AN AERODYNAMIC STUDY OF CAR SPOILER USING CFD

MUHAMMAD ZAID BIN NAWAM

A thesis submitted
in fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering with Honours

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2018

DECLARATION

I declare that this thesis entitled “An Aerodynamic Study of Car Spoiler Using CFD” is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : ....

Name : Muhammad Zaid Bin Nawam

Date : ....

APPROVAL

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

Signature : ....

Supervisor's Name : Dr. Mohd Afzanizam Bin Mohd Rosli

Date : ....

DEDICATION

To those who have been supporting me throughout my 4 years of study.

Thank you.

ABSTRACT

Aerodynamic design of an automotive vehicle influences the performance, safety, and maneuverability of the vehicle. The factor of rising price of the fuel besides the world vision on reducing the greenhouse effect are some of the aspects that boost the needs of refining the vehicle aerodynamics. There is a fact that the design of the vehicle affects 11% of fuel to overcome the drag force on a high-speed driving. Furthermore, the shape of the vehicle especially sedan type has greatly affected by the lift force. Undesirable high value of lift force on an automotive vehicle could cause lack of stability and safety. In this study, the improvisation of the aerodynamic characteristics of a vehicle is discussed in term of rear- wing spoiler design. Four spoiler designs are proposed, analyzed and compared to each other. The drag and lift coefficient value of the vehicle with attached spoiler, the pressure and velocity distribution are evaluated in determining the best spoiler design. The flow characteristics were analyzed using ANSYS FLUENT®. The CFD technique used is validated by using the Ahmed body as a benchmark as the experimental result on the model is available. Grid independence study is conducted in determining best mesh resolution and turbulence model for this study. The generic sedan car as a base model and the spoilers are created in CATIA®. The base model is attached with various spoiler designs and tested to gather its data and result. The finding of the study shows that without a spoiler, the drag and lift coefficient of the car are 0.19230 and 0.03957. While when attached with spoiler, there is an increment in the drag coefficient of the car in range of 26.68% to 150.89%. However, the lift coefficient value shows a declination in range of 1.39% to 1992.87%. The best spoiler design is spoiler 3. The best spoiler design should have the least drag coefficient increment and considerably higher negative lift coefficient.

ABSTRAK

Rekabentuk kenderaan automotif mempengaruhi prestasi dan keselamatan kenderaan. Faktor kenaikan harga minyak dan visi untuk mengurangkan kesan rumah hijau menyebabkan perlunya penambahbaikan aerodinamik kenderaan. Terdapat satu fakta bahawa reka bentuk kenderaan memberi kesan kepada 11% bahan bakar untuk mengatasi daya seret pada kelajuan tinggi. Selain itu, bentuk kenderaan terutamanya jenis sedan banyak dipengaruhi oleh daya angkat. Nilai angkat yang tinggi pada kenderaan automotif boleh menyebabkan kekurangan kestabilan dan keselamatan. Dalam kajian ini, penambahbaikan ciri-ciri aerodinamik kenderaan dibincangkan dari segi reka bentuk spoiler sayap belakang. Empat reka bentuk spoiler dicadangkan, dianalisis dan dibandingkan. Nilai pekali seretan dan angkat kenderaan dengan spoiler, pengagihan tekanan dan kelajuan dinilai dalam menentukan reka bentuk spoiler terbaik. Kajian kebergantungan grid dijalankan dalam menentukan resolusi mesh dan model aliran terbaik untuk kajian ini. Ciri-ciri aliran dianalisis menggunakan ANSYS FLUENT®. Teknik CFD yang digunakan disahkan dengan menggunakan model Ahmed sebagai rujukan. Kereta sedan sebagai model asas dan spoiler dihasilkan menggunakan CATIA®. Model asas dipasang dengan pelbagai reka bentuk spoiler dan diuji untuk mengumpulkan data dan hasilnya. Dapatan kajian menunjukkan bahawa tanpa spoiler, pekali seret dan angkat kereta adalah 0.19230 dan 0.03957. Manakala, dengan spoiler yang dilampirkan, terdapat peningkatan dalam pekali seret kereta di antara 26.68% hingga 150.89%. Walau bagaimanapun, nilai pekali angkat menunjukkan penurunan sebanyak 1.39% hingga 1992.87%. Spoiler 3 telah dipilih sebagai bentuk spoiler terbaik .Rekabentuk spoiler yang terbaik sepatutnya mempunyai peningkatan pekali seret paling rendah dan penguranagan pekali angkat yang berpatutan.

ACKNOWLEDGEMENT

First and foremost, I would like to express my deepest appreciation to all those who provided me the possibility to complete this report. My special gratitude to my final year project supervisor, Dr. Mohd Afzanizam Bin Mohd Rosli, whose contribution in stimulating suggestions and encouragement, helped me to coordinate my project especially in writing this report.

I have to appreciate the guidance given by other supervisor as well as the panels especially in our presentation that has improved our presentation skills thanks to their comment and advices.

My appreciation to all lecturers and my friends for their knowledge sharing, encouragement and helps. Thank you so much.

Special thanks to my mother, Salasiah Binti Jamaludin and my family for encouragement and support during my study.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF APPENDICES

LIST OF ABBREVIATION

2D	-	2 Dimension
3D	-	3 Dimension
CFD	-	Computational Fluid Dynamics
C_D	-	Drag Coefficient
C_L	-	Lift Coefficient
Eq.	-	Equation
JK PSM	-	Jawatankuasa Projek Sarjana Muda
k-ε	-	k-epsilon
k-ω	-	k-omega
SST	-	Shear Stress Transport
UTeM	-	Universiti Teknikal Malaysia Melaka

LIST OF PUBLICATION

Nawam. M. Z., M. Rosli. M. A., & Rosli. N. A. S. (2018). Simulation study on the effect of rear-wing spoiler on the aerodynamic performance of sedan vehicle. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences.* Submitted on 6^{th} June 2018.

CHAPTER 1

1 INTRODUCTION

This chapter covers the introduction of the project entitled “Simulation Study of Aerodynamic in Car Spoiler”. In this chapter it will cover project background which consists of introduction of aerodynamic and spoiler development, the problem statement, objective and scope of the project.

1.1.1 Project Background

The present car industry has provided the users with many types of cars ranging from sports, sedan, and truck to a luxury car. Back to the old days, where the aerodynamic design of a car is not a crucial thing, the common shape of the car is like a box. In the country of Malaysia, the trend of a box-like shaped car could be seen in the production of the first national car in the mid-1980s that is Proton Saga (Jomo K. S., 2003). This trend of a box-shaped car has continued for several years and several models in Proton production. However, in the present car industry, aerodynamic design and shape have become a very important aspect. The old box-shaped cars could be hardly seen in the new-production car nowadays. The development in the car industry has led to a more economical and efficient vehicle. The factors of the development are due to the advancing technologies besides the factor of energy-efficiency.

In this project, the main discussion is mainly focusing on the aerodynamic. According to (Sandra May, 2015), aerodynamics is the way of air moves and flows through things. Aerodynamic forces including drag and lift forces are the important aspects and values that determine the quality of the aerodynamic design of the vehicle. The drag force is a sort of force that is trying to stop or slows down the vehicle movement. It may be in term of tire friction with the road or may be in term of air drag. Drag forces play an important role in affecting the value of the car drag's coefficient. Drag coefficient as referred to (Sandra May, 2015) is a value that engineers and aerodynamicists use to model all of the complex dependencies of body shape or design and flow condition on car drag. These values of drag in term of force or coefficient give a big impact on the car economy and efficiency. Lift force, on the other hand, is a force that acts perpendicularly to the motion of the vehicle. It is a sort of forces that potentially lift the vehicle upward. Lift coefficient is described as a value that is used to simulate all the complexity of dependencies of the flow behavior and condition and its design that contributed and affected the lift. While the drag coefficient gives impact in efficiencies, the lift values affect the stability of the vehicle. Thus, these two values must be in the best range to produce a good design of a vehicle.

Aerodynamic of the car has now been in the top priority for the car designer. There are abundance of study and research in the automotive industry that found the advantages and cruciality of the aerodynamic. (Mazyan, 2013) has discussed the effect of attaching rear spoiler in order to improve the aerodynamic design of the Ahmed body. The finding of the simulation clearly shows that there is an improvement in the aerodynamic drag behavior acting on the car. The result of 10% decrement in the aerodynamic drag could give a big impact on the performance and efficiency of the car. A study by (G, Mukkamala, & Kulkarni, 2014) found that 50% of the vehicle fuel consumption on a standard highway speed is contributed by the aerodynamic performance of the vehicle.

The development in the car industry and technology also has produced a more high-performance car that can achieve a speed as high as 457.86km/h such as produced by Koenigsegg Agera RS. As we know, there are many factors involved in achieving this record. However, a good aerodynamic design must be a crucial factor that contributed to the achievement. Without a good aerodynamic design of the car, the high speed as that could be nearly impossible. In the other hand, a car with that speed without proper aerodynamic design could be dangerous and impractical to be driven.

In today's car design development, designers and engineers have come out with many ideas and solution to the aerodynamic matters. Some of them are by changing car's design in term of body frame as seen in the evolution of car chassis and body from the tear-drop car until the streamline design. In addition, other than changing the body shape, the other alternative done by the engineers are by adding a device to improve the aerodynamic properties of the car. There are many types of these aerodynamic devices such as bumpers, side skirt and spoilers. In this paper, none of the car chassis and body design will be discussed. The main thing that will be focused is about the aerodynamic device that is the rear-wing spoiler.

The rear-wing spoiler is a device that is added to the rear of the car to improve the car's aerodynamic and stability by adding more pressure on the rear of the car. (Cheng & Mansor, 2017) states that generally there are two types of common rear spoiler equipped in most of the vehicle especially car. The type of the spoilers are free-standing wing spoiler and the other type is strips. In both sedan and hatchback car type, the common spoiler used is strips. Sometimes they are factory standard equipped and sometimes it could be purchased aftermarket. While the free–standing wing spoiler is commonly used for a sedan car as it requires a sufficient area to be attached to a car. In the early 20s, many car users install the rear spoiler for the purpose of decoration. However, the rear spoiler today is not only for that purpose. The adding of the rear spoiler could affect the lap time for a race car. Besides, the stability of road car could have a big difference with a rear spoiler.

1.2 Problem Statement

When a vehicle is moving, it is cutting through an air. By imagining walking in the swimming pool, it is the same thing happens when we move or walk in the air, just the effect is not so significant compared to moving in water. But when saying about a high-speed movement even in the air, for example, 100km/h that is an average speed of today's car, the design of the car plays an important role for the efficiency and stability of the movement. Too many drag forces on the car body could cause high waste in engine energy consumption while too much lift forces could cause the car lacks in stability and dangerous to be driven.

Usually, the lift force affects most on the rear of the car especially a sedan car because of its shape. Moving at high speed causes the high-speed air to move following the body shape of the car causing high pressure on the front hood follows by front windshield. But arriving at the rear of the car, the air suddenly loses its track and causing the air flow to be turbulence and producing a low-pressure region at the car's rear. This low-pressure region is undesired as it could cause an undesired drag and lift and the tires could lose its traction to the road and causing an accident.

In conclusion, the problem statement for this research are as follows:

1. How to obtain C_D and C_L?
2. What is the effect of attaching a rear spoiler?
3. What is the best spoiler design?

1.3 Objectives

The objectives of the project are:

i. To design a spoiler for a sedan car.
ii. To predict the C_D and C_L of the car without attaching rear spoiler using ANSYS-Fluent®.
iii. To predict the C_D and C_L of the car with rear spoiler using ANSYS-Fluent®.

1.4 Scope of Project

Aerodynamic characteristic depends on many factors, this study focuses on external devices that is rear-wing spoiler on a sedan car. The flow of this project mainly based on the design, development and numerical simulation of the effects of rear-wing spoiler. There are four spoiler designs will be considered by modifying the shape of the spoiler. The finding of this study is discussed in term of C_D and C_L value, pressure and velocity distribution.

CHAPTER 2

2 LITERATURE REVIEW

This chapter will give a generic view about the background and development of car spoiler along with the concept and theories of aerodynamics that relate with this study.

2.1 Introduction

Car spoilers, vortex generator, tail plate and similar functioning devices were once a mere product. However, in today scenarios, spoilers had become a must equipped device in a most performance car.

In 1899, a Belgium race car driver, Camille Jenatzy breaks the records on 100km/h barrier for the first time in a race car (Katz, 2006). As shown in the Figure 2-1, the car that Jenatzy drives is the car that is described as the pioneer of the streamlined car in the automotive industry. This electro-powered race car has a torpedo shape to reduce the total drag acting on the car.

