AERODYNAMICS BEHAVIOUR OF PERSONA CAR USING COMPUTATIONAL FLUID DYNAMICS (CFD)

MOHD SHAIFULLAH BIN SHAHRUDDIN

BACHELOR OF MECHANICAL ENGINEERING UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

BORANG PI	ENGESAHAN STATUS TESIS*
JUDUL: <u>AERODYNAMI(</u> <u>USING COMPUT</u> SESI	CS BEHAVIOUR OF PERSONA CAR FATIONAL FLUID DYNAMICS (CFD) PENGAJIAN: <u>2009/ 2010</u>
Saya <u>MOHD SHAIFU</u>	LLAH BIN SHAHRUDDIN (870812-01-5275) (HURUF BESAR)
mengaku membenarkan tesis (Sa Perpustakaan dengan syarat-syar	rjana Muda/ Sarjana / Doktor Falsafah)* ini disimpan di at kegunaan seperti berikut:
 Tesis adalah hakmilik Unive Perpustakaan dibenarkan me Perpustakaan dibenarkan me pengajian tinggi. **Sila tandakan (√) 	ersiti Malaysia Pahang (UMP). embuat salinan untuk tujuan pengajian sahaja. embuat salinan tesis ini sebagai bahan pertukaran antara institusi
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 TERI	HAD
	Disahkan oleh:
(TANDATANGAN PENULIS)	(TANDATANGAN PENYELIA)
<u>398, JALAN MERANTI,</u> <u>81550, GELANG PATAH,</u> <u>JOHOR BAHRU, JOHOR</u>	MUHAMAD ZUHAIRI SULAIMAN (Nama Penyelia)
Tarikh: 24 NOVEMBER 2009	Tarikh : 24 NOVEMBER 2009
CATATAN: * Potong yang tic ** Jika tesis ini SU berkuasa/organ dikelaskan seba • Tesis dimaksud Penyelidikan, a penyelidikan, a	lak berkenaan. JLIT atau TERHAD, sila lampirkan surat daripada pihak isasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu agai atau TERHAD. Ikan sebagai tesis bagi Ijazah doktor Falsafah dan Sarjana secara tau disertasi bagi pengajian secara kerja kursus dan tau Laporan Projek Sarjana Muda (PSM).

UNIVERSITI MALAYSIA PAHANG FACULTY OF MECHANICAL ENGINEERING

We certify that the project entitled *Aerodynamics Behaviour of Persona Car using Computational Fluid Dynamics (CFD)* is written by *Mohd Shaifullah bin Shahruddin*. We have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfilments of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Examiner

Signature

AERODYNAMICS BEHAVIOUR OF PERSONA CAR USING COMPUTATIONAL FLUID DYNAMICS (CFD)

MOHD SHAIFULLAH BIN SHAHRUDDIN

Report 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 Automotive Engineering.

Signature:Name of Supervisor: MUHAMAD ZUHAIRI SULAIMANPosition: LecturerDate:

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 SHAIFULLAH BIN SHAHRUDDIN
ID Number	: MH06041
Date	: 24 NOVEMBER 2009

To my beloved father mother

Shahruddin bin Ahmad Mohaini binti Yahya

ACKNOWLEDGEMENTS

First of all, I want to thank The Almighty Allah SWT for the beautiful life that has been given to me in the past 22 years and the present. I am very thankful to be given the time and chances to finally complete this research. I am grateful and would like to express my sincere gratitude to my supervisor Mr. Muhamad Zuhairi Sulaiman 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.

I also would like to express my gratitude to the Faculty of Mechanical Engineering and Universiti Malaysia Pahang, for their assistance in supplying the relevant literatures.

I am also obliged to express my appreciation towards my beloved mom and dad and also my family members for their enduring patience, moral and financial supports. My fellow friends should also be recognised for their support. My sincere appreciation also extends to all my colleagues and others who have provided assistance at various occasions. Their views and tips are useful indeed. Unfortunately, it is not possible to list all of them in this limited space. Thank you to all. Thank you for everything. May God bless all of you.

ABSTRACT

An aerodynamic characteristic of a car is of significant interest in reducing car accidents due to wind loading and in reducing the fuel consumption. On the limitations of conventional wind tunnel experiment and rapid developments in computer hardware, considerable efforts have been invested in the last decade to study vehicle aerodynamics computationally. This report presents a numerical simulation of flow around Proton Persona car using commercial fluid dynamics software FLUENT for 2D simulation and COSMOSFloWorks for 3D simulation. The study focuses on CFD-based lift and drag coefficient prediction on the car body and the air flow pattern around the car body using Computational Fluid Dynamics (CFD) software. A three dimensional computer model of a Proton Persona was used as the base model in this study. The wind speed selected in this study ranges from 80 km/hr to 140 km/hr with increment of 20 km/hr. After numerical iterations are completed, the aerodynamic data and detailed complicated flow behaviour are visualized clearly. The drag and lift coefficient of Proton Persona have been estimated by using a mathematical equations. The pressure and velocity distributions along the surface of the car also have been analyzed. From the results obtained, it was found that highest speed occurs where the pressure is lowest, and the lowest speed occurs where the pressure is highest. Therefore it satisfies the Bernoulli's principle. In addition, the flow pattern around the model showed very similar with the previous works, emphasizing in findings.

ABSTRAK

Ciri aerodinamik sesebuah kereta adalah sangat penting dalam pengurangan kadar kemalangan kereta yang disebabkan oleh beban angin dan dalam mengurangkan penggunaan minyak. Oleh sebab terdapat had-had bagi eksperimen terowong angin konvensional dan pembangunan pantas dalam perkakasan komputer, ikhtiar yang banyak telah dilaburkan dalam dekad yang lalu untuk mengkaji aerodinamik kenderaan secara pengkomputeran. Laporan ini membentangkan satu penyerupaan berangka aliran sekitar kereta Proton Persona menggunakan perisian komersial dinamik bendalir FLUENT untuk simulasi 2D dan COSMOSFloWorks untuk simulasi 3D. Kajian itu menumpukan pada ramalan pekali daya angkat dan seretan pada badan kereta dan corak aliran udara sekitar badan kereta itu menggunakan perisian Computational Fluid Dynamics (CFD). Satu model komputer tiga dimensi, Proton Persona telah digunakan sebagai model asas dalam kajian ini. Kelajuan angin yang pilih dalam julat kajian ini adalah daripada 80 km/jam hingga 140 km/jam dengan tambahan setiap 20 km/jam. Selepas iterasi berangka siap, data aerodinamik dan sifat aliran rumit yang terperinci dipaparkan dengan jelas. Pekali seretan dan daya angkat Proton Persona telah dianggarkan dengan menggunakan persamaan matematik. Pengagihan tekanan dan halaju sepanjang permukaan kereta itu juga telah dianalisis. Daripada keputusan yang diperolehi, didapati kelajuan tertinggi itu berlaku di mana tekanan terendah, dan kelajuan terendah berlaku di mana tekanan tertinggi. Oleh itu ia memuaskan prinsip Bernoulli. Seperkara lagi, corak aliran sekitar model itu menyerupai dengan kajian terdahulu, mengukuhkan lagi penemuan.

