SIMULATION AND ANALYSIS DRAG AND LIFT COEFFICIENT BETWEEN SEDAN AND HATCHBACK CAR

MOHD KHALIL AZINGAH BIN SALLEH

Thesis submitted in fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

SUPERVISOR'S DECLARATION

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

Signature :
Name of Supervisor : Dr Yusnita Rahayu
Date : 20 November 2009

STUDENT'S DECLARATION

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

Signature :
Name : Mohd Khalil Azingah bin Salleh
ID Number : MH 07006
Date : 25 November 2009

Dedicated to my beloved parents
Mr. Salleh Bin Ahmad
Mrs. Fatimah Binti Ahmad
And
All my sisters and brothers

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Dr. Yusnita Rahayu for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, and his strong conviction for science. I appreciate his consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naive mistakes, and his commitment to my future career. I also would like to express very special thanks to Dean Faculty of Mechanical Engineering, Associate Prof. Dr. Rosli Bin Abu Bakar for his advice and suggestion.
I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

I would also like to acknowledge with much appreciation the crucial role of the staff in Mechanical Laboratory, who gave a permission to use the computer laboratory to design and run the analysis and also the plastic laboratory to make mould measurement and to use all the necessary tools in the laboratory. Special thanks too goes to Mr. Azizuddin Bin Abd Aziz against as the Final Year Coordinator, who has give some advice and share his knowledge on this final year project.

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

ABSTRACT

This thesis presents research about the difference coefficients barriers and lift coefficients for two basic design types of sedan and hatchback cars. The objective of this thesis is to identify drag and lift coefficient.. Process simulation and analysis for both the model design was conducted with computer-aided drawing software and analyzed using software COSMOSFloworks. From the results of coefficients can be concluded that barriers and lift coefficients for both types of design is different. Restriction coefficient and lift coefficient for the hatchback design is much lower than the sedan design. This means that the hatchback design is more efficient and aerodynamics value is higher. This assessment and differences between the two designs is very meaningful for determining the design of more efficient designs in the car now. Results also able to enhance the security features on the car at once can reduce fuel consumption used. Results are also able to improve the design of the car in early development in the future.

ABSTRAK

Tesis ini membentangkan penyelidikan berkenaan perbezaan pekali hambatan dan pekali angkat bagi dua jenis rekabentuk asas kereta iaitu sedan dan hatchback. Objektif tesis ini ialah mengenalpasti pekali hambatan dan pekali angkat.. Proses simulasi dan analisis bagi kedua-dua model rekabentuk ini dijalankan dengan perisian lukisan bantuan komputer dan dianalisis menggunakan perisian COSMOSFloworks. Dari hasil keputusan dapat disimpulkan bahawa pekali hambatan dan pekali angkat bagi kedua-dua jenis rekabentuk ini adalah berbeza. Pekali hambatan dan pekali angkat bagi rekabentuk hatchback adalah lebih rendah berbanding rekabentuk sedan. Ini bermakna rekabentuk hatchback adalah lebih efisyen dan nilai aerodinamiknya lebih tinngi.Keputusan penilaian dan perbezaan antara kedua-dua rekabentuk ini amat bermakna bagi menentukan rekaan yang lebih efisyen dalam rekabentuk kereta sekarang. Keputusan juga berupaya meningkatkan lagi ciri-ciri keselamatan pada kereta sekaligus dapat mengurangkan penggunaan bahan api yang digunakan. Keputusan ini juga berupaya memperbaiki rekabentuk kereta tersebut di awal pembangunan pada masa hadapan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

C_D	Drag Coefficient
C_L	Lift Coefficient
F_L	Lift Force
F_D	Drag Force
Cp	Pressure Coefficient
Df	Friction Drag
b	Width
l	Length

LIST OF ABBREVIATIONS

CAD	Computer-aided drafting
CAE	Computer-aided engineering
FE	Finite element
RANS	Reynolds-averaged Navier –Stokes equation
DNS	Direct numerical simulation

CHAPTER 1

INTRODUCTION

1.1 Project Background

The importance of aerodynamics to several type car bodies model needs a development of drag and lift estimation to know how much the car performance on the road against air resistance beside to improve the stability, reducing noise and fuel consumption.

1.2 Problem Statement

Drag and lift will cause many problems on the performance of car model like instability, noise and fuel consumption. Thus, in this project the CAD models of sedan and hatchback bodies was simulated and analyze of their aerodynamics especially on the drag and lift 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 determine average Drag Coefficient, C_D of sedan and hatchback car
2. To determine average Lift Coefficient, C_L of sedan and hatchback car

1.4 Project Scopes

1. Study of aerodynamics on road vehicle
2. Analyze the project with CFD for various car speeds
3. Determine where the regions of separated and reversed flow of our car design by using Cosmos Flow simulation software.
4. Determine how the trajectory of the flow through our car design is.

1.5 Summary

For the end of simulation and analysis, the objectives for this experiment will achieved by follow the scopes of project.

CHAPTER 2

FUNDAMENTAL OF AERODYNAMICS

2.1 Theory of Aerodynamics

In this section, the fundamental of aerodynamics is 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 are studied.

2.1.1 Bernoulli's Equation

Aerodynamics play main role to defined road vehicle's characteristic like handling, noise, performance and fuel economy. 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;

- Viscous effects are assumed negligible
- The flow is assumed to be steady
- The flow is assumed to be incompressible
- The equation is applicable along streamline

$$p + \frac{1}{2}\rho v^2 = constant \qquad (2.1)$$

From equation (2.1) shows the increasing of velocity will case the decrease in static pressure and vice 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 ;

Static pressure + Dynamic Pressure = Stagnation Pressure.

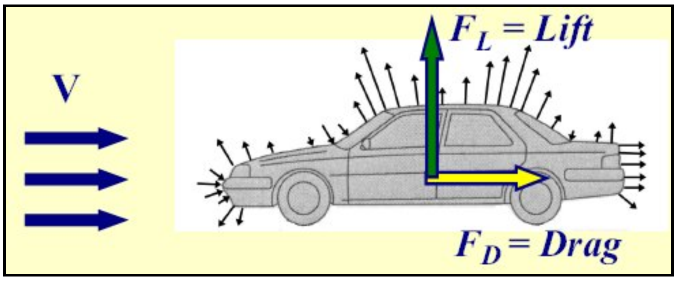


Figure 2.1 Drag and lift force due to pressure from velocity distribution [Adam Heberly, 1999]

2.1.2 Pressure, Lift and Drag Coefficient

Drag can generate by two main perspectives: [Clancy, L.J. (1975)]

- From the vehicles (body)
- 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.

2.1.2.1 Pressure Coefficient

The equation for coefficient of pressure (*Cp*) due to dynamic pressure can derive as: [John D.Anderson. (2007)]

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

The equation of dynamic pressure defined as; [John D.Anderson. (2007)]

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

In term of local velocity, the pressure coefficient (only valid for incompressible flow) can derive as ; [John D.Anderson. (2007)]

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

The form of equation (2.4) is from the relation equation (2.2) and equation (2.5) as shown below ; [John D.Anderson. (2007)]

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

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*.
[Clancy, L.J. (1975)]

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: [John D.Anderson. (2007)]

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

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; [John D.Anderson. (2007)]

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

(2.7)

Besides that, the drag coefficient, C_{df} can derive from friction drag, D_f, on a flat plate as: [John D.Anderson. (2007)]

$$C_{d\,f} = \frac{D_f}{\frac{1}{2}\rho v^2 b.l} \qquad (2.8)$$

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: [John D.Anderson. (2007)]

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

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).

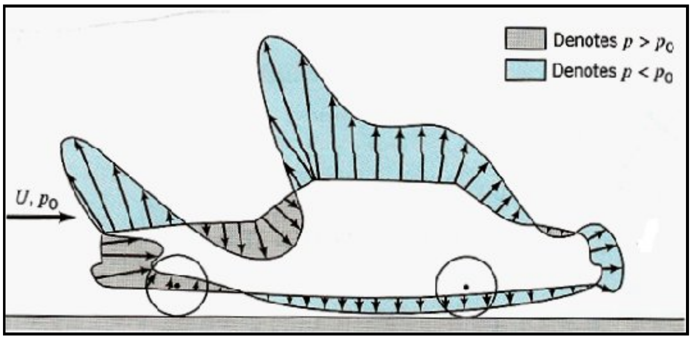


Figure 2.2: Pressure distributions on the surface of an automobile
[J. Katz (2006)]

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 .

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 variation of boundary layer thickness is shown in Figure 2.3:

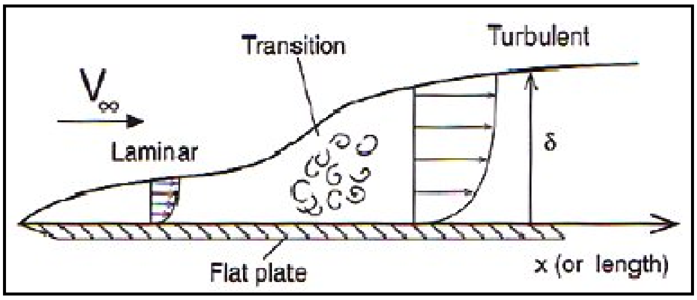


Figure 2.3: Variation of boundary layer thickness along flat plate
[J. Katz (2006)]

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.

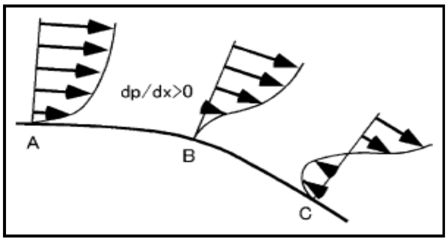


Figure 2.4: Schematic of velocity profile around a rear end.
[W. H. Hucho (1998)]

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). [W. H. Hucho (1998)]

2.1.5 Shape Dependence

As discussed before the drag was depend on the shape of vehicle. 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, $0<C_D<1$ and $v< v_\infty$, the drag, $D=\frac{1}{2}\,\rho v_\infty^2 C_D.A$ that was defined from equation (2.7). Thus from the definition we can conclude that area of frontal area projected of composite body as: [John D.Anderson. (2007)]

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

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

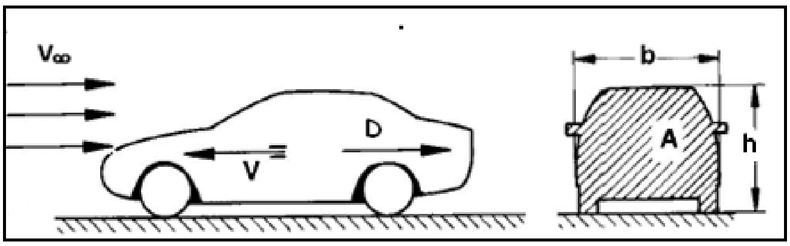


Figure 2.5: The relationship frontal area on vehicle body against the normal flow of velocity. [J. Katz (2006)

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. The influences of various drag coefficients on velocity shown in Figure 2.6.