Figure 2-1: Jenatzy's car (La Jamais Contente) (Adam Lohonyai, 2014)

The post-world war era shows a great development in car industry focusing on the aerodynamic design of the car. This era has produced a most significant design of a teardrop design car shape. In 1921, Edmund Rumpler surprised visitors at the Berliner Automobile Ausstellung with his revolutionary car design named 'Tropfenwagen' or a 'Teardrop Vehicle' as shown in Figure 2-2. This car could achieve a markable speed of 185km/h that was pretty fast in that era. Besides, the efforts in reducing the drag acting on the car had shown a great achievement with its tested drag coefficient in 1979, which resulted in only 0.28 of C_D value (Li Yanlong Yang Zhigang, 2013) which is quite impressive comparing with today's technology and achievement.

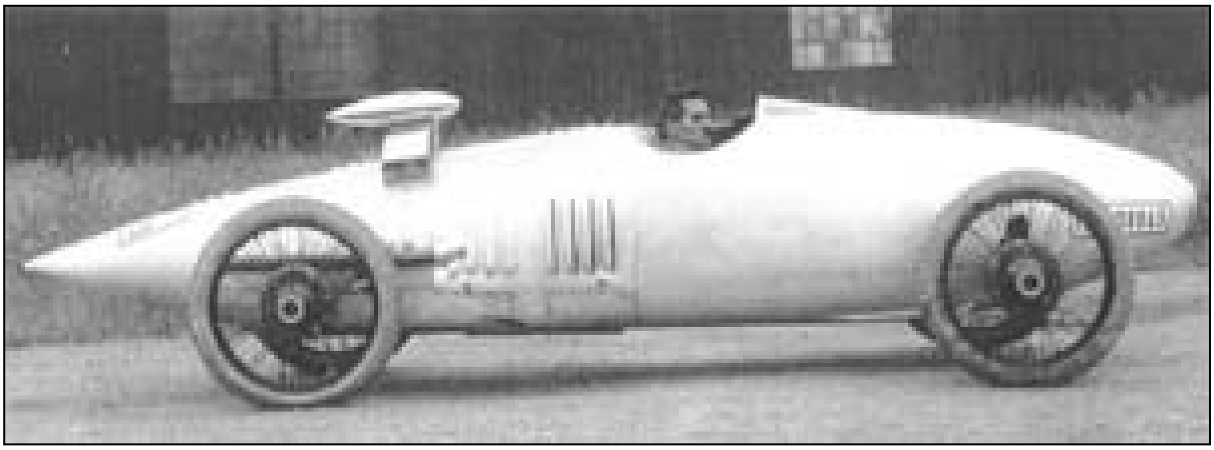


Figure 2-2: Benz-Tropfenwagen (Mattijs Diepraam, 2005)

Edmund Rumpler, the designer of the Tropfenwagen held the first aerodynamic testing in a wind tunnel. The Tropfenwagen showed an impressive result in the test. Few years later, in 1929 a grandson of Opel founder, Fritz Von Opel has come out with a new concept of rocket race car that utilizes jet engine and employ inverted wings (Katz, 2006) as shown in Figure 2-3. The wings equipped at the side of the car are in a negative angle of attack in order to create a large down force. However, this aggressive innovation is totally ignored. In the same era, a Hungarian engineer, Paul Jaray has developed an innovation of smooth surfaces of the car's body, headlamps, tapered rear end and cambered windshields as shown in Figure 2-4. It is to be admitted that his innovation managed to achieved a low value of drag coefficient that are ranging from 0.29 to 0.30 (Katz, 2006). However, (Syon, 2008) states that due to the conflicts of political and economic turmoil in the 1930s causing the idea being just a short-lived experiment. Still, it inspires others to follow his steps. Due to the great achievement, much of Paul's innovations were adopted by many car manufacturers such as Audi and BMW.

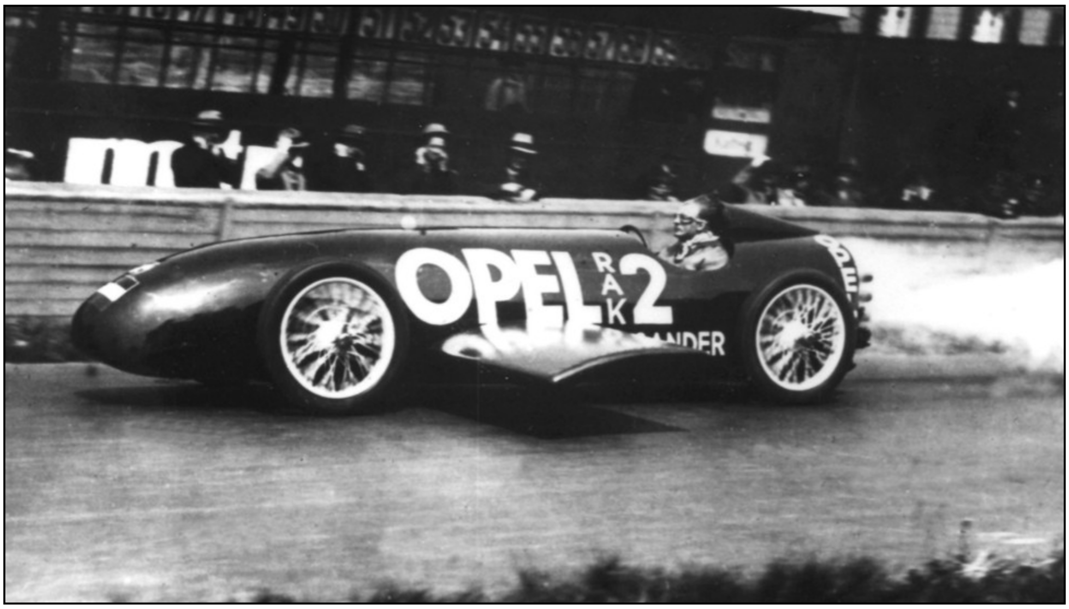


Figure 2-3: Opel-RAK (Michael Ballaban, 2014)

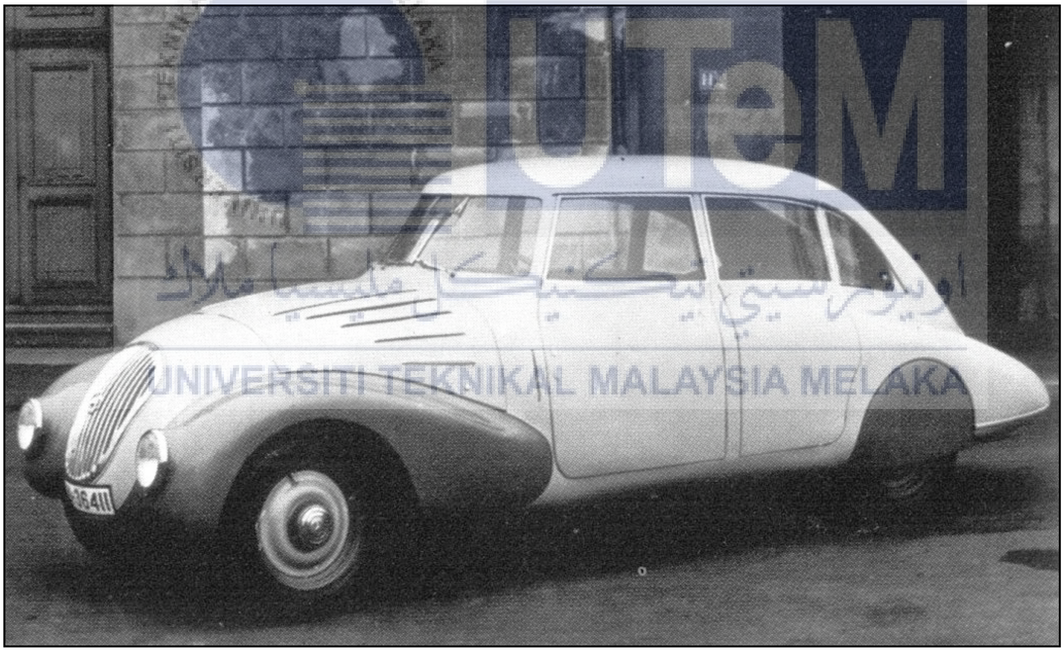


Figure 2-4: Paul Jaray's Streamline Car (Alex Oagana, 2010)

In the early years of 1930s, a German aerodynamicist Wunibald Kamm brought the concept of aerodynamics in cars, with the use of airfoils resulted in Kammback cars. One of the car that applied Kammback design is Ford GT as shown in Figure 2-5. Kammback is designed based on the streamline shape but only its back is slightly truncated. Kammback design achieve a lesser drag coefficient comparing to the complete air foil or streamline shape.

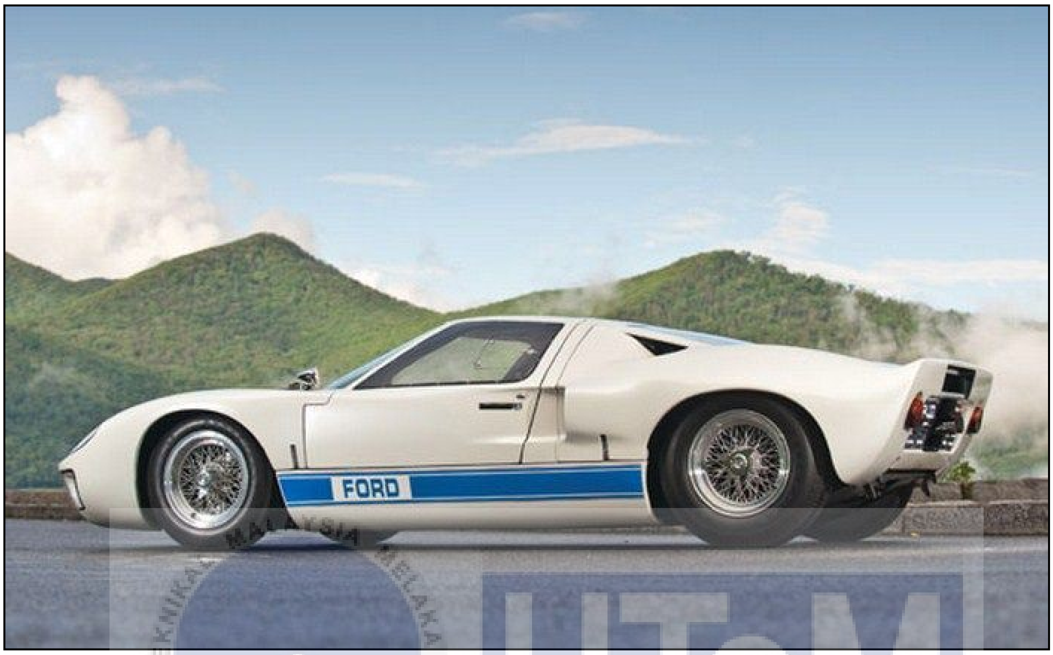


Figure 2-5: Ford GT (David Villarreal, 2012)

The development of the car designs throughout the era was majorly caused by optimization of aerodynamic features of the car. The modification and development in terms of curvatures, spoiler's location and height may be included in the detail changes in the car design. Besides, aggressive innovation such as equipment of inverted wings also takes place. The development of car shapes is also significant as shown in the above figures.

The detail optimization had caused and resulted in the dramatic change of the performance of the car. However, the car manufacturer's main focus is in the reduction of the drag. Hence, the shape optimization was given priority. In 1930s, the re-evaluation and study on the drag reduction by aero-dynamists was conducted. Starting from that, car designs became more realistic as we can see in today's car design. In 1983, Audi 100 as shown in figure 2-6 was the first car with a conventional design that achieved a drag coefficient as low as 0.3 (Julian Edgar, 2011).

Figure 2-6: Audi 100 (Jonathan Crouch, 2006)

Current State of Art (Julian Edgar, 2011)

- The current state of art in aerodynamics utilizes both the detail and the shape optimization
- The reasonable and acceptable value of coefficient of drag for modern cars could be in range of 0.25 to 0.35
- Reasonable future drag coefficient should be under 0.25

The development of car spoilers comes along with the evolution of car's shape and detail improvisation. Besides, there are many crucial and important principles, theories, and concepts that comes along with the development. For example, the effect of velocity, pressure around the cars body etc. Due to that reason, it is important to discuss and review about them to get the overview of the working principles behind the significant of car spoilers.

2.2 General Concepts

In this section, the literature review had been divided into a few categories to provide a clearer view of the literature review. Every part should make the concepts of the theory to be easier to be understood.

2.2.1 Lift concept

Aerodynamic lift is described as a vertical force that lift a body onto the air. The concept of aerodynamic lift lies on the principle of pressure difference that act on the object.