TABLE OF CONTENTS

SUPERVISOR'S DECLARATION	i
STUDENT'S DECLARATION	ii
DEDICATION	iii
ACKNOWLEDGEMENTS	iv
ABSTRACT	v
ABSTRAK	vi
TABLE OF CONTENTS	vii
LIST OF TABLES	х
LIST OF FIGURES	xi
LIST OF SYMBOLS	xiii
LIST OF ABBREVIATIONS	xiv

CHAPTER 1 INTRODUCTION

1.1	Project Background	1
1.3	Problem Statement	2
1.3	Objectives	2
1.4	Scopes of Study	2

CHAPTER 2 LITERATURE REVIEW

2.1	Introduction	3
2.2	History Of Automotive Aerodynamics Technology	3
2.3	Aerodynamics Theory	5
	2.3.1 Bernoulli's Theorem	5

Page

	2.3.2	Aerodynamics Drag	7
	2.3.3	Aerodynamics Lift	8
	2.3.4	External Flow	9
	2.3.5	Pressure Distributions	10
	2.3.6	Separation Flow	12
	2.3.7	Formation of Vortex Shedding	13
2.4	Turbu	lent Models	14
	2.4.1	Classification of Turbulent Models	14
	2.4.2	Reynolds-Averaged Navier-Stokes (RANS) Models	14
	2.4.3	Computation of Fluctuating Quantities	15
	2.4.4	Governing Equations	15
2.5	Comp	utational Fluid Dynamics (CFD)	17
	2.5.1	Definition	17
	2.5.2	Advantages and Disadvantages	18
	2.5.3	Elements	19
	2.5.4	CFD in Automotive Industry	19

CHAPTER 3 METHODOLOGY

3.1	Introduction	20
3.2	Flow Chart Methodology	21
3.3	Modelling Geometry	22
3.4	2D Simulation Setup	23
	3.4.1 Mesh Generation	23
	3.4.2 Fluent Setup	24
	3.4.3 Defining the Models	24
	3.4.4 Solver	25
	3.4.5 Viscous	25
	3.4.6 Defining the Material Properties	25
	3.4.7 Boundary Conditions	26
	3.4.8 Executing the Fluent Code	26
3.5	3D Simulation Setup	27
	3.5.1 COSMOS FloWorks setup	27
	3.5.2 Frontal Area	29

CHAPTER 4 RESULTS AND DISCUSSION

4.1	Introduction	30
4.2	Calculation of Reynolds Number	31
4.3	2D Analysis Result	33
	 4.3.1 Drag and Lift Analysis 4.3.2 Sample Calculation for Drag and Lift Coefficient 4.3.3 Pressure, Velocity and Flow Pattern Analysis 4.3.4 Pressure Distribution Analysis 	33 36 37 41
4.4	 3D Analysis Result 4.4.1 Important Parameter of flow analysis 4.4.2 Data Collection and Analysis 4.4.3 Sample Calculation for Drag and Lift Coefficient 4.4.4 Contour Plot of Velocity and Pressure 	42 42 42 46 47
	4.4.5 Flow Trajectories of Velocity Analysis	49

CHAPTER 5 CONCLUSION AND RECOMMENDATIONS

5.1	Conclusion	53
5.2	Further Study Recommendations	54
REFI	ERENCES	55
APPI	ENDICES	56
A1	Gantt Chart for FYP 1	56
A2	Gantt Chart for FYP 2	56
В	Proton Persona Drawing	57

LIST OF TABLES

Table No.	Title	Page
3.1	Variables and convergence criteria for FLUENT simulation of free layers apparatus.	27
4.1	Table of various velocities, drag and lift data	33
4.2	Table of various velocities and percentages of C_D and C_L changes (2D)	35
4.3	Data collection for analysis with various velocities	42
4.4	Table of various velocities and percentages of C_D and C_L changes (3D)	45

LIST OF FIGURES

Figure No.	Title	Page
2.1	History of vehicle dynamic in passenger car	4
2.2	Venturi Effect	6
2.3	Flow around a vehicle (schematic)	10
2.4	Flow field and pressure distribution for a vehicle-shaped body in two-dimensional flow (schematic)	10
2.5	Separation of the boundary layer flow at a wall (schematic)	12
2.6	Major locations of flow separation on a car	13
2.7	Regimes of flow around circular cylinder	14
2.8	Car Aerodynamic Simulation	19
3.1	Flow Chart Methodology	21
3.2	Drawing of Proton Persona	22
3.3	Computational grid for FLUENT CFD modeling	24
3.4	Computational domain and boundary condition	28
3.5	Grid refinement around the Proton Persona model	28
3.6	The frontal area of Proton Persona	29
4.1	Variation of C_D over speed range of Persona car	33
4.2	Variation of C_L over speed range of Persona car	34
4.3	Graph of the C_D and C_L percentage changes against velocity	35

4.4	The pressure contour with several of velocities inlet	37
4.5	The velocity contour with several of velocities inlet	38
4.6	Separation flow at the front end of Persona car	39
4.7	Reattachment flow at the front edge hood of Persona car	39
4.8	Reverse flow and vortex generation at rear end of the Persona car	40
4.9	The pressure distribution along the Persona car at velocity 80 km/h	41
4.10	Graph of drag coefficient, C _D against various velocities	43
4.11	Graph of lift coefficient, C _L against various velocities	44
4.12	Graph of the C_D and C_L percentage changes against velocity	45
4.13	The velocity contour with several of velocities inlet	47
4.14	The pressure contour with several of velocities inlet	48
4.15	The isometric view of trajectories velocity flow of 140 km/h	49
4.16	The side view of trajectories velocity flow of 140 km/h	50
4.17	Separation flow at the front end of the car	50
4.18	Reattachment flow at the roof of the car	51
4.19	Reverse flow and vortex generation at the rear end of the car	52

LIST OF SYMBOLS

А	Area		
C _D	Drag Coefficient		
C _L	Lift Coefficient		
p	Pressure		
ρ	Density		
V	Velocity		
F _D	Drag Force		
F_L	Lift Force		
μ	Dynamic Viscosity		
C _P	Pressure Coefficient		
p∞	Initial Pressure		
V_{∞}	Initial Velocity		
k	Turbulence kinetic energy		
ε	Kinetic energy dissipation rate		
$ au_{ij}$	Shear-stress tensor		
S _{ij}	Shearing-rate tensor		
σ_k	Prandtl number of the turbulence kinetic energy		
	Prandtl number of the turbulence kinetic energy dissipation rate		

LIST OF ABBREVIATIONS

- 2-D Two Dimensional
- 3-D Three Dimensional
- CAD Computational Aided Design
- CFD Computational Fluid Dynamics
- DNS Direct numerical simulation
- DSM Differential stress models
- EVM Eddy-viscosity models
- FYP Final year project
- LES Large-eddy simulation
- NLEVM Non-linear eddy-viscosity models
- RANS Reynolds-Averaged Navier-Stokes
- Re Reynold Number
- RNG Renormalization Group
- RSTM Reynolds-stress transport models
- SOC Second-order closure models

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

This project is about to study and analysis aerodynamics behaviour of Persona Car using Computational Fluid Dynamics (CFD). This project involves drawing and simulating a geometrical model of Proton Persona by using aerodynamic simulation software. By the end of the project, we should be able to determine aerodynamic lift, drag and flow characteristics of the car. Actually, there are other methods to analysis aerodynamic behavior of a car such as wind tunnel but for this project the analysis is based on aerodynamic simulation software.

Aerodynamics is a branch of dynamics concerned with studying the motion of air, particularly when it interacts with a moving object. Aerodynamics is a subfield of fluid dynamics and gas dynamics, with much theory shared between them. Understanding the motion of air (often called a flow field) around an object enables the calculation of forces and moments acting on the object. Typical properties calculated for a flow field include velocity, pressure, density and temperature as a function of position and time. In automotive field, the study of the aerodynamics of road vehicles is very important. The main concerns of automotive aerodynamics are reducing drag (though drag by wide wheels is dominating most cars), reducing wind noise, minimizing noise emission and preventing undesired lift forces at high speeds. For some classes of racing vehicles, it may also be important to produce desirable downwards aerodynamic forces to improve traction and thus cornering abilities.

1.2 PROBLEM STATEMENT

The study of aerodynamics in the automotive industry is important to improve fuel economy as well as vehicle comfort and safety. Instead of wind tunnel testing which is high in cost, the alternative method to study aerodynamics by using Computational Fluid Dynamics (CFD). CFD is widely use by researchers to apply in various type of field. This technology provide enormous amount of information as well as high economical efficiency. It is also very useful for initial prediction of new design of a vehicle. The problem about this project is to analysis aerodynamic characteristic such as aerodynamic drag, lift and flow behavior of Proton Persona car by using CFD software.

1.3 OBJECTIVES

- 1. To estimate the drag coefficient, C_D and lift coefficient, C_L of the Persona car.
- 2. To study the air flow behavior around the body of Proton Persona car using Computational Fluid Dynamics (CFD) software.