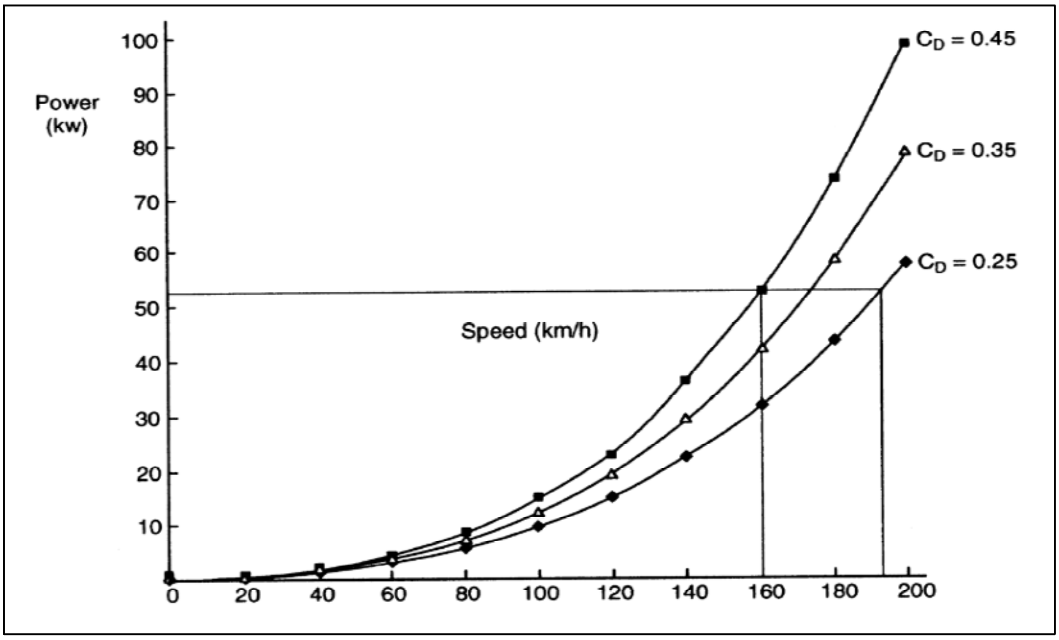


Figure 2.6: The influence of drag coefficients on velocity and spent power on road.

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.

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. The requirement that influences concept of cars today can show at Figure 2.7.

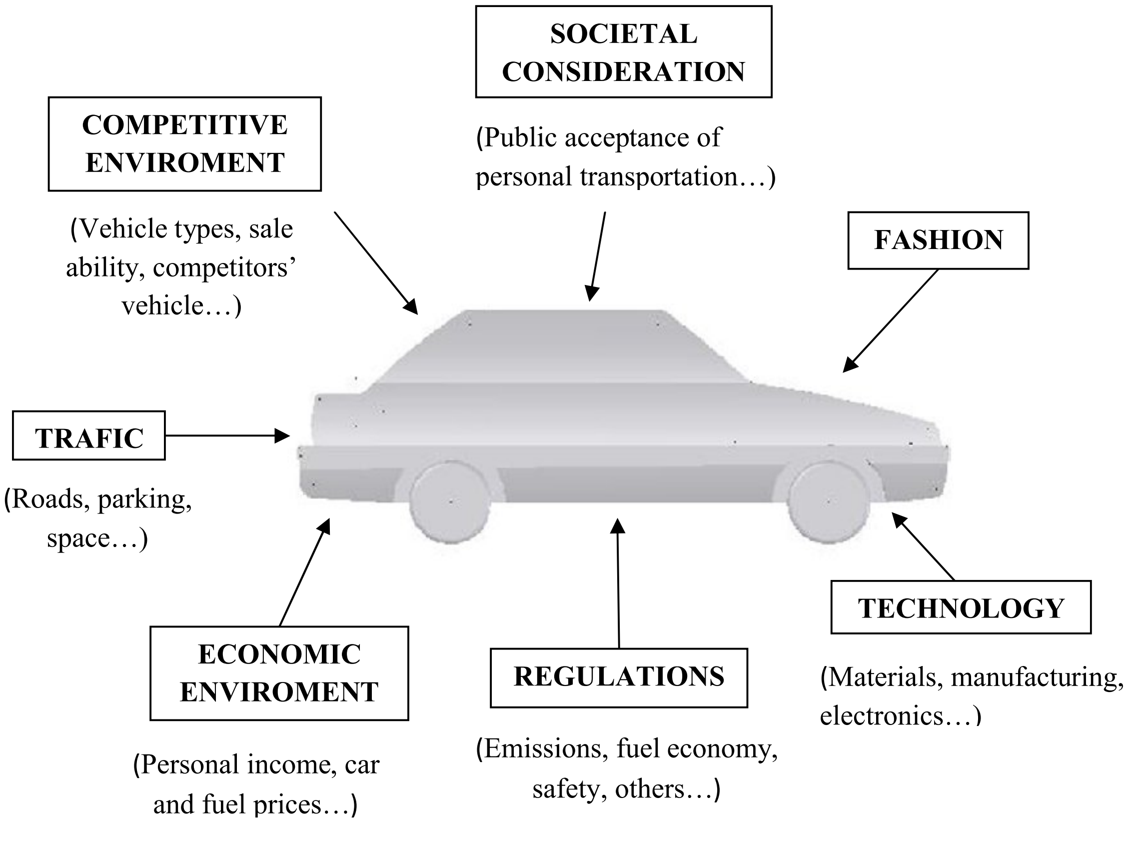


Figure 2.7: The concept of car is influenced by many requirements of very difference nature. [J. Katz (2006)]

If looked at the history, the vehicle body's shape is made according to the aeronautical practice and from the naval architecture adapted shape. 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.

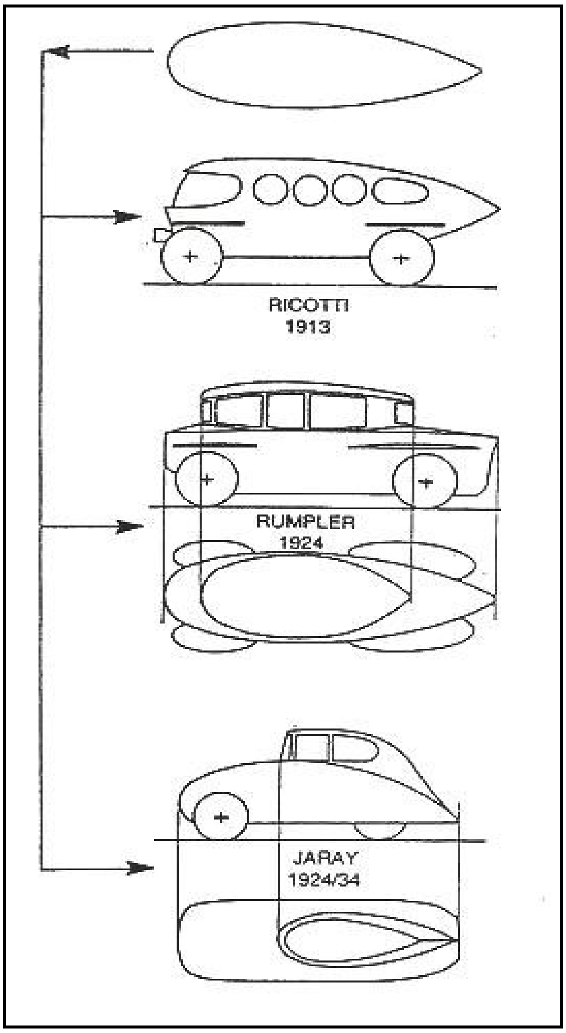


Figure 2.8: The early attempts to apply aerodynamic to road vehicle consisted of the direct transfer of shapes originating from aeronautical and marine practice.

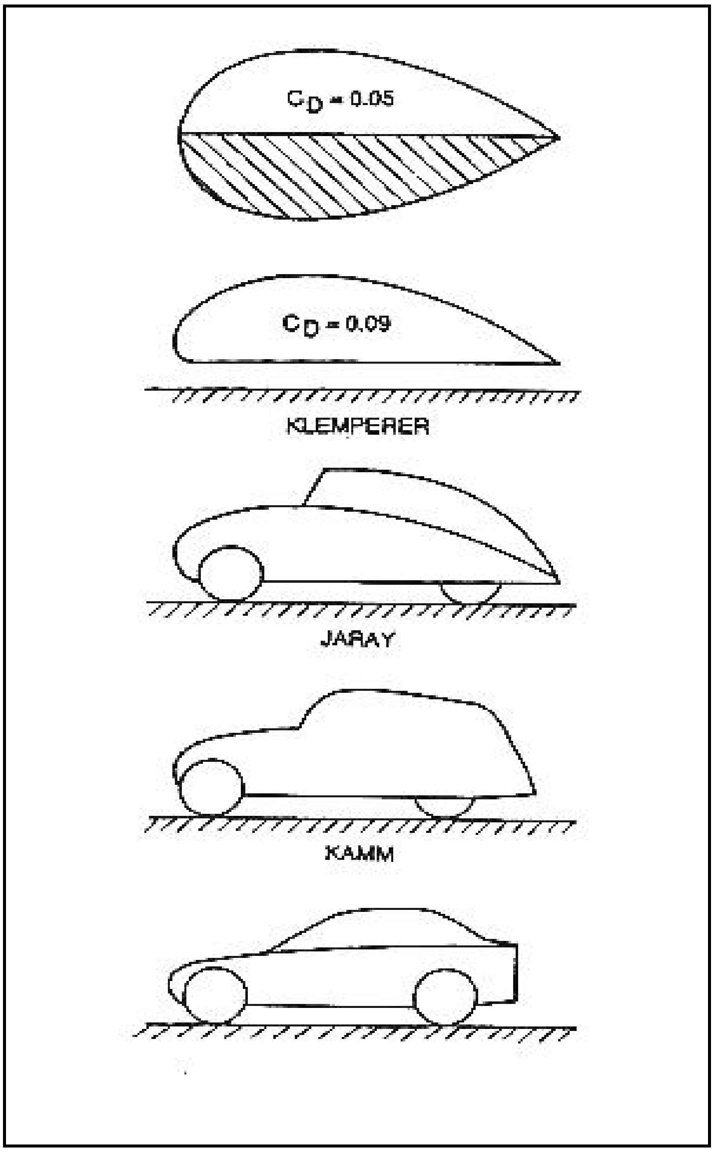


Figure 2.9: Klemperer recognized the flow over body revolution.