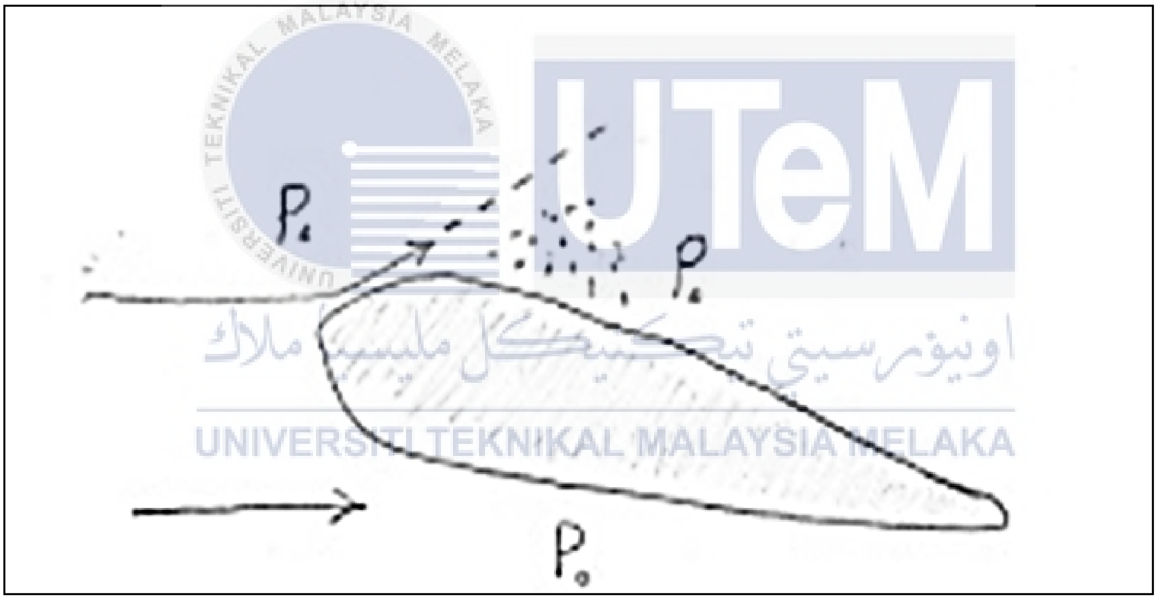


Figure 2-7: Airfoil shape (Joseph F. Alward, 1997)

Figure 2-7 shows a cross-section of an airfoil shape. When the air around the airfoil object is in rest condition, the pressure around the body is equal to the atmospheric pressure that is P_0. As soon as the wind starts moving through the airfoil object, logically the wind will stream straight to the above line as shown on the straight broken-line. Right below the broken-line, there are dots that represent the particle of air that is in rest condition.

However, as soon as the wind strikes the airfoil object, the onrushing wind stream collides with the air that is at rest condition causing the air particles at rest join the moving wind stream leaving the airfoil object’s surface. The process of sweeping and combining of both group of air is called as entrainment process (Joseph F. Alward, 1997). This process could change the expected stream direction as shown above to the reality outcome of the wind stream direction as shown in figure below. As shown below, Figure 2-8, it is clearly can be seen that the flow of air separated into two directions after the entrainment process occurs.

Figure 2-8: Airfoil shape (Joseph F. Alward, 1997)

The particles of air on the upper level above the airfoil object have entrained into the stream leaving a few air particles near the airfoil object surface. This cause a lower pressure region in the region P. Thus, the wind in relatively higher-pressure region, P_0 will be pushed towards the lower pressure region P. This bending effect of the air stream that follow the airfoil contour is called “Coanda Effect” (Chao et al., 1992).

While there are relatively low and high-pressure region on the upper part of the airfoil body, the pressure under the airfoil body is still same as the atmospheric pressure if there are no air stream flow under the body. Thus, the under-body region is relatively higher pressure compared to the upper surface of the airfoil body hence creating pressure difference between the lower and the upper part of the airfoil. As we know, the formula of lift force:

$$FF_{LL}=\Delta PP\times AA \qquad \text{Eq.1}$$

Thus, the pressure difference between lower region (P_0) and upper region (P) will produce a lift force on the airfoil body. The force F_1 is shown in the Figure 2-9 below:

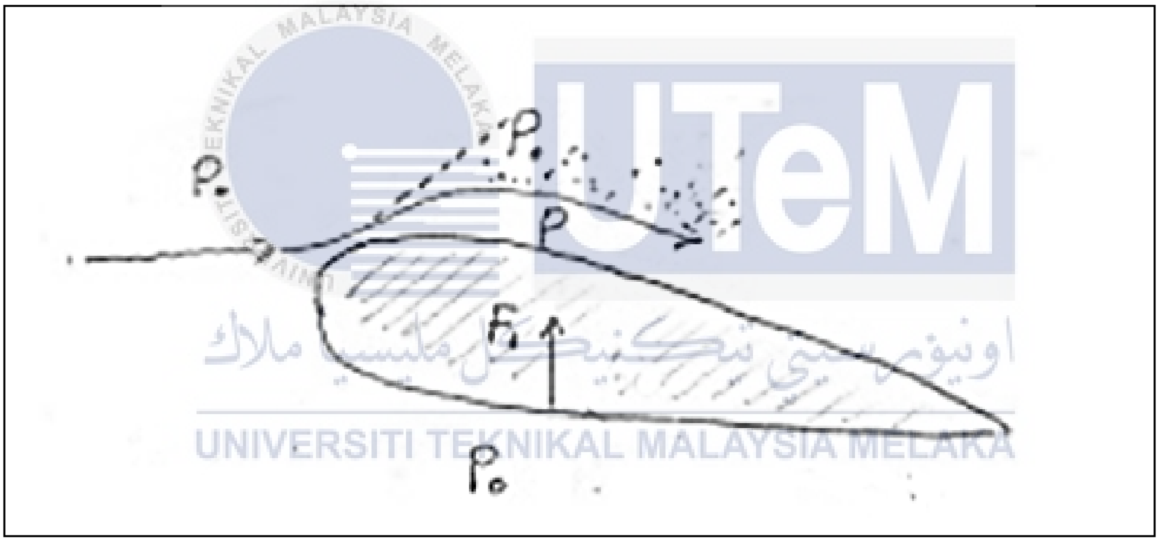


Figure 2-9: Airfoil shape (Joseph F. Alward, 1997)

2.2.1.1 Lift Concept on Car

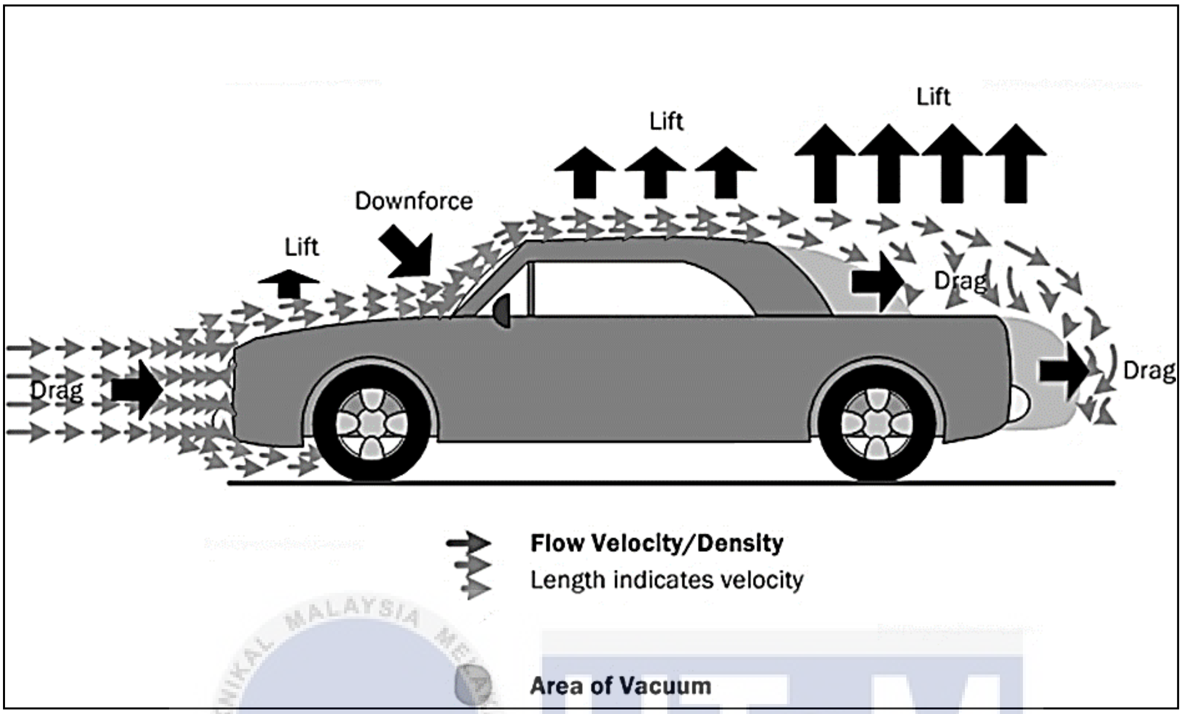


Figure 2-10: Aerodynamic Lift on Car (Adam Lohonyai, 2014)

In case of car aerodynamic, the concept of the lift is same as the lift concept explained above. When the car is moving at certain speed, the turbulence flow of the air creates a high-pressure region on the frontal and upper body of the car while the lower part of the car has relatively lower pressure. Thus, the pressure difference could create undesirable lift force. The aerodynamic forces that act on the car can be seen in the figure 2-10. In aeronautic industries, the aerodynamic lift is desired. However, in automobile industries, such lift force is undesired as it could cause a lack in stability and traction thus affecting the performance of the car. The presence of vacuum or low-pressure area at the back of the car is the main issue that plays a big role in affecting the drag and lift coefficient value of an automotive vehicle.

$$F_L = \frac{1}{2}\rho v^2 C_L A \qquad \text{Eq.2}$$

Where F_L is the Lift force

ρ is the density of the fluid.

v speed of the object

C_L is the lift Coefficient

Equation above shows an aerodynamic lift equation (Joseph F. Alward, 1997). Based on the equation, it is clearly shows the relation between lift force and velocity. The relation shows that, higher lift coefficient value will affect in higher lift force. This equation will be used in later chapter to obtained the lift force exerted on the car body.

2.2.2 Drag concept

(Glenn Elert, 2017) describes aerodynamic drag force as a force that resists the motion of an object through the medium of a fluid. There are two types of drag that acts on a moving body:

1. Pressure-drag
2. Skin friction-drag

In the present modern-day passenger sedan cars, the coefficient of drag ranges from 0.3–0.5 for any wind angle at zero degrees (L.E. Mayfield, n.d.).

2.2.2.1 Pressure Drag

In a moving object, the main type of pressure that acts on a moving object is pressure drag. Pressure drag is mainly caused by the air particles being more pushed together on the front-facing surfaces and more spaced out on the back surfaces. This is caused when the air stream separates away from the surface and begin to swirl (Dr Mark Jermy and Lindsey Underwood, 2011). Figure below shows the pattern of air flow on an airfoil object.

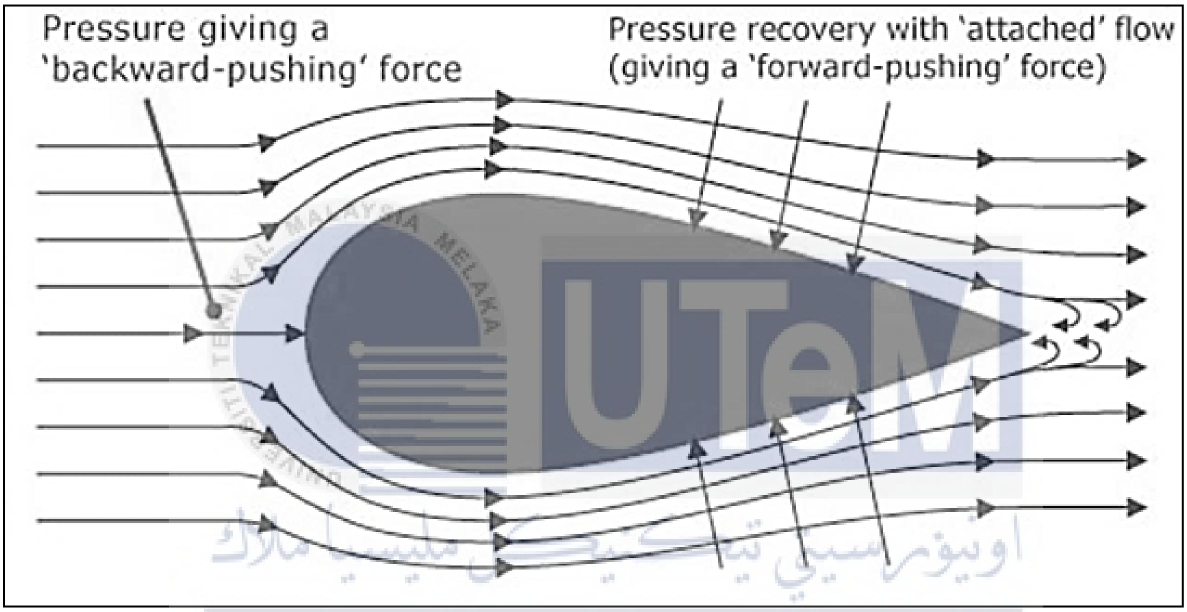


Figure 2-11: Airfoil shape (Dr Mark Jermy and Lindsey Underwood, 2011)

As we see on the above Figure 2-11, we can see that there is relatively higher-pressure region act on front of the object while the back of the object is relatively lower pressure region. Thus, the pressure difference will cause a drag force opposing the motion of the object. The equation below shows the relation of pressure difference with the drag force.

$$FF_{DD} = \Delta PP \times AA \quad \text{Eq.3}$$

Clearly, the equation shows that as the pressure difference increase, the drag force acting on the object would be increase too. Though, the drag force acting on an object depends on the object shape. As shown in the above figure, airfoil shape makes the lower pressure region at the back of the object smaller. Thus, reducing the lower pressure region means reducing the drag force acting on it. Comparing to figure below, the low-pressure region at the back of the circle object is relatively larger region. Hence, the pressure difference between the frontal and back facing region would be higher thus creating higher value of drag force.

