AERODYNAMICS OPTIMIZATION OF REAR CAR

MOHD NAZIRUL MUBIN BIN MOHAMMAD ZAKI

UNIVERSITY MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS ◆

JUDUL: AERODYNAMICS OPTIMIZATION OF REAR CAR

SESI PENGAJIAN: 2007/2008

Saya, **MOHD NAZIRUL MUBIN B. MOHAMMAD ZAKI (860329435211)**
(HURUF BESAR)

mengaku membenarkan tesis (Sarjana Muda / ~~Sarjana / Doktor Falsafah~~)* ini disimpan di perpustakaan dengan syarat-syarat kegunaan seperti berikut:

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

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

Disahkan oleh:

____________________________ ____________________________

(TANDATANGAN PENULIS) (TANDATANGAN PENYELIA)

Alamat Tetap:

No 13, Jalan Semilang 3, Taman Seri Putra, 42700 Banting, Selangor.

ZAMRI BIN MOHAMED
(Nama Penyelia)

Tarikh: **10 NOVEMBER 2008** Tarikh: **10 NOVEMBER 2008**

CATATAN: * Potong yang tidak berkenaan.

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

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

AERODYNAMICS OPTIMIZATION OF REAR CAR

MOHD NAZIRUL MUBIN BIN MOHAMMAD ZAKI

Report is submitted in partial fulfillment
of the requirements for the award of the degree of
Bachelor of Mechanical Engineering with Automotive

Faculty of Mechanical Engineering
Universiti Malaysia Pahang

NOVEMBER 2008

SUPERVISOR'S DECLARATION

We hereby declare that we have checked this project and in our opinion this project is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive

Signature: ……………………………………….
Name of Supervisor: En. Zamri Bin Mohamed
Position: Lecturer
Date: 10 November 2008

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

STUDENT'S DECLARATION

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

Signature: ……………………………………………

Name: Mohd Nazirul Mubin bin Mohammad Zaki

ID Number: MH05042

Date: 10 November 2008

To my beloved father mother

Mr Mohammad Zaki bin Rajikin
Madam Musliah binti Mohd Nor

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor En. Zamri Bin Mohamed for his brilliant ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, and his belief that a Bachelor program is only a start of a life-long learning experience. I appreciate his consistent support from the first day I applied to PSM course to these concluding moments. I am truly grateful for his progressive vision about my work progressing, his tolerance of my naive mistakes, and his commitment to my future career. I also would like to express very special thanks again to my supervisor for his suggestions and co-operation throughout the study. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

My sincere thanks go to all staff of the Faculty of Mechanical Engineering, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to my fellow friends for their excellent co-operation, inspirations and supports during this study.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. Special thanks should be given to my fellow members. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

Automotive aerodynamic is the study of air flow characteristic around and through a vehicle during motion. In this project, a flow characteristic study is including pressure distribution on the car concentrating only at rear body. For any current car that has received aerodynamic attention, the contribution of the fore body to drag is usually small. The objectives of this project are to estimate the drag and find the optimization of rear shape of body at constant velocity 110 km/h by using aerodynamic simulation software that is Computational Fluid Dynamic (CFD). From the analysis by CFD, the force value is created as goal and from force value; drag coefficient can be estimated by using a mathematical equation. Before the analysis can performed, a geometrical model with different shape of the rear body must be created first by using Computer Aided Design software (CAD). By using design and shape variation by means of multiple rear slant angle and rear bonnet, analysis was done in CFD. From the data, coefficient of drag, C_D was calculated for every shape and in order to get the optimization shape, the shape that has lowest value of C_D was combined in a one model and then the model was simulate again in CFD. After simulation was done, the result gives the lowest value of C_D and the objective to find the optimization of rear shape was successful achieved.

ABSTRAK

Automotive aerodynamik adalah satu kajian tentang ciri-ciri atau sifat aliran udara pada sekitar badan kereta dalam berkeadaan bergerak. Di dalam projek ini, kajian adalah meliputi cirri-ciri aliran udara dan juga pembahagian tekanan yang dikenakan pada kereta tetapi tumpuan lebih diberikan pada bahagian belakang kereta sahaja. Untuk sebarang keadaan kereta yang telah menerima perhatian aerodinamik, sumbangan rintangan daya pada badan depan kereta adalah biasanya kecil. Objektif utama dalam projek ini adalah menentukan nilai pekali rintangan angin dan mencari bentuk belakang kereta yang optimum pada kelajuan 110 km/j dengan menggunakan perisian aerodynamic (CFD). Daripada analisis yang dijalankan dengan menggunakan (CFD), pekali bagi rintangan daya boleh ditentukan dengan persamaan matematik. Sebelum menjalankan simulasi, sebuah CAD model kereta dengan rekabentuk belakang kereta yang berbeza-beza diperlukan dan dibina terlebih dulu dengan menggunakan perisian CAD. Dengan menggunakan model kereta yang pelbagai bentuk, analisis telah berjaya selesai dijalankan dengan menggunakan perisian CFD . Daripada data yang diperoleh, pekali rintangan daya dikira bagi setiap bentuk dan dalam usaha untuk mencari bentuk belakang kereta yang optimum, bentuk-bentuk yang mempunyai nilai pekali terendah digabungkan dalam sebuah model kereta dan simulasi dijalankan semula dan hasilnya, pekali bagi bagi rintangan udara telah diperoleh.

TABLE OF CONTENTS

CHAPTER 2 LITERATURE REVIEW

CHAPTER 2 METHODOLOGY

LIST OF TABLES

LIST OF GRAPH

LIST OF FIGURE

LIST OF SYMBOL

ρ	Density
p	Pressure
v	Vehicle speed
D	Drag force
L	Lift force
D_A	Aerodynamic Drag force
C_D	Drag Coefficient
A	Frontal Area
L_A	Aerodynamic lift force
C_L	Lift coefficient
P_r	Air temperature
Re	Reynold number

LIST OF ABBREVIATIONS

CAD	Computer-aided design
CFD	Computational fluid dynamic
3-D	Three dimensional
Re	Reynolds number
Ma	Mach number
Fr	Froude number
ε/l	Relative roughness
BLM	Base-line model

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

This project is about study and analysis of rear car shape to find the optimization of rear shape and C_D value of a car. This project involves sketching, drawing and simulating a geometrical model using different design parameter shape of rear slant angle and rear bonnet shape by using aerodynamic simulation software. This project is very important because it will study of the characteristic of flow of air around and through a vehicle. The flow over the part of any body moving through the air is easier to manage than the over the rear. To see the effect and to understand this flow, it must visualize a car moving through the air like experimental in wind tunnel but for this project, the analysis is based on aerodynamic simulation software.

Aerodynamics is the branch of dynamics that deals with the motion of air and other gaseous fluids and with the forces acting on bodies in motion relative to such fluids. Aerodynamics affects the motion of a large airplane, a rocket, or a kite flying high in the sky. Most everyday things are either caused by aerodynamic effects or in general obey the aerodynamic laws. A car driven in a road is affected by aerodynamic forces created and even in nature; birds can fly in air because of this relative motion between their wings and air. The aerodynamics of road vehicles will be expounded. Forces created by the relative motion of the vehicle through air that is drag force, lift force and down force.

The main concerns of automotive aerodynamics are reducing drag, and preventing undesired lift forces at high speeds. In this project, it will explain how the aerodynamics is influenced by changes in body shape especially for rear slant angle roof and rear bonnet of road vehicles around the body using aerodynamic simulation software.

1.2 Problem Statement

The problem about this project is to analysis the flow occurred around of car with different shape of rear car body. It has been found that there is various design of body shape for roof and rear bonnet made from different car makers. Each shape will influence of the total drag of the vehicles. Drag will cause many problems on the performance of vehicles like instability, noise and vibration, also fuel consumption. So, the study of flow around of a vehicle is important to prove it and determine the optimization of the rear shape of vehicles. Using CFD 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 Limitation

The main limitation for this project is to create a good geometrical model for a car. There is a quite difficult to get an exact measurement and dimension every inch of a car whereas CAD software like SolidWorks need an accurate dimension to build a model. So for the whole of a car body because the exact measurement. Beside that, many parameters must be created first before can be analyzed in this project.

This project is only considering optimization for the rear body car with only three parameters there are rear slant angle, rear end bonnet angle and rear side bonnet angle. Also, this value of C_D is only valid for normal maxima highway speed that is 110km/hour or approximately 30.56m/s. The expectation result in this project may different from the actual.

1.4 Objective

1. To estimate the drag coefficient, C_D optimization by using difference design of shape or rear body of car.
2. To analysis and study of air flow characteristic on the rear bonnet and roof shape using Computational Fluid Dynamic (CFD) software.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

The aim of this chapter is to give some overview information about the motorcycle dynamometer which in the subject of inertia roller. In this chapter, the explanations and some of aerodynamic histories, the previous research and findings, the theories are included. With a reference from various source such as journal, thesis, references books, literature review has been carry out to collect all information related to this project. Also it describe about the CFD software and performing analysis by this software.

2.2 ROAD VEHICLES AERODYNAMICS

2.2.1 History of Automotive Aerodynamic Technology

The road vehicle industry started taking into consideration the aerodynamics of the vehicles in the early 1900. In the first stages of the development of self-propelled vehicles, the shapes and designs of the vehicles was inherited from horse driven carriages. The first automobiles however were moving with low speeds on bad roads. There was no need in examining the aerodynamic nature of a vehicle. The driver and passenger could be protected from wind, rain and mud with the simple, traditional design of horse-drawn carriages. The increase of automobile speed resulted in the exposure of drivers and passengers to the airstreams. As a result the introduction of structural parts such as windscreens was developed to protect the occupants from airstream effects. Gradually the development increased always according to the needs of both the physics (aerodynamic effects) and art properties (shape, style) around a vehicle.

A brief overview of the history of vehicle aerodynamics is summarized in Figure 2.1. During the first two periods, the aerodynamic development was done by individuals who most of them came from outside the car industry. What they tried to do was to carry over the aerodynamic principles of the aerospace and naval industry to cars. The aircraft and ship designers found their originals from nature, birds and fish, upon which their designs were based. They tried to borrow shapes from aircrafts and ships because at that time they looked progressive to them. However it soon turned out to be the wrong approach and new concepts were introduced. In the later two periods the disciplines of car aerodynamics was taken over by the car companies. The aerodynamic studies for car design were integrated to product design. Teams, not individual inventors, have been responsible for aerodynamics and they still are.

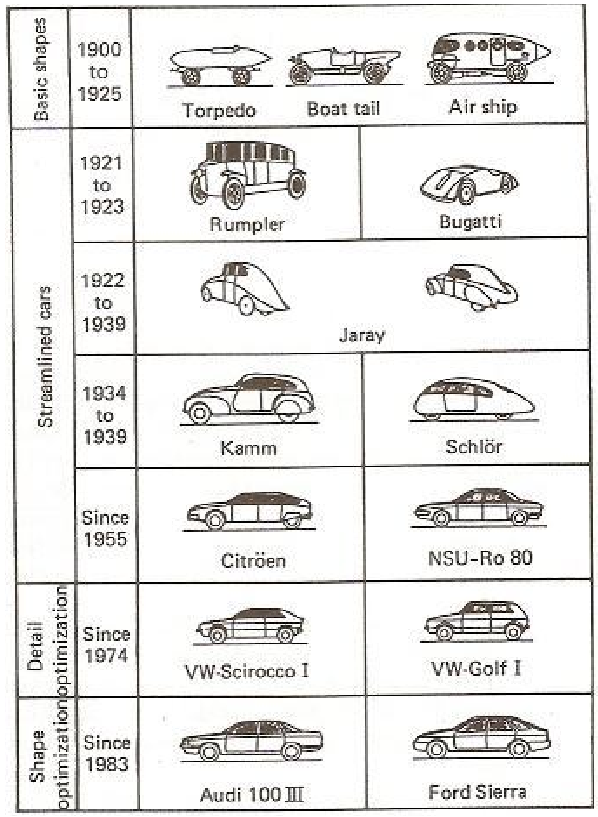


Figure 2.1: History of vehicle dynamic in passenger car

The first automobile to be developed according to the aerodynamic principles was a torpedo-shaped vehicle that had given it a low drag coefficient but the exposed driver and out of body wheels must have certainly disturbed its good flow properties. However they ignored the fact that the body was close to the ground in comparison to aircrafts and underwater ships flown in a medium that encloses the body. In a car like this, the ground along with the free-standing wheels and the exposed undercarriage causes disturbed flow.

As the years pass the studies on aerodynamic effects on cars increase and the designs are being developed to accommodate for the increasing needs and for economic reasons. The wheels developed to be designed within the body, lowering as a result the aerodynamic drag and produce a more gentle flow. The tail was for many years long and oddly shaped to maintain attached the streamline. The automobiles became developed even more with smooth bodies, integrated fenders and headlamps enclosed in the body. The designers had achieved a shape of a car that differed from the traditional horse-drawn carriages. They had certainly succeeded in building cars with low drag coefficient. (Dr. V Sumantran and Dr. Gino Sovran, 1996, Wolf-Heinrich Hucho, 1998)

2.2.2 Rear end/rear bonnet

Three types of rear end are common for cars, squareback, fastback, and notchback; in highly simplified form they are sketch in Figure 2.2. The main parameters for each shape are outlined; only those dominant for each type are shown. The flow separates at the rear of a car because the body is truncated. Two types of separation occur, characterized by the terms “quasi-two-dimensional” and “three dimensional”. Depending on the rear geometry, these two types of separation can be interacting.
Both forms of separation are governed by specific parameters. For quasi-two-dimensional separation, these parameters are boat-trailing, which is defined by the angles of the rear end. The dependence of the two separation types on their governing shape parameters will be discussed separately. Their interaction will then be considered when describing results from the specific car development programs. (Wolf-Heinrich Hucho, 1998, Thomas D. Gillespie, 1992)

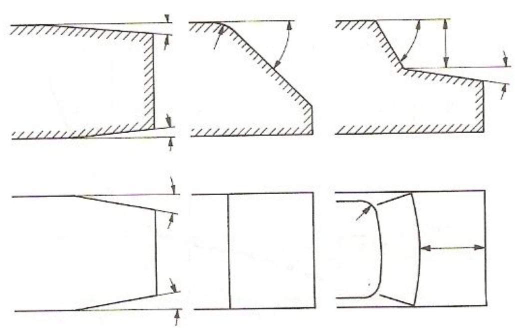


Figure 2.2: The three traditional types of car back and their essential geometric parameters

2.2.3 Boat-Tailing

An aim of shape development is to make the static pressure at the end of a vehicle's body, as high as possible, and the base itself, where this base pressure acts, as small as possible. This requires drawing in (tapering) the rear, a technique called "boat-tailing"; Figure 2.3 shows the extent to which the drag of a body of revolution can be reduced by tapering. The specific optimal angle depends on the upstream history of the flow. Extending the rear end encounters a saturation effect; with increasing length the positive effect on drag becomes progressively weaker. If the rear end is properly truncated, which is called "bob-talling," very little drag-reduction potential is lost. This finding confirms the idea of W. Kamm.

In record vehicles, there is considerable freedom in the choice of rear-ending length; the rear can therefore run out gently. In contrast, for standard production cars boat-tailing must be contained within a given overall length or it is achieved at the expense of interior space. The extent to which drag can be reduced by pulling in the sides of a real car is shown in Figure 2.4 shows how the flow in fact follows the tapered contour. All three examples have in common the fact that the selected boat-tail angle of l0' is only about half as high as the optimum angle for bodies of revolution given in Figure 2.3. (Wolf-Heinrich Hucho, 1998)

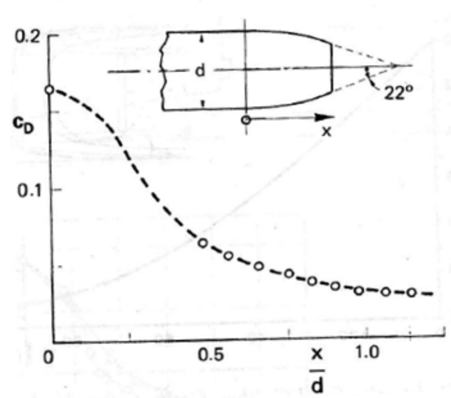


Figure 2.3: Drag reduction of a body of revolution by boat-tailing

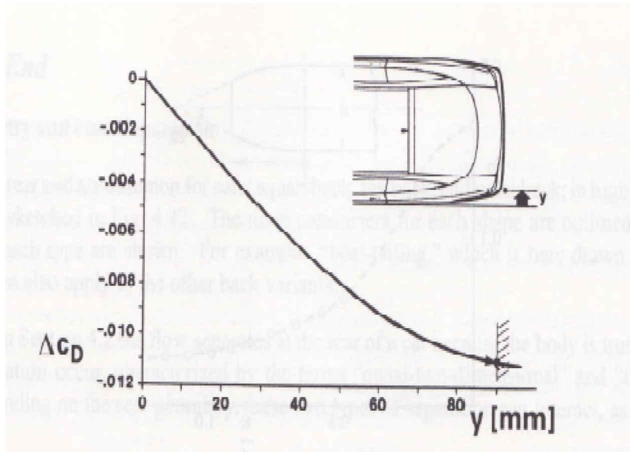


Figure 2.4: Boat-tailing applied to the Opel Calibra Coupe

2.2.4 Pressure Distribution on Car's Body

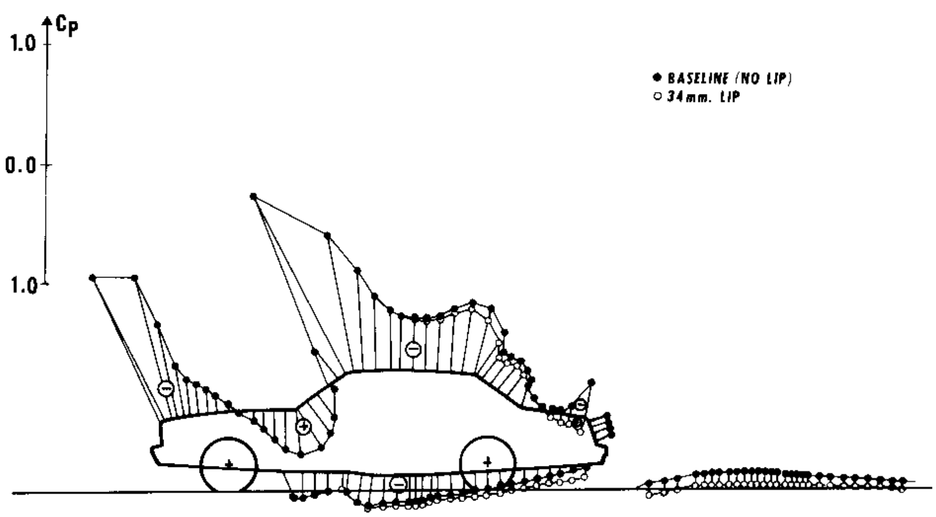


Figure 2.5: Pressure distribution along the centerline of a car. (Thomas D. Gillespie, 1992)

Figure 2.5 show experimentally measured pressure plotted perpendicular to the surface that distributed along the body of car. The pressure is indicated as being negative or positive with respect to the ambient pressure measured some distance from the vehicle. Note that a negative pressure is developed at the front edge of the hood as the flow rising over the front of the vehicle attempts to turn and follow horizontally along the hood. The adverse pressure gradient in this region has the potential to stall the boundary layer flow creating drag in this area.

Near the base of the windshield and cowl, the flow must be turned upward, thus high pressure is experienced. The high-pressure region is an ideal location for inducting air for climate control system, or engine intake, and has been used for this purpose in countless vehicles in the past. The high pressure is accompanied by lower velocities in this region, which is an aid to keeping the windshield wipers from being disturbed by aerodynamic forces.

Over the roof line the pressure goes negative as the air flow tries to follow the roof contour. The pressure remains low down over the backlite and on the trunk because of the continuing curvature. It is in this area that flow separation is most likely. Design of the angles and details of the body contour in this region require critical concern for aerodynamics. Because of the low pressure, the flow along the sides of the car will also attempt to feed air into this region and may add to the potential for separation. The flow along the sides is drawn up into the low-pressure region in the rear area, combining with flow over the roof to form vortices trailing off the back of the vehicle. (Dr. V Sumantran and Dr. Gino Sovran, 1996, Bruce R. Munsan. Donald F. Young and Theodore H. Okiishi, 1996, Thomas D. Gillespie, 1992)

2.3 AERODYNAMICS THEORY

2.3.1 Bernoulli’s Equation for Pressure

The gross flow over the body of a car is governed by the relationship between velocity and pressure expressed in Bernoulli’s Equation. The surface pressure distribution of a given body must be known in order to evaluate the aerodynamic loads on it. (Bernoulli's Equation assumes incompressible flow, which is reasonable for automotive aerodynamics, whereas the equivalent relationship for compressible flow is the Euler Equation.) The equation is: (Thomas D. Gillespie, 1992)

$$P_{static} + P_{Dynamic} = P_{total}$$

$$P_s + \rho V^2 = P_t \qquad (2.1)$$

Where ρ is density of air and V is velocity of air.

Daniel Bernoulli's equation defines the physical laws upon which most aerodynamic concepts exist. This equation is absolutely fundamental to the study of airflows, and any attempt to improve the flow field around a Formula 1 car is governed by the natural relationship between the fluid (air), speed and pressure. Bernoulli's equations were derived using the assumptions that the air density does not change with pressure i.e. air remains incompressible, and does not take into account any height term as it is not relevant in this situation. (Luca Iaccarino, 2003). Bernoulli's equation states that the static plus the dynamic pressure of the air will be constant (P_t) as it approaches the vehicle. Visualizing the vehicle as stationary and the air moving (as in wind tunnel), the air streams along lines, appropriately called streamline. A bundle of streamlines forms a streamtube. The smoke streams used in a wind tunnel allow streamtubes to be visualized. (Thomas D. Gillespie, 1992)

2.3.2 Drag and Lift

It is now known that there are two basic categories of aerodynamic forces acting on the vehicle. The first is pressure, which acts normal (perpendicular) to the surface and is responsible for a vehicle's lift and part of the drag. The second is shear force, which acts parallel to the body's surface and contributes only to drag. The resultant forces due to these contributions can be divided into three components: moment, drag and lift coefficients but here only drag and lift will be concentrated upon since the force due to moment is only important in cases of strong cross winds and when overtaking which are not conditions investigated in the present study. The direction of the drag force is parallel to the vehicle's motion and points toward the back of the vehicle; the side force is positive and points out from the side of the vehicle and finally the lift acts upwards, normal to the ground, and vice versa in terms of downforce. The reason for defining the drag and lift as non dimensional coefficients is that the value of the coefficients is "independent of speed and will be related to the vehicle's shape only". (Dr. V Sumantran and Dr. Gino Sovran, 1996)

Aerodynamic forces are very important especially at speeds beyond 200 km/hr, but more importantly, to obtain the data in a non dimensional form the measured forces must be divided by the square of the velocity. "The important conclusion is that these coefficients are independent of vehicle speed and only depend on vehicle shape" (Dr. V Sumantran and Dr. Gino Sovran, 1996, Luca Iaccarino, 2003). The definitions of lift C_l and drag C_D are given below:

$$C_D = \frac{Fd}{½\rho V^2\ A} \qquad\qquad C_l = \frac{Fl}{½\rho V^2\ A} \qquad\qquad (2.2)$$

Fd is the drag and L is the lift, and A is the frontal area.

2.3.3 Boundary Layer

For this project, only external flow will be considered. In still air, the undisturbed velocity V_∞ is the road speed of the car. Due to fluid viscosity, the fluid velocity near a stationary surface is zero as explained previously, while a thin layer exists where the velocity parallel to the plate gradually increases to that of the outer velocity. This layer of rapid change in the tangential velocity is called the boundary layer, and its thickness increases with the distance along the plate. The boundary layer exists on all surfaces such as on the car body. The thickness of the boundary layer may vary along the surface as for example the "thickness is only several mm at the front of a car travelling at 100 km/hr, and can be several cm thick toward the back of a streamlined car". A thicker boundary layer creates more viscous friction and can also lead to flow separation which leads to additional drag and a loss of downforce in the case of a wing section. There are two types of boundary layer: laminar and turbulent. Normally it starts laminar and develops into a turbulent boundary layer. The length of the boundary layer and the Reynolds number in that area determine whether it will develop into a turbulent boundary layer, and the area where it changes form laminar to turbulent is called the transition region. In the turbulent boundary layer the momentum loss, surface friction and drag are all greater since this is the thickest part of the layer. The variation in the boundary layer thickness along a flat plate may be seen in Figure 2.6.

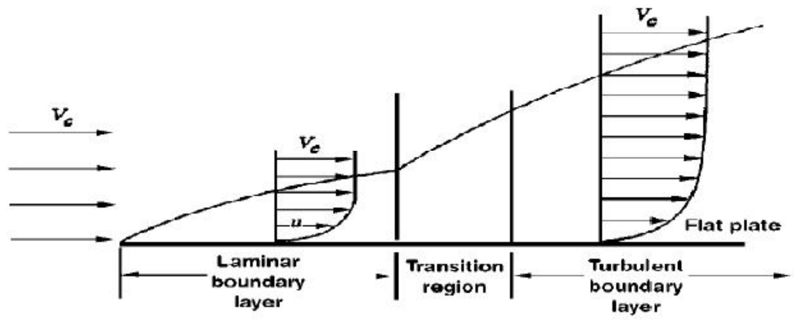


Figure 2.6: Boundary layer along a thin plate

The pressure distribution imposed by the external flow determines if the boundary layer flow will be laminar or turbulent. If the pressure increases in the direction of the flow, the flow can separate, and again if the flow travels along a curved surface the flow can separate and then reattach itself if the curvature is large enough. When flow separates and reattaches, this means that the boundary layer has turned turbulent as the latter can stay attached at higher pressure gradients. However, when separation occurs without reattachment the drag increases. Therefore, it is custom practice to introduce vortex generator along a surface to induce turbulence in the flow enabling it to remain attached for longer. (Luca Iaccarino, 2003, Bruce R. Munsan. Donald F. Young and Theodore H. Okiishi, 1996)

2.3.4 Flow Separation

At some point the flow near the surface may actually be reversed by the action of the pressure as shown in Figure 2.7. The point where the flow stops is known as the separation point. At this point, the main stream is no longer attached to the body but is able to break free and continue in a more or less straight line. Because it tries to entrain air from the region behind the body, the pressure in this region drops below the ambient. Vortices from and the flow is very irregular in this region.

The phenomenon of separation prevents the flow from simply proceeding down the back side of a car. The pressure in the separation region is below that imposed on the front of the vehicle, and the difference in these overall pressure forces is responsible for "form drag". The drag forces is arising from the action of viscous friction in the boundary layer on surface of the car is "friction drag". (Thomas D. Gillespie, 1992, Wolf-Heinrich Hucho, 1998)

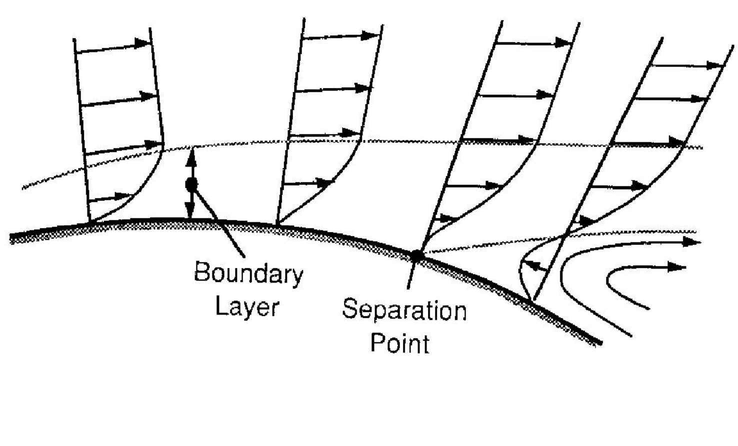


Figure 2.7: Flow Separation in an adverse pressure gradient

2.3.5 Flows over Vehicle

In fluid mechanical term, road vehicles are bluff bodies in very close proximity to the ground and their detailed geometry is extremely complex. Internal and recessed cavities which communicate freely with external flow (i.e. engine compartment and wheel wells, respectively) and rotating wheels add to their geometrical and fluid mechanical complexity. The flow over a vehicle is fully three-dimensional. Boundary layers are turbulent. Flow separation is common and maybe followed by reattachment. Large turbulent wakes are formed at the rear, and in many cases they interact with longitudal vortices shed from the afterbody. As a typical for bluff bodies, drag is primarily pressure drag. Accordingly, the avoidances of separation or, if this is not possible, its control is among the main objectives of vehicle aerodynamics. (Wolf-Heinrich Hucho, 1998)

To understand this flow work, we can visualize a car moving through the air. A fluid may exert forces and moments on a body in and about various directions. Flow over immersed bodies / external flow would produce two forces on the bodies that is drag force and lift force. The force a flowing fluid exerts on a body in the flow direction is called drag, or force to resist movement in the direction of travel while the components of the pressure and wall shear stress in the direction normal to the flow (perpendicular) tend to move the body in that direction, and their sum is called lift.

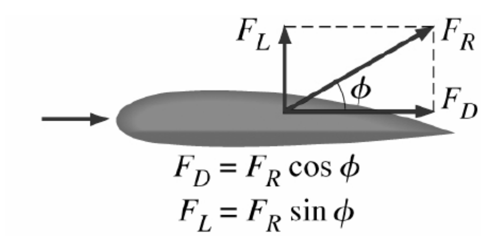


Figure 2.8: Flow over a body

In vehicle aerodynamics, drag is due to both friction drag and pressure drag for a cylinder normal to the flow (mostly pressure drag). The pressure drag is proportional to the frontal area and to the difference between the pressures acting on the front and back of the immersed body. The pressure drag is usually dominant for blunt bodies and becomes most significant when the velocity of the fluid is too high. Friction drag is the components of the wall shear stress force in the direction of flow. Friction drag is zero for a flat surface normal to flow, and friction drag is maximum for a flat surface parallel to flow. Thus it depends on the orientation of the body as well as the magnitude of the wall shear stress. Thus, in case blunt body friction drag is less at higher Re, and may be negligible at very high and the drag in such cases is mostly due to pressure drag (higher *Re*). It can say that at low *Re*, most drag is due to friction drag and at higher *Re*, most drag is due to pressure drag. The shape of an object also changes the amount of drag. Most round surfaces have less drag than flat ones. Narrow surfaces usually have less drag than wide ones. The more air that hits a surface, the more drag it makes. C_D depends on many factors such as shape of body (round, square, etc), Reynolds number of the flow (laminar or turbulent flow), surface roughness (smooth with laminar flow or rough turbulent flow) and the influence of other bodies or surface nearby (any object disturbing the flow). (Bruce R. Munson, Donald F. Young, Theodore H. Okiishi, 2006)

Aerodynamic Drag Coefficient:

$$C_d = \frac{Fd}{½\rho V^2\, A} \tag{2.3}$$

Where:

F_d = drag force

A = A is frontal area (the projected area.)

ρ = is the density of the medium through which the object is moving

V = is the velocity of the object

In this project, the investigation numerically the flow around the body for the base on roof shape, which is rear slant several angle, included the shape of rear bonnet.

Results are will be compared with experimental data. The study of three-dimensional flow around a ground vehicle has become a subject of significant importance in the automobile industry. One obvious way of improving the fuel economy of vehicles is to reduce aerodynamic drag by optimizing the body shape. Execution of good aerodynamic design under stylistic constraints requires an extensive understanding of the flow phenomena and, especially, how the aerodynamics is influenced by changes in body shape. The flow region which presents the major contribution to a car's drag is the wake flow behind the vehicle. The location at which the flow separates determines the size of the separation zone, and consequently the drag force. Clearly, a more exact simulation of the wake flow and of the separation process is essential for the accuracy of drag predictions. A real-life automobile is very complex shape to model or to study experimentally. However, the simplified vehicle shape will generates fully three-dimensional regions of separated flow which may enable a better understanding of such flows. (Emmanuel Guilmineau, 2007)

Figure 2.9: Body view (a) 25º rear slant (b) 35º rear slant

2.4 COMPUTATIONAL FLUID DYNAMIC (CFD)

2.4.1 CFD as a Tool for Aerodynamics Simulation

Computational Fluid Dynamics, CFD, emerges as an advanced investigative means in the fields of automobile, aeronautics and aerospace. It reduces the wind tunnel experiments. In this project, it is stated that CFD is applied to the automotive aerodynamics with Finite Element Method, FEM. By that we get the pressure field and velocity field around the vehicle, then acquire air drag coefficient, C_D .and compare it with the wind tunnel experimental results if necessary.

Automotive aerodynamics characteristics are very important due to the increasing improvement of the automobile. At present, wind tunnel is a chief research means in our country, but Computational Fluid Dynamics, CFD, has developed very fast in some countries, with the appearance of high performance computer and accurate analytic method. As a modern method, CFD can not only shorten the automobile design period, but also greatly reduce the wind tunnel experiments.

In this project, it will study certain of car. In order to obtain the accurate result with less time and cost, adopt simplified model of the car, which can efficiently stand out the main facets of the problem. On the CAD platform such as Solid Work the geometrical model is set up, then is imported into the preprocessor software ALGOR. After meshing it, it will get the finite element model, which is exported into CFD software (FLUENT) to precede fluid analysis. On the base of analysis result it can display the pressure contour and the velocity vector of the external field of the body. By the postprocessor we acquire the air drag coefficient, $C_{D.}$

For subsequent aerodynamic studies, individual engine parts are not modeled. In this study, the air cooling vents and the engine space above the subframe will be closed. Using this finished base case, numerous types of vehicles can be generated on the same template for simulation in a short time period. A half model with a centerline symmetry plane is sufficient for initial design studies, but the model car in this study will constructed as a full model with an asymmetric geometry (where the asymmetries are mostly confined to the underbody). This is done to enable detailed predictions of the flow field around and under the vehicle. The wind tunnel geometry around the model is a rectangular enclosure. It is of such a dimension that the adverse pressure effects between the vehicle and the wall are minimized. (Werner Seibert, Robert Lewis, 2004, Sun Yongling, Wu Guangqiang, Xieshuo)

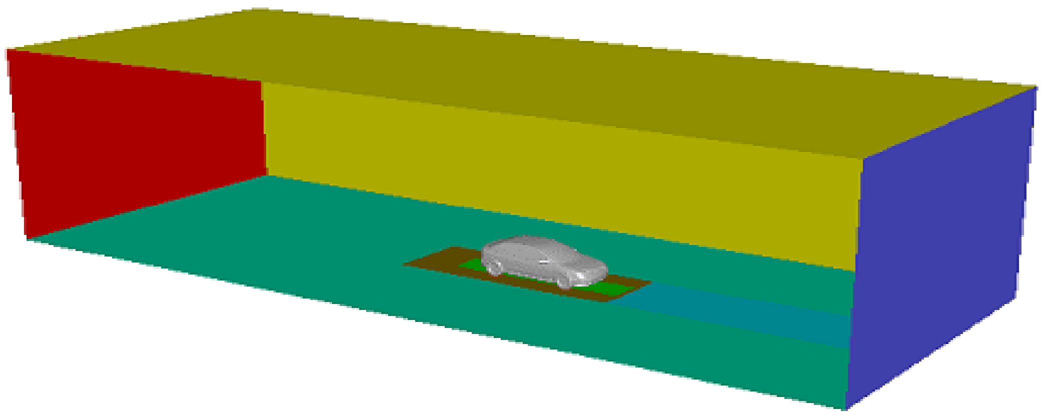


Figure 2.10: CFD analysis in boundary condition

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

Generally, this project involved in designing, simulating and analyzing result. The model of car that was used in this project is Proton Saga (BLM). Virtually the drawing process is designed by SolidWork. Before all parameter design prepared, the original (standard model) must be analyzed first to ensure that all the design can be analyzed in CFD, the testing must be performed first because CFD software is very sensitive, the simulation result can be error if even small hole found at the model. Then, after the result from the testing model not showing any error, all parameter design can be prepared based on that model the. Then, all design than has been analyzed using Computational Fluid Dynamic software (CFD) by using Cosmos FloWorks and all the analysis done at FKM's computer lab.

3.2 Flow Chart Methodology

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, a terminology of works and planning show in the flow chart at Figure 3.1. This is very important because to make sure the goal or what purpose this project is not wrong.

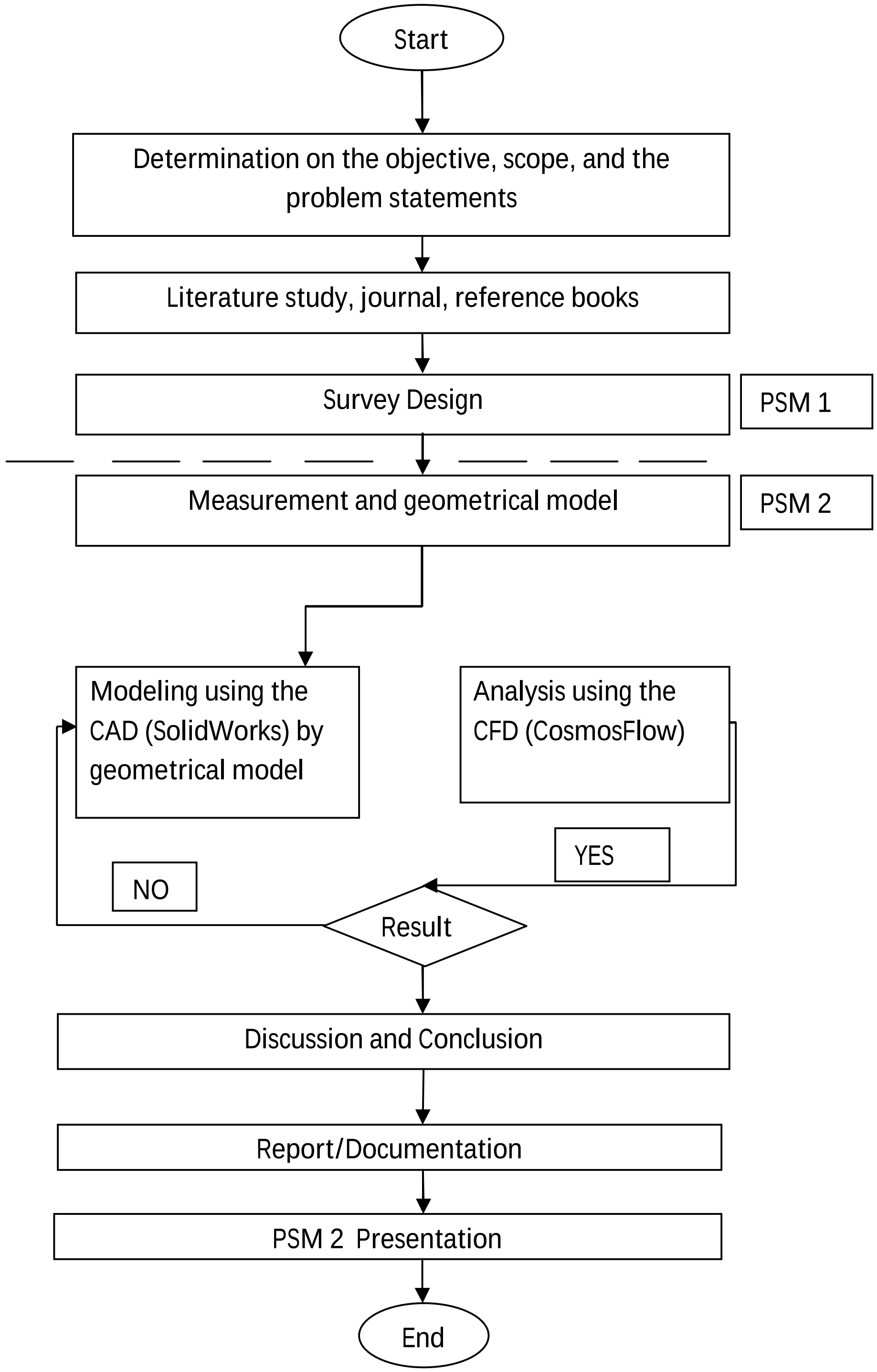


Figure 3.1 Flowchart of the Overall Methodology

3.3 Data Collecting

The data of dimension for Proton Saga BLM body was taken from the brochure then modeling the body by SolidWorks software. The dimension is taken as accurate as possible but some points have to take as approximately because of some limitation for example lack of better tool. The dimension is very important for Proton Saga BLM body to simulating the model in CFD analysis to get the appropriate or expected value of drag coefficient.

3.4 CAD Modeling

After collecting the dimension of the car body, the CAD model was starting to build in SolidWorks software assisted by supervisor. Most of time spend during this project is CAD modeling because after a model have created, it must test first in order to get the best value of C_D. After that, the model must be refining as an improvement from the previous model. The clearance from the ground to the lowest point of the model is about 163mm approximately. This value is taken from standard model of sedan car. The ground clearance value is also important because it will influence the total value of drag.

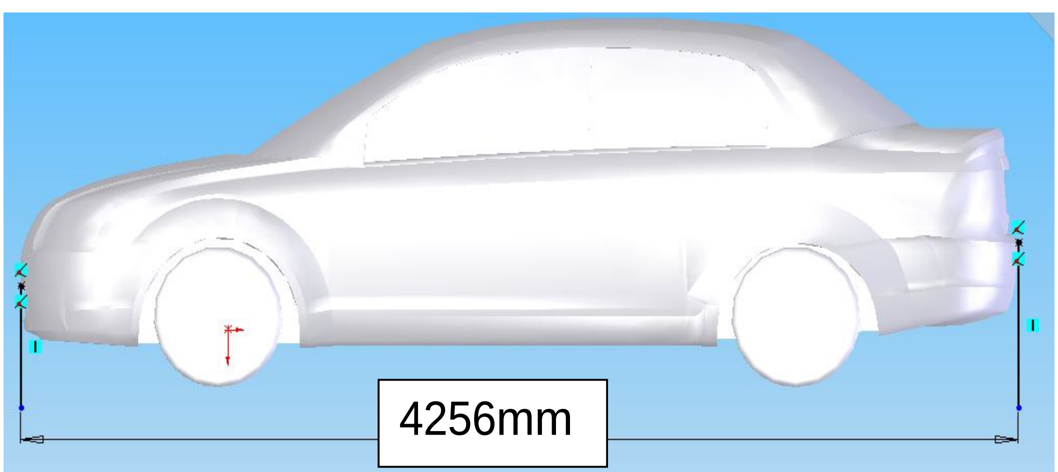


Figure 3.2 Side view CAD model of Proton Saga BLM

3.5 CFD Analysis

After the car body was modeled in CAD, the next stage is importing the CAD model into CFD software. CFD analysis in this project will be run inside the COSMOSFloWorks software. This is sometimes called solving. At this stage, the car speeds will analyze at constant value that is 110km/h. This value was selected because this is maxima speed in Malaysia highway. The boundary condition for this analysis is external flow with adiabatic wall with roughness value of 0 micrometer. Beside that, the types of flow considered are laminar and turbulent flow for CFD model analysis. For reference axis, it must be select as x-direction. The reference axis is chosen so that an angular velocity vector may be aligned with the reference axis. The most important in this stage is the goal setting; the goal must be set as component of force on x-direction and also velocity parameter must be correct in direction and also the positive and negative sign must be correct.

3.6 Frontal Area Measuring

Before calculating the drag coefficient, the frontal area must be known first. From SolidWorks software, the frontal area can be constructed by sketching the projected area of the model 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. The area of Proton Saga BLM is about $1.915m^2$.

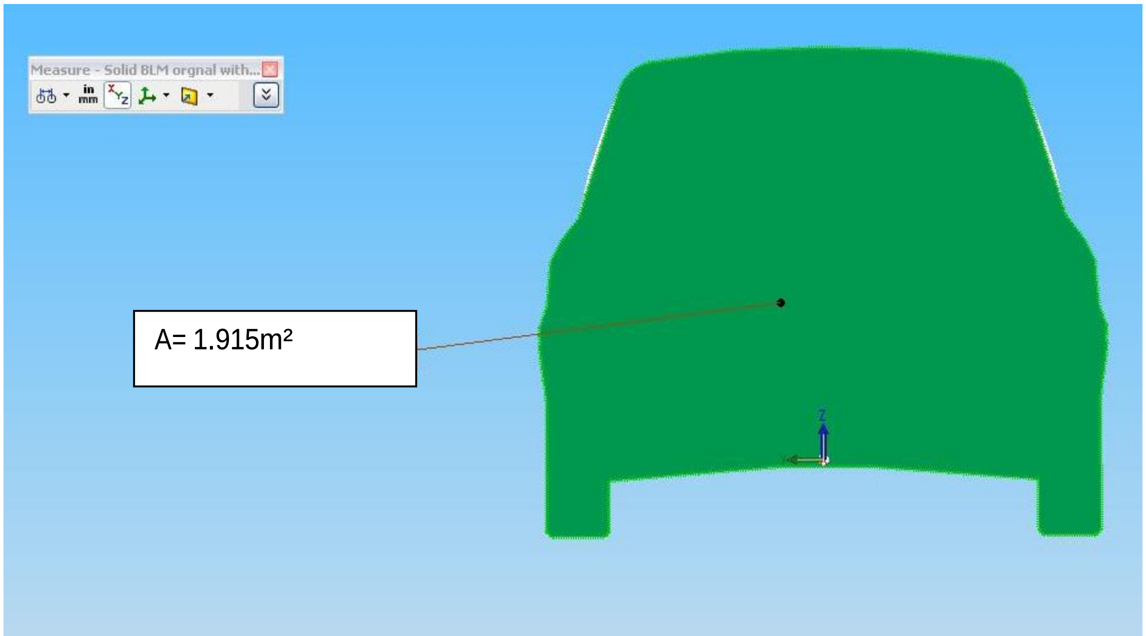


Figure 3.3: The frontal area projected of CAD Proton Saga BLM model

CHAPTER 4

RESULT AND DICUSSION

4.1 INTRODUCTION

This chapter discusses about the designing, and also the analysis result. For the first thing, the design is obtained from a few concept designs by considering the dimensions and previous research. For this project, the analysis of the car model is only consider the sedan type of car that means the model required a minimal length of distance of rear bonnet to make this model look as sedan type. So, the length is about ± 350mm.

After the various designs shape was build in CAD software (SolidWorks), and afterwards the model analyzed by using the COSMOSFloWorks software (CFD). The result of the experiments will be discussed in this chapter.

One simulation in CFD for each shape taken about 30 minutes and drag force was selected as goal in the simulation. Then, the force value from the simulation insert to the formula to get the C_D value and afterwards simulation was repeated for all shape that was set up before.

4.2 DATA COLLECTION AND ANALYSIS

4.2.1 Important Parameter of flow analysis

All the important parameters that involve in this analysis for simulation was defined before run the calculation of CFD. The reference points of flow analysis are detailed below:

(a) Unit system: SI unit (m-kg-s)

(b) Analysis type: Air

(b) Ambient parameter conditions

i. Ambient Pressure: 101325 Pa

ii. Ambient Temperature: 300.15 K

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

(c) Analysis type: External Flow

(d) Wall Condition: Adiabatic (Incompressible Flow)

4.2.2 Drag Coefficient Calculation

From the data collection of Table 4.1 and Graph 4.1, the drag coefficient, C_D was calculated based on the equation (2.3) and also the parameter that involved in this analysis such as density of air, frontal area and velocity of air. All the calculation is including each designs of shape of rear slant angle.

4.2.3 Sample Calculation of Coefficient of Drag, C_D

Below is the sample of calculation for drag coefficient, C_D.

1. Drag Coefficient for 45º rear slant angle:

$$C_D = \frac{373.176}{\frac{1}{2}(1.225)(1.915)(30.56)^2}$$

$$= 0.340668622$$

2. Drag Coefficient for 37º rear slant angle:

$$C_D = \frac{363.626}{\frac{1}{2}(1.225)(1.915)(30.56)^2}$$

$$= 0.331950523$$

4.2.4 Data Collection for Rear Slant Angle

The drag forces data collections that were getting started from CFD analysis for various angle of rear slant angle was listed in the Table 4.1 and the graph from the data has been plotted in Graph 4.1. The rear end angle was specified in this analysis is 45º, 43º, 40º, 37º, and 35º.

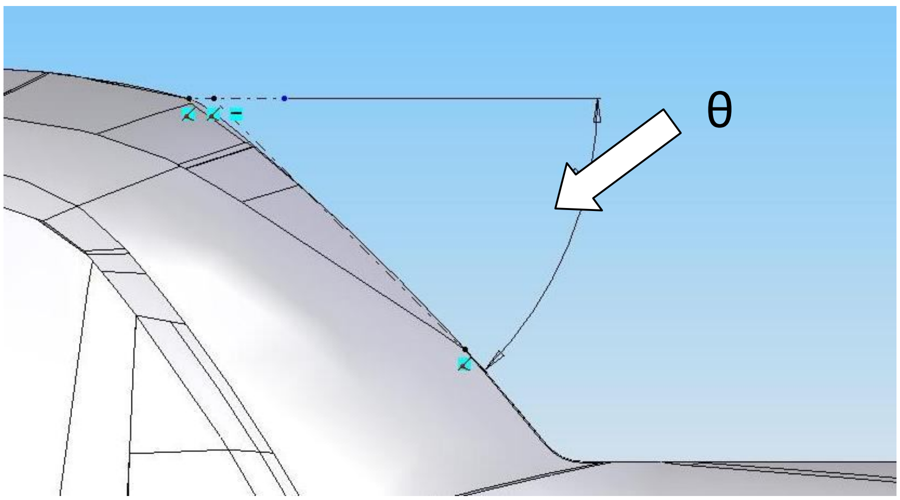
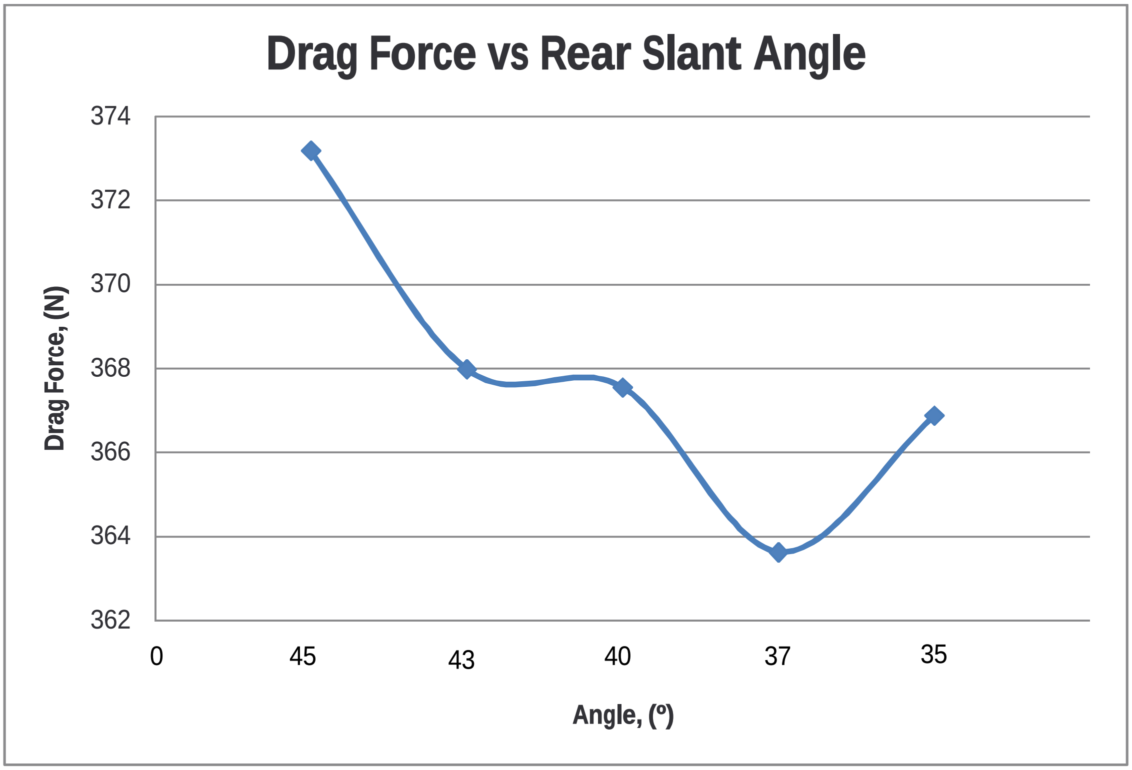


Figure 4.1: Rear slant angle of Proton Saga BLM

Table 4.1: Table of various rear slant angles and drag forces

Angle, (º)	**Drag Force, (Newton)**
45	373.176
43	367.98
40	367.546
37	363.626
35	366.877

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



Graph 4.1 Graph drag force, D against rear slant angle

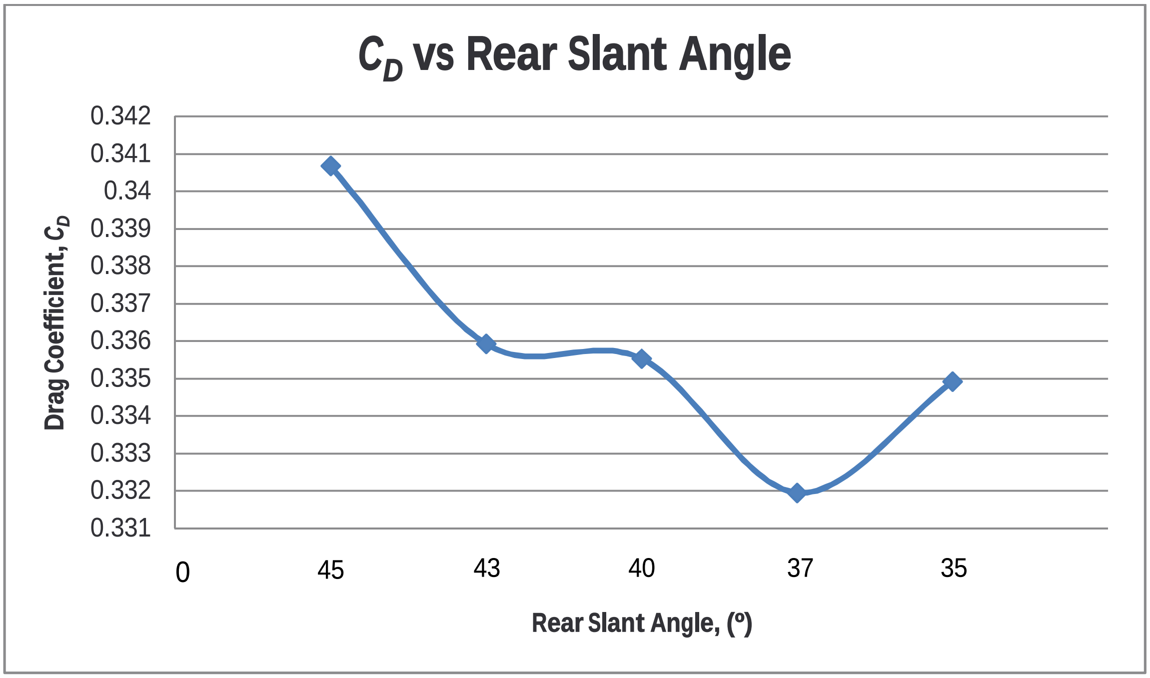
From the Graph 4.1 above shown the trend of the point that drag force decrease with decrease of rear slant angle, but after at one point, drag force increased again. Thus, it means the shape or the rear slant angle influecnced with total force of drag.

4.2.4.1 Drag Coefficient, C_D on Rear Slant Angle

After the C_D was calculated for every angle, the value of C_D was listed in a table 4.2 below.

Table 4.2: Table of various rear slant angles and C_D

Angle, (°)	**Coefficient of drag, C_D**
45	0.340668622
43	0.335925246
40	0.335529052
37	0.331950523
35	0.334918328



Graph 4.2 Graph drag coefficient, C_D against rear slant angle

From the graph 4.2 above, the trend of the result of drag coefficient can be observed clearly. It shown drag coefficient, (C_D) slightly decreases to the decrease of the rear slant angle from 45º to 37º. The analysis is starting with the 45º rear slant angle and from calculation, it gives the coefficient of drag is 0.340668622. After first analysis was done, simulation was proceeding with second shape that is 43º rear slant angle and C_D dropped to 0.335925246. It decreases about 0.004743376. Then, there is just a little bit difference changes in C_D value between 43º and 40º. After that, C_D is become dropped again to 0.331950523 after the simulation done. But after the simulation done for the 35º rear slant angle, the C_D become increase to 0.334918328 and from this result, the lowest C_D is at 37º rear slant angle. Notice that after the each simulation was done, the differences of C_D on each design parameter is very small. This happen because only small changes angle are considering in this simulation. But the important thing is the forces values were clearly show it decrease when the angle decreases until reach 37º angle.

4.2.4.2 Pressure Distribution on Vehicle

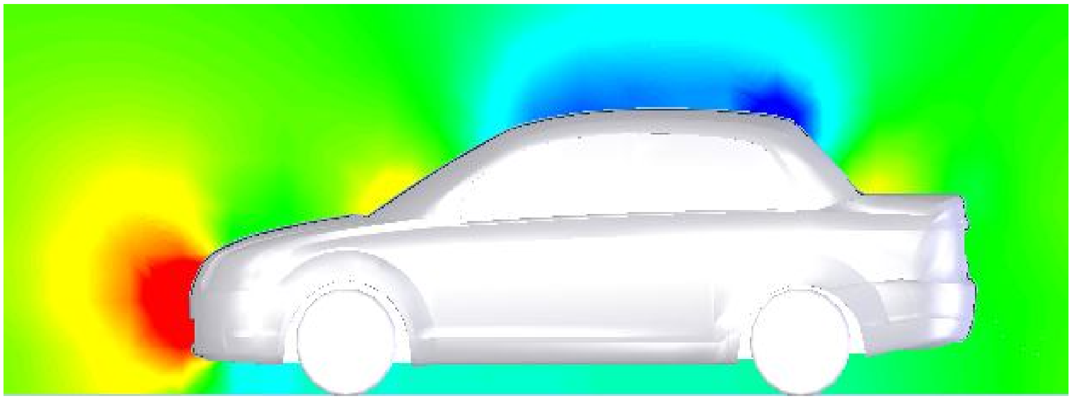


Figure 4.2: Pressure distribution on 45º rear slant angle

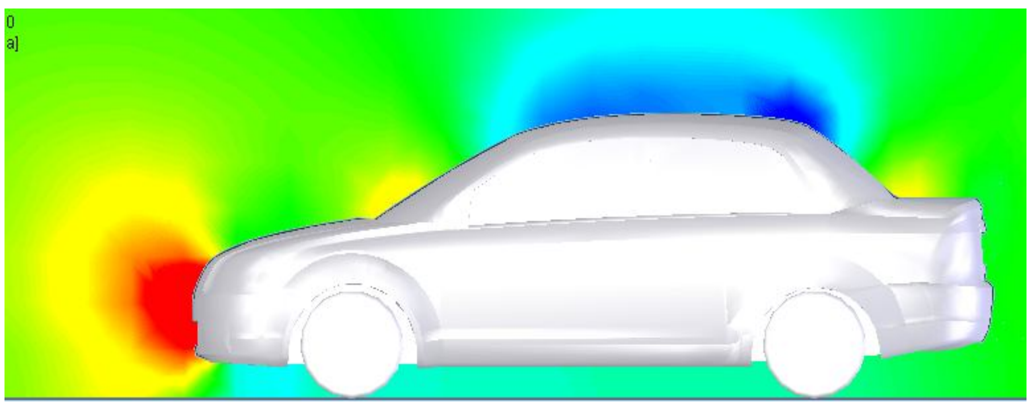


Figure 4.3: Pressure distribution on 37º rear slant angle

From figure 4.2 and 4.3 above, nothing much differences on pressure distribution between 45º and 37º. The comparison between these two shapes is based on the results that give the highest and lowest $C_{D.}$ The reason of there is nothing big differences on the pressure distribution is because nothing much shape was change between two models and it just only involved rear slant angle is different and the different of each angle also too small. This pressure differences cannot be observed on this scale. From the figures above, the red color region in front hood of the car is indicated as highest pressure acting on it. This happen because most of the air flows is hit on this area first before it hit to the other area. The high pressures are accompanied by lower velocities in this region due to Bernoulli theorem. The dark blue color region indicated low pressure because of the velocity of the air flow decrease when it tries to follow the roof contour. The yellow color indicated a pressure that below of the high pressure (red color) and above of the atmospheric or ambient pressure that represent as green color, (101.325kpa).

4.2.4.3 Contour Plot of Velocity

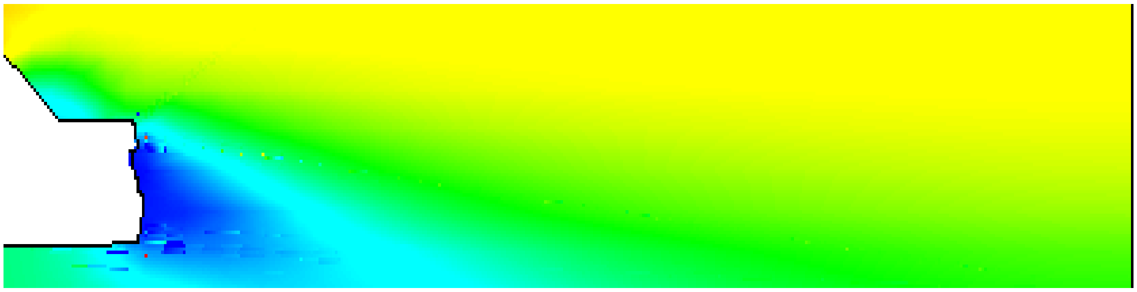


Figure 4.4: Contour plot for 45º rear slant angle

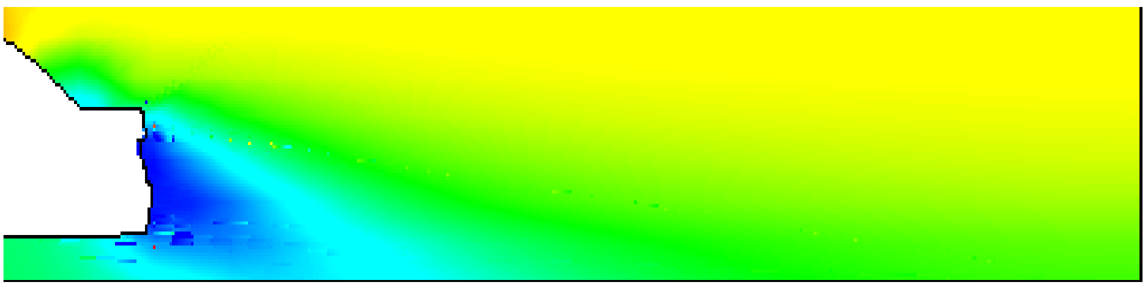


Figure 4.5: Contour plot for 37º rear slant angle

Figure above is showing the comparison of wake region and separation area between 45º and 37º rear slant angle. Comparison between this different is chosen because of the C_D value for 45º is highest while 37º is lowest. If focus on the wake region at the rear of car, the region area for 45º is bigger than 37º. While the separation area and wake region area actually affects drag directly, the extent to which the flow is forced to turn down behind the vehicles affects the aerodynamic lift at the rear. Separation must occur at some point, and the smaller the area, generally the lower the drag. Flow control that minimizes the separation area generally results in more aerodynamic lift at the rear because of the pressure reduction as the flow is pulled downward. (Thomas D. Gillespie, 1992). Besides that the blue color regions that occur at the behind of the car indicate that this region is vacuum and the velocity at this point is 0m/s.

4.2.5 Data for Rear Bonnet Angle

The bonnet shape was specified with five changes at the rear end of the bonnet. The differences between each design are 5º and the range is between 0º to 20º. The length of the bonnet is remain constant and the change is only involved the angle between the lower point of rear windshield to the end point of bonnet. The first simulation starting with 0º; this is the original shape without any changing at rear bonnet and then it followed by 5º, 10º, 15º, 20º degree as shown in figure below. The result of the all simulation is shown in Table 4.3.

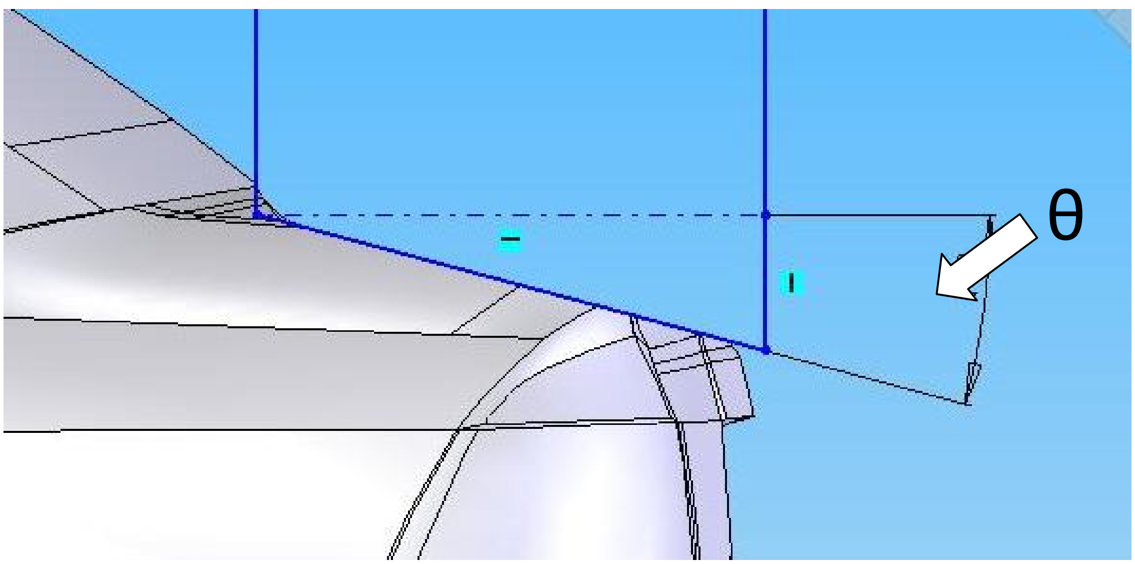
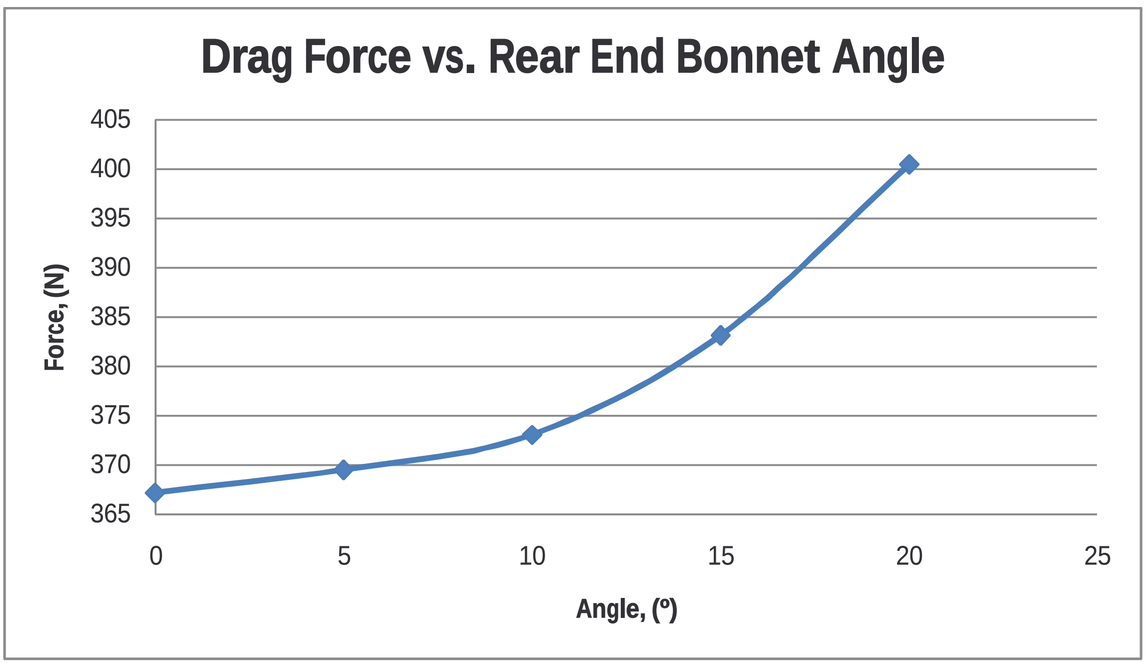


Figure 4.6: Angle of rear end bonnet, θ

Table 4.3: Table of various rear end bonnet angles and drag forces

Angle, (º)	**Drag Force, (Newton)**
0	367.187
5	369.5185
10	373.075
15	383.157
20	400.4715

The data from Table 4.3 was plotted into the graph of drag forces, *D* against the velocity, *V*. Then, the graph series of *D* vs. *V* was shown in Graph 4.3 below.



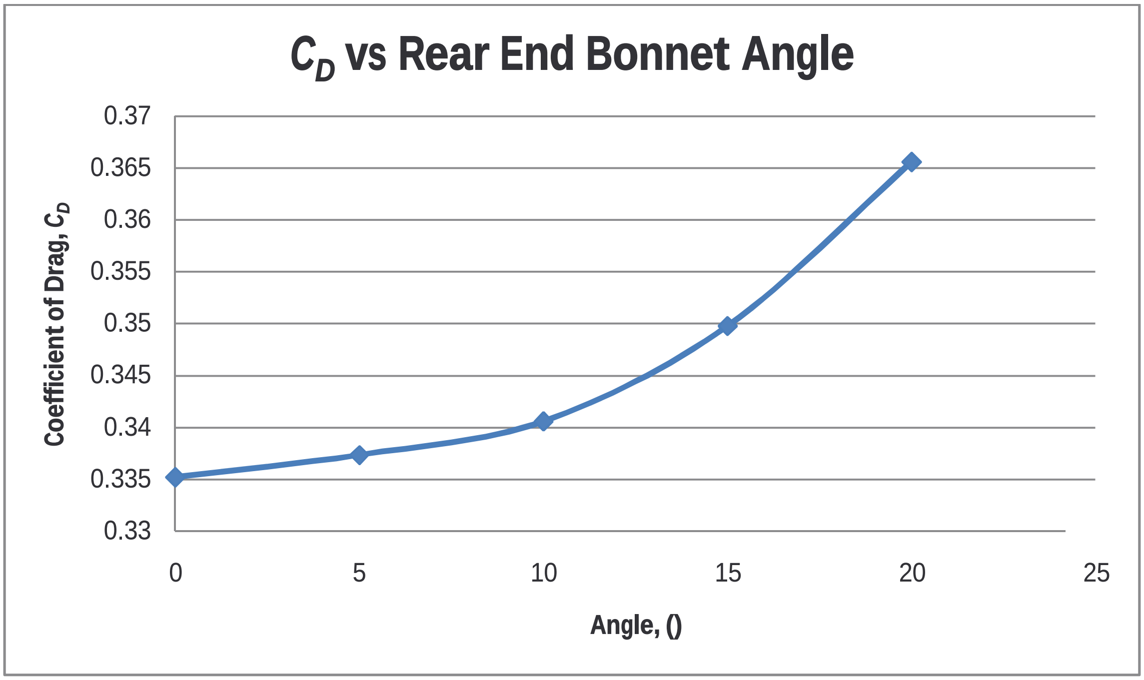
Graph 4.3: Graph drag force against rear end bonnet angle

4.2.5.1 Drag Coefficient, C_D on Rear End Bonnet Angle

After the C_D was calculated for every angle, the value of C_D was listed in a table 4.4 below.

Table 4.4: Table of various rear end bonnet angles and C_D

Angle, (°)	**Coefficient of drag, C_D**
0	0.335201324
5	0.337329727
10	0.34057642
15	0.349780177
20	0.36558641



Graph 4.4 Graph drag coefficient, C_D against rear end bonnet angle

From the data on table and graph above, the trend of the result can be observed clearly which is C_D is increase uniformly with the increase of rear end bonnet angle. The maximum point of C_D is 0.36558641 (force created is 400.4715N) that occur in 20º rear end angle meanwhile the lowest point of C_D is 0.335201324(force created is 367.187N) that occur at 0º rear end angle (original shape of Proton Saga). From the observation of the graph also, the difference between one point to another point is very small just from ± 0.002 in and the differences between maximum point to lower point is just 0.03039. The small differences in each point in this result already expected from beginning because of this of analysis just consider the very small part on the whole of the vehicle and it is not enough parameter if the analysis just only considers the rear end bonnet in order to get bigger differences to each point.

4.2.5.2 Surface Plot Pressure Distribution

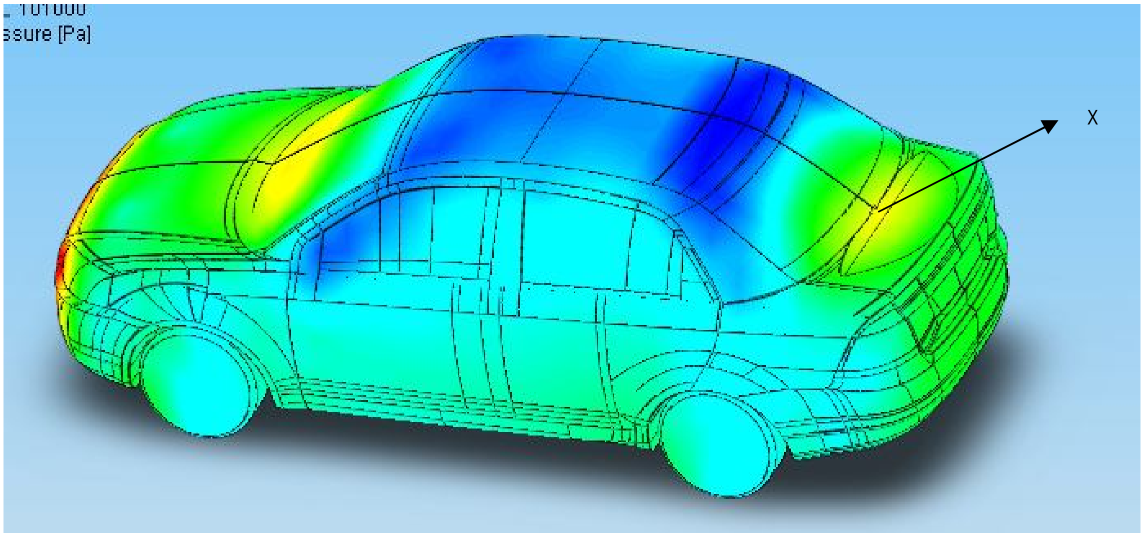


Figure 4.7: Surface plot pressure distribution on 0º rear end bonnet

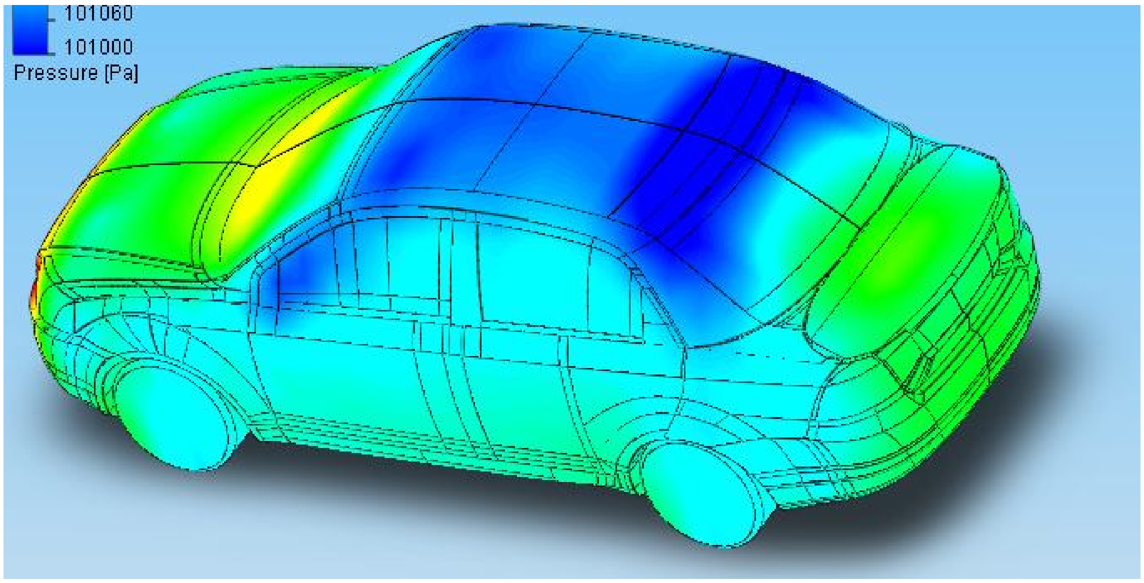


Figure 4.8: Surface plot pressure distribution on 20º rear end bonnet

Two figures above show the comparison of surface pressure between 0º and 20º rear end bonnets. These two gives the highest and lowest C_D. All the surface pressure between these two models is same except for the area at rear bonnet. The 'x' sign at Figure 4.8 show that the pressure at that region for 0º model is bigger than 20º rear model. It means that from 0º to 20º rear end bonnet angle, the pressure at that area in decreasing with increasing of angle of rear bonnet. The reductions of pressure from 0º to 20º at the area will cause the drag increasing directly. In real case, car manufacturer are trying to prevent designing rear body that can gives low pressure at the behind because it will give more drag force and then of course td C_D will increase.

4.2.5.3 Contour Plot of Velocity

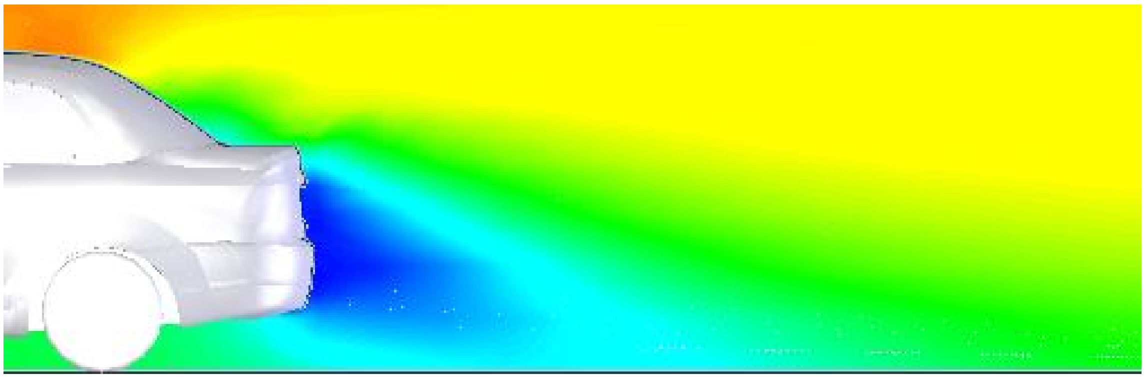


Figure 4.9: Flow on 0º rear end bonnet

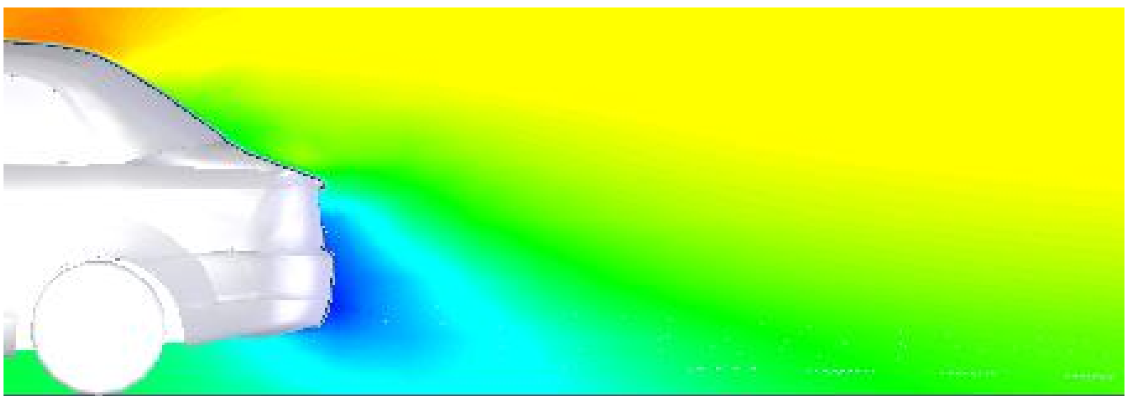


Figure 4.10: Flow on 0º rear end bonnet

Figure 4.9 and Figure 4.10 above show the wake area at the behind of the car. From the observation in the figures, there are also some differences at the size of wake region between two models which is wake region for 20º is bigger than 0º. Even tough drag for 20º is higher than 0º, the pressure at rear portion at bonnet at 20º is lower than 0º and also the separation area for 20º is bigger than 0º. All these factors are contributed the increasing of drag for 20º angle of rear end bonnet.

4.2.5.4 Trajectories Velocity Flow Analysis

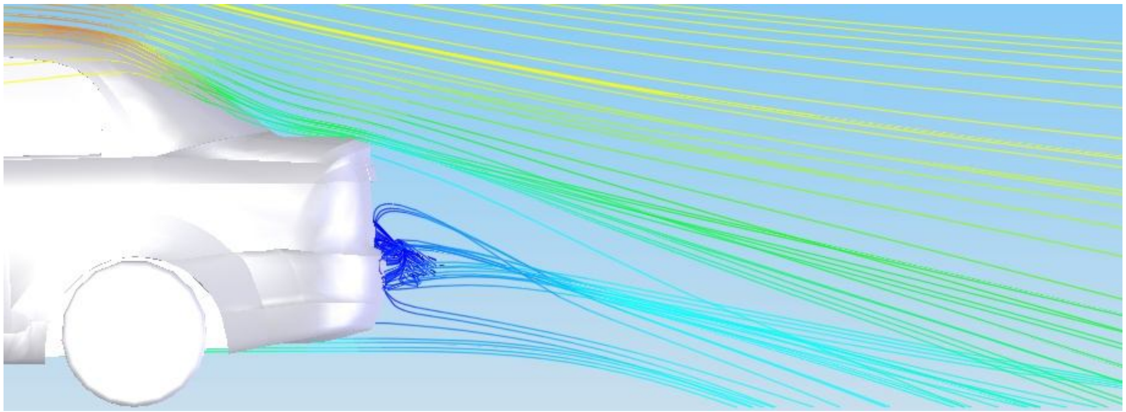


Figure 4.11: Trajectories velocity flow for 0º rear end bonnet

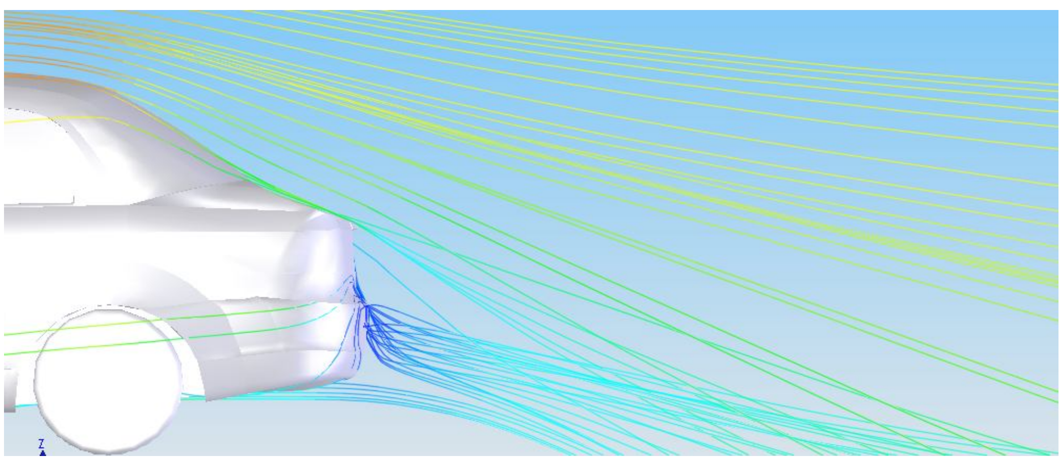


Figure 4.12: Trajectories velocity flow for 20º rear end bonnet

Figures 4.11 and Figure 4.12 also show the differences flow trajectories characteristic between 0**º** and 20**º** rear end angles. On 0**º,** the air flow follows the contour of the rear body (rear windshield) until it hit the flat bonnet while on 20**º** rear end bonnet a little bit difference because the air flow are straightly go down because of the contour of bonnet the is likely connected from rear mirror of vehicle.

4.2.6 Data Collection for Rear Side Bonnet Distance

Rear side bonnet also become one of the part that consider the factor that influence the drag of coefficient. In this part of analysis, five changes was specified with the differences between each distance is 20mm and range from 100mm to 180mm. The figure below is show the part at rear side bonnet that has been changed.

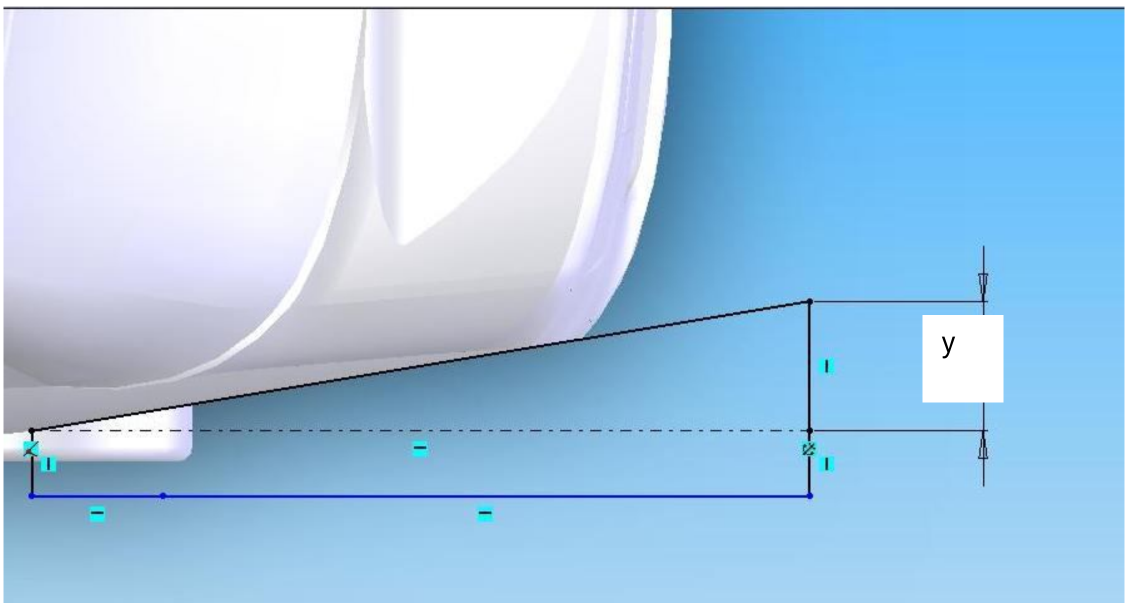
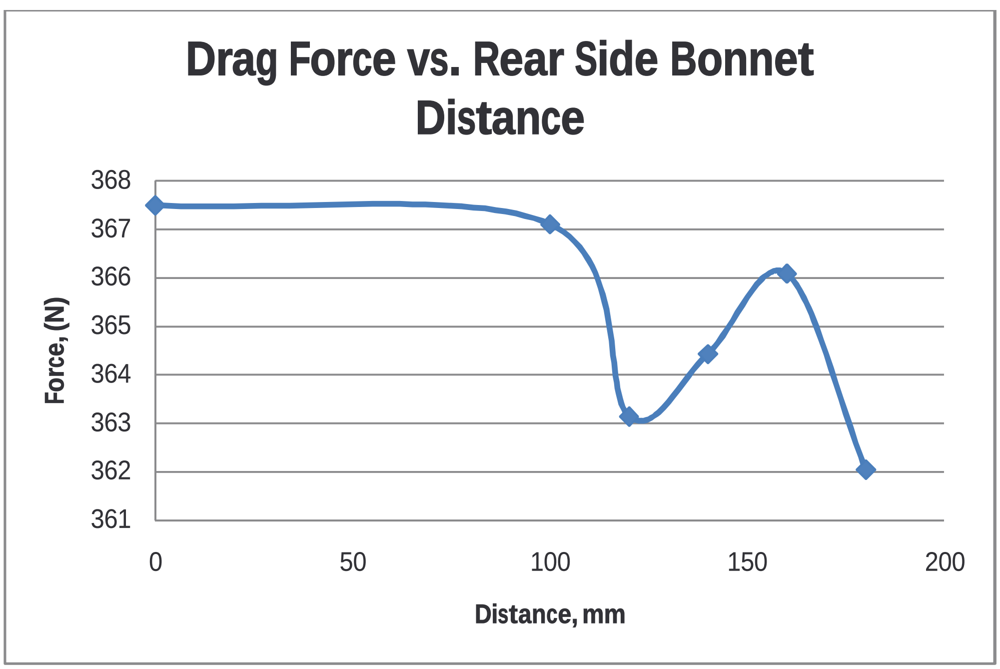


Figure 4.13: Rear side bonnet distance represent as y

Table 4.5: Table of various rear side bonnet distance and drag forces

Distance,y (mm)	**Drag Force, (Newton)**
0	367.497
100	367.107
120	363.145
140	364.435
160	366.092
180	362.051

The data from Table 4.4 was plotted into the graph of drag forces, *D* against the velocity, *V*. Then, the graph series of *D* vs. *V* was shown in Graph 4.4 below.



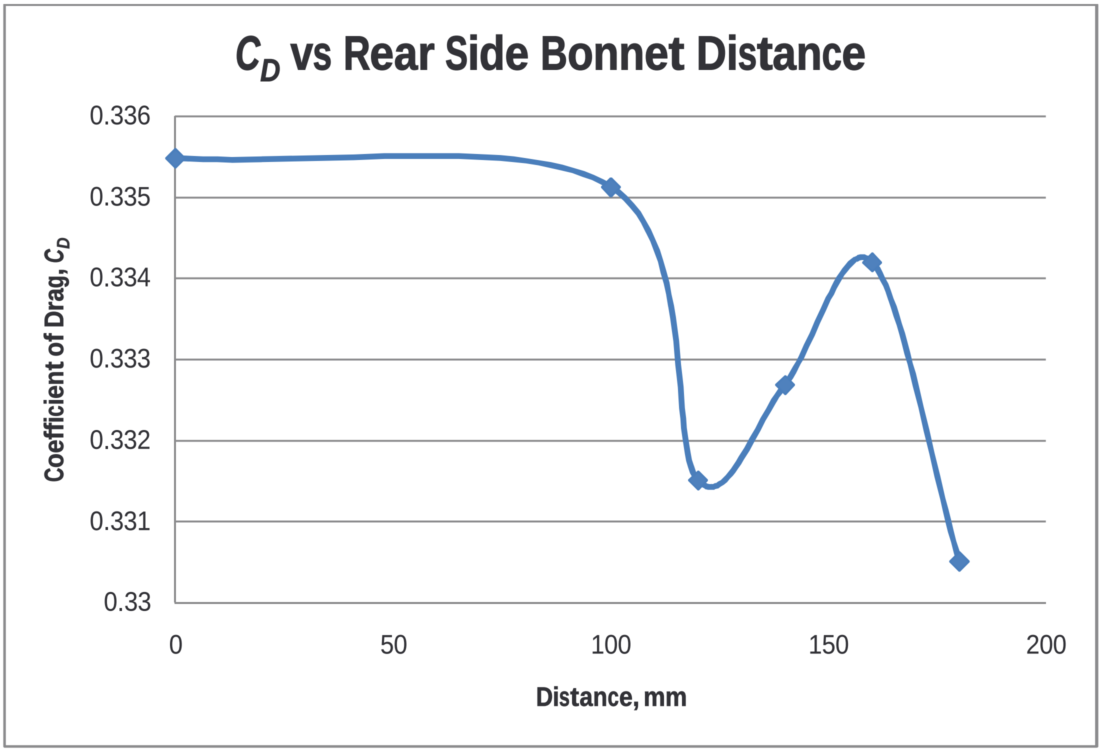
Graph 4.5: Graph drag force against rear side bonnet distance

4.2.6.1 Drag Coefficient, C_D on Rear End Bonnet Distance

After the C_D was calculated for every angle, the value of C_D was listed in a table 4.5 below.

Table 4.6: Table of various rear side bonnet distances and C_D

Distance,y (mm)	**Coefficient of drag, C_D**
0	0.33548432
100	0.335128293
120	0.331511423
140	0.332689051
160	0.33420171
180	0.330512721



Graph 4.6: Graph Drag Coefficient, C_D against Rear side bonnet distance

From the graph above, the highest value of C_D is 0.33548432 with y=0mm of change, which means this is the original shape of this car. Then the C_D values 0.335128293 become decrease for the next two points until the third point (y=120mm) 0.331511423. Then, C_D value increase again for fourth point (y=140mm) and fifth point (y=160mm). For the last point that is y=180mm, the C_D value 0.330512721 decreases to become as a lowest point. The difference at each C_D is small; just ± 0.001 only and this trend of result were expected before the simulation was begun. What is the problem in this result is at the fourth and fifth point which is that two points are not follow the trend of point that supposed to be decrease from first point until the last point. This causes the graph that appear is not uniformly curvy as expected from beginning.

4.2.6.2 Surface Plot Pressure Distribution

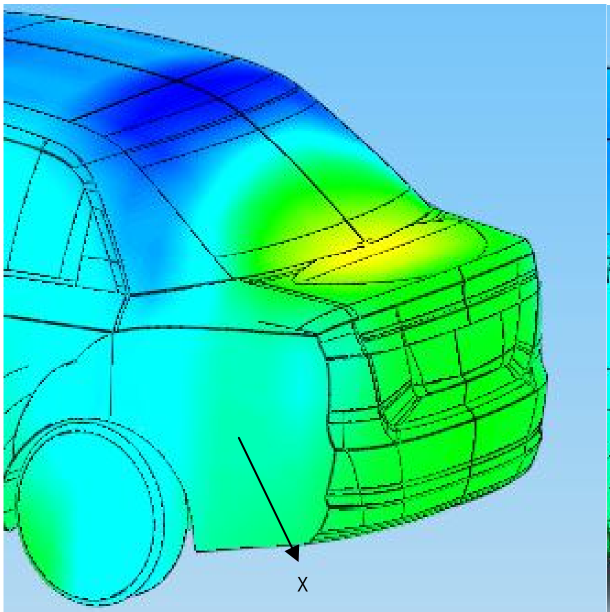


Figure 4.14: Pressure on y=180mm

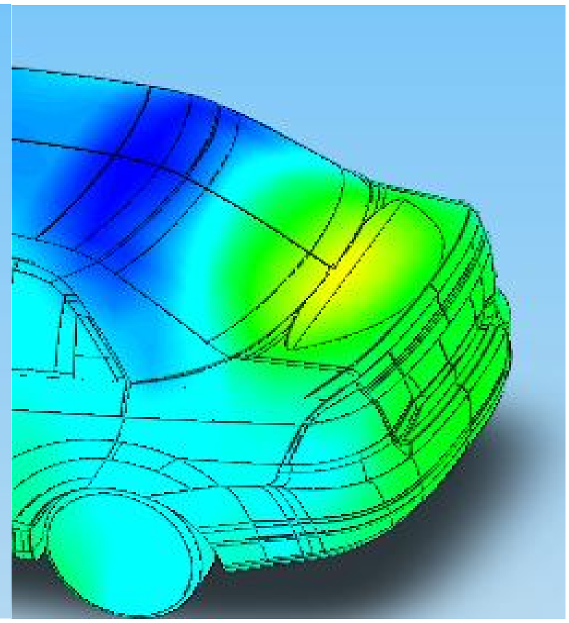


Figure 4.15: Pressure on y=0mm

From the Figure 4.14 and Figure 4.15, it shows surface pressure distribution between y=0mm and y=180mm rear side bonnet that occur in that portion. The comparison involves between this two shapes because of y=180mm shape give the lowest C_D while y=0mm gives the highest C_D. From the observation, surface pressure at y=0mm shape at the side portion show this area occurs negative pressure (decrease pressure respect to the ambient pressure). While the surface pressure at y=180mm shape (‘x’ sign) show that the pressure is at ambient pressure. In other words, the pressure different between these two shapes resulting the pressure drag differences and then the C_D also different (see appendix D for the scale of pressure). It can clearly state that reduction pressure on the surface area means will increase drag and also C_D.

4.2.6.3 Contour Plot of Velocity

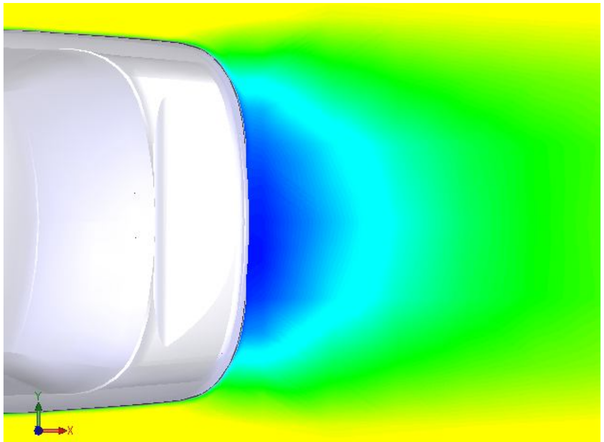


Figure 4.16: Flow on y=0mm

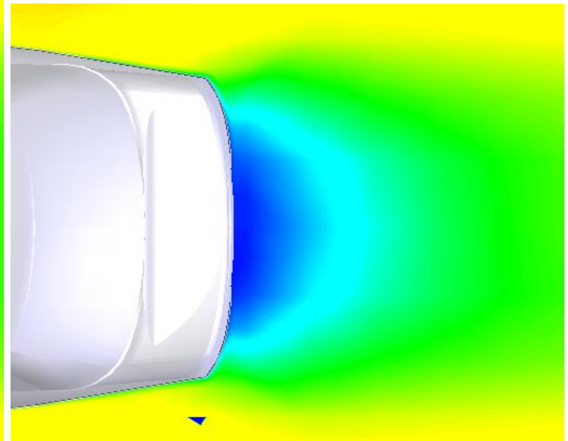


Figure 4.17: Flow on y=180mm

The Figure 4.16 and Figure 4.17 show the wake region and separation area size at the rear of the vehicle. From the figures, the differences clearly observed which wake region and separation area for y=0mm is bigger than 180mm rear shape. The size of the separation area and wake region are influenced the total aerodynamic drag which is the bigger the area, generally the bigger the drag. This statement proved with look at the drag force value in the Table 4.4 which is drag force for 0mm is bigger than y=180mm. The colors also indicate that the differences of air velocity at the behind of vehicle. The dark blue color indicates at that area, the velocity is 0 m/s.

4.2.7 Optimization

After all the simulation for every shapes was done, the last simulation is combined all three design type that gives the lowest drag of coefficient C_D value in order to find the optimization of rear shape of the vehicle. This simulation combined 37º rear slant angles, 0º of rear end bonnet and y=180mm rear side distance. After the simulation done, the result gives the force value 357.76 N and after the value calculated in order to find C_D, it gives C_D result approximately 0.327. This C_D value is better and success to reduce the drag in about ± 0.01.

Table 4.7: Table of drag force and C_D for optimization

Force, (N)	357.76
C_D	**0.326595511**

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

After all the analysis were done, as the conclusion from this project, the objective in this project was done successfully achieved which to estimate optimization shape of rear car for a sedan car by using Proton Saga BLM model by using CFD software. The second objective also successfully achieved that is study of characteristic of air flow around of vehicles during in motion.

From the result , it can conclude the lowest C_D for rear slant angle at 37º the lowest C_D for rear bonnet angle at 0º and the lowest C_D for rear side bonnet distance is at y=180mm. The optimization of rear body is a combination of all three shapes above.

From the contour plot of the velocity in every shape of vehicles, the analysis of characteristic of air flow could be done by observing the visualization. Even though the patent of visualization for each shape is quite similar, but by comparing result with highest and lowest value of C_D by using the contour plot, the differences can clearly be observed in order to study and analyze the reason of decreasing or increasing of drag force. It is clearly observed that the wake region which occurs at rear vehicle is due to larger area. Hence, it gives a high drag force.

The pressure distribution on the surface area of the vehicle can be observed in every shapes after simulation and from the data, analysis can done in order to explain the drag force differences in every shapes. The reduction of pressure at the rear of the car portion such as at the bonnet will increase the drag and then the C_D also will increase. More over, the flow trajectories also give the significant value of the analysis that which is characteristic of the path line such as stream line and turbulence flow can be observed around the vehicle.

5.2 Further Study Recommendation

After some study and analysis in this project, some recommendations were list below to improve this analysis of project to give a better result in order to get the value of C_D. The recommendations are:

1. Use software that better and high performance such as FLUENT software to perform the simulation analysis of aerodynamics drag and compare the result.
2. Refine the geometry model by using 3D scanner in order to get the accuracy model geometry and result in CFD.
3. Use the variation of velocity such as using range 70 km/h to 110km/h.

REFERENCES

1. Akiyoshi Yamada and Shigo Ito, 1993, *Computational Analysis of Flow around a Simplified Vehicle-Like Body*.
2. Bruce R. Munsan. Donald F. Young and Theodore H. Okiishi. *Fundamental of Fluid Mechanics*. Fifth Edition. John Wiley & Sons (Asia), Inc. 2006.
3. Dr. V. Sumantran and Dr. Gino Sovran. *Vehicle Aerodynamics*. Society of Automotive Engineers, Inc. 1996.
4. Emmanuel Guilmineau. *Computational Study of Flow around a Simplified Car Body*. Elsevier Ltd. 0167-6105. 2007.
5. *FundamentalsGuide*. COSMOSFloWorks 2003. COSMOS. 2003
6. Ing. Andreas Kleber. *Simulation of Air Flow Around an OPEL ASTRA Vehicle with FLUENT*. JA132. 2001.
7. Luca Iaccarino. *Cranfield University Formula 1 Team: An Aerodynamics Study of the Cockpit*. School of Engineering. Cranfield University. August 2003.
8. Sun Yongling, Wu Guangqiang, Xieshuo, Numerical Simulatin of The External Flow Field Around a Bluff Car.
9. Thomas D. Gillespie, *Fundamental of Vehicle Dynamics*, Society of Automotive Engineers, Inc.
10. Wolf-Heinrich Hucho. *Aerodynamic of Road Vehicle.* Fourth Edition. Society of Automotive Engineers, Inc. 1998.

APPENDIX A

PROJECT GANTT CHART

Appendix A1: Gantt chart for FYP 1

Activities/ Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Project Title, objective and scope														
Literature review														
Identify problem statement														
Define objective and scope														
Methodology														
Presentation preparation														
FYP 1 presentation														

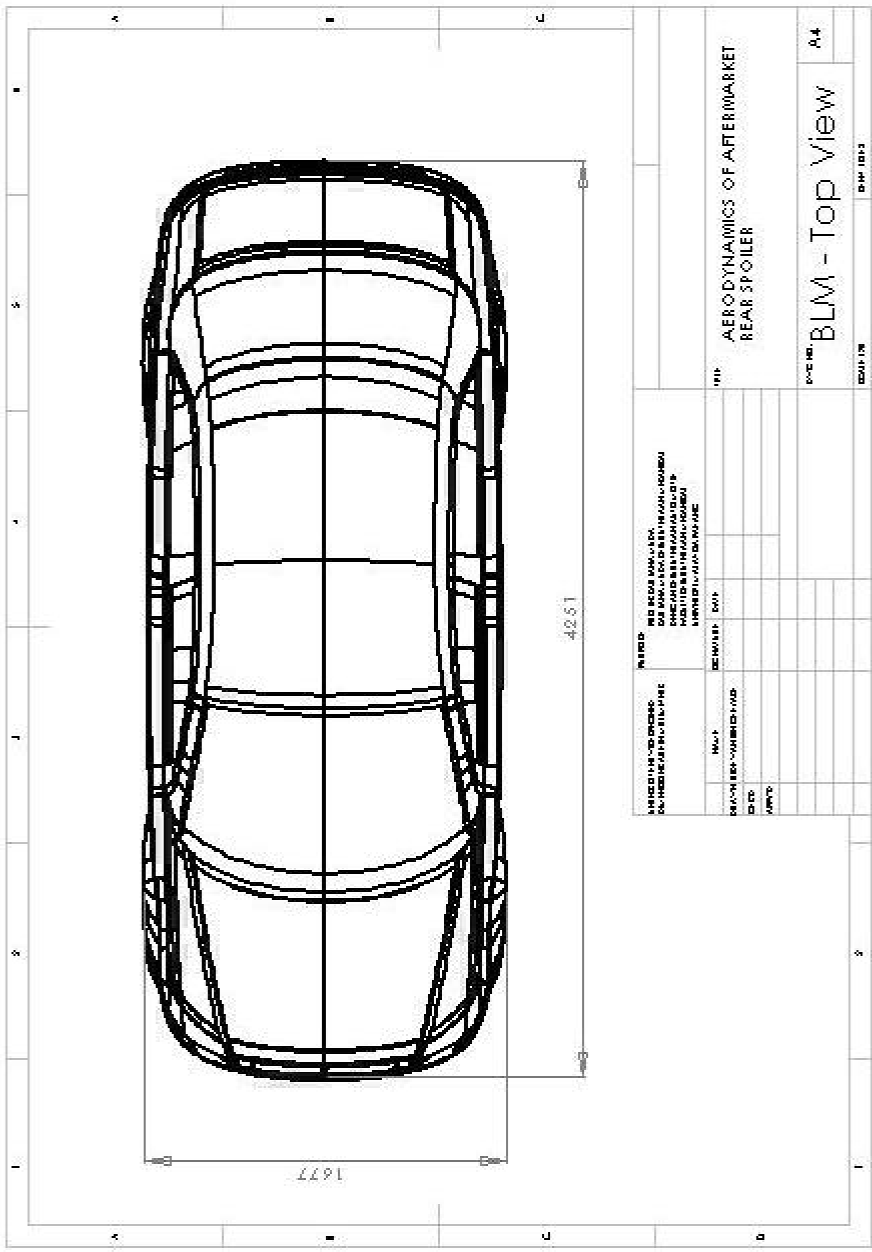
Appendix A2: Gantt chart for FYP 2

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

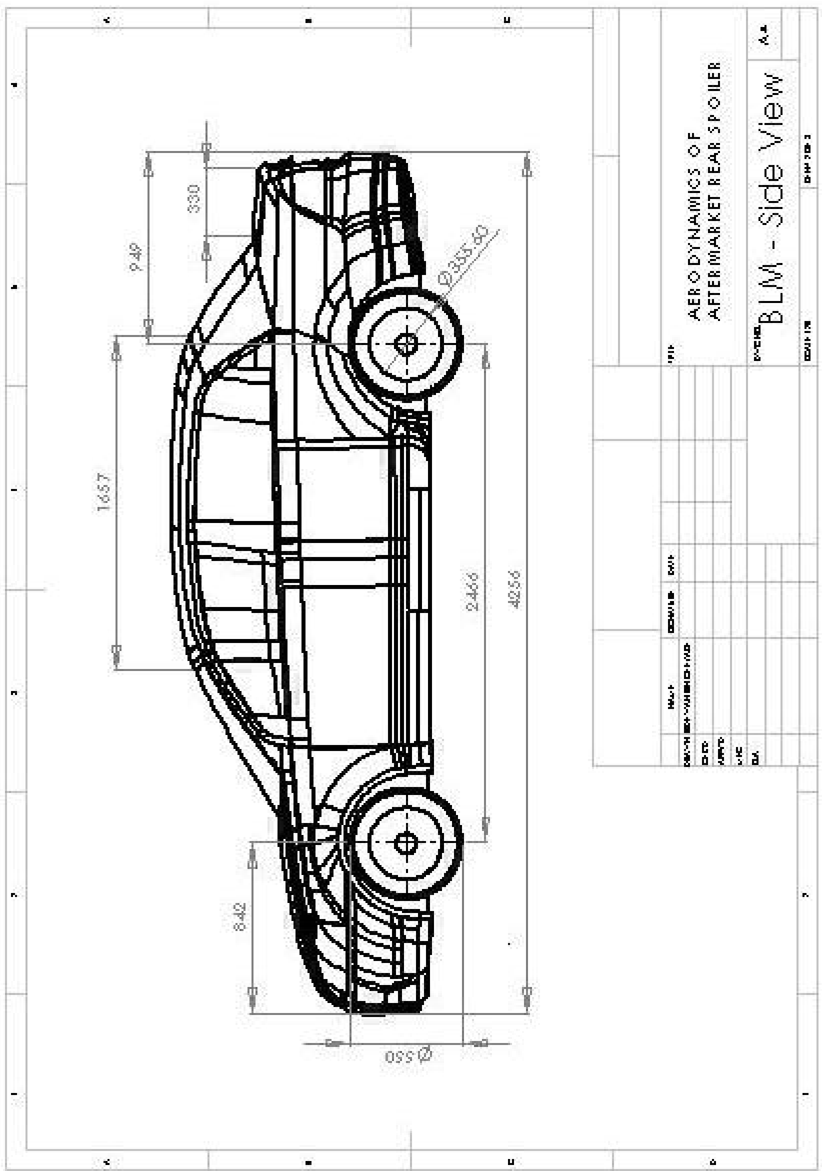
APPENDIX B

DIMENSION OF CAD MODEL

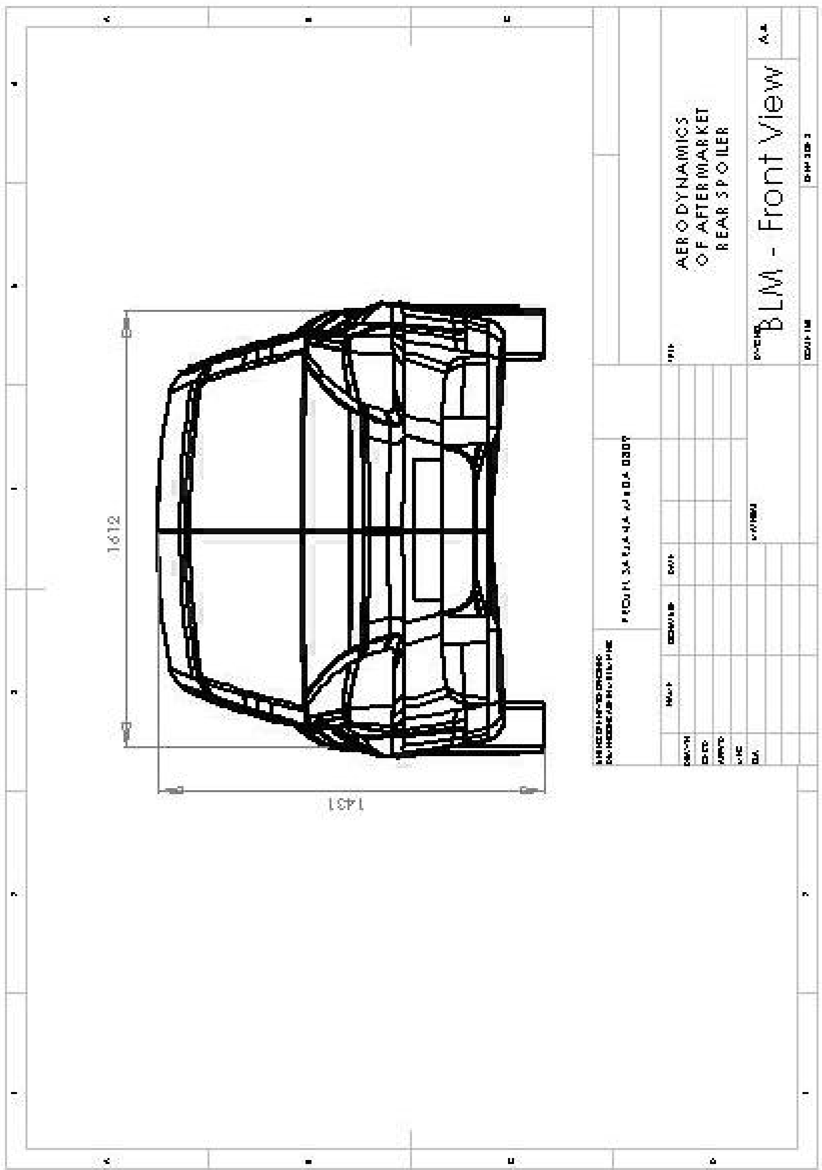
Appendix B1: Top View of BLM



Appendix B2: Side View of BLM



Appendix B1: Front View of BLM



APPENDIX C

COSMOSFLO WORKS ANALYSIS STEP

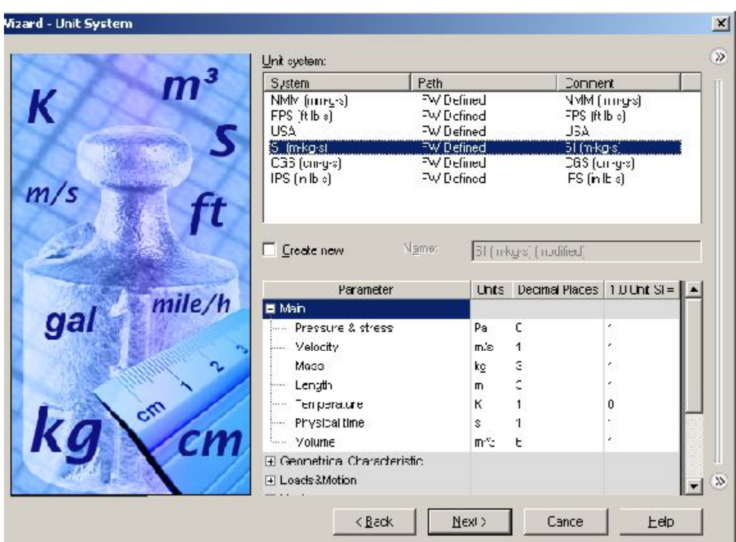
Appendix C

Create a COSMOSFloWorks Project

1. Click FloWorks, Project, Wizard...
2. Configuration name for the project. Select Create new to create a new configuration and name it Test1. Click Next.

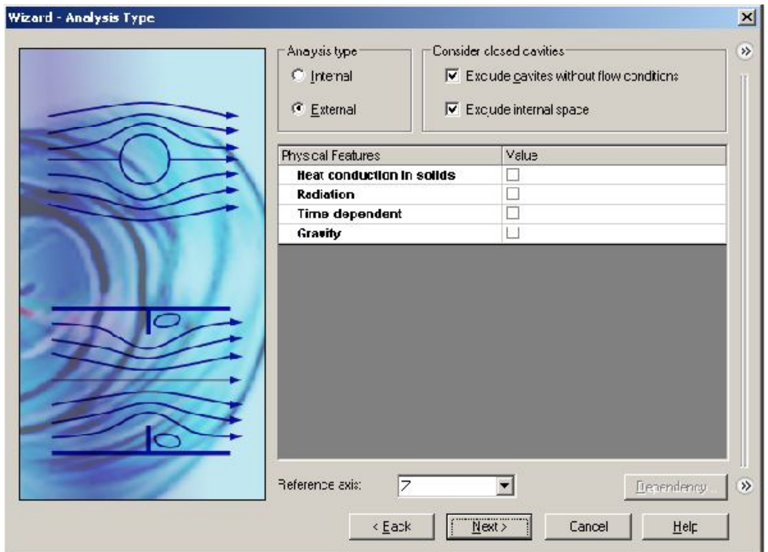
3. Unit System.

Choose SI (m-kg-s) in the Unit system area. Click Next.



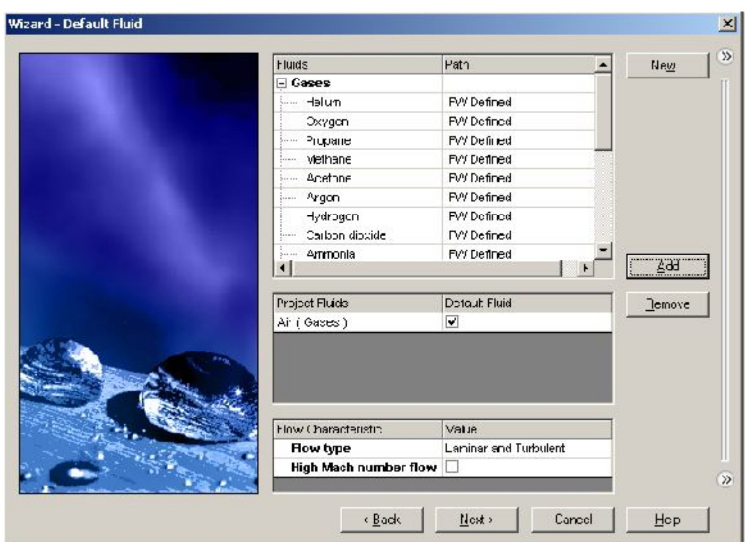
4. Analysis Type and Physical Features.

Select External as the Analysis type. Select the Exclude cavities without flow conditions and the Exclude internal space check boxes. For Reference axis, select X. The reference axis is chosen so that an angular velocity vector may be aligned with the reference axis.



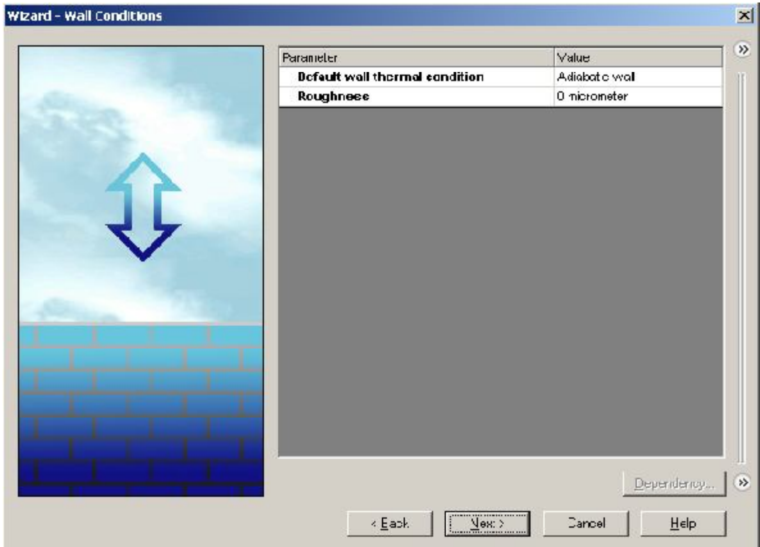
5. Fluid.

Under Gases, Select Air, then click Add. TIP: You can also double-click Air or drag and drop a fluid from one list to another.



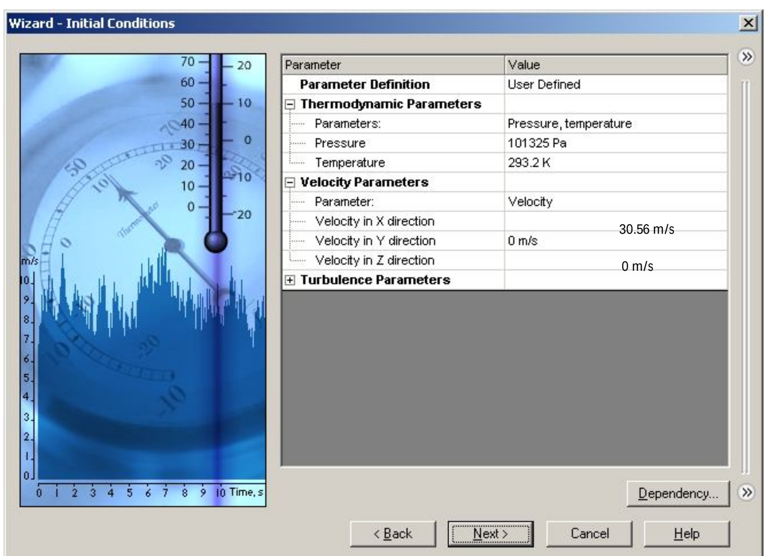
6. Wall Conditions.

Default Adiabatic wall and Roughness value of 0 micrometer are acceptable. Click Next.



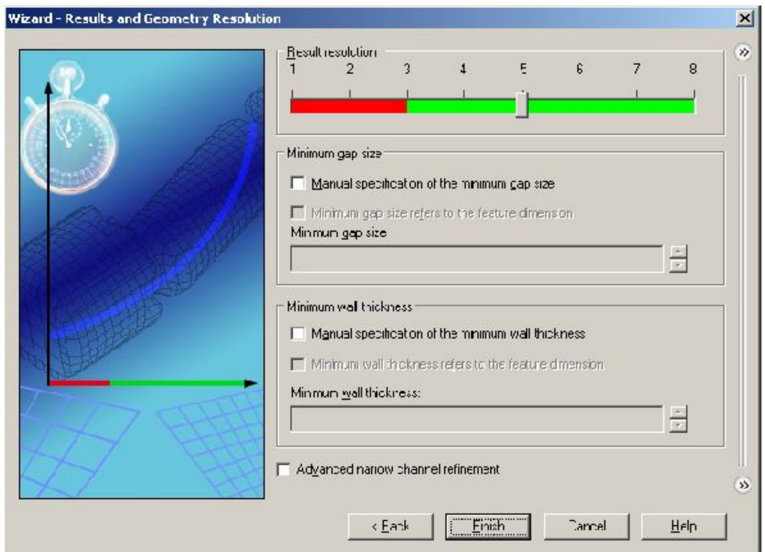
7. Initial and Ambient Conditions.

Under Velocity Parameters, double-click the value of X component and type 30.56 m/s. Click Next.



8. Results and Geometry Resolution.

Set the Result resolution to 3 which will yield acceptably accurate results in a reasonable amount of time. Click Finish.



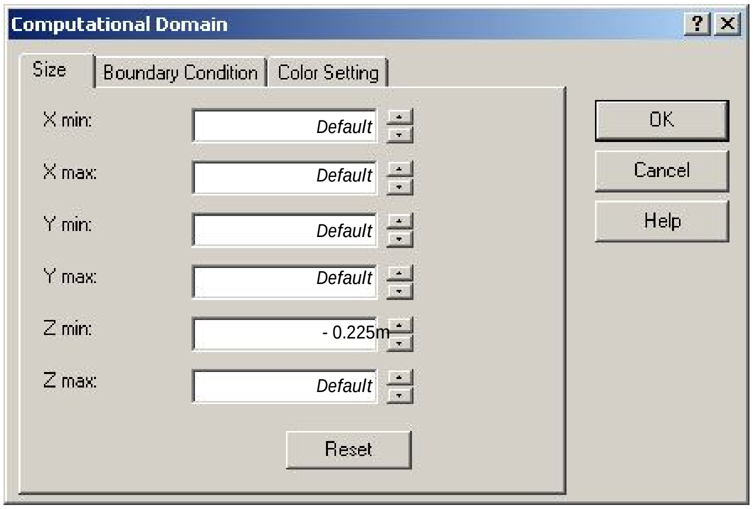
Modifying the Computational Domain

1. Show the analysis tree.

Click on the COSMOSFloWorks analysis tree tab. Expand the Input Data listing.

2. Computational domain settings.

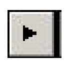
Right-click Computational Domain and select Edit Definition. On the Size tab, use the default value except for the z

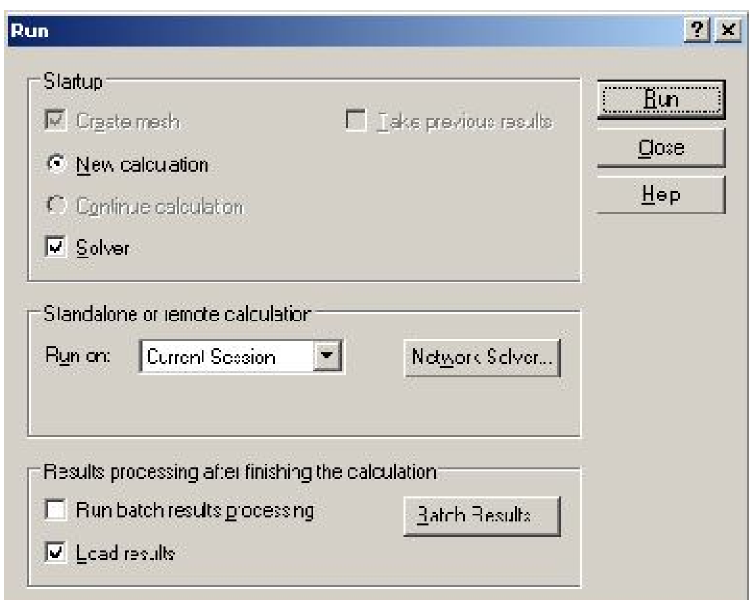


Click OK.

Running the Analysis

This starts the calculation for the current project.

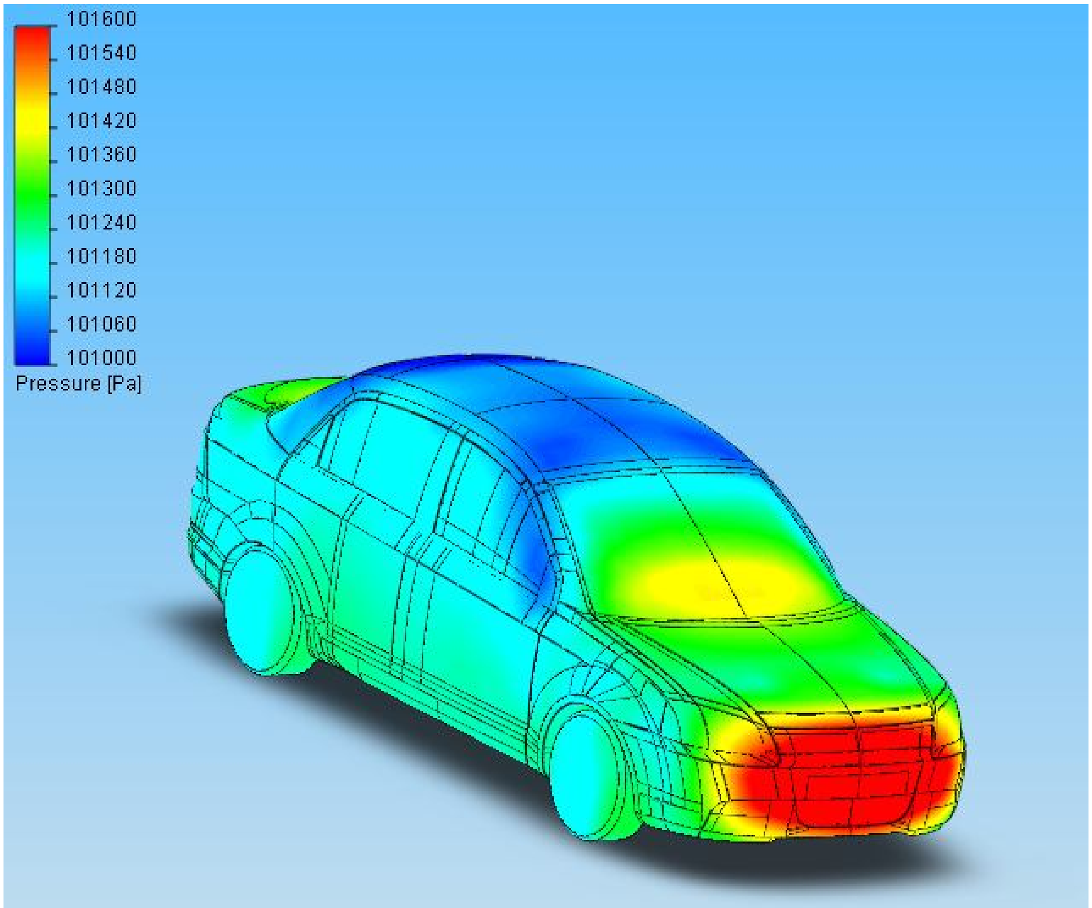
1. Run the analysis. Right-click Test1 and click Run…, or click FloWorks, Solve, Run, or click Run Solver  on the COSMOSFloWorks toolbar.



APPENDIX D

SURFACE PLOT PRESSURE DISTRIBUTION

Appendix D1: Surface Plot Pressure Distribution



Appendix D2: Surface Plot Pressure Distribution