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. A brief overview of history on vehicle aerodynamic was summarized in Figure 2.10 below. [W. H. Hucho (1998)]

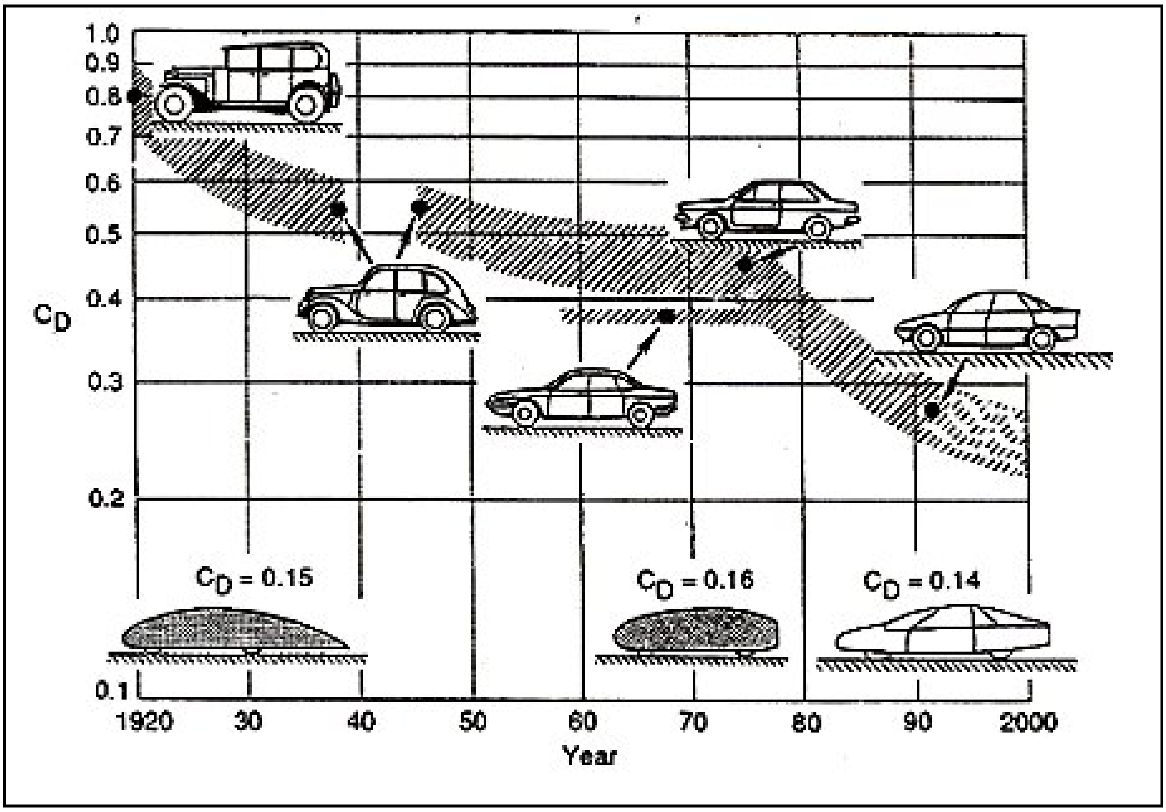


Figure 2.10 The drag history of cars using a logarithmic scale for drag emphasizes how difficult it is achieve very low drag values.

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. This is because;

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

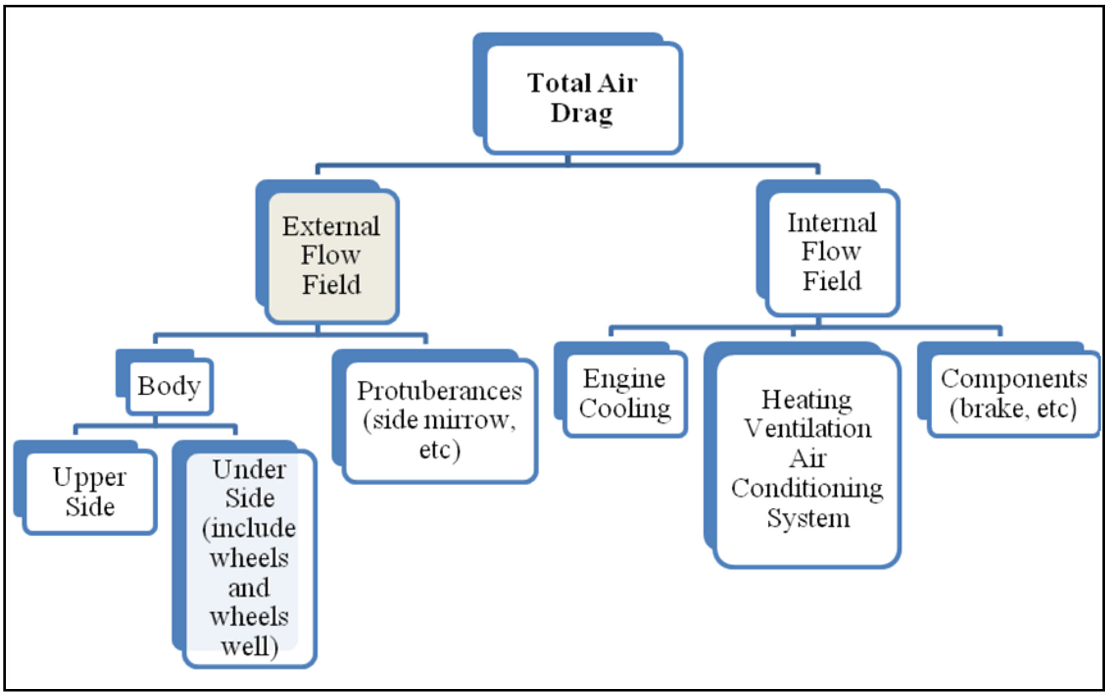


Figure 2.11: Breakdown of drag according to the locations of generation.

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 [W. H. Hucho (1998)]:

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

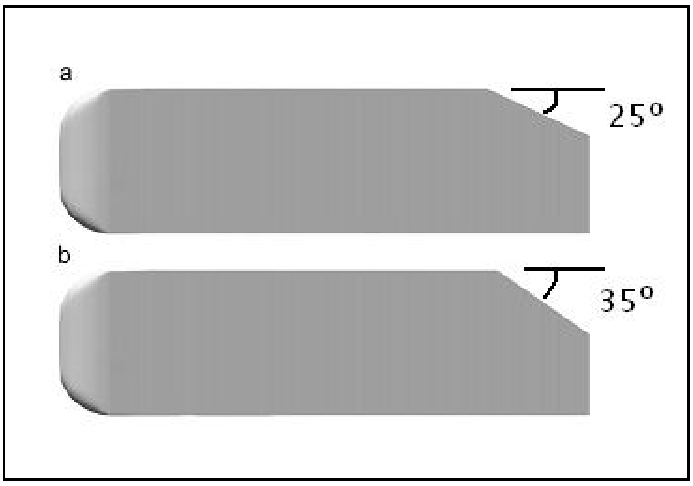
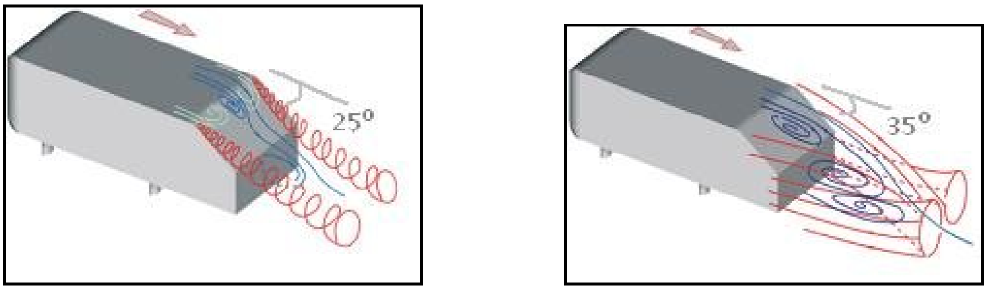


Figure 2.12: Ahmed body view (a) 25° rear slant; (b) 35° rear slant



(a) (b)

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

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

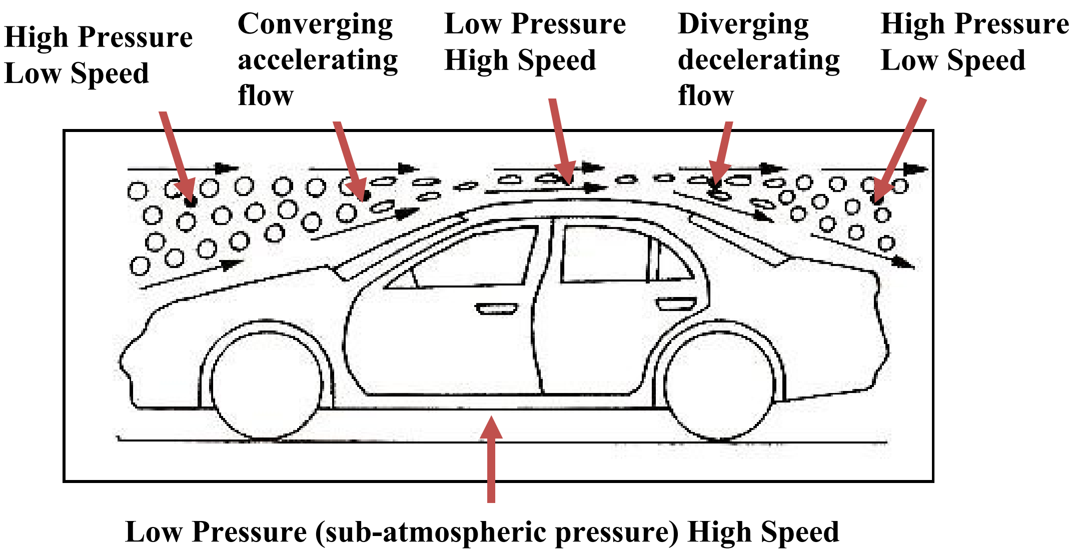


Figure 2.14: The relative velocity of air and pressure condition over the upper profile of a moving car. [J. Katz (2006)]

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.

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.