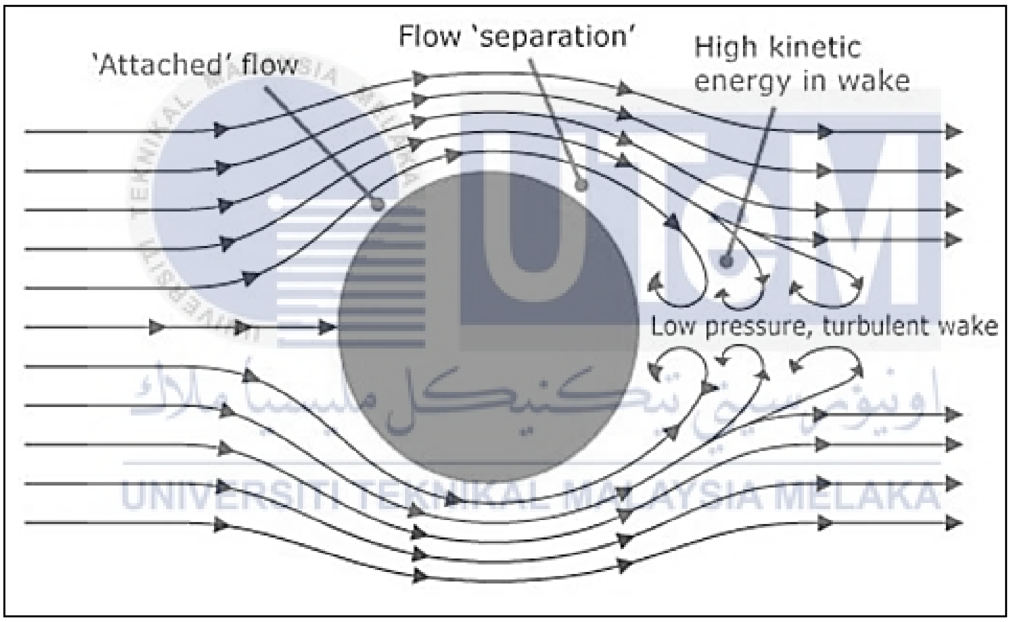


Figure 2-12: Ball shape (Dr Mark Jermy and Lindsey Underwood, 2011)

2.2.2.2 Skin Friction Drag

As the air stream move over a rough surface of object, the particles of air that are closest to the object surface will collide with the surface. This cause the motion of the air particles slow down. The slowed down air particles then will collide with other air particles in the onrushing air stream. Due to the collision, the movement of the onrushing air stream will also slow down. However, the air particles that flow further away from the rough surface would not be affected by the slowing down air particles. Figure below shows the air flow that is further from the car is not affected by the flow near the car.

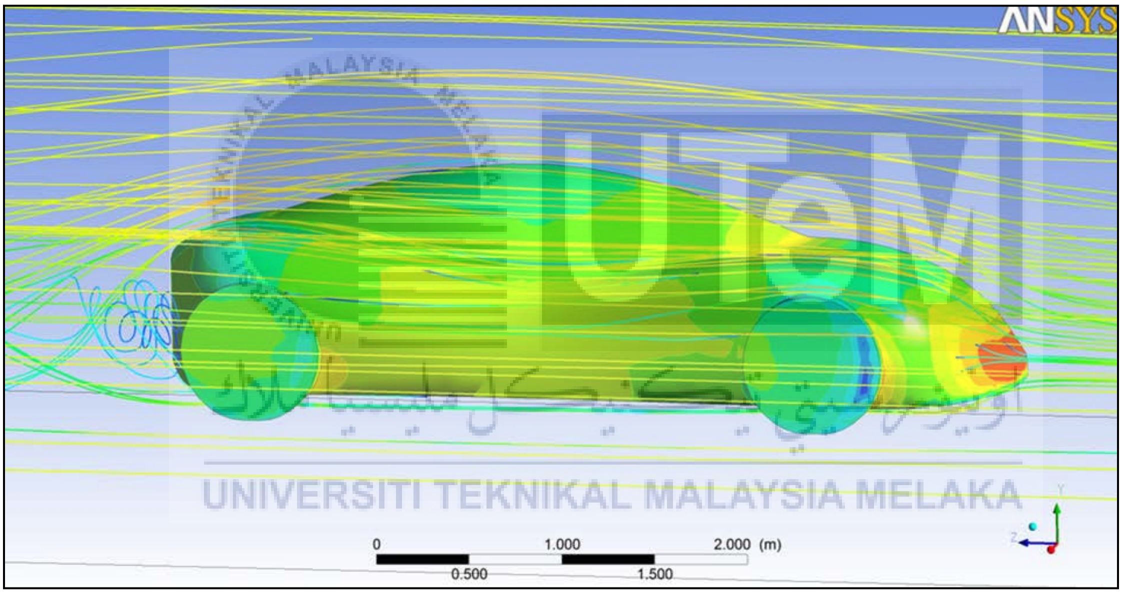


Figure 2-13: Airflow on car (Dio Rhapsody, 2016)

Equation below shows an aerodynamic drag equation (Dr Mark Jermy and Lindsey Underwood, 2011). Based on the equation, it is clearly shows the relation between drag force and frontal area of the object. This means that sleeker or slimmer body is less drag compare to wider body. However, the velocity and density are also proportional to the drag force. This equation will be used in later chapter to obtained the net force acting on the x direction on the car body along with the viscous forces.

$$F_D = \frac{1}{2}\rho v^2 C_D A \qquad \text{Eq.4}$$

Where F_D is the drag force

ρ is the density of the fluid

v is the speed of the object in the fluid

C_D is the drag Coefficient

A is the cross-sectional area

2.2.3 Ahmed Body

(G et al., 2014) states that the Ahmed body is employed by (Ahmed et. al. 1984). It is a simplified ground vehicle geometry which enables us to understand the fully three-dimensional flow around real-life automobiles as shown in Figure 2-14 and 2-15. The drag force experienced by this body is greatly influenced by the wake formed at the rear. Furthermore, it has been found that 40% of the total drag acting on a car body is influenced and concentrated at the rear (Chainani, Perera, & Noel, 2008). The experimental data for the Ahmed body that is used in this study is obtained from (Meile, Brenn, Reppenhagen, Lechner, & Fuchs, 2011).

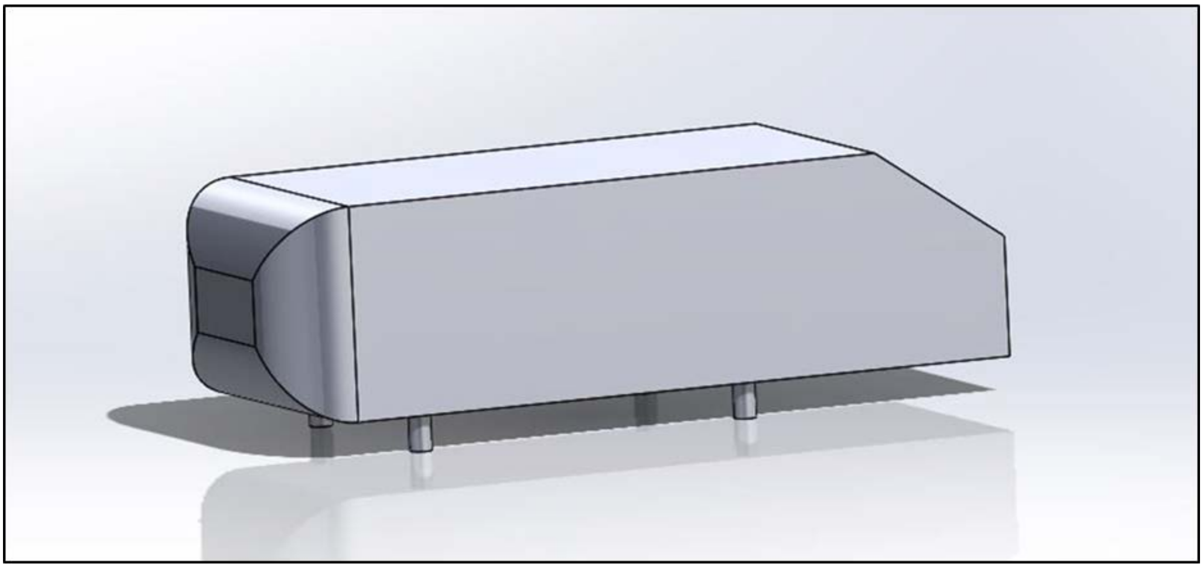


Figure 2-14: Ahmed body (Colby Mazzuca, 2017)

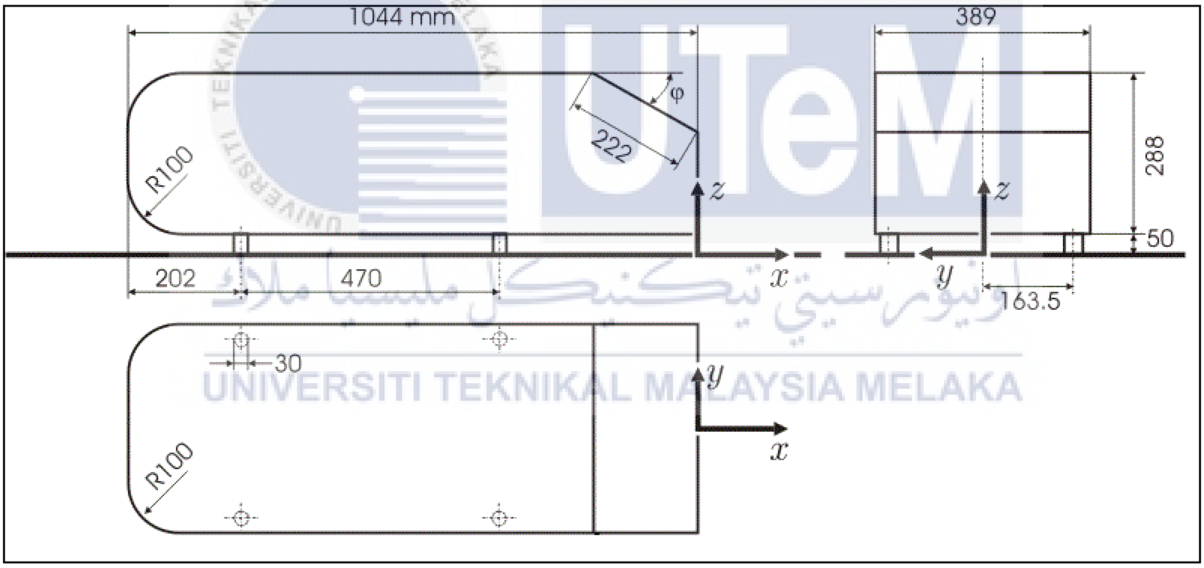


Figure 2-15: Ahmed body (Hinterberger & Rodi, 2004)

In this study, the Ahmed body is used to verified the CFD Ansys-Fluent technique in obtaining the result especially the drag and lift coefficient.

2.2.4 Aerodynamic Device - Rear Spoilers

The rear spoiler is a device that is added to the rear of the car to improve the car's aerodynamic and stability by adding more pressure on the behind of the car. There is two type of rear spoiler design for a car. They are strips and free-standing wing spoiler (Cheng & Mansor, 2017) as shown in Figure 2-16 and 2-17. The strips design is commonly used in both sedan and hatchback car. While the free–standing wing spoiler commonly used for a sedan car as it requires a sufficient area to be attached to a car. The addition of the rear spoiler could affect the lap time for a race car. Besides, the stability of road car could have a big difference with a rear spoiler.

Figure 2-16: Free-standing wing spoiler (Michael Grayen, 2016)

Figure 2-17: Strips spoiler (Michael Grayen, 2016)

The rear spoilers are devices that increase the stability of the car, reduce the drag and regulate the pressure difference resulting in the better performance of the car. The spoilers constitute of the front and the rear spoilers. However, the rear spoilers contribute to a major aerodynamic stability of the car (Hu & Wong, 2011). (Bansal & Sharma, 2014) states that the effect of adding a tail plate on a passenger car are reduction of 3.87% in drag coefficient and 16.62% in lift coefficient. The aerodynamic devices – rear spoilers also act as a diffuser.

