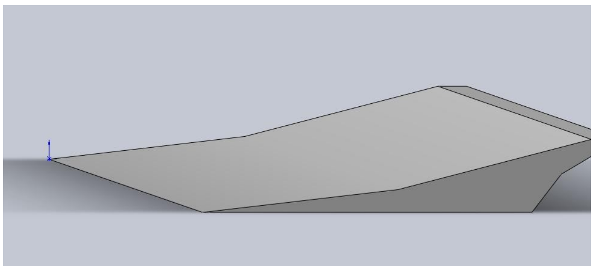
Model 1



Model 2



Model 3



Model 4

Since the car body and the fairing 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 car speeds will analyze at 110 km/h for every simulation according to the maximum speed at the highway.

RESULTS AND DISCUSSION

Table 4.1: Table of analysis for proton Iswara model with and without fairing

	Unfairing	Model 1	Model 2	Model 3	Model 4
Drag force (N)	394.054	388.477	390.783	388.540	390.850

The data from table 4.1 is an average of the 3 times of running of simulation in CFD. The entire model is in 10.5^{o} of angle taken randomly. To get the good result average value is collected. All the data from Table 4.1 was plotted into the graph of drag forces, *D* against the type of fairing. Then, the graph series was shown in Figure 4.1 below.

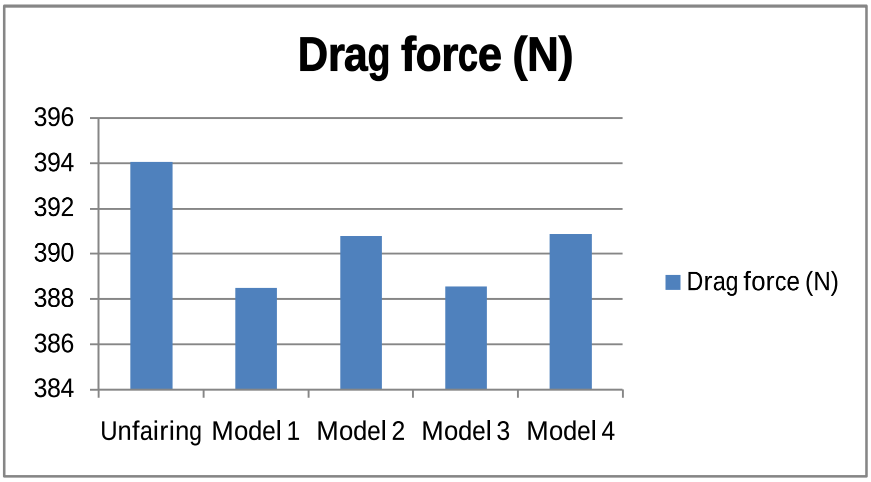


Figure 4.1 Graph drag forces, *D* against type of fairing

From the Figure 4.1 above shown the proton iswara without fairing have higher of drag force with 394.054 N. In the first simulation, all model is randomly taken with angle of 10.5 as shown in table 4.1. Fairing model 1 and model 3 is the best model that can reduce drag force from 394.054 to 388.477 by model 1 and 388.540 by model 3. So that, this two models is been selected to proceed in the next step of simulation.

After past the first step of model selection simulation, the model that been selected is been running again in the CFD to be analyze with different angle of the model. In this steps fairing model 1 and model 3 is been chosen to analyze with 7 angle of model. From table 4.2, fairing model 1 is the best model that can reduce more drag force than the fairing model 3.

Table 4.2: Table of various angle and drag forces

Angle	Model 1	Model 3
8.5□	388.627	388.725
9.0□	383.860	383.860
9.5□	383.724	383.757
10.0□	383.628	383.633
10.5□	388.477	388.540
11.0□	388.684	388.730
11.5□	388.693	388.763

Since the coefficient has been calculated for every angle, the drag coefficient and angle was listed and plotted in Table 4.3 and Figure 4.2.

Table 4.3: Table of various angle and drag coefficients

Angle of fairing model 1	Coefficient of drag
8.5□	0.369
9.0□	0.365
9.5□	0.365
10.0□	0.364
10.5□	0.369