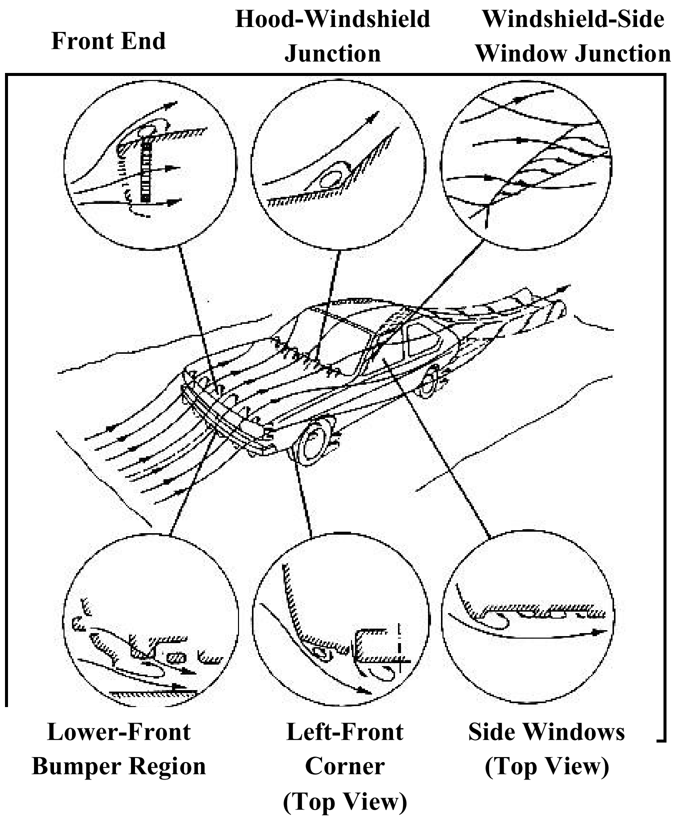


Figure 2.15: Flow around a car, and major of locations of flow separation. [J. Katz (2006)]

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.

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. The fundamental equation of in viscid flow was simplified on the vehicle body as shown in Figure 2.17 below as a two-dimensional flow, this simplification actually from three-dimensional flow around a car's body shaped. [J. Katz (2006)]

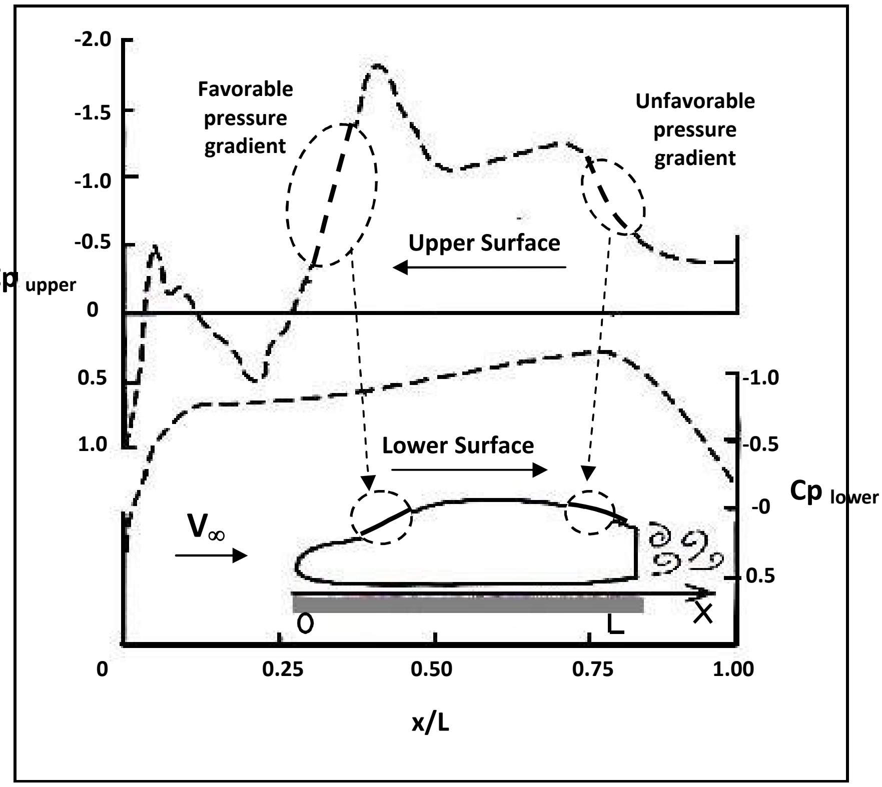


Figure 2.16: Pressure Coefficient Distribution over an automobile shape.

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. 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.

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. 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 . CFD is the validated computational codes create by programmer as a branch of Fluid Mechanic that have complex mathematical basis. 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 . There are based on:

- Laplace's equation,
- Reynolds-averaged Navier-Stokes equation (RANS),
- Instantaneous Navier-Stokes equation, called direct numerical simulation (DNS)
- 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 .

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 [W. H. Hucho (1998)]:

- The continuity equation
- The momentum equations
- 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 : [John D.Anderson. (2007)]

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

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

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

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:

- Preprocessing- discretization of body surface or the computational domain
- Solving- aerodynamic computation
- 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 for an example results plot (contours, vector, isolines) and process results (XY plots, goals, report, etc) .

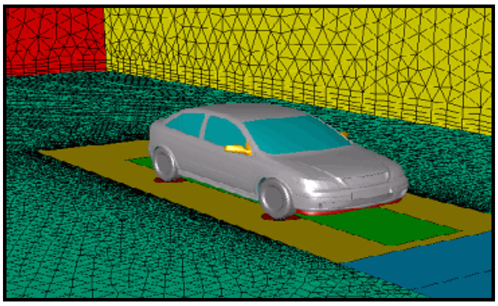


Figure 2.17: The 3D-hybrid grid

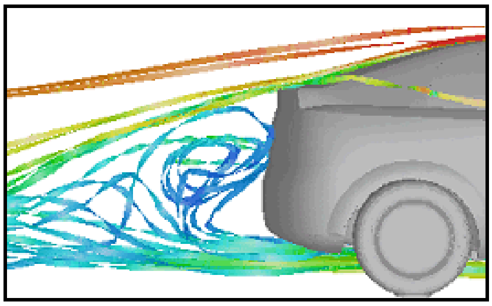


Figure 2.18: Flow analysis after a design study (pathlines)

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.

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 :

- Small Solid Feature Refinement
- Curvature Refinement
- Narrow Channel Refinement
- Square Difference Refinement

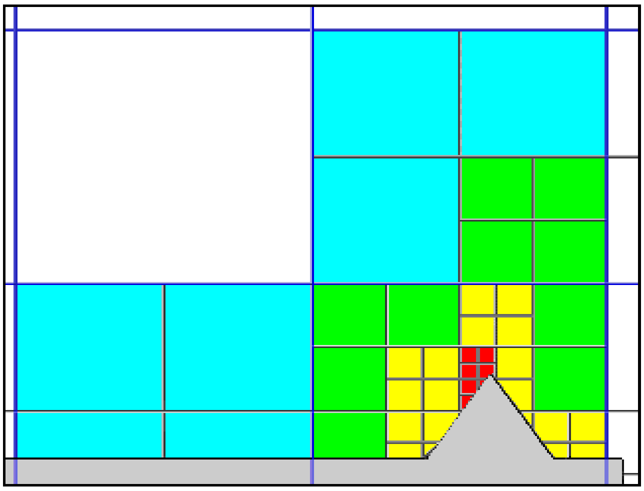


Figure 2.19: Fluid cell refinements due to the Cell Mating rule

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 .

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.

CHAPTER 3

PROJECT METHODOLOGY

3.1 Introduction

For the methodology in determine of drag and lift coefficient for Sedan and Hatchback 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, drag and lift 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 analysis, a terminology of works and planning show in the flow chart at Figure 3.1.

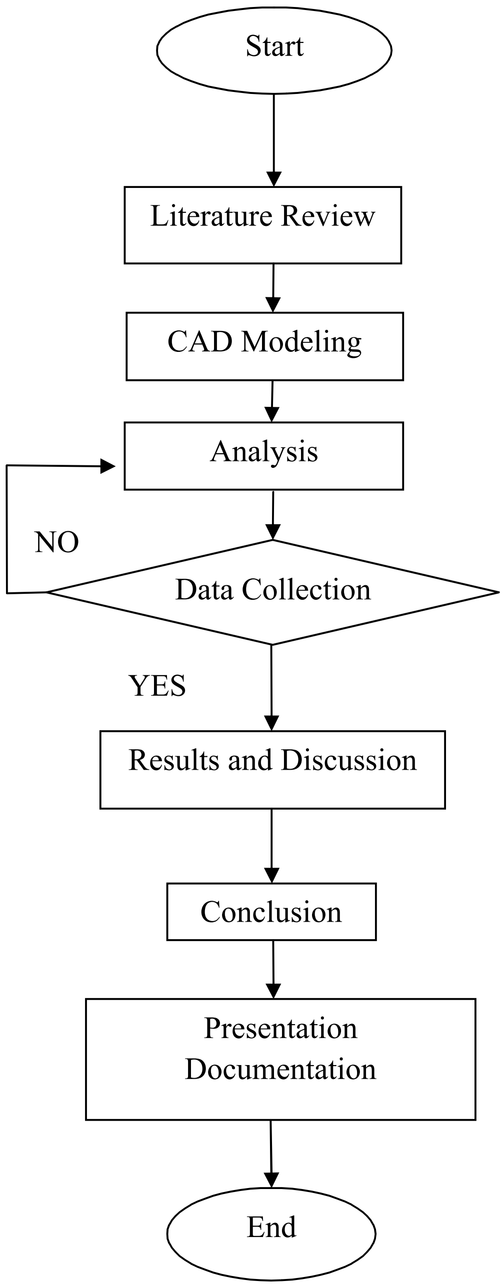


Figure 3.1: Flowchart of the Overall Methodology

3.3 Literature Review

The progress for this project will start with gather information by research and literature review via internet, journal, reference books, supervisor and other relevant academic material that related to this project. The literature is more about the aerodynamics of car, the drag and lift coefficient and CosmosWork software. The detail literature review can be referring to chapter two.