Rear spoilers provide the following advantages (Laha & Ramezanpour, 2AD):

- Increases the tires performance and capability to produce required forces
- Offering stability at a very high speed
- Better traction therefore generating fuel efficiency
- Improves braking performance
- Better cornering stability

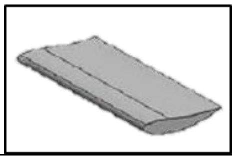
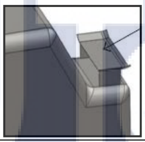
2.2.5 Dimensional Analysis and Similitude

(Hassan, n.d.) states that similitude is method of predicting prototype conditions from the observation of a smaller scale model. Similitude involves the use of the dimensionless parameters obtained in the dimensional analysis. Generally, there are two approaches that can be used. The first and common method is the Buckingham π-theorem. This method contains a series of steps in ensuring the dimensional homogeneity. The second method is by extracting the dimensionless parameters that affecting a particular flow situation from the differential equations and boundary conditions that are required to describe the phenomenon being investigated. In aerodynamic testing of a vehicle, testing using an actual size of the real vehicle requires high amount of time and energy consumption. This including the simulation and experimental method of testing. Usually in experimental testing, it is required to deal with a quite large size of object. The example of these object may be a truck, submarine, and may be a building. Large capacity experiment will require high cost. Besides, high energy consumption also is required. Undoubtedly, it also demands a larger space. Furthermore, simulation testing using a CFD software on a complex and large scale of object also would affect the processing time. Due to that reason, dimensional analysis and similitude technique is used to counter the barrier and limitations.

2.2.6 Comparison between Literature Review

The table below shows the comparison between the results of the previous study and research.

Table 2-1: Comparison of literature review

Title	Design	% Reduction		Remark
		C_D	C_L	
A Numerical Study on Rear-spoiler Of Passenger Vehicle (2017) Xu-xia Hu, Eric T.T. Wong		1.74	3.90	Sedan
Influence of rear-roof spoiler on the aerodynamic performance of hatchback vehicle (2016) S. Y. Cheng, M. Shuhaimi		3	488	Ahmed Body
Drag Reduction of Passenger Car Using Add-On Devices (2014) Ram Bansal and R. B. Sharma		2.02	6	Sedan
		3.87	16.62	Sedan
CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction (2013) R. B. Sharma, Ram Bansal		3.87	16.62	Sedan
A Numerical Study on Rear-spoiler Of Passenger Vehicle (2012) Mustafa Cakir		17.24	7.66	Sedan
		6.47	20.72	Sedan

The comparison above were done to ensure that the attachment of the rear spoiler could give a better improvement to the car. Besides, it is also to ensure the uniqueness of my research and not repeating the previous research and study. Lastly, the purpose of this comparison to ensure that this research result could be better than the previous research.

CHAPTER 3

3 METHODOLOGY

This section contains a description and justification of the methodology used to achieve the stated objective of the study. This chapter includes on the way of this research work is carried out in detail in designing new spoiler. The action plans of conducting this study and the reason and rationale for the application of every procedures and technique used to identify, select, process and analyses information applied in understanding the problem are described. The flowchart below simplifies the flow of the research work.

3.1 Flowchart

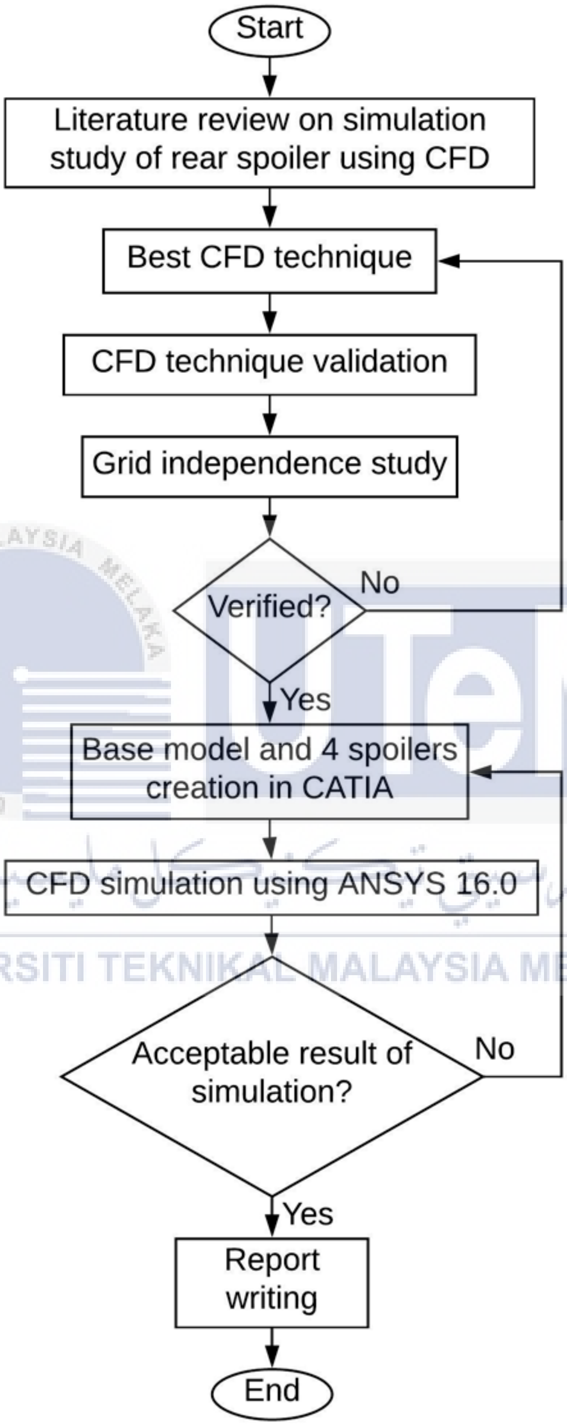


Figure 3-1: Flow chart

3.1.1 Literature review

Admittedly, in the study of aerodynamics of a car, there are uncountable numbers of research had been done on this topic. Yet, the researches that had been done may be differs in a few parameters and objective of the study. Due to that reason, all articles, journals, books, and trusted websites were reviewed in order to find and determine the most suitable and best way of conducting this study. Some of the study conducted also emphasized on the best method in conducting research and study related to the field aerodynamic of a car, especially in aerodynamic device such as spoiler. This act of reviewing the study and research done by others is to ensure that the method taken in this study doesn't contrary and opposing the common and traditional method.

3.1.2 Draft paper

In literature review section, there are many journal and article that had been studied and reviewed. The best research paper that is relate closest to the objective of this research was selected. The journal or article were then studied and modified by changing the parameters and conditions to suit the objective of this research. The purposes of conducting this draft paper were to ensure that there are no important data and information that should be included in a research of aerodynamics is lost or forgotten.

3.1.3 Best CFD technique

This research focused on the uses of CFD solver ANSYS-Fluent® software as a medium to study the air flow around the car body. Since ANSYS-Fluent® using the numerical method to solve the governing equation which related to fluid flow around car body, it is chosen as the main platform in aerodynamic study in this research.

In this section, there are various tutorials and related journals on the best way of conducting CFD simulation such as (Lanfrit, 2005) were studied and reviewed. This is done to find the best and precise method in obtaining a reliable result. The resources of references in this section may come from various video tutorial and guide from Ansys academic support website itself.

3.1.4 Verification of CFD technique

In previous section, in the pursuit of the best CFD technique in testing the aerodynamic performance, various tutorial and guide were reviewed. Next, the chosen method was tested by doing a simulation on Ahmed body (a simplified car model). Ahmed body was used to verify the CFD technique because it is the benchmark in verification of CFD technique in aerodynamic study. Besides, it is a simplified model of a car in a small scale. Thus, verification of various CFD technique using Ahmed body could save time instead of using a more complex and large car model. Then, the results obtained were compared with the experimental data from previous study done by other researchers. Once the result is in an acceptable range of the experimental results, the next step in the methodology could be continued. However, if the results do not satisfy the range of acceptable results, the CFD method and technique should be reviewed again until the results are acceptable and verified.

Besides, a grid convergence study is also conducted to test two parameters. The parameters including the mesh resolution and the turbulence model used. In order to examine the mesh resolution, there are three benchmarks simulated. The benchmark simulation for mesh resolution study are as follows:

- Benchmark #1 : Base model with coarse mesh resolution (81,470 cells)
- Benchmark #2 : Base model with medium mesh resolution (91,397 cells)
- Benchmark #3 : Base model with fine mesh resolution (1,830,143 cells)

There are many turbulence models could be utilized in ANSYS-Fluent®. In order to choose the best model that suits the research conducted, a grid convergence study on the turbulence model uncertainties is conducted. The benchmark simulations for model uncertainties study are as follows:

- Benchmark #4 : Base model (Realizable k-ε)
- Benchmark #5 : Base model (SST k-ω)
- Benchmark #6 : Base model (Spalart-Allmaras)

3.1.5 Base model and four spoiler designs creation in CATIA

In this section, the generic sedan passenger car as a model is created. The dimension of the base line is created based on the Ahmed body geometry in order to ensure the relevance of the geometry while saving the processing time. The dimension of the base model is shown in Figure 3-2 while the Ahmed body dimension is shown in Figure 3-3.

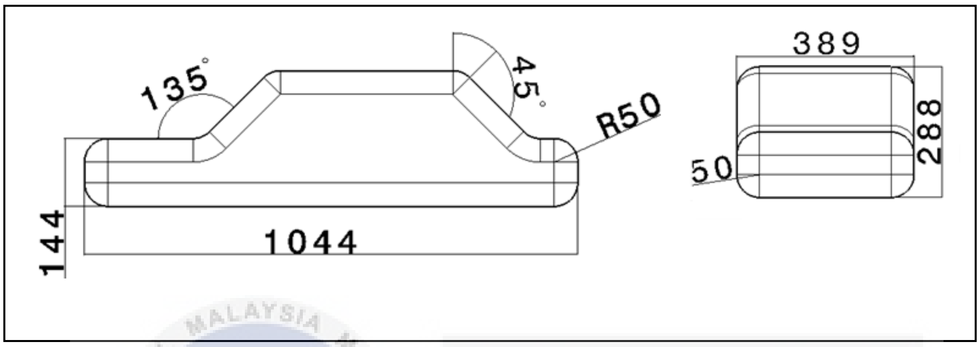


Figure 3-2: Dimension of the base model

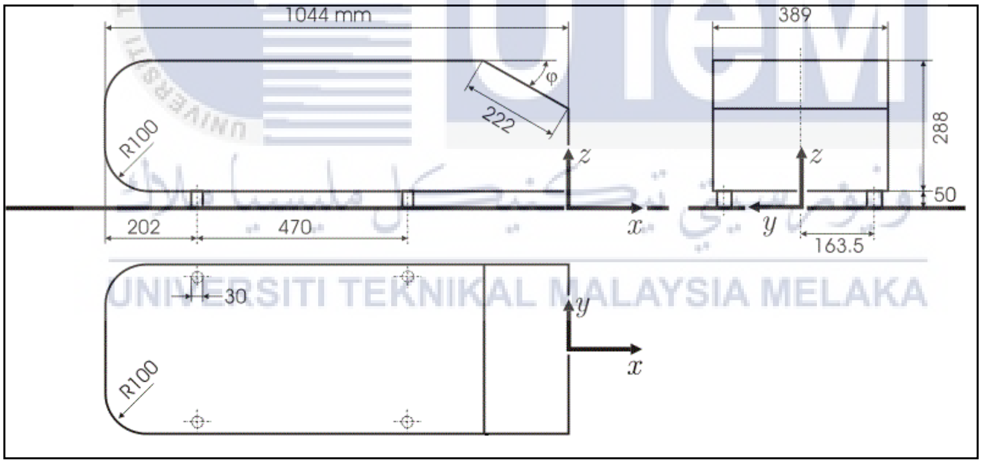


Figure 3-3: Ahmed body dimension

Besides, there are expected four designs of rear-wing spoilers going to be produced. The parameters and guides in designing the new spoilers were based on the literature review done. The aim of this section is to design a best spoiler comparing with a present spoiler in the market and study done. The CAD software that were used in designing the spoilers were CATIA®. CATIA® was chosen to be the platform of drawing the spoilers as it is the most familiar software used by mechanical engineering students in UTeM. The created spoilers' dimensions are shown in Figure 3-4, Figure 3-5, Figure 3-6 and Figure 3-7.