11.0□	0.369
11.5□	0.369

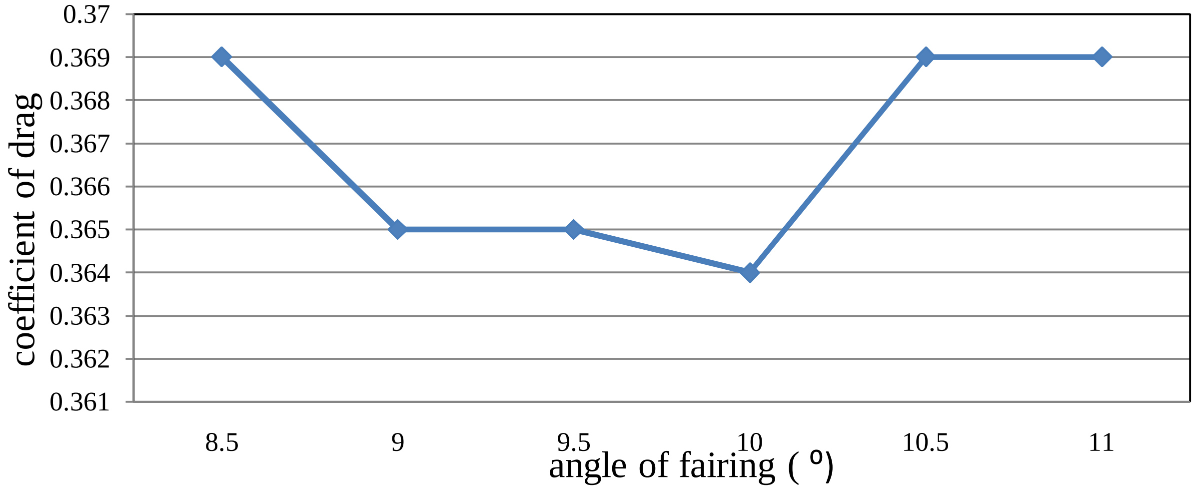


Figure 4.2 Graph of Drag Coefficient, C_D against angle of model

The Figure4.2 shown that the value of drag coefficient is decreasing when the angle of model is increased from 0.369 to 0.364 but at the value of 10.5º, the value of drag coefficient will increasing to 0.369 back. It is because at this angle, the fairing did not diverge the airflow but it block the airflow. So the simulation is stop at this angle.

The best fairing angle that can reduce effectively of drag coefficient is the fairing angle 10.0º according to the Figure 4.2.

In the CFD result, surface of of the windscreen is been analyze due to the colour of the pressure as shown in Figure 4.1 below. This figure shown that the different of proton Iswara without fairing and when adding fairing model 1 with angle 10.0º.

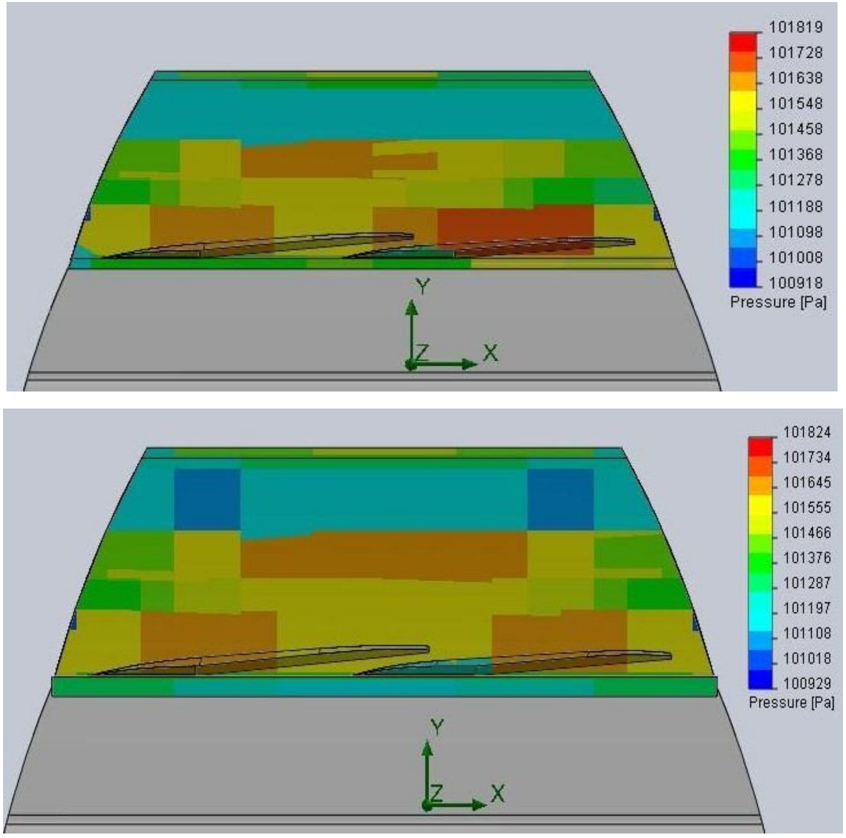


Figure 4.3 Different surface plot of pressure

From the Figure 4.1, surface of proton Iswara without fairing is high of presure with more red and orange colour. . So, when the proton Iswara is adding fairing model 1 with angle 10.0^{o} the pressure surface plot colour is decreasing and more light colour appear at the front windscreen. That mean, the pressure at the front windscreen is decreasing when fairing model 1 with angle 10.0^{o} is adding at the front hood of the car.

Table 4.4 comparison of the drag coefficient

Model	Unfairing	Model 1 (10.0□)
Drag coefficient	0.374	0.364

$$\text{percentage of drag reducing} = \frac{0.374-0.364}{0.374} \times 100$$

$$= 2.67\ \%$$

From the simulation, value of drag coefficient that can be reducing is 2.67% with adding fairing model 1 with angle of 10.0^{o} to model proton Iswara at front hood.

CONCLUSION

As a conclusion from this project, aerodynamics drag for Proton Iswara's body is 0.374 of drag coefficient at velocity 110km/h analysis. By adding the fairing at the front hood will improve the aerodynamic drag and make airflow smooth over the car body.

For passenger car proton Iswara fairing model 1 with angle of 10.0^{o} is the best model from 4 models that have been simulate in CFD. Reducing of 0.01 of drag coefficient with 2.67% percentage of reducing drag from unfairing model will improve the aerodynamics of car.

By reducing the pressure at the windscreen will make the wind noise also can be reduced. Then, the airflow will smooth and the performance of the car greater than before.

REFERENCES

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

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

[3] I.M.Awang, C.Y.Leong, A.R.Abu-Bakar, Abd-Ghani,R.Abd-Rahman and M.Z. Md- Zain.*Modelling and Simulation of automotive wiper noise and vibrations using Finite Element Method,* University Teknologi Malaysia report.2008

[4] Van Setters,R., Service manager, Intercontinental Truck Body Inc., B.C., Canada,1992.

[5] Shewood, A.W., ”Wind Tunnel Test of Trailmobile Trailers”, University of Maryland Wind Tunnel Report Nr. 35, 1953