3.3 CAD Modeling

The data of dimension for Sedan and Hatchback body was collecting from the internet and measuring then modeling the body by SolidWorks software. The model of the Sedan and Hatchback car was collecting by random. Data collecting of dimension as accurate as possible is very important for the both body to simulating the model in CFD analysis to get the appropriate or expected value of drag and lift force. The dimension of Sedan Car show in Figure 3.2 below:

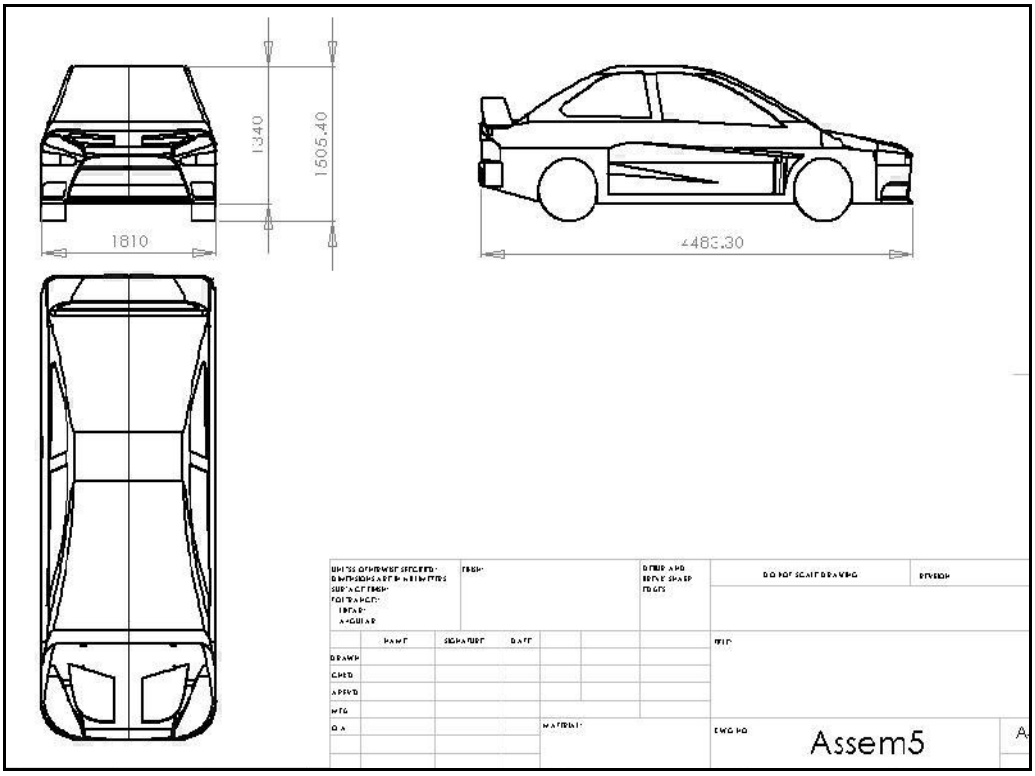


Figure 3.2: The dimension of Sedan Car

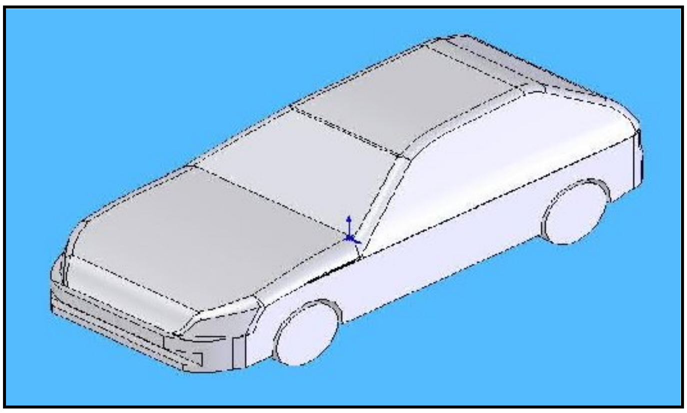


Figure 3.3: The Hatchback car in Solidwork type.

Figure 3.4: The Sedan car in Solidwork type.

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.

3.4 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 COSMOSWorks software. At this stage, the various car speeds will analyze at ranging 40 km/h to 140 km/h for every 20 km/h. The boundary condition for this analysis is external flow with adiabatic wall. Beside 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.

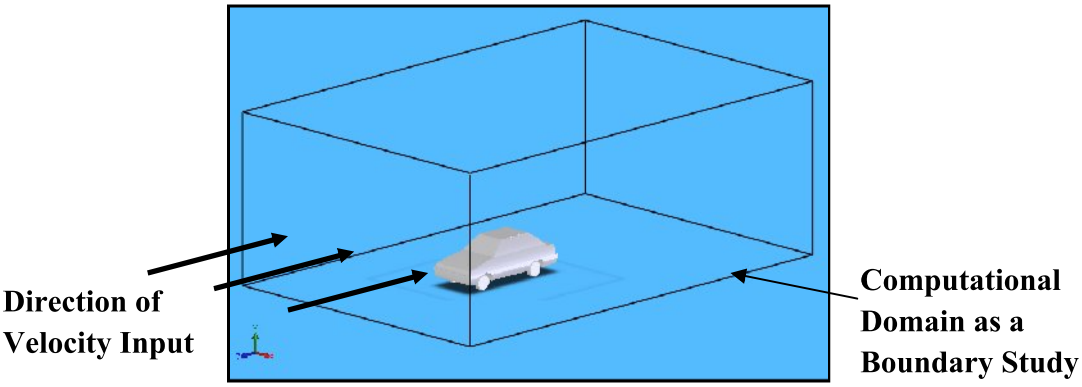


Figure 3.5: Boundary Condition of CFD analysis

3.4.1 Refinement

During analyzing the car model, the basic mesh were refining by strategy refinement that able in the COSMOSWorks. 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. .

3.4.2 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.

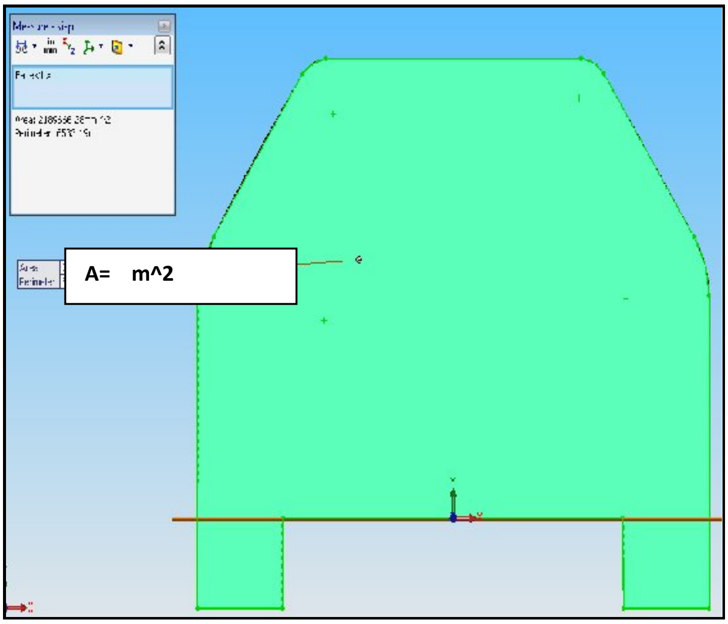


Figure 3.6: The frontal area projected of CAD model

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.

3.5 Presentation And Documentation

Lastly, the final report writing and prepare the presentation. The presentation process will be held two times, at the last first semester and at the last second semester. First presentation should be present about project title, problem statement, objective, project scope, literature review and project methodology. Second presentation must be showing the result and the project outcome. The presentation session will be judge by three person of panel from Mechanical Engineering Faculty (FKM). For the report or documentation, the report draft chapter one until chapter three need to submit at the first semester. Second semester, the report must be complete and need to banding. All the report writing is guided by FKM thesis format and also guidance from supervisor. All task scheduled is take around fourteen weeks to complete.

3.6 Summary

Method of this project should be done by details and constant to get a nearly precise result. This method is standard method which used in majority of researcher.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Data Collecting

4.1.1 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 140km/h for sedan and hatchback car are listed in the Table 4.1 and 4.2 and the graph from the data has been plotted in Graph 4.1, 4.2, and 4.4.

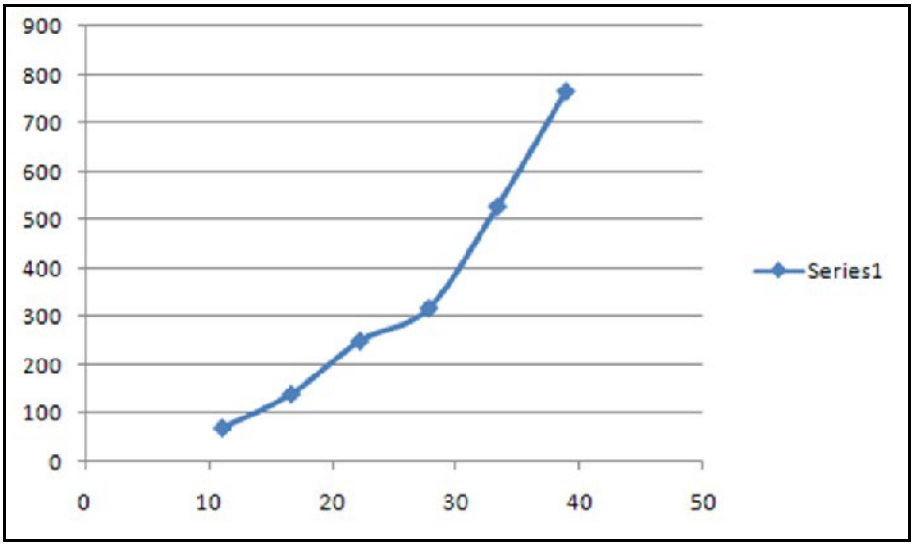
Table 4.1: Table of various velocities, drag forces, drag coefficient, Lift force and Lift coefficient for sedan car.

Velocity (km/h)	Velocity (m/s)	Drag Force (N)	Drag Coefficient	Lift Force (N)	Lift Coefficient
40	11.11	65.085	0.4165	69.843	0.4469
60	16.67	146.228	0.4166	139.606	0.4468
80	22.22	257.943	0.4326	249.169	0.4936
100	27.78	423.213	0.4351	316.721	0.4941
120	33.33	5390207	0.4833	526.178	0.5741
140	38.87	791.076	0.4835	763.530	0.5891

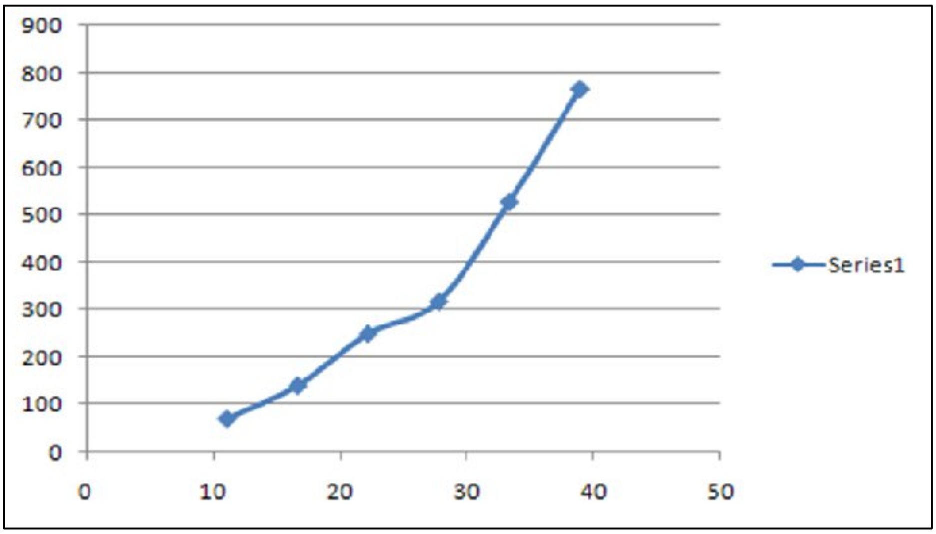
Table 4.2: Table of various velocities, drag forces, drag coefficient, Lift force and Lift coefficient for hatchback car

Velocity (km/h)	Velocity (m/s)	Drag Force (N)	Drag Coefficient	Lift Force (N)	Lift Coefficient
40	11.11	65.49	0.4038	81.68	0.5037
60	16.67	148.35	0.4063	190.35	0.5214
80	22.22	268.79	0.4144	348.63	0.5375
100	27.78	425.15	0.4193	552.55	0.5450
120	33.33	636.29	0.4360	767.38	0.5258
140	38.87	824.49	0.4149	1089.11	0.5487

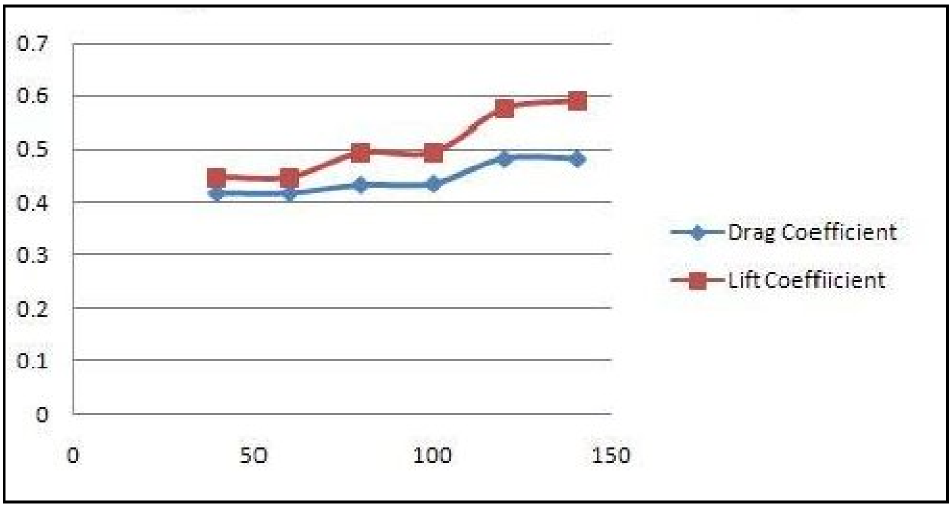
The data from Table 4.1 and Table 4.2 was plotted into the graph of drag forces, *D* against the velocity, *V*. for both bodies. Then, the graph series of *D* VS *V* was shown in Graph 4.1, 4.2 and 4.4 below.



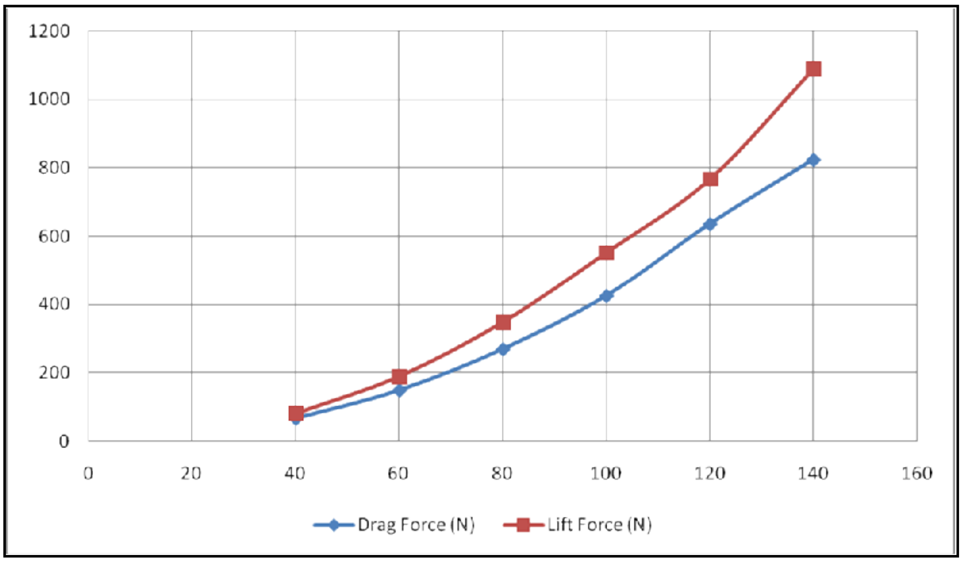
Graph 4.1: Graph Drag Force C_D against Velocity, V for sedan car



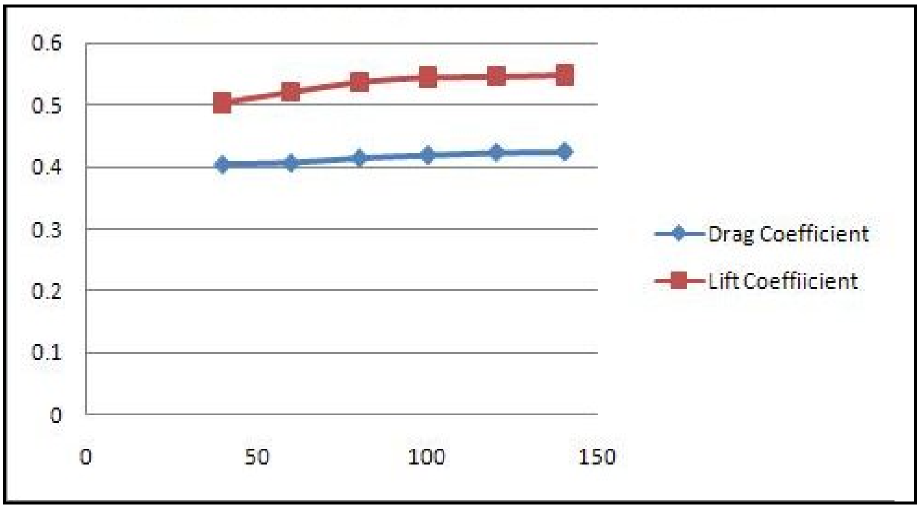
Graph 4.2: Graph Lift Force C_L against Velocity, V for sedan car



Graph 4.3: Graph Drag and Lift Coefficient against Velocity, for sedan car



Graph 4.4: Graph Drag and Lift Force against Velocity for hatchback car



Graph 4.5: Graph Drag and Lift Coeficient against Velocity for hatchback car

From the Graph 4.3 and 4.5 above shown the curve of drag and lift coefficient were proportial with the velocity of the car for Sedan and Hatchback car . It shows clearly the drag forces, *D* was function to the square of velocity from 40 km/h up to 140km/h. Thus, it means the increasing of velocity would affected to the increasing of drag and lift coefficient .

4.1.2 Value of Projected Area

The value of projected area was getting from the CAD software. The projected area means as frontal area of the Sedan's body that has overall height and width While, the value of projected area were measured is about 2.11 m^2 while the hatchback is about 2.189 m^2directly from the CAD's Software.

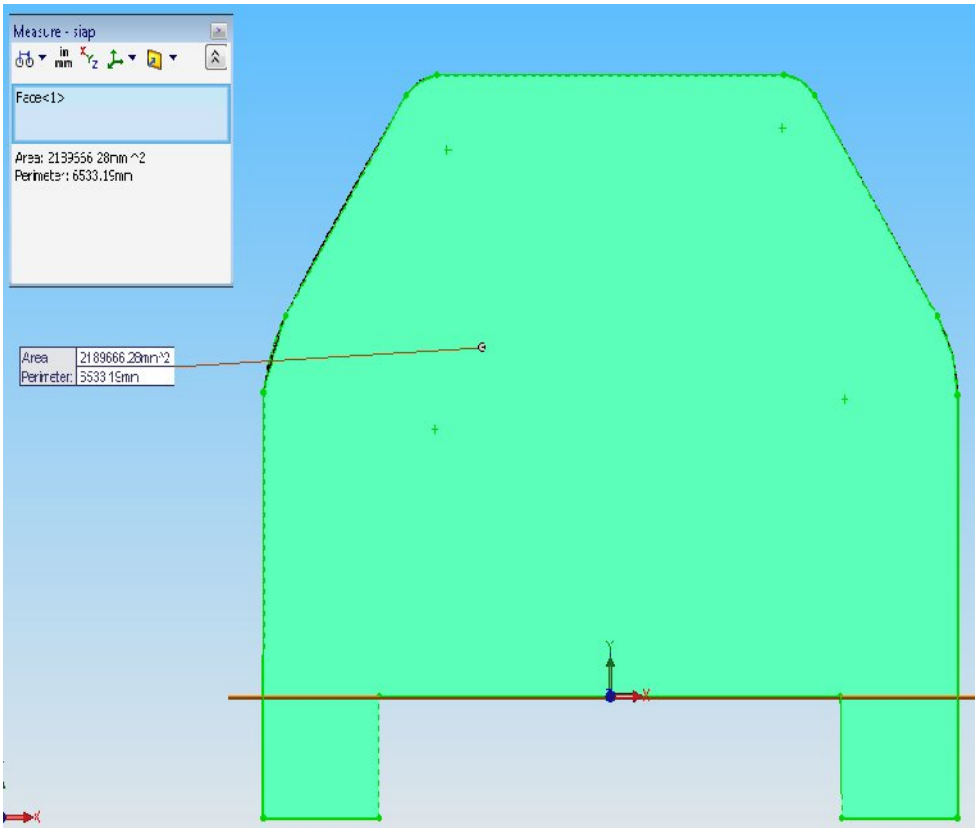


Figure 4.1: Frontal Area from the Simulation

4.2 Data Analysis

4.2.1 Calculation of Drag and Lift Coefficient

From the all data collections of Table 4.1, and Table 4.2**,** reference point measurement, and projected area of the car. The drag coefficient and lift coefficient was calculated base on the equation (2.6). The calculation were including for every speed of analysis where is between 40 km/h to 140km/h plotted into a graph.

4.2.2 Sample Calculation for Drag and Lift Coefficient

Therefore, drag and lift coefficient for every velocity of car were calculated from the equation (2.6) and (2.9).