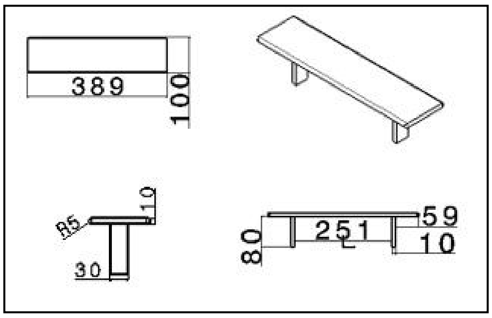


Figure 3-4: Generic model and dimension of first spoiler

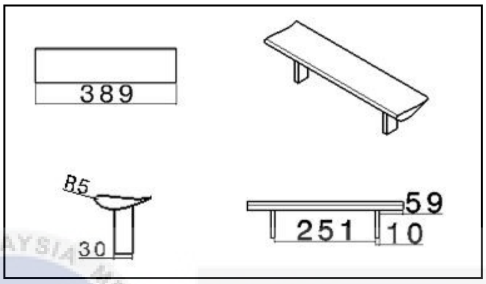


Figure 3-5: Generic model and dimension of second spoiler

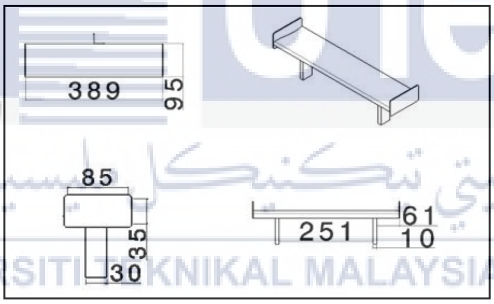


Figure 3-6: Generic model and dimension of third spoiler

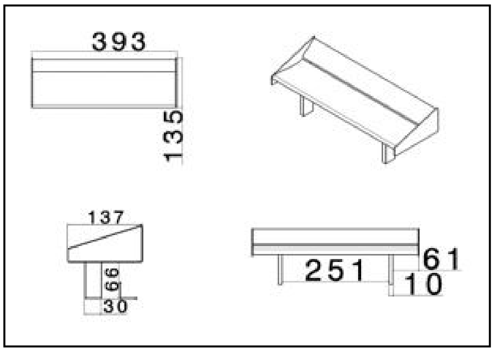


Figure 3-7: Generic model and dimension of forth spoiler

3.1.6 CFD simulation using ANSYS-Fluent 16.0

In this section, the analysis was first conducted on the base model of the car. Once the analysis on the car results had been obtained and validated, the analysis was then done to the base model with addition of each design of the rear-wing spoilers. The results of each analysis were then analyzed and compared to each other.

3.1.6.1 Pre-Processing

Pre-processing basically is an initial process before the analysis could be run on the model.

3.1.6.1.1 Model Clean-Up and Preparation

Generally, the model of the car geometries is always close to the real or actual dimension and complexity. The detail including the shape of every component of the car such as headlamps, side mirrors and others. However, the complexity of the car geometry could be harder to be analyzed and solved by the software. In order to solve the limitation, geometry clean-up using the repair tools could be utilized. Same goes with the designed spoilers in CATIA®, each of them had to be designed to obey ANSYS-Fluent® geometries constraint in order to reduce alternation and modification that had to be done in the ANSYS-Fluent®.

3.1.6.1.2 Virtual Wind Tunnel

The virtual wind tunnel could be simulated by using the enclosure tools in the Design Modeler. However, there are guides to be followed to build the enclosure around the car's body. The best practice as stated in (Lanfrit, 2005), it states that length of the enclosure in front of the car model must be at least three times length of the car. While at the back of the car, the enclosure should have a minimum length of 5 car model. The volume and displacement of the car model should not exceed 1-1.5% of the total cross sectional of the enclosure. The enclosure setup done on the model is shown in Figure 3-8. Due to limited computer resources and times, the enclosure built divide the object to half. Means that, the simulation will be calculated for just one side of the car.

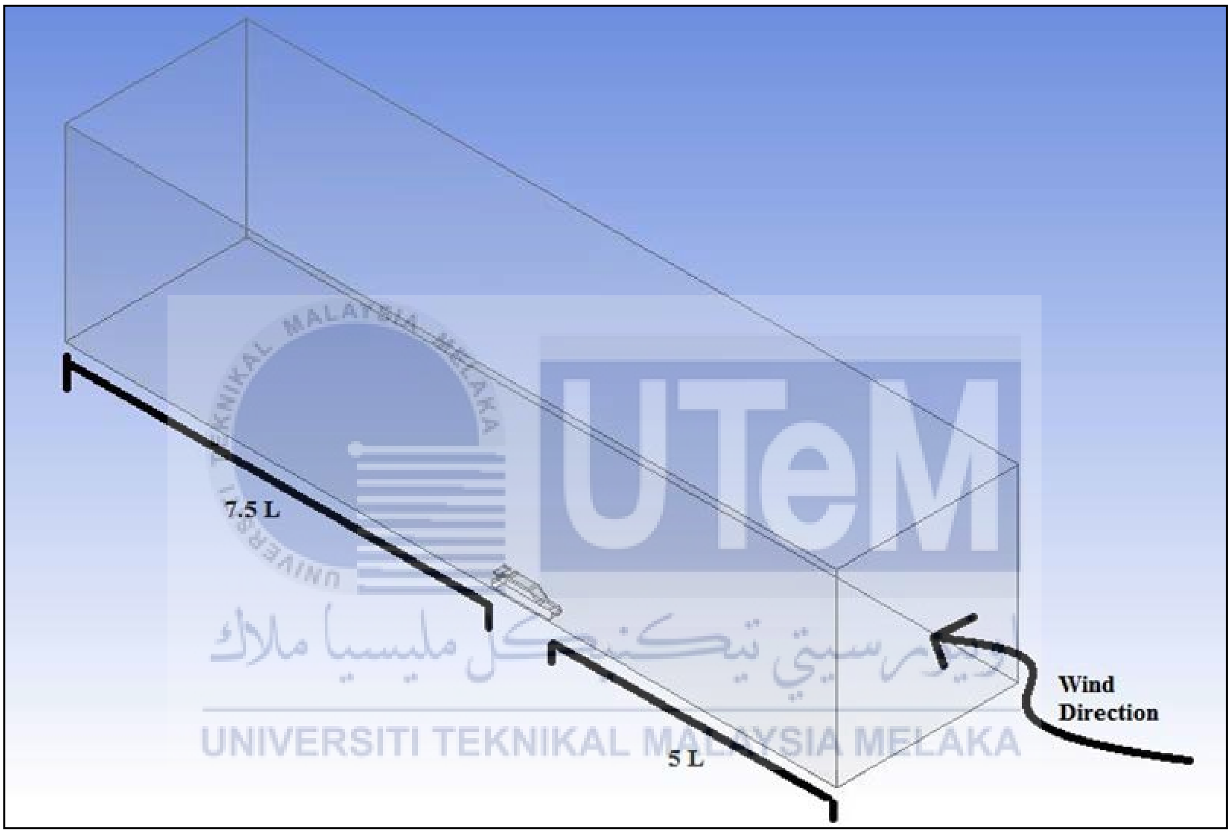


Figure 3-8: Virtual wind tunnel and vehicle orientation

The surfaces of the enclosure or the air domain are renamed so that the numerical solver of ANSYS-Fluent® would recognize the boundary and appropriate boundary condition would be applied automatically. The named selection of the air domain faces is shown in the Figure 3-9.

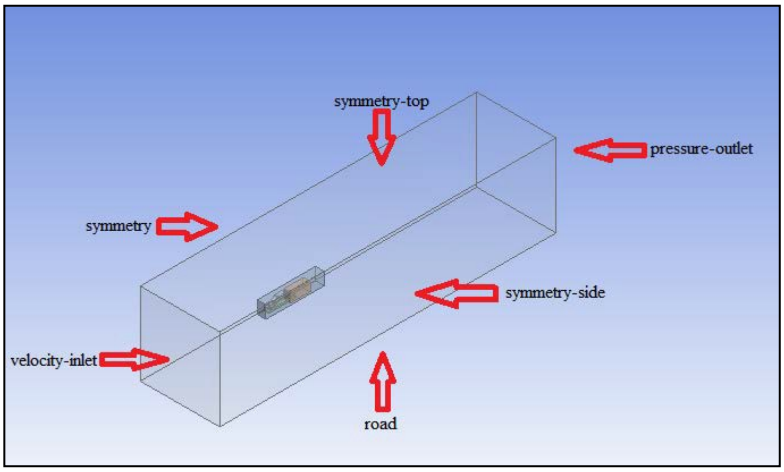


Figure 3-9: A virtual wind tunnel with named boundary

In addition, in order to control the surrounding meshing near the car body, it is a best practice to add a box into the enclosure. Also, the dimension of the box also should adhere and follow the guides given. Based on (Lanfrit, 2005), it states that the box should extend for about half length of the car to the front, sides and top while extending for about a car length to the back. Later in the meshing process, the meshing around the surrounding of the car could be more controllable. However, addition of the box is not a mandatory process. Based on Figure 3-10, there 3 body geometry are created.

- Car box
- Undercar box
- Wake box

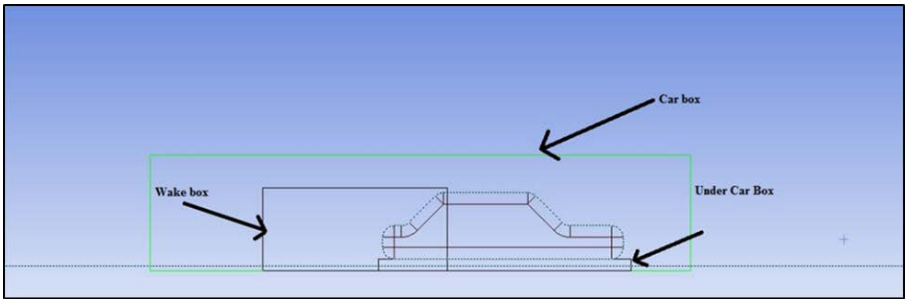


Figure 3-10: Body created for mesh sizing

In this study, there are 11 different situations that will be simulated including 6 benchmarks and 5 cases. The geometrical box created for all those cases in the Ansys Design Modeler are shown in Table 3-1 below.

Table 3-1: Geometrical box created for each simulation

Simulation		Car box	Wake box	Under car box
Benchmark	#1	-	-	-
	#2	-	-	-
	#3	√	√	√
	#4	√	-	-
	#5	√	-	-
	#6	√	-	-
Case	#1	√	√	√
	#2	√	√	√
	#3	√	√	√
	#4	√	√	√
	#5	√	√	√

3.1.6.2 Meshing

In meshing the car model, the strategy of utilizing the internal boxes is the most common strategy used in previous researches and studies such as conducted by (Cakir, 2012). This strategy utilized the boxes that is created in the pre-processing tools. Furthermore, this strategy is common as it can combine different meshing technique in a single model, means that finer meshing could be used in the surrounding near the car's body while coarser meshing is used in the further surrounding. This could save more time comparing to setting all the meshing resolution to fine size. There are other meshing strategies such as using adaption function. However, this method is not the best method due to several limitations. Based on the meshing done in Figure 3-11, there 3 body sizing are created.

- Car box : The meshing size is limited at 15mm
- Undercar box : The meshing size is limited at 10mm
- Wake box : The meshing size is limited at 10mm

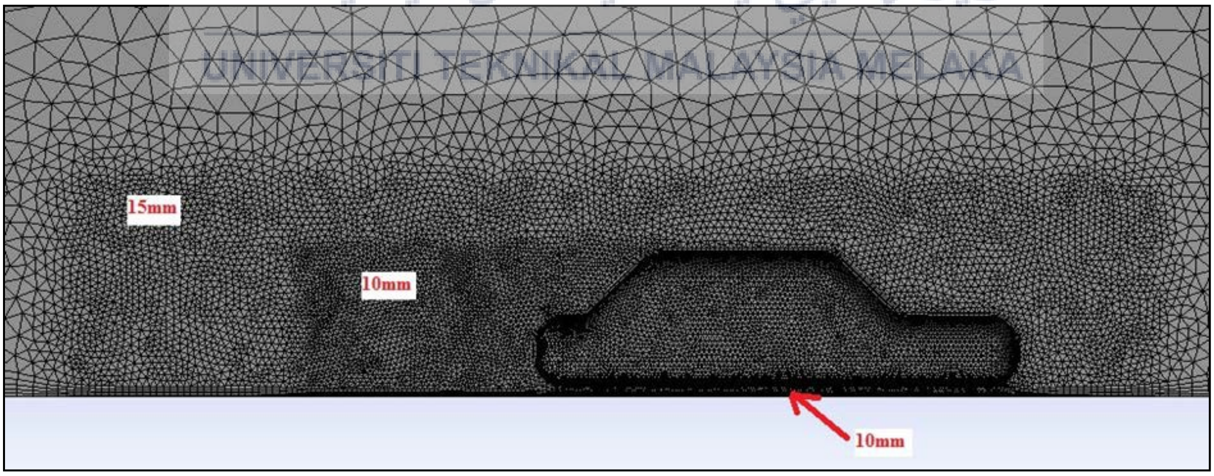


Figure 3-11: Body sizing generated

The mesh generated is still incomplete as no inflation layer around the vehicle body. The inflation layer is important in this study as the airflow involve in turbulent flow. The purpose of the inflation layer is to capture the boundary layer form on the surface of the car and road surface where no-slip condition was set. According to (Lanfrit, 2005), the inflation is enabled using program-controlled inflation. The result of the inflation layer generated is shown in Figure 3-12.