1.4 SCOPES OF STUDY

The scopes of this project covered study and analyses the drag and lift coefficient and also the flow behaviour of Proton Persona using CFD software. The model geometry will be created using CAD software, SolidWorks. The analysis of this model geometry will be based on 2-D and 3-D simulations which will be developed using two different softwares which are FLUENT 6 and COSMOSFloWorks respectively. The simulations only involve the external flow of the car.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

In this chapter, the explanations and some history of aerodynamics, theories, the previous research and findings are included. With reference from various sources such as journal, thesis, reference books, literature review has been carried out to collect all information related to this project. CFD software and performing analysis by this software also described.

2.2 HISTORY OF AUTOMOTIVE AERODYNAMICS TECHNOLOGY

Aerodynamics and vehicle technology have merged only very slowly. A synthesis of the two has been successful only after several tries. This is surprising since in the neighboring disciplines of traffic technology, naval architecture, and aeronautics the cooperation with fluid mechanics turned out to be very fruitful. Of course, the designers of ships and airplanes were in a better position. They found their originals in nature from fish and birds. From these natural shapes they took many essential features. The automobile had no such originals. Hence its designers tried to borrow shapes from ships and airplanes, which must have appeared progressive to them. Very soon this turned out to be the wrong approach. Only when it broke away from these improper originals did aerodynamics make a breakthrough in the automobile.

Another reason for the early repeated failures of aerodynamics with vehicles is that it started far too early. The first automobiles were pretty slow. On the bad roads of those days streamlined bodies would have looked ridiculous. Protecting driver and passengers from wind, mud and rain could be accomplished very well with the traditional design of horse-drawn carriages. Later the prejudice that streamlined bodies were something for odd persons overrode the need for making use of the benefits of aerodynamics for economical reasons.

A brief overview of the history of vehicle aerodynamics is summarized in Figure 2.1. During the first two of the total four periods, aerodynamic development was done by individuals, most of them coming from outside the car industry. They tried to carry over basic principles from aircraft aerodynamics to cars. Later, during the remaining two periods, the discipline of vehicle aerodynamics was taken over by the car companies and was integrated into product development. Since then, teams, not individual inventors, have been (and are) responsible for aerodynamics.

Basic shapes	1900 to 1925	Torpedo Boat t	ail Air ship
Streamlined cars	1921 to 1923	Rumpler	Bugatti
	1922 to 1939	Jaray	
	1934 to 1939	Kamm	Schlör
	Since 1955	Citröen	NSU-Ro 80
Detail optimization	Since 1974	VW-Scirocco I	VW-Golf I
Shape optimization	Since 1983	Audi 100 III	Ford Sierra

Figure 2.1: History of vehicle dynamic in passenger car [1].

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 horsedrawn carriages. They had certainly succeeded in building cars with low drag coefficient [1].

2.3 AERODYNAMICS THEORY

2.3.1 Bernoulli's Theorem

Bernoulli's theorem implies that if the fluid flows horizontally so that no change in gravitational potential energy occurs, then a decrease in fluid pressure is associated with an increase in fluid velocity. If the fluid is flowing through a horizontal pipe of varying cross-sectional area, for example, the fluid speeds up in constricted areas so that the pressure the fluid exerts is least where the cross section is smallest. This phenomenon is sometimes called the Venturi Effect (Figure 2.2), after the Italian scientist G.B. Venturi (1746–1822), who first noted the effects of constricted channels on fluid flow [1].



Figure 2.2: Venturi Effect [1].

Bernoulli's principle can be applied to various types of fluid flow, resulting in what is loosely denoted as Bernoulli's equation (Eq. 2.1). In fact, there are different forms of the Bernoulli's equation for different types of flow. The simple form of Bernoulli's principle is valid for incompressible flows (most liquid flows) and also for compressible flows (gases) moving at low Mach numbers. More advanced forms may in some cases be applied to compressible flows at higher Mach numbers.

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

Where the p, ρ , v are pressure, density and velocity, respectively.

Bernoulli's principle is equivalent to the principle of conservation of energy. This states that in a steady flow the sum of all forms of mechanical energy in a fluid along a streamline is the same at all points on that streamline. This requires that the sum of kinetic energy and potential energy remain constant. If the fluid is flowing out of a reservoir the sum of all forms of energy is the same on all streamlines because in a reservoir the energy per unit mass (the sum of pressure and gravitational potential ρgh) is the same everywhere.

Fluid particles are subject only to pressure and their own weight. If a fluid is flowing horizontally and along a section of a streamline, where the speed increases it can only be because the fluid on that section has moved from a region of higher pressure to a region of lower pressure; and if its speed decreases, it can only be because it has moved from a region of lower pressure to a region of higher pressure. Consequently, within a fluid flowing horizontally, the highest speed occurs where the pressure is lowest, and the lowest speed occurs where the pressure is highest.

2.3.2 Aerodynamics Drag

The force on an object that resists its motion through a fluid is called drag. When the fluid is a gas like air, it is called aerodynamic drag (or air resistance). When the fluid is a liquid like water it is called hydrodynamic drag. Drag is a complicated phenomena and explaining it from a theory based entirely on fundamental principles is exceptionally difficult.

Fluids are characterized by their ability to flow. In semi-technical language, a fluid is any material that can't resist a shear force for any appreciable length of time. This makes them hard to hold but easy to pour, stir, mix, and spread. As a result, fluids have no definite shape but take on the shape of their container. Fluids are unusual in that they yield their space relatively easy to other material things at least when compared to solids.

Fluids may not be solid, but they are most certainly material. The essential property of being material is to have both mass and volume. Material things resist changes in their velocity and no two material things may occupy the same space at the same time. The portion of the drag force that is due to the inertia of the fluid is the resistance to change that the fluid has to being pushed aside so that something else can occupy its space is called the pressure drag [1].

$$C_d = \frac{F_d}{1/2 \rho v^2 A}$$
 (2.2)

 $\begin{array}{ll} \mbox{Where } F_d = \mbox{Drag force} & C_d = \mbox{Drag coefficient} \\ \rho = \mbox{fluid density} & A = \mbox{frontal area} \\ v = \mbox{Velocity} & \end{array}$

2.3.3 Aerodynamics Lift

Lift is normally of little importance in passenger cars as their speed is usually too low to produce much lift. It was noticed early on that something strange happened at high speeds: the car seemed to be lifting off the ground. Lift can be serious, particularly in racing cars. It has a serious effect on the control and handling of the car [1].

Lift occurs because the airflow over the top of a car is faster than across the bottom. This occurs to some degree in all cars. As the speed increases, the pressure decreases, according to Bernoulli's theorem. The top of the car therefore has a lower pressure than the bottom, and the result is a lifting force.

The amount of lift generated by an object depends on a number of factors, including the density of the air, the velocity between the object and the air, the viscosity and compressibility of the air, the surface area over which the air flows, the shape of the body, and the body's inclination to the flow, also called the angle of attack.

$$C_{L} = \frac{F_{L}}{1/2\rho V^{2}A}$$
(2.3)

Where $F_L = lift$ force $\rho = fluid$ density v = Velocity C_L = Lift coefficient A = frontal area

2.3.4 External Flow

The external flow around a vehicle is shown in Figure 2.3. In still air, the undisturbed velocity v_{∞} is the road speed of the car. Provided no flow separation takes place, the viscous effects in the fluid are restricted to a thin layer of a few millimeters thickness, called the boundary layer. Beyond this layer the flow can be regarded as inviscid, and its pressure is imposed on the boundary layer. Within the boundary layer the velocity decreases from the value of the inviscid external flow at the outer edge of the boundary layer to zero at the wall, where the fluid fulfills a no-slip condition. When the flow separates (Figure 2.3 shows separation at the rear only) the boundary layer is "dispersed" and the flow is entirely governed by viscous effects. Such regions are quite significant as compared to the characteristic length of the vehicle. At some distance from the vehicle there exists no velocity difference between the free stream and the ground. Therefore, in vehicle-fixed coordinates, the ground plane is a stream surface with constant velocity V_{∞} , and at this surface no boundary layer is present. This fact is very important for the simulation of flows around ground vehicles in wind tunnels. The boundary layer concept is valid only for large values of

$$\operatorname{Re}_{l} = \frac{\rho V L}{\mu} > 10^{4}$$
(2.4)

This dimensionless parameter is called the Reynolds number. It is a function of the speed of the vehicle v, the viscosity μ and density ρ of the fluid, and a characteristic length *L* as defined in Figure 2.3. The character of the viscous flow around a body depends only on the body shape and the Reynolds number. For different Reynolds numbers entirely different flows may occur for one and the same body geometry. Thus the Reynolds number is the dimensionless parameter which characterizes a viscous flow.