- Drag Coefficient for velocity of car is 40km/h for sedan car:

$$C_D = \frac{F_D}{\frac{1}{2}\rho A V^2}$$

$$C_D = \frac{65.085}{1/2 * 1.20 * 2.11 * 11.11^2}$$

$$= 0.4039$$

- Drag Coefficient for velocity of car is 40km/h for hatchback car:

$$C_D = \frac{F_D}{\frac{1}{2}\rho A V^2}$$

$$C_D = \frac{65.49}{1/2 * 1.20 * 2.11 * 11.11^2}$$

$$= 0.4038$$

- Lift coefficeint for for velocity of car is 40 km/h for Sedan car:

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

C_L = 69.843

$$1/2 * 1.20 * 2.11 * 11.11^2$$
$$= 0.4469$$

- Lift coefficeint for for velocity of car is 40 km/h for Hatchback car:

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

$$C_L = \frac{81.68}{1/2 * 1.20 * 2.11 * 11.11^2}$$
$$= 0.5227$$

4.2.3 Average of Drag and Lift 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 4.3 and Graph 4.4.

Table 4.3: Table of average drag and lift coefficients for sedan car.

Velocity, (km/h)	**Drag Coefficient, C_D**	**Lift Coefficient, C_L**
40	0.4165	0.4469
60	0.4166	0.4468
80	0.4326	0.4936
100	0.4351	0.4941
120	0.4833	0.5741
140	0.4835	0.5891
Average of C_D	0.4446	0.5074

Table 4.4: Table of average drag and lift coefficients for hatchback car.

Velocity, (km/h)	**Drag Coefficient, C_D**	**Lift Coefficient, C_L**
40	0.4038	0.5227
60	0.4063	0.5215
80	0.4144	0.5379
100	0.4193	0.5451
120	0.4360	0.5259
140	0.4149	0.5488
Average of C_D	0.4157	0.5336

The Graph 4.3 and 4.4 also shown that the average of drag coefficient is proportionally increase to the increasing of velocity.

4.3 Countour Plot of Velocity and Pressure

The contour plot of velocity and pressure were figured in Figure 4.2 and Figure 4.3 for sedan car and Figure 4.4 and Figure 4.5 for hatcback car. The contour plot of velocity and pressure were shows to analyzed the characteristic of wake region and the pressure's boundary layer around the both type body.

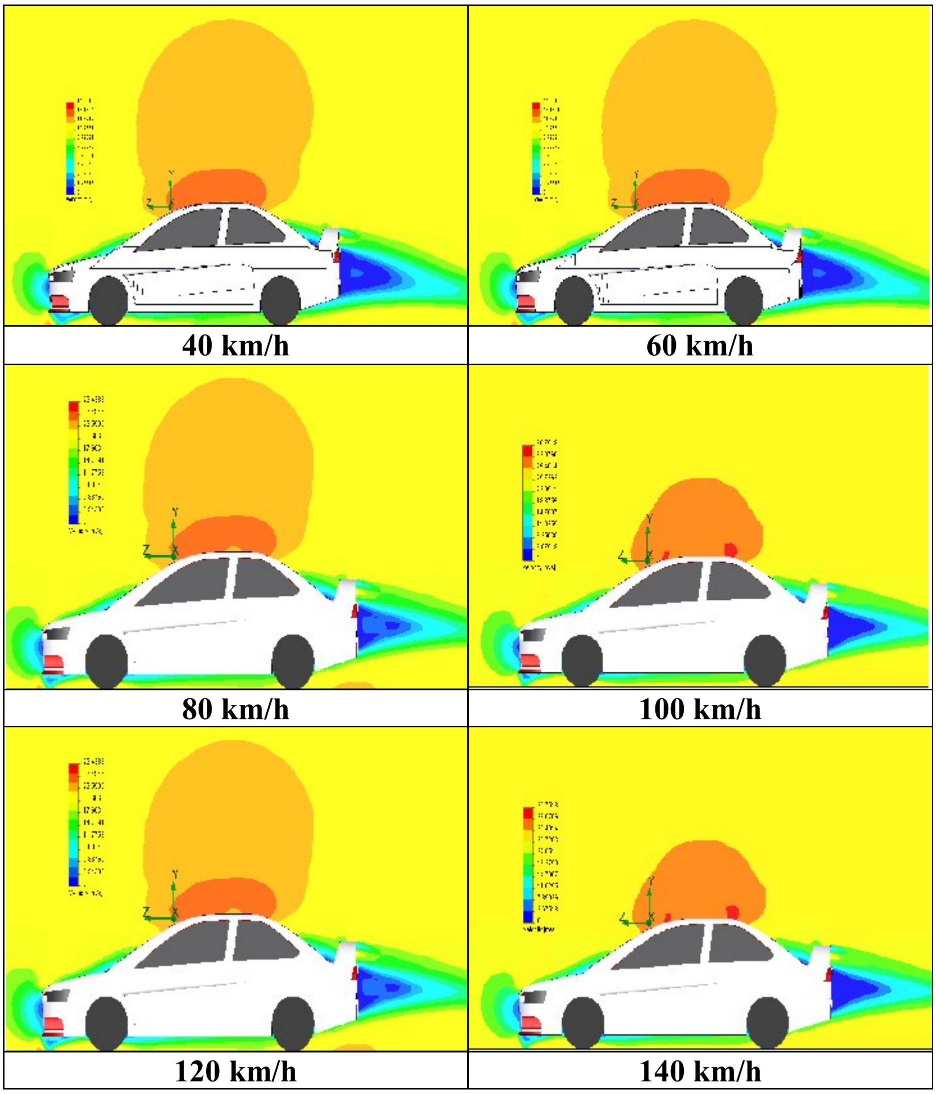


Figure 4.2: The contour plot of velocity for the ranging velocity's analysis for sedan car.

From the figure above was demonstrating the patent of the contour plot of velocity quite same for every velocity of analysis for sedan car. The red and blue color from the contour plot of velocity shows high and low velocity.

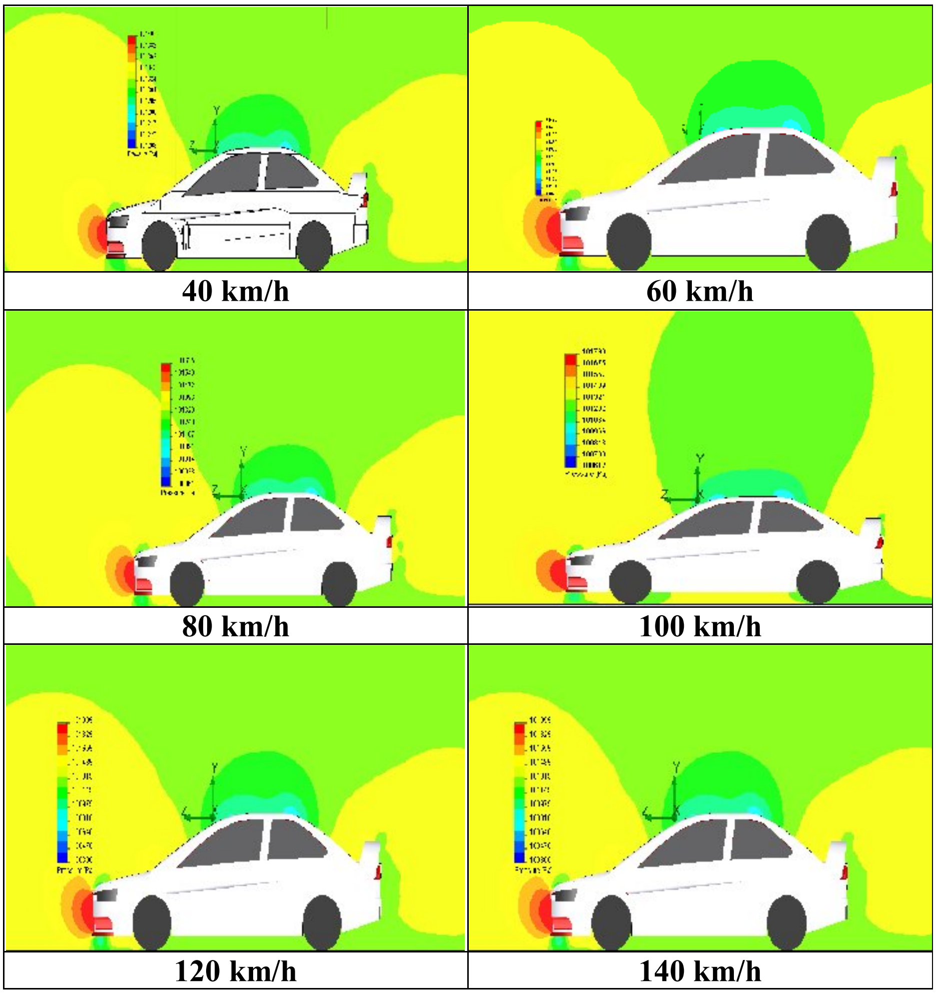


Figure 4.3: The contour plot of pressure for the ranging pressure's analysis for sedan car.

Figure 4.3 show the contour plot of pressure for Sedan car. If looked detail, the big slight differences contour plot of velocity at the high velocity of 140km/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.

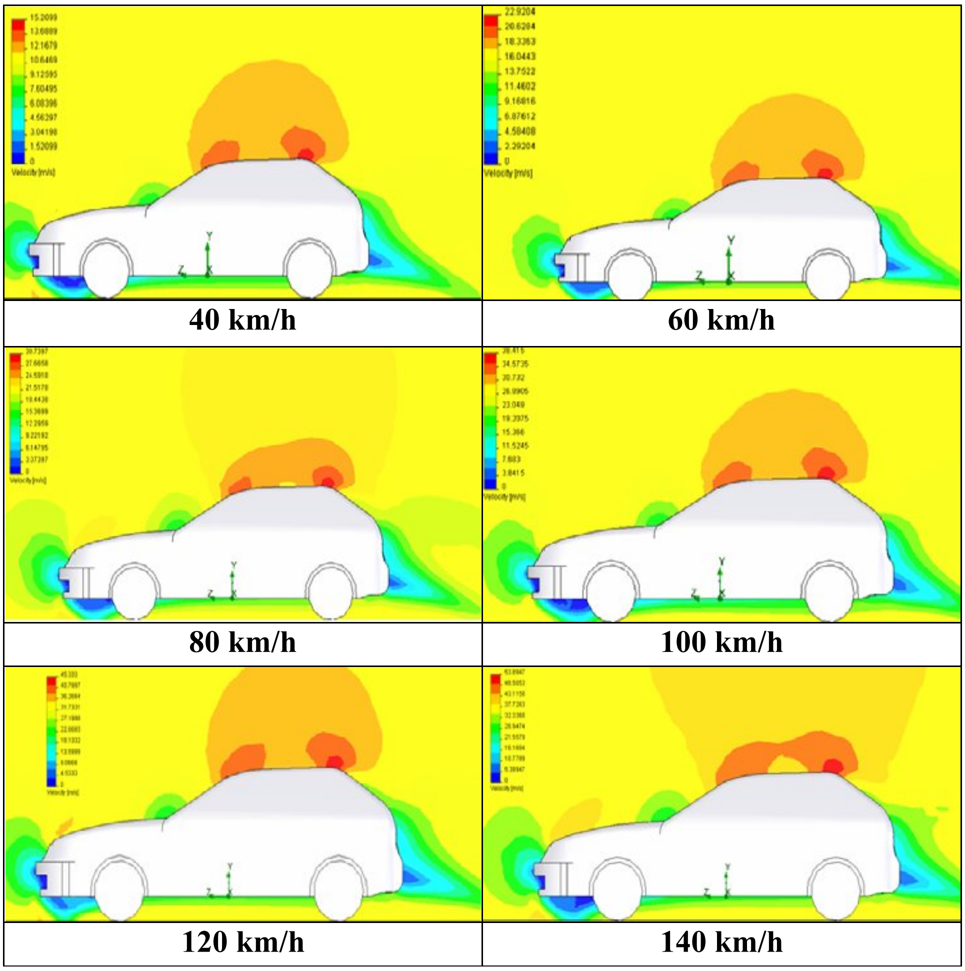


Figure 4.4: The contour plot of velocity for the ranging velocity's analysis for hatchback car.