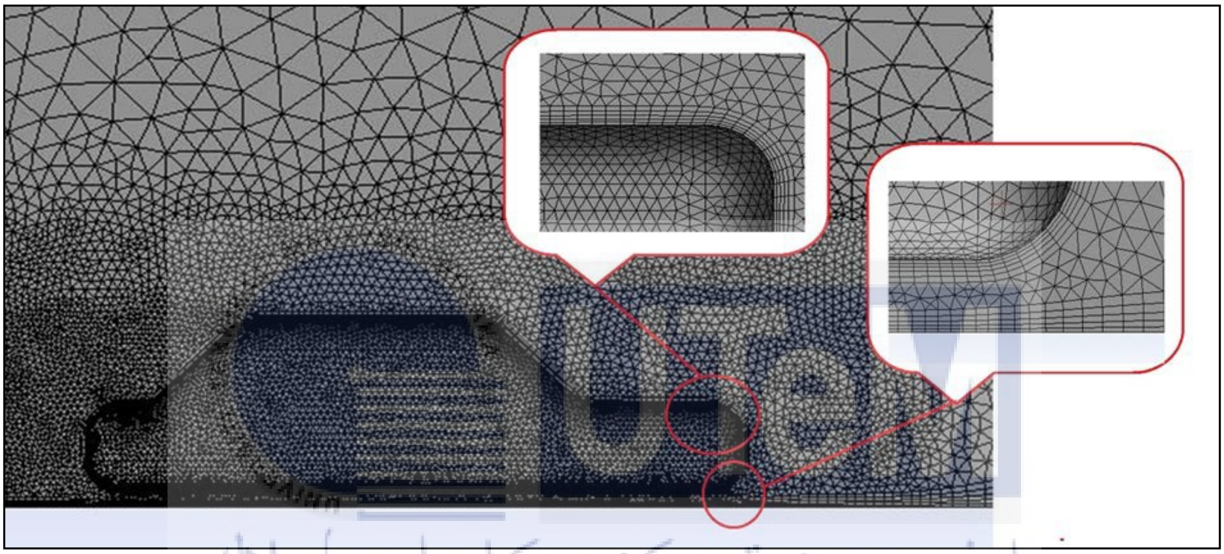


Figure 3-12: Inflation layer on car and road surfaces

In this study, there are total 11 different situations that will be simulated including 6 benchmarks and 5 cases. The meshing setup for all those cases are shown in Table 3-2 below.

Table 3-2: Mesh sizing parameters

	Case					Benchmark					
	#5	#4	#3	#2	#1	#6	#5	#4	#3	#2	#1
Global Mesh Sizing Settings											
Use adv. size fun.	On: Proximity and curvature									Default	Default
Relevance center	Coarse										
Initial size speed	Active assembly										
Smoothing	High										
Transition	Slow										
Span angle center	Fine										
Curvature nor. angle	12.0										
Min. size	1mm										
Max. size	250mm										
Growth rate	1.2 (20%)										
Inflation											
Use automatic inflation	Program Controlled										Default
Inflation option	First Aspect Ratio										
First aspect ratio	5										
Maximum layers	5										
Growth rate	1.2 (20%)										

3.1.6.3 Fluent Setup

This section is where the properties, boundary condition, material and others are set before the calculation could be run.

3.1.6.3.1 Properties

The properties of the materials were set as following (Hinterberger & Rodi, 2004):

Fluid : Air

Density : 1.225 kg/m^3

Viscosity : 1.7894e-05 kg/ms

Solid : Aluminium

Density : 2719 kg/m^3

3.1.7.2 Boundary Conditions

The setup for boundary condition is shown in Table 3-3.

Table 3-3: Boundary Physics

Boundary	Type	Remark
Car	Wall	-
Pressure-outlet	Pressure-outlet	-
Road	Wall	-
Symmetry	Symmetry	-
Symmetry-side	Symmetry	-
Symmetry-top	Symmetry	-
Velocity-inlet	Velocity-inlet	V_{air} = 40m/s

3.1.7.3 Turbulence Modelling

For an external aerodynamic study, ANSYS-Fluent recommended the users to use the Realizable k-epsilon turbulence model. (Lanfrit, 2005) states that based on industrial applications of this model, it is possible to achieve good results in terms of integral values such as drag coefficient which in range of 2-5%. In addition, it is also very stable and fast converging while allowing a large number of calculations in a small-time frame. However, in this research, the turbulence model of k-ω and Spalart-Allmaras are also tested in the grid independence study. The turbulence model used for each simulation is shown in Table 3-4.

Table 3-4: Turbulence model for each simulation of benchmark

<table>
<tr><th>Simulation</th><th>No.</th><th>Turbulence model</th><th>Simulation</th><th>No.</th><th>Turbulence model</th></tr>
<tr><td rowspan="6">Benchmark</td><td>1</td><td>SST k-omega</td><td rowspan="6">Case</td><td>1</td><td rowspan="6">Realizable k-epsilon</td></tr>
<tr><td>2</td><td>Spalart-Allmaras</td><td>2</td></tr>
<tr><td>3</td><td>Realizable k-epsilon</td><td>3</td></tr>
<tr><td>4</td><td rowspan="3">Realizable k-epsilon</td><td>4</td></tr>
<tr><td>5</td><td>5</td></tr>
<tr><td>6</td><td>6</td></tr>
</table>

3.1.8 Post-Processor

ANSYS-Fluent allow the users to view the result of the simulation in more graphical after the calculations finished. The graphical results including (Cakir, 2012):

1. Domain geometry and grid display
2. Vector plots
3. Line and shaded contour plots
4. 2D and 3D surface plots
5. Particle tracking
6. XY plots and graph of results

CHAPTER 4

4 RESULT AND DISCUSSION

4.1 Simulation Results of Case #1, Case #2, Case #3, Case #4 and Case #5

In this section, the computational results of the following cases are presented and discussed.

Case 1: results for the base model

Case 2: results for the base model with attached spoiler 1

Case 3: results for the base model with attached spoiler 2

Case 4: results for the base model with attached spoiler 3

Case 5: results for the base model with attached spoiler 4

Every benchmarks and cases are tested and simulated in ANSYS-Fluent® to obtain graphical results that show the aerodynamic physical properties that affect the model. The colored figures define the aerodynamic physical properties at an instant position. On the first half of this chapter, five cases including the base model and four attached spoilers are discussed while on the other half of the chapter, six benchmarks that are used in the grid independence study are discussed.

4.1.1 Convergence history

In this study, the convergence criteria were having default residual values that is below $1x10^{-3}$. The residuals convergence plot history for all five cases are given in Figure 4-1, Figure 4-2, Figure 4-3, Figure 4-4 and Figure 4-5.

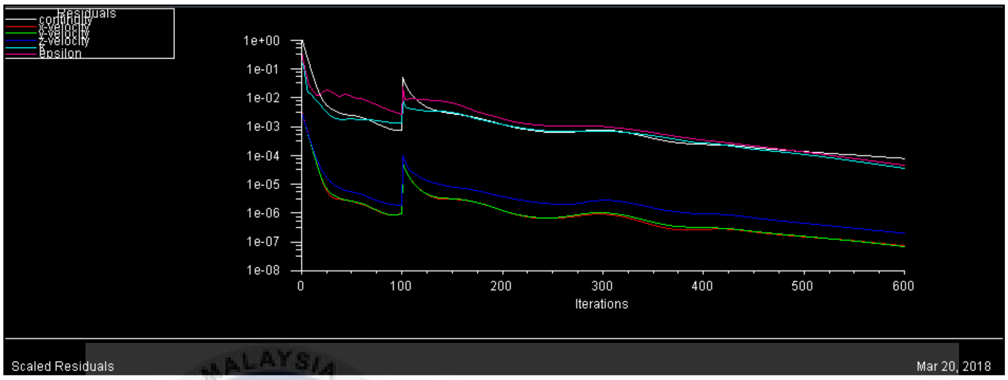


Figure 4-1: The scaled residuals convergence history for case #1

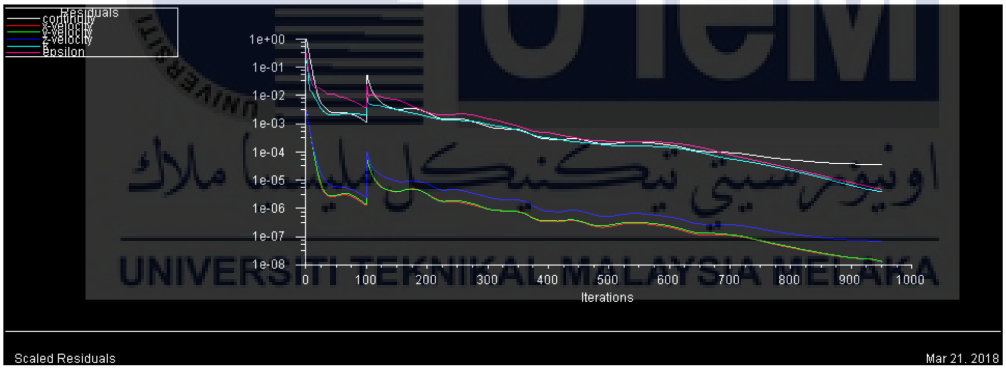


Figure 4-2: The scaled residuals convergence history for case #2

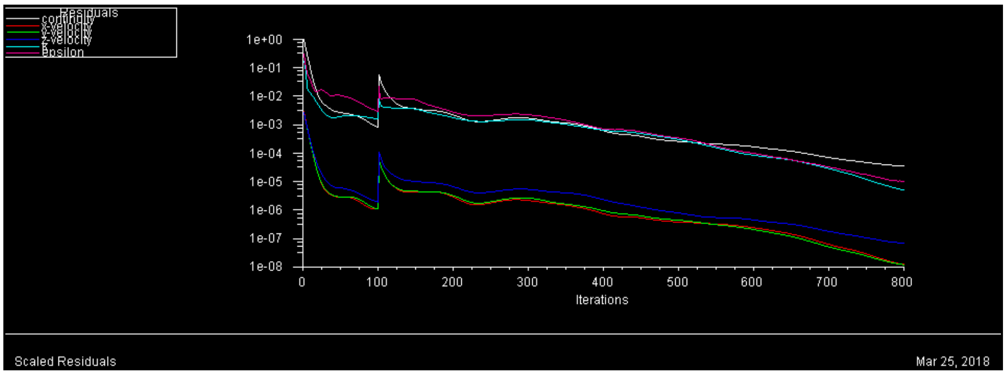


Figure 4-3: The scaled residuals convergence history for case #3

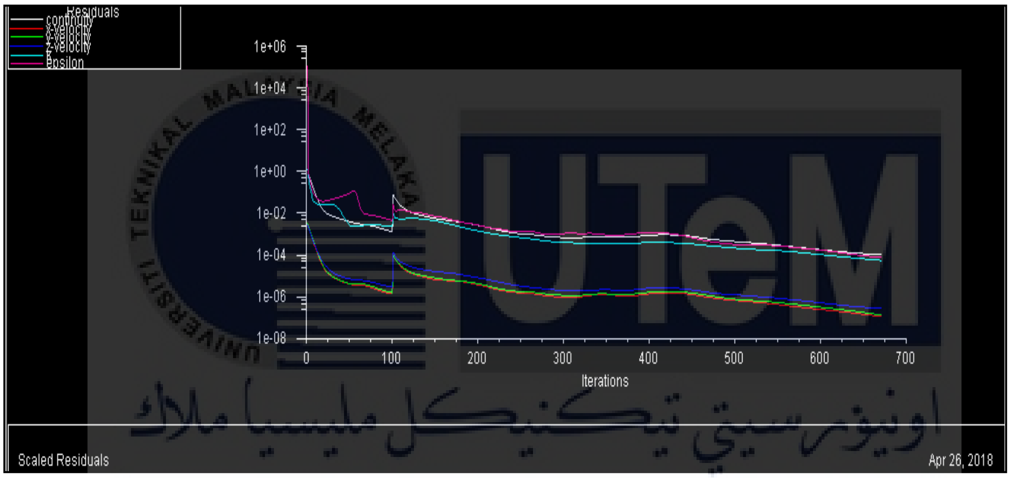


Figure 4-4: The scaled residuals convergence history for case #4

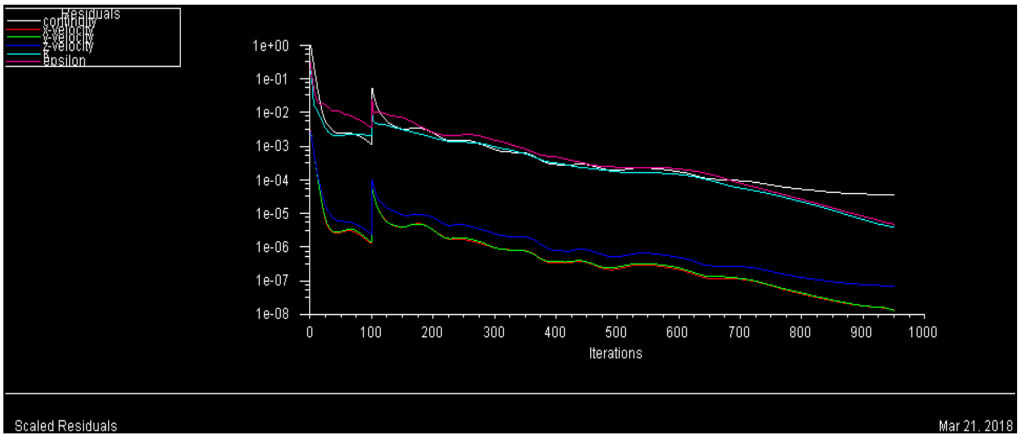


Figure 4-5: The scaled residuals convergence history for case #5

As shown in the Figure 4-1, Figure 4-2, Figure 4-3, Figure 4-4 and Figure 4-5, the residuals graphs are jumped off at 101st iteration. The phenomena are the effect of the scheme changing from the first order upwind to second order upwind in order to converge faster.

4.1.2 Colour contour analysis

The pressure distribution analysis on the surface of the vehicle is done using Bernoulli's equation. The contour of static pressure distribution shown in Figure 4-6, Figure 4-7. Figure 4-8, Figure 4-9, Figure 4-10 indicates the level of pressure effect on the surface of the car. Observation on the contour of the pressure distribution around the base model in Figure 4-6 obviously shows that there are two areas with positive pressure. The areas are at the front of the car's body and the other one is at the area between the front hood and the windshield. On the other hand, there are three areas with negative pressure can be found on the generic car model. The areas are at the front and rear end of the roof and at a small area of the front hood. However, when the spoilers are attached to the car, the areas of positive pressure increase at the spoiler area. This improves the lift coefficient of the car as more pressure on the upper part of the car. This result on higher negative lift coefficient of the car.

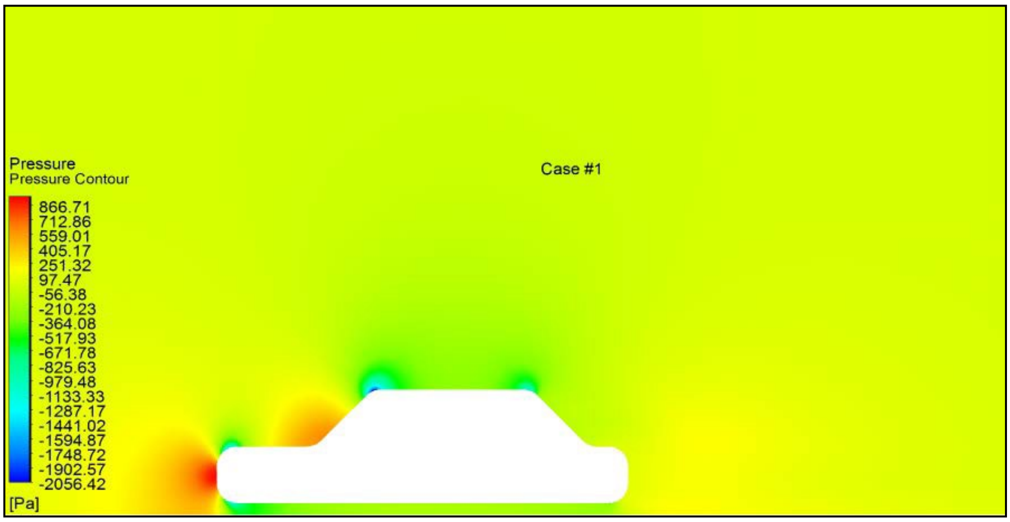


Figure 4-6: Distribution of pressure on case #1 (without spoiler)

Figure 4-7: Distribution of pressure on case #2

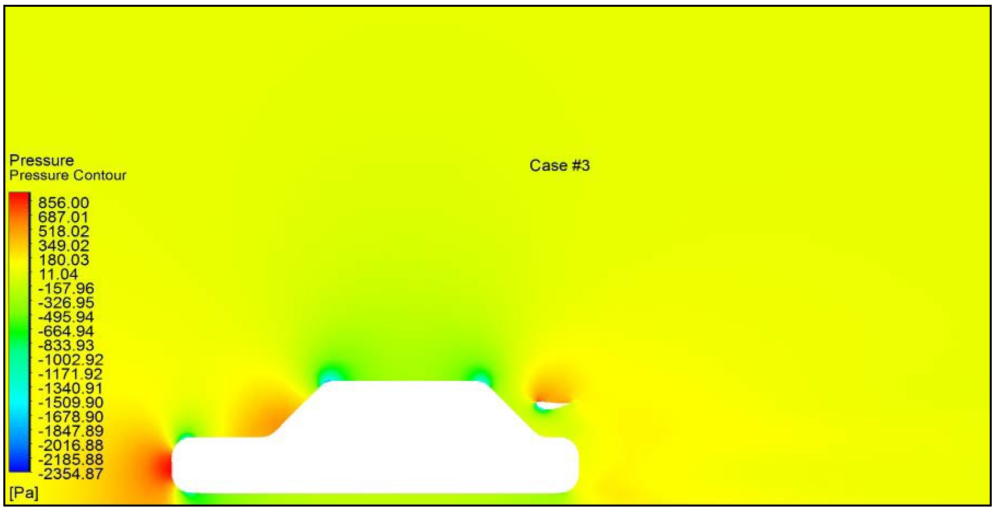


Figure 4-8: Distribution of pressure on case #3

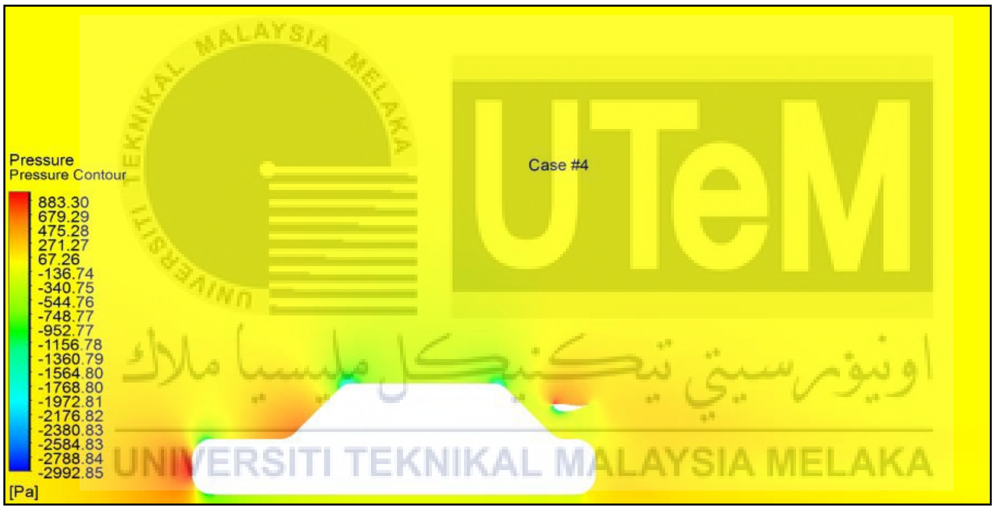


Figure 4-9: Distribution of pressure on case #4

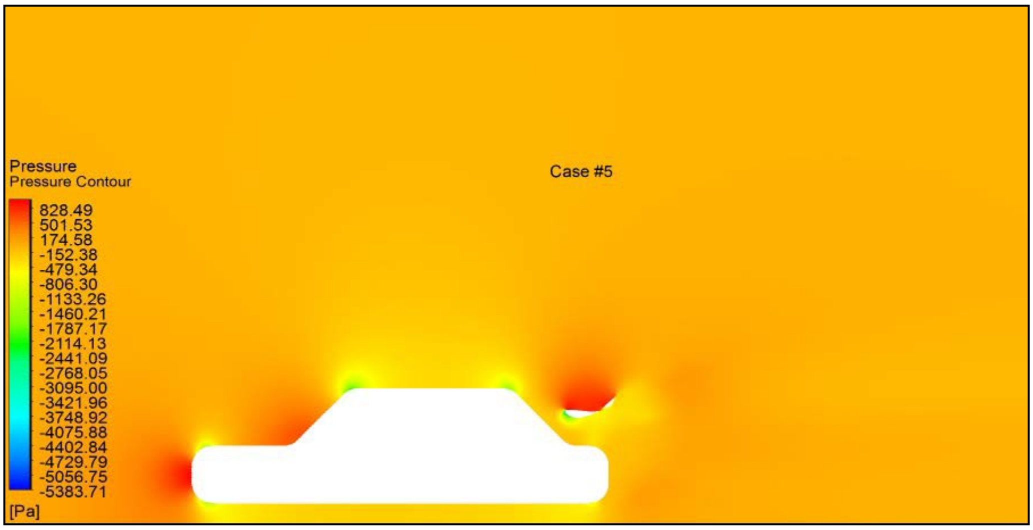


Figure 4-10: Distribution of pressure on case #5

The concept of pressure drags happen when there are two different areas with different level of pressure. In this case, the area on the bottom of the car's body is relatively higher than the pressure level at the top of the car. The factor of the pressure difference could contribute on the high lift coefficient of the car. The high lift coefficient could cause pitching moment on the rear of the car to lift up from the ground which will make the rear wheels traction on the road hence causing the car to lost control. In the simulation of the base model of the sedan car, the final predicted drag and lift coefficient are 0.19230 and 0.03957.

The airflow over the car body creates a velocity distribution which resulting in the aerodynamic loads acting on the body of the generic model. The velocity distribution indicates the amount of velocity at different portions of the vehicle. In addition, the purpose of the rear spoiler is to spoil the velocity direction to reduce the lift force on the car thus improving the aerodynamic performance of the car. Moreover, the velocity value is directly proportional to the value of the drag and lift coefficient. Thus, the velocity distribution around the car gives a big impact on the aerodynamic performance of the car. The airflow separation shown in Figure 4-11, Figure 4-12, Figure 4-13, Figure 4-14 and Figure 4-15 causing the airflow to become turbulent, hence lower pressure region formed at the back of the car. This will cause a negative effect on the car aerodynamic performance and stability. The addition of the spoiler as shown in Figure 4-12, Figure 4-13, Figure 4-14 and Figure 4-15 could make the air “see” becomes longer, smoother flow from the roof to the spoiler, that helps to delay the airflow separation. Thus, the aerodynamic performance of the car could be better.

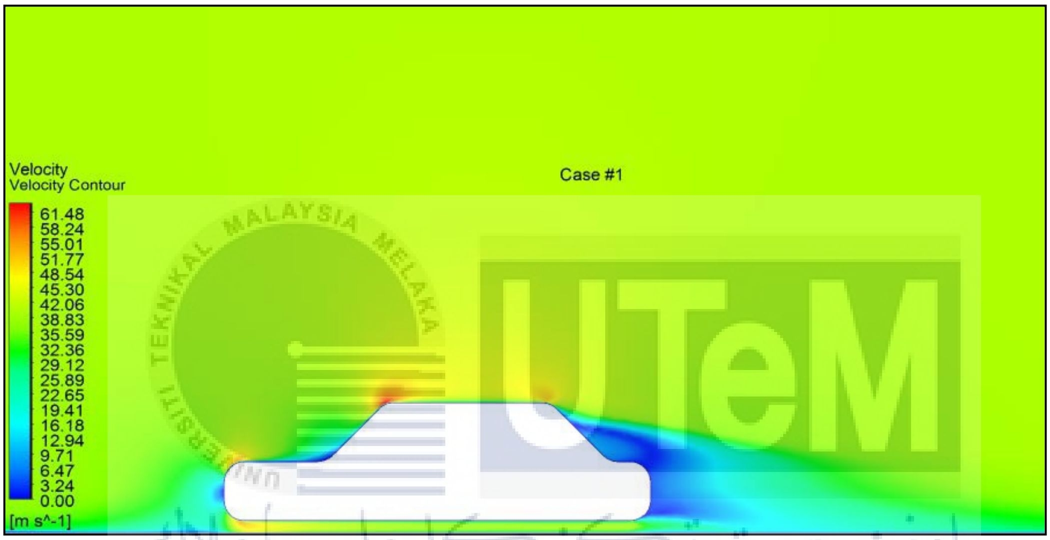


Figure 4-11: Distribution of velocity on case #1 (without spoiler)