Figure 2.3: Flow around a vehicle (schematic) [2].

2.3.5 Pressure Distributions



Figure 2.4: Flow field and pressure distribution for a vehicle-shaped body in twodimensional flow (schematic) [2].

The two-dimensional flow around a vehicle-shaped body is shown in Figure 2.4. This flow is a considerable simplification of a three-dimensional flow around a vehicle, and may be regarded as a qualitative picture of the flow at the longitudinal cross-section of a car. The upper part of the figure indicates the streamlines. Three stagnation points occur: in the nose region, in the cove between hood and windshield (scuttle), and at the trailing edge. The pressure distribution on the contour is drawn schematically as $c_p(x/l)$ in the lower half of the figure, where

$${}^{\mathbf{C}}\mathbf{p} = \frac{\mathbf{p} - \mathbf{p}_{\infty}}{\frac{1}{2}\,\rho\,\mathbf{v}_{\infty}^2} \tag{2.5}$$

is the dimensionless pressure coefficient. Leads to

$$c_{p} = \frac{p - p_{\infty}}{\frac{1}{2}\rho v_{\infty}^{2}} = 1 - (\frac{v}{v_{\infty}})^{2}$$
(2.6)

In the stagnation points of the flow field, v = 0, Eq. (2.6) yields $c_p = 1$. At the lower surface of the vehicle, the pressure is higher than the free-stream pressure, $c_p > 0$, but for very small ground distances even suction, $c_p < 0$, may be present. At the upper surface, high pressures, $c_p > 0$, are observed in the region of the scuttle, whereas high suction, $c_p < 0$, is found on the cabin roof. On the rear part of the vehicle's upper surface a steep pressure rise occurs and it is in this region where considerable differences exist between the real flow of a viscous fluid and the inviscid flow shown here. The pressure distribution in Figure 2.4 indicates that the pressure level on the upper side of the vehicle is much lower than on the lower side. This means that a net upward lift force acts on the vehicle [2].

2.3.6 Separation Flow

Laminar and turbulent boundary layer flow strongly depends on the pressure distribution which is imposed by the external flow. For a pressure increase in flow direction the boundary layer flow is retarded, especially near the wall, and even reversed flow may occur. This behavior is shown schematically in Figure 2.5. It can be seen that between forward and reverse flow a dividing streamline leaves the wall. This phenomenon is called separation. For the separation point A, the condition

holds. Turbulent boundary layers can withstand much steeper adverse pressure gradients without separation than laminar boundary layers. This is because the turbulent mixing process leads to an intensive momentum transport from the outer flow towards the flow adjacent to the wall. For a pressure decrease in flow direction there exists no tendency to flow separation.



Figure 2.5: Separation of the boundary layer flow at a wall (schematic) [2].



Figure 2.6: Major locations of flow separation on a car [2].

2.3.7 Formation of vortex shedding

Different Re will affect the formation of vortex shedding over bluff bodies. For example, the Figure 2.7 below shows the formation of vortex shedding (circular cylinder) by varied the Reynolds Number value, no separation occur at Re<5. For 5<Re<40, separation occurs and two symmetric fixed eddies (vortex) are formed in the wake on two side of the cylinder. For 40<Re<190 the wake is laminar. The wake is turbulent and remain turbulent at Re>300. For Re above 3.5×10^6 , the boundary layer is fully turbulent or unsteadiness [3].



Figure 2.7: Regimes of flow around circular cylinder [4].

2.4 TURBULENT MODELS

2.4.1 Classification of turbulent models

Nowadays turbulent flows may be computed using several different approaches. Either by solving the Reynolds-averaged Navier-Stokes equations with suitable models for turbulent quantities or by computing them directly [10]. The main approaches are summarized below:

2.4.2 Reynolds-Averaged Navier-Stokes (RANS) Models

• Eddy-viscosity models (EVM)

One assumes that the turbulent stress is proportional to the mean rate of strain. Further more eddy viscosity is derived from turbulent transport equations (usually k + one other quantity).

• Non-linear eddy-viscosity models (NLEVM)

Turbulent stress is modelled as a non-linear function of mean velocity gradients. Turbulent scales are determined by solving transport equations (usually k + one other quantity). Model is set to mimic response of turbulence to certain important types of strain.

• Differential stress models (DSM)

This category consists of Reynolds-stress transport models (RSTM) or secondorder closure models (SOC). One is required to solve transport equations for all turbulent stresses.

2.4.3 Computation of fluctuating quantities

• Large-eddy simulation (LES)

One computes time-varying flow, but models sub-grid-scale motions.

• Direct numerical simulation (DNS)

No modeling what so ever is applied. One is required to resolve the smallest scales of the flow as well.

2.4.4 Governing Equations

To account for the turbulence effect on the flow field, Reynolds time averaging technique was employed on the Navier-Stokes equation to yield the Reynolds Averaged Navier-Stokes (RANS) equation which can be mathematically expressed as

$$\frac{\partial \tilde{\mathbf{u}}_{i}}{\partial t} + \tilde{\mathbf{u}}_{j} \frac{\partial \tilde{\mathbf{u}}_{i}}{\partial x_{j}} = -\frac{1}{\rho} \frac{\partial \tilde{p}}{\partial x_{i}} + \frac{\partial}{\partial x_{j}} \left(v \frac{\partial \tilde{\mathbf{u}}_{i}}{\partial \tilde{\mathbf{u}}_{j}} - \tilde{\tau}_{ij} \right); \quad i=1,2,3; \quad j=1,2,3; \quad (2.8)$$

where the bar on top of the variables implies that the variables are the time-averaged quantities. In Eq. (2.8), v is the effective viscosity while τ_{ij} is the shear-stress tensor. Eq. (2.8) is impossible to resolve due to the appearance of the Reynolds stress. To bring closure to the above equation, the Reynolds stress term is modeled through the means of k- ε turbulence modeling technique [13].

The $k-\varepsilon$ turbulence model is usually applied to simulate air flow fields in mechanical ventilation system and in other modern engineering applications. In the early stage of research, turbulence model was only applied for incompressible high Reynolds number flows but it was later experimentally proven that air flows next to solid walls were associated with low Reynolds numbers. Therefore, the development and testing of low Reynolds number turbulence models have been a topic for extensive research. A remedy to this approach is the introduction of a wall function into the modeling so that the airflow within the entire computational domain can be calculated at the same time even if the Reynolds number near the walls is low while that far away from the wall is high. The turbulence model used in this work is the RNG $k-\varepsilon$ turbulence model because of its good prediction of complex flows. The complete formulation of the RNG $k-\varepsilon$ turbulence model is given in Einstein summation convention as follows:

$$\frac{\partial(\rho k)}{\partial t} + u_i \frac{\partial(\rho k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{\mu_1}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + G - \rho \epsilon; \quad i=1, 2, 3,$$
(2.9)

$$\frac{\partial(\rho\epsilon)}{\partial t} + u_{i}\frac{\partial(\rho\epsilon)}{\partial x_{i}} = \frac{\partial}{\partial x_{i}} \left(\frac{\mu_{1}}{\sigma_{\epsilon}}\frac{\partial\epsilon}{\partial x_{i}}\right) + C_{1\epsilon}\frac{\epsilon}{k}G - C_{2\epsilon}^{*}\rho\frac{\epsilon^{2}}{k}; \quad i=1, 2, 3,$$
(2.10)

where k is the turbulence kinetic energy, ε is the kinetic energy dissipation rate, σ_k is the Prandtl number of the turbulence kinetic energy, and σ_{ε} is the Prandtl number of the turbulence kinetic energy dissipation rate. Rests of the variables are calculated based on the following equations.

$$G=2\mu_t S_{ij} S_{ij}, \tag{2.11}$$

$$C_{2\varepsilon}^* = C_{2\varepsilon} + C_{2\varepsilon}$$
 (2.12)

$$C_{2\varepsilon}^{'} = \frac{C_{\mu}\rho\eta^{3}(1-\frac{\eta}{\eta_{0}})}{1+\beta\eta^{3}},$$
 (2.13)

$$\mu_1 = \rho C_{\mu} \frac{k^2}{\epsilon}, \qquad (2.14)$$

 $\eta = Sk/\epsilon$, and (2.15)

$$\mathbf{S} = \sqrt{2\mathbf{S}_{ij}} \mathbf{S}_{ij,} \tag{2.16}$$

where S_{ij} the shearing-rate tensor, and g_i is the body force in the x_i direction. The coefficient of the model are modified based on such that $C_u = 0.085$, $\sigma_{\varepsilon} = 0.719$, $\sigma_k = 0.719$, $C_{I\varepsilon} = 1.42$, $C_{2\varepsilon} = 1.68$, $\beta = 0.012$, and $\eta_0 = 4.38$.

2.5 COMPUTATIONAL FLUID DYNAMICS (CFD)

2.5.1 Definition

Computational Fluid Dynamics (CFD) is an analysis of fluid flow, heat transfer and associated phenomena in physical systems using computers. The Navier-Stokes equation complemented with mass and energy conservation conditions describe the flow and the thermal behaviour of fluids. They are a set of non linear differential equations that can be analytically solved only under particular conditions, when they can be simplified [4].

Without these simplifications is generally impossible to solve the problem by analytical means. However, progress has been made during the last 30 years by developing a number of mathematical models to facilitate the numerical solution and reduce the calculation time. These models have given rise to a number of sophisticated numerical tools for fluid dynamic calculations. Some of these tools have developed further into commercial packages of which Fluent, StarCD and CFX are the most important products currently on the market. These codes are now widely used in many industrial, high tech and research applications.

CFD can be used to improve understanding of furnace behaviour & interactions, evaluate furnace or new technology performance, provide conceptual designs, identify potential operational problems, and also as the guide experiments.

2.5.2 Advantages and Disadvantages

The advantages of CFD are firstly it is relative low cost by using physical experiments and tests to get essential engineering data for design can be expensive. Secondly, CFD simulations are relatively inexpensive, and costs are likely to decrease as computers become more powerful. CFD simulations can be executed in a short period of time. Quick turnaround means engineering data can be introduced early in the design process and be able to simulate realistic conditions. Many flow and heat transfer processes cannot be easily tested such as hypersonic flow. CFD provides the ability to theoretically simulate any physical condition.

The disadvantages of CFD are the models are models of the reality which mean CFD solutions can only be as accurate as the physical models on which they are based. Besides, solving equations on a computer invariably introduces numerical errors. As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model. For example flow in a duct with sudden expansion. If flow is supplied to domain by a pipe, a fully-developed profile should be use for velocity rather than assume uniform conditions [10].
2.5.3 Elements

All CFD codes contain three main elements. Firstly, a pre-processor, which is used to input the problem geometry, generate the grid, define the flow parameter and the boundary conditions to the code. Secondly, a flow solver, which is used to solve the governing equations of the flow subject to the conditions provided. There are four different methods used as flow solvers are finite difference method, finite element method, finite volume method, and spectral method. And lastly, a post-processor, which is used to massage the data and show the results in graphical and easy to read format [10].

2.5.4 CFD in Automotive Industry

The Automotive Industry is perhaps the most important user of CFD techniques because its needs are complex and varied. The need for rapid prototyping and use of analytical tools during all stages of design and development, plus the tight schedules to reduce the time-to-market create one of the most challenging environments for CFD. External aerodynamics, under-hood air flow and thermal management, induction and engine in-cylinder flow, fuel injection and combustion, climate control and passenger comfort, aero-acoustic noise prediction are the major fields of application. Famous examples can be found in the Formula 1 sport domain where all major racing teams have one of the CFD code developer company as a strategic supplier [4].



Figure 2.8: Car Aerodynamic Simulation [10].

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

In order to get a better understanding of the aerodynamic behavior of the Proton Persona, a computational fluid dynamics (CFD) model was developed using commercially available software. In this case, SolidWorks will be use to model the geometry of the car and then it will be export to FLUENT to do the analysis within a two-dimensional and COSMOSFloWorks for three-dimensional analysis. The three-dimensional analysis cannot be analysis using FLUENT because of time consuming and high performance computer is required to do the analysis. So COSMOSFloWorks as the replacement software which is less computer requirement but still acceptable for 3-D aerodynamic analysis.



Figure 3.1: Flow Chart Methodology

3.3 MODELING GEOMETRY

The geometry of this project which is Proton Persona has been drawn by using SolidWorks. The vehicle length (L), width (W), and height (H) are about 4.477 m, 1.725 m, and 1.438 m, respectively. For 2D analysis, the cross section of the side face of the Persona car was selected and then the file was saved in IGES file (*.igs) in order to import the geometry to Gambit to generate mesh after that. And for 3D analysis, the geometry is directly analyze using COSMOSFloWorks that is already in the SolidWorks software. The details about the dimension of Proton Persona are shown in Figure 3.2.



*All dimension in millimeter (mm)

Figure 3.2: Drawing of Proton Persona

3.4 2D SIMULATION SETUP

3.4.1 Mesh Generation

To set up for 2D model meshing, the first step is to import the IGES file from the SolidWorks. Then the virtual wind tunnel will be draw and after the consideration of the size of the car, the duct that have dimension 18 m in length and 4 m in height need to be drawn. In order to do that, the first step is to place nodes (points where the grid lines of the mesh connect) on the edges. Total x-y coordinates of the duct the geometry was entered as vertices into Gambit. From here, the vertices were connected to create the edges of the 2-D model. The next step in Gambit is to take the edges and create faces. Then, the surface of the car need to be subtract so that only the outer of the car is to be meshed. Once the vertices, edges, and faces are created, actual meshing process can begin.

A few different options for mesh generation are available within Gambit, including those consisting of triangular elements or quadrilateral elements. In this case the triangular grid is applied to the entire domain. The mesh size function was created which is the start size is set to 1, the growth size is set to 1.3 and then the size limit is set to 500. The edge of the car was selected as the source of meshing and the face of the wall as the attachment. After taking a few moments, the mesh generates 66694 triangular cells to the entire domain (Figure 3.3).

After meshing, boundary zones are created on the geometry. These zones are used later by FLUENT to specify the boundary conditions. For this study, the top and bottom of the duct along with the edges of the car geometry were specified as separate zones called "walls." A "wall" is defined as a surface that is assumed to be solid that no fluid can flow through. The front of the duct was specified as a "velocity inlet," and the rear of the duct was specified as a "pressure outlet." A "velocity inlet" is used to define the velocity and scalar properties of the flow at inlet boundaries and a "pressure outlet" is used to define the static pressure at flow outlets. It is also noted that the zone types (wall, velocity inlet, etc.) can be changed within FLUENT as well, as long as zones are defined. Once the mesh and zones are created, the mesh is then imported into FLUENT.



Figure 3.3: Computational grid for FLUENT CFD modeling.

3.4.2 FLUENT Setup

The first steps taken after importing the mesh geometry into FLUENT involve checking the mesh/grid for errors. Checking the grid assures that all zones are present and all dimensions are correct. It is also important to check the volume and make sure that it is not negative. If the volume is shown as negative, there is a problem with the grid, since volume cannot be negative. The grid can also be displayed to ensure that the mesh generation is qualitatively reasonable. When the grid is checked completely and free of errors, a scale and units can be assigned.

3.4.3 Defining the Models

To run the cases, the model properties must be set. Model properties include the internal FLUENT solver type, material fluid properties, type of viscous and grid boundary conditions. The following settings were used to create the model in FLUENT.

Solver options include Pressure Based and Density Based, along with sub-options under each solver such as steady/unsteady and implicit/explicit. For this study, the options chosen were:

- Density Based
- Implicit
- Steady
- 2-D

3.4.5 Viscous

The viscous model option gives the user the choice between different turbulence models such as inviscid, laminar, Spalart-Allmaras, k-epsilon, k-omega, Reynolds Stress, Detached Eddy Simulation and Large Eddy Simulation. For this simulation, the k-epsilon model was used to capture turbulent flow.

3.4.6 Defining the Material Properties

This section of the input contains the options for the materials chosen as the working fluid. For this case, the working fluid is the air. Properties that can be specified in this section are density, and viscosity. For this study, the following options were chosen:

- density (air) incompressible ideal gas 1.225 kg/ m³
- viscosity (air)– 1.7894×10^{-5} kg/ms

3.4.7 Boundary conditions

Boundary conditions for the faces and volumes will be configured in Gambit. The flow conditions will be set as steady state, turbulent and incompressible. The velocity of the air at the inlet will be set in the range of 80 km/hr to 140 km/hr with increment of 20 km/hr. The boundary inlet pressure is left to be as default pressure which is 101325 Pa. Surface of the car will set as wall. Residual conditions were all set to 10^{-3} for convergence criteria. On the floor, free slip condition is applied in the region between the inlet and the front wheels to prevent the boundary layer from developing. The ceiling and the sidewall of the numerical domain are supposed to be free-slip on the surface.

3.4.8 Executing the FLUENT Code

Each case must be initialized before the FLUENT code begins iterating toward a converged solution. Initializing the case essentially provides an initial guess for the first iteration of the solution. In the initialization process, the user must specify which zones will be provided with initial conditions. For the modeling performed in this study the option chosen was to compute from velocity inlet. The final initialization step is for the user to enter the maximum number of iterations, after which the simulation begins. For the modeling performed in this study, the number of iterations ranged between 5000 and 10000 depending on the case being run and how long it took to converge.

Five different model properties were monitored by FLUENT's solver and checked for convergence. This criterion requires that the scaled residuals decrease to 10^{-3} for all equations. At the end of each solver iteration, the residual sum for each of the conserved variables is computed and stored, thus recording the convergence history. Table 3.1 is a list of variables and their respective convergence criteria.

Variable	Convergence Criterion
Continuity	0.001
X-velocity	0.001
Y-velocity	0.001
k	0.001
epsilon	0.001

Table 3.1: Variables and convergence criteria for FLUENT simulation of free layers apparatus.

If the solution converges, the results can be analyzed. If the solution does not converge within the given number of iterations, one can request additional iterations or check the results given at that point to determine whether additional iterations will converge toward a physical solution.

3.5 3D SIMULATION SETUP

3.5.1 COSMOS FloWorks setup

The present configuration of simulation is used COSMOS Floworks 2009 software. The Proton Persona is design using SolidWorks before export to Floworks. For the analysis type, external flows analyses deal with flows over or around a model and both cavities without flow conditions and internal flow is applied and all physical features such as gravitation, rotation and radiation is ignored. Air is used as fluid domain and turbulence flow is set.

For the computational domain, the size of the rear part or outflow is 9 m, extends to 2 times of body lengths. This consideration is applied to make sure that the outlet flow condition does not affect the near-body wake. The inlet flow is placed at 4.5 m from front part (1 of body length), both width and height is set about 5 body lengths corresponding to 11.107 m and 6.245 m. At the inflow section, a uniform velocity is applied at range 80 to 140 km/h with increment of 20 km/h. The turbulence model used was k- ϵ model where the

turbulence energy, k is 1 J/kg and the turbulence dissipation, ε is 1 W/kg. At both side boundaries and the top and bottom boundaries, free-slip conditions are applied (see Figure 3.4).



Figure 3.4: Computational domain and boundary condition



Figure 3.5: Grid refinement around the Proton Persona model

Figure 3.5 shows meshing of computational domain and grid refinement of the model. In grid or meshing part, 46 blocks (x-axis), 32 blocks (y-axis) and 106 blocks (z-axis) is set for the computational domain. The grid refinement is applied on bluff model by using the initial mesh condition. The level refinement is set to level 2, with the refinement criterion is 1.5 and unrefinement criterion is 0.15. The refinement strategy is set so that the refinement will occur at 30, 50 and 70th iterations. The overall meshing shows the number of cells is 126456, fluid cells 114347, solid cells 4978 and partial cells is 7131.

3.5.2 Frontal Area

The frontal area is one of the important factors in study of aerodynamics. The frontal area will contribute to the wake disturbances at the rear end of the car. This value is getting from the SolidWorks software itself. This value will be use in calculating the drag and lift coefficient for both 2D and 3D analysis of Proton Persona. So, the projected area of this car is 1.97m².



Figure 3.6: The frontal area of Proton Persona

CHAPTER 4

RESULTS AND DISCUSSION

4.1 INTRODUCTION

This chapter discusses about the analysis result of Proton Persona consist of 2D and 3D simulations. Both simulations using same model geometry that was build in CAD software (SolidWorks) but the software use is different. For 2D simulations, the software used is FLUENT 6 while COSMOSFloWorks for 3D simulations.

Both 2D and 3D result will be discussed separately. But the scope of discussion will be the same for both simulations. The scope involving aerodynamic drag and lift, the pressure and velocity distribution, and the flow pattern along the surface of the Persona car will be discuss in this chapter. All the simulations and result will be base on the four different inlet velocities which is 80, 100, 120 and 140 km/h.

4.2 CALCULATION OF REYNOLDS NUMBER

In order to ensure the flow is turbulence, the calculation of Reynolds Number is applied. To calculate the Reynolds Number, the length, L is representing by the overall length of Proton Persona car. From Eq.2.4:

$$Re = \frac{\rho v L}{\mu}$$

Where;

Length, L = 4.477 m Density of air, $\rho = 1.225 \text{ kg/m}^3$ Viscosity of air, $v = 1.7894 \text{ x } 10^{-5} \text{ kg/ms}$

Calculation for air speed, v = 22.22 m/s @ 80 km/h)

 $Re = \frac{(1.225)(22.22)(4.477)}{1.7894 \times 10^{-5}}$ $= 6.81 \times 10^{6}$

Calculation for air speed, $v = 27.78 \text{ m/s} \oplus 100 \text{ km/h}$)

 $\operatorname{Re}=\frac{(1.225)(27.78)(4.477)}{1.7894\times10^{-5}}$

$$= 8.51 \times 10^{6}$$

Calculation for air speed, v = 33.33 m/s @ 120 km/h)

$$Re = \frac{(1.225)(33.33)(4.477)}{1.7894 \times 10^{-5}}$$
$$= 10.22 \times 10^{6}$$

Calculation for air speed, v = 38.89 m/s @ 140 km/h)

$$\operatorname{Re} = \frac{(1.225)(38.89)(4.477)}{1.7894 \times 10^{-5}}$$
$$= 11.92 \times 10^{6}$$

According to Akbari (1999), the flow is fully turbulent at $\text{Re} \ge 3.5 \text{ X} \ 10^6$. Since all the Reynolds Number calculated above 3.5 X 10^6 , hence, all the simulation is fully turbulent. So the choice of turbulent model can be made.

4.3 2D ANALYSIS RESULT

4.3.1 Drag and Lift Analysis

After the analysis iterations are converged, the reports of forces were generated from the FLUENT. The force vector is (1 0 0) for drag force and (0 1 0) for lift force. The car surface is taken as the wall zones that to be calculated. The data collections for various velocities ranging from 80 km/h to 140 km/h are listed in the Table 4.1 and the graph has been plotted from the data.

Velocity (km/h)	$\mathbf{F}_{\mathbf{D}}(\mathbf{N})$	$\mathbf{F}_{\mathbf{L}}\left(\mathbf{N}\right)$	CD	CL
80	191.17951	1950.4744	0.320908275	3.27400868
100	292.58619	3051.5183	0.314207843	3.27702064
120	415.96073	4395.6165	0.310319727	3.27926753
140	559.71524	5986.9596	0.306703637	3.28063657
Average			0.313034871	3.27773335

Table 4.1: Table of various velocities, drag and lift data



Figure 4.1: Variation of C_D over speed range of Persona car



Figure 4.2: Variation of C_L over speed range of Persona car

Figure 4.1 shows the variations of C_D over the Persona car speed ranges from 80 km/h to 140 km/h. The graph shown that the drag coefficient is reversely proportional to the velocity which means that as the velocity increase, the drag coefficient is decrease. From the graph, it shown that the drag coefficient is decreasing significantly as the speed of Persona car is increases. Table 4.2 below show that the percentage of decrement is 2.09% as the velocity going up to 100km/h. Then the percentage is continued decrease about 1.24% and 1.17% to the velocity of 120 km/h and 140 km/h respectively. The significant different value of drag might come from large build change of wake disturbance at the rear end of the car as the velocity increase.

Figure 4.2 shows the variations of C_L over the Persona car speed ranges from 80km/h to 140 km/h. The graph trend shown is differently with the Figure 4.1. From the graph, the lift coefficient is directly proportional to the velocity which means that as the velocity increase, the lift coefficient is increase too. From the graph, it shown that the lift coefficient is increase rapidly but the truth is that the increment is only in the small size. From Table 4.2, the percentage of increment is only 0.09% as the velocity going up to

100km/h. Then the percentage is continued increase about 0.07% and 0.04% to the velocity of 120 km/h and 140 km/h respectively. It was found from this figure that the change in C_L is negligible as the speed of the Persona increases.

Table 4.2: Table of various velocities and percentages of C_D and C_L changes (2D)

Velocity	Percentage of C _D changes	Percentage of C _L changes
(km/h)	%	%
80	0	0
100	2.087958484	0.091911612
120	1.237434344	0.068517947
140	1.165278824	0.041731083



Figure 4.3: Graph of the C_D and C_L percentage changes against velocity

Since the value of the drag and lift coefficients is different with various velocity, the average is taken and the value is 0.313 for the drag coefficient and 3.278 for the lift coefficient as shown in Table 4.1. The estimation of the drag coefficient is acceptable as according to Klemperer (1992), the standard sedan car have 0.300 for the drag coefficient. But for the lift coefficient, the value is too large for standard sedan car. The high in value of this is might come from error in simulations setup. The small gap of the ground clearance and the present of the wheels during the simulations might contribute to high in pressure of the underbody compare to the top of the car. This large pressure differential between the top and the underbody of the car is responsible for the large lift force to the Persona car. Since there is no real data of drag and lift coefficient of the Proton Persona get from the literature, the validation of the value is just based on the theory.

4.3.2 Sample Calculations for Drag and Lift Coefficient

Sample calculations for analysis 80km/h. From Eq.2.2:

$$C_{\rm D} = \frac{F_{\rm D}}{1/2\rho V^2 A}$$

$$C_{\rm D} = \frac{191.17951}{\frac{1}{2} (1.225) (22.22^2) (1.97)}$$

$$= 0.320908275$$

From Eq.2.3:

$$C_{L} = \frac{F_{L}}{\frac{1}{2}\rho V^{2}A}$$
$$C_{L} = \frac{1950.4744}{\frac{1}{2}(1.225)(22.22^{2})(1.97)}$$

= 3.27400868

Sample calculation of percentages of C_D and C_L changes

$$=\frac{0.320908275 - 0.314207843}{0.320908275} \times 100\%$$

= 2.09%

4.3.3 Pressure, Velocity and Flow Pattern Analysis



Figure 4.4: The pressure contour with several of velocities inlet



Figure 4.5: The velocity contour with several of velocities inlet

From the Figure 4.4 and 4.5, the contour is developing from velocity 80 to 140 km/h which means that the pressure is increase as the inlet velocity increase. The pressure and velocity value is indicated by the contour which is increasing as the contour color change from blue to red. Every each case show that the highest pressure is at the front end of the car. At the same time, from the velocity contour, it shows that at that point the velocity is the lowest. It also known as the stagnation point where the separation flow occur on that region (see Figure 4.6).



Figure 4.6: Separation flow at the front end of Persona car

The flow is accelerated due to reattachment flow at front edge of the hood (see Figure 4.7) contribute to pressure decrement. Then separation flow occur again at the hood-windshield junction contribute to pressure increment. Due to upstream line curvature from the front windshield to the roof, the separation flow reattaches and the flow is accelerating along the roof. Hence the pressure is decreasing along the roof. It is indicated by color contour change from orange to green (see Figure 4.4).



Figure 4.7: Reattachment flow at the front edge hood of Persona car



Figure 4.8: Reverse flow and vortex generation at rear end of the Persona car

But then the reverse action happen where the pressure contour color change back from green to yellow along the slanted rear windshield of the car (see Figure 4.4). It means that the velocity is decreasing and the same time the pressure is increasing along the surface. Then the velocity is continued to decrease at the rear end of the car that is indicated by the blue contour (see Figure 4.5). In this region the reverse flow and vortex generation happen. This region is also known as the wake region.

The structure of the wake is a consequence of the flow around the body and can offer insight into drag-producing mechanisms. The wake behind the body is characterized by a separation zone as shown in Figure 4.8. A recirculating vortices are clearly visible and the separation bubble closes at the rear end of the car. This large wake disturbance is responsible for the high drag coefficient of this car.

Along the underbody of the car, the airflow is seemly slower than across the top of the car. It is indicated by the blue contour along the underbody. The bottom of the car therefore has a higher pressure than the top. The large different of the pressure between the bottom and the top of the car contribute to high lift coefficient of this car.

4.3.4 Pressure Distribution Analysis



Figure 4.9: The pressure distribution along the Persona car at velocity 80 km/h

Figure 4.9 shows pressure distributions on the surface of the car at 80 km/h. The results base on the pressure distribution along the surface of the car consists of upper and lower surface. For the upper surface, the highest Cp plotted is at the front end of the car. It is observed that the pressure decreases to have minus-peaks on the leading edge of the hood and roof, and also minus-peaks at the joint of the roof and the slanted surface where the flow is accelerated. These are the regions where the peaks are due to strong streamline curvature. But then the pressure increase back as the velocity is decrease at the rear end of the car. For the lower surface, the Cp is decreasing from the front through the rear of the car due to acceleration of velocity underbody. The lowest Cp plotted at the bottom of the wheels where the velocity accelerates rapidly as the area getting small.

4.4 3D ANALYSIS RESULT

4.4.1 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
- (c) Ambient parameter conditions
 - i. Ambient Pressure: 101325 Pa
 - ii. Ambient Temperature: 300.15 K
 - iii. Density of Air: 1.225 kg m-3
- (d) Analysis type: External Flow
- (e) Wall Condition: Adiabatic (Incompressible Flow)

4.4.2 Data Collection and Analysis

The drag and lift forces data collections that were getting started from CFD analysis for various velocities ranging from 80 km/h to 140 km/h was listed in the Table 4.3 and the graph from the data has been plotted in Figure 4.10 and 4.11.

Table 4.3: Data collection for analysis with various velocities

Velocity				
(km/h)	$\mathbf{F}_{\mathbf{D}}(\mathbf{N})$	$F_{L}(N)$	CD	CL
80	162.916	47.441	0.273465982	0.079617135
100	260.42	96.1033	0.279664623	0.103221694
120	375.641	139.215	0.280239947	0.103837978
140	514.121	190.256	0.281719649	0.10425934
Average			0.27877255	0.097734037



Figure 4.10: Graph of drag coefficient, C_D against various velocities

Figure 4.10 shown that the drag coefficient is directly proportional to the velocity which means that as the velocity increase the drag coefficient is increase too. The same trend of graph also shown by lift coefficient (see Figure 4.11). For velocity ranging between 80 km/h to 140 km/h from the graph, illustrating the drag coefficients were small slightly change of increasing velocity. Table 4.4 and Figure 4.12 show that the percentage of increase about 0.21% and 0.53% to the velocity of 120 km/h and 140 km/h respectively.



Figure 4.11: Graph of lift coefficient, CL against various velocities

The lift coefficient also shows the changes with increase the velocity. The increment were significantly when the velocity change from 80 to 100 km/h which by 22.87 %. Then the Figure 4.12 shown that there is small percentage increment as the velocity change from 100 to 140 km/h which by 0.59% and 0.40% respectively.

Velocity	Percentage of C _D changes	Percentage of C _L changes
(km/h)	%	%
80	0	0
100	2.216455086	22.86782816
120	0.205296754	0.59350501
140	0.525239226	0.404148446

Table 4.4: Table of various velocities and percentages of C_D and C_L changes (3D)



Figure 4.12: Graph of the C_D and C_L percentage changes against velocity

Since the value of the drag and lift coefficient were calculated, the average of drag and lift coefficient for this 3-dimensional analysis of Proton Persona body that ranging 80 km/h to 140km/h is 0.279 and 0.098 respectively as shown in Table 4.3.

4.4.3 Sample Calculation for Drag and Lift Coefficient

Sample calculation for analysis 80km/h. From Eq.2.2:

$$C_{\rm D} = \frac{F_{\rm D}}{1/2\rho V^2 A}$$

$$C_{\rm D} = \frac{191.17951}{\frac{1}{2} (1.225)(22.22^2)(1.97)}$$

$$= 0.273465982$$

From Eq.2.3:

$$C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A}$$

$$C_{L} = \frac{47.441}{\frac{1}{2} (1.225)(22.22^{2})(1.97)}$$

= 0.079617135

Sample calculation of percentages of C_D and C_L changes:

 $=\!\frac{0.279664623\text{-}\ 0.273465982}{0.279664623}{\times}100\%$

= 2.22%

4.4.4 Contour Plot of Velocity and Pressure



Figure 4.13: The velocity contour with several of velocities inlet

If looked detail from Figure 4.13, the big slight differences contour plot of velocity at the high velocity of 140km/h from others. The clear differential can be seen 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 the body.

At 120 km/ h, the wake region or separation region also clearly different where the intensity of blue color is more than the others in addition have a big of region. It shows the

base pressure drag is high and affected to the increasing of drag force. At the same time, it shows rationalization of analysis before from Figure 4.13.



Figure 4.14: The pressure contour with several of velocities inlet

For the Figure 4.14, 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 4.14. 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.

4.4.5 Flow Trajectories of Velocity Analysis

The detail analysis of the flow on the body by trajectories velocity flow analysis diagram were figured in the Figure 4.15. The figure was shown as a general flow on the Proton Persona body and the velocity of 140km/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 4.15: The isometric view of trajectories velocity flow of 140 km/h



Figure 4.16: The side view of trajectories velocity flow of 140 km/h

Shown at the Figure 4.16 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 (see Figure 4.17).



Figure 4.17: Separation flow at the front end of the car



Figure 4.18: Reattachment flow at the roof of 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 (see Figure 4.18). 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.



Figure 4.19: Reverse flow and vortex generation at the rear end of the car

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 at Figure 4.19, the wake region is due to the streamline flow at the trailing edge of side and roof of the car. Therefore, it's 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 RECOMMENDATIONS

5.1 CONCLUSION

Computational fluid dynamics (CFD) simulations of the turbulent flow field around a 2-D and 3-D of Proton Persona car were presented. The commercial CFD code Fluent with the $k-\varepsilon$ turbulence model was used for the 2-D simulations of aerodynamics while COSMOSFloWorks was used for the 3-D simulations. After all the analysis was done, as the conclusion from this project, the objective in this project was done successfully achieved which to estimate drag and lift coefficient of Proton Persona. The second objective also successfully achieved that is study of air flow behavior around of the vehicles during in motion.

Based on the cases considered in this work, the drag coefficient of Proton Persona from 2-D analysis is 0.313 and lift coefficient is 3.278. While for 3-D analysis, the drag and lift coefficient is 0.279 and 0.098 respectively. The estimation of the drag coefficient for both analyses is acceptable as according to Klemperer (1992), the standard passenger car have 0.300 for the drag coefficient. But for the lift coefficient, the value is quite far from the theory that might come from the error in simulation setup. Since there is no real data of drag and lift coefficient of the Proton Persona get from the literature, the comparison of the value is just based on the theory.

From the contour plot of the velocity in 2-D and 3-D simulations, 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.

From the contour plot also, the pressure contour is seems slightly in contrast to the velocity contour. It does mean that the highest speed occurs where the pressure is lowest, and the lowest speed occurs where the pressure is highest. Therefore it satisfies the Bernoulli's principle.

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. Moreover, 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 RECOMMENDATIONS

Some recommendations were list below to improve this analysis of project:

- 1. Use high performance computer to simulate 3D analysis using FLUENT.
- 2. Design and analysis on how to reduce drag and lift coefficient of Proton Persona.
- 3. Construct the experimental analysis by wind tunnel to validate the simulation analysis.
REFERENCES

- [1] Hucho, W. H. 1998. *Aerodynamic of road vehicle*. Fourth Edition. Society of Automotive Engineers, Inc.
- [2] Sumantran V. and Sovran, G. 1996. *Vehicle aerodynamics*. Society of Automotive Engineers, Inc.
- [3] Akbari, M.H. 1999. *Bluff-body Flow Simulations Using Vortex Methods*. Ph.D Thesis. Mcgill University Montreal, Canada.
- [4] N., Villafranca, J.L. 1992. Wake Similarity and Vortex Formation for Two-Dimensional Bluff Bodies. Ph.D Thesis. University Of Notre Dame, Indiana.
- [5] Battistin M. 2005. *A Future for Computational Fluid Dynamics at Cern*. Archamps, France: Ts Department.
- [6] Singh S.N., Rai L., Puri P. and Bhatnagar A. 2004. *Effect of moving surface on the aerodynamic drag of road vehicles*. Indian Institute Of Technology, Delhi, India.
- [7] Theera, K. and Kittichaikarn, C. 2003. *Numerical Analysis Of Flow Over Car Spoiler*. Kasetsart University, Thailand.
- [8] Tsubokura M., Kobayashi, T., Nakashima, T., Nouzawa, T., Nakamura, T., Zhang, H., Onishi, K. and Oshima N. 2007. *Computational visualization of unsteady flow around vehicles using high performance computing*. 38(5): 981-990
- [9] Basara B. 1999. Numerical Simulation of Turbulent Wakes Around A Vehicle. *Asme Fluids Engineering Division Summer Meeting*. California, USA.
- [10] Sodja, J. 2007. *Turbulence models in CFD*. University of Ljubljana: Department of physics, Faculty for mathematics and physics.
- [11] Versteeg, H. K., Malalasekera, W. 2007. An Introduction to Computational Fluid Dynamics The Finite Volume Metho. Pearson, Prentice Hall.
- [12] Hovermann, F. 2003. Development of a New Apparatus to Measure Flame Spread Through a Free Stratified Fuel/Air Mixture. *Numerical Modeling and Experimental Results*, Rowan University.
- [13] Tsai, C.H., Fu, L.M., Tai, C.H., Huang, Y.L. and Leong J.C. 2008. *Computational aero-acoustic analysis of a passenger car with a rear spoiler*. Elsevier Inc.

APPENDIX A1 GANTT CHART FOR FYP 1

Projectactivities	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14
Identify Froblam														
Objective and Scope														
Literature review														
Developing model geometry in Solidwork														
Learning FLUENT														
Report Writing														
Presentation Preparation														
FYP1 Fresentation														

APPENDIX A2 GANTT CHART FOR FYP 2

Project activities	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14
Refining model geometry in SOLIDWORK														
Simulation using FLUENT & COSMOS														
Discussion and Conclusion														
Thesis writing														
Presentation Preparation														
FYP2 presentation														

APPENDIX B PROTON PERSONA DRAWING