At 100 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.

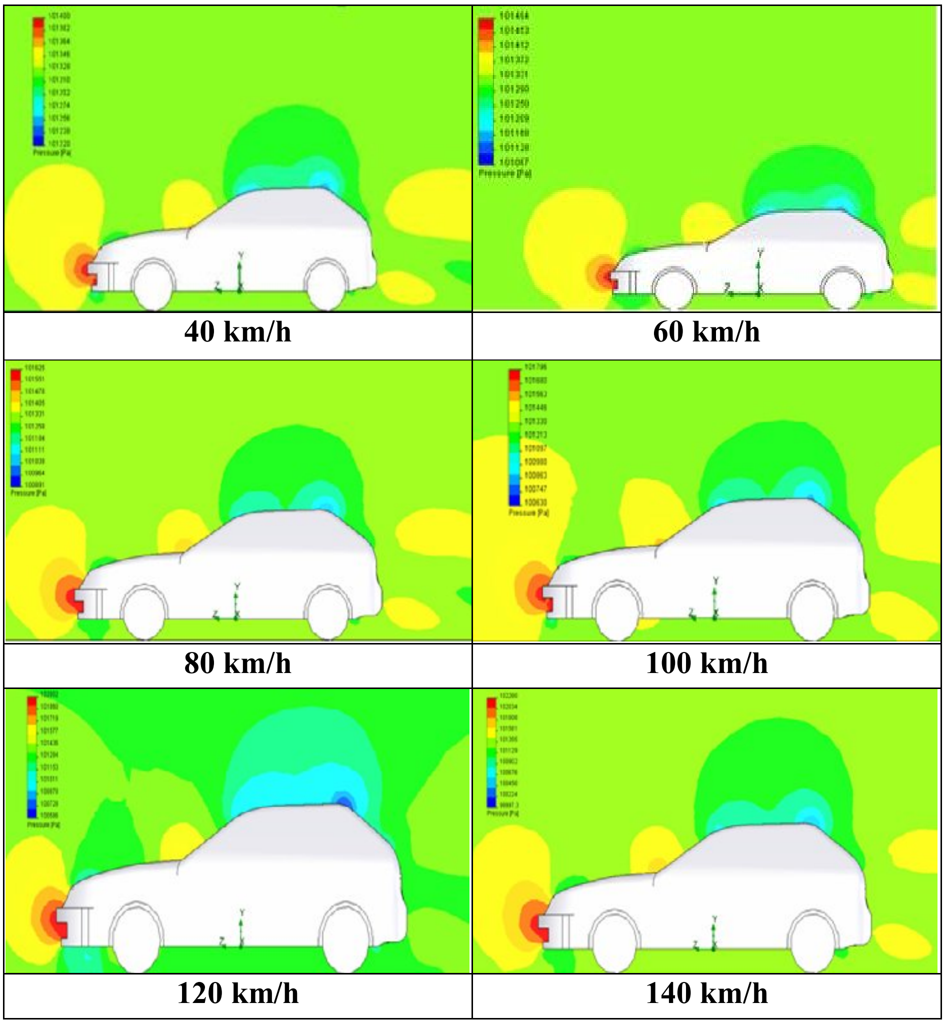


Figure 4.5: The contour plot of pressure for the ranging pressure's analysis for hatchback car.

For the Figure 4.5, 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 4.6.

The local pressure depicts clearly difference for the velocity of 140km/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. Besides that, the big red color of stagnation point means as high dynamics pressure also give significant value for the aerodynamics drag.

4.4 Trajectories Velocity Flow Analysis

The detail analysis of the flow on the body by trajectories velocity flow analysis diagram were figured in the Figure 4.5 and Figure 4.6. The figure was shown as a general flow on the sedan and hatchback body and the example pressure 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.

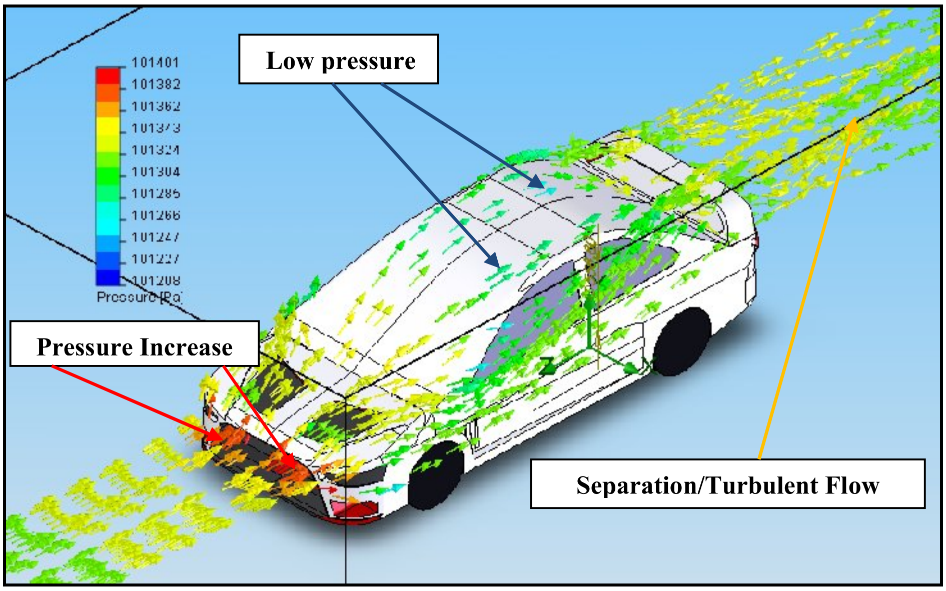


Figure 4.5: The isometric view of trajectories pressure flow of 40 km/h for sedan car.

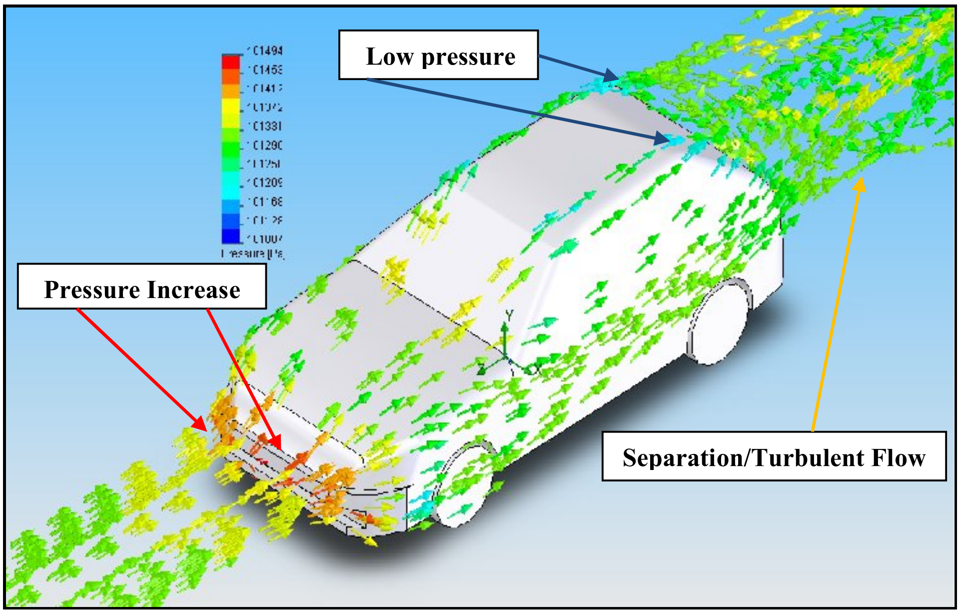


Figure 4.6: The isometric view of trajectories pressure flow of 40 km/h for hatchback car.

Shown at the Figure 4.5 and 4.6 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.

CHAPTER 5

CONCLUSION AND FUTURE RECOMMENDATION

5.1 INTRODUCTION

The project was done successfully with archiving the project objective. However, during the project stage was occur a few problem and the problem need to solve in way to done the project. Besides that, the project was running guidance by the scope of project. It is because limitation of time and equipment to done the project. But, the project can be continued based on future recommendation in way to achieve the better result for the project. The project that has better result can be significant to the next researcher and as the references.

5.2 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. The analysis shows the drag and lift coefficient for hatchback car is lower than sedan model car. That is mean the hatchback car is more efficient and suitable in fuel consumption

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 60 km/h and 140km/h, the plot of velocities obviously depicts variation. For the plot of pressure, the difference clearly shows at 140km/h.

Moreover, the trajectories flow was also give the significant value of analysis to the Sedan and hatchback body. The streamline or flow of velocity and pressure can looked next to the whole body. 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.

5.3 FUTURE RECOMMENDATION

The drag and lift coefficient of Sedan and Hatchback car can be reduce by a few method in aerodynamics field. The research of future method for these reductions can be used to improve of future car. The research also perhaps can decrease of fuel consumption as main as issues in automotive field. Also the safety of the car can be increase by the research of the aerodynamics aspects.

There is several future methods would like to research to reduce the Drag Coefficient to improve future car.

i. Profile Edge Rounding or Chamfering
ii. Bonnet Slope and Windscreen Rake
iii. Roof and Side Panel Cambering
iv. Rear Side Panel Taper
v. Underbody Rear End Upward Taper (diffuser)
vi. Rear End Tail Extension

So, for future research, hopefully these example methods can be simulate by using any software such as Solidwork Cosmos for analyzed the aerodynamics of the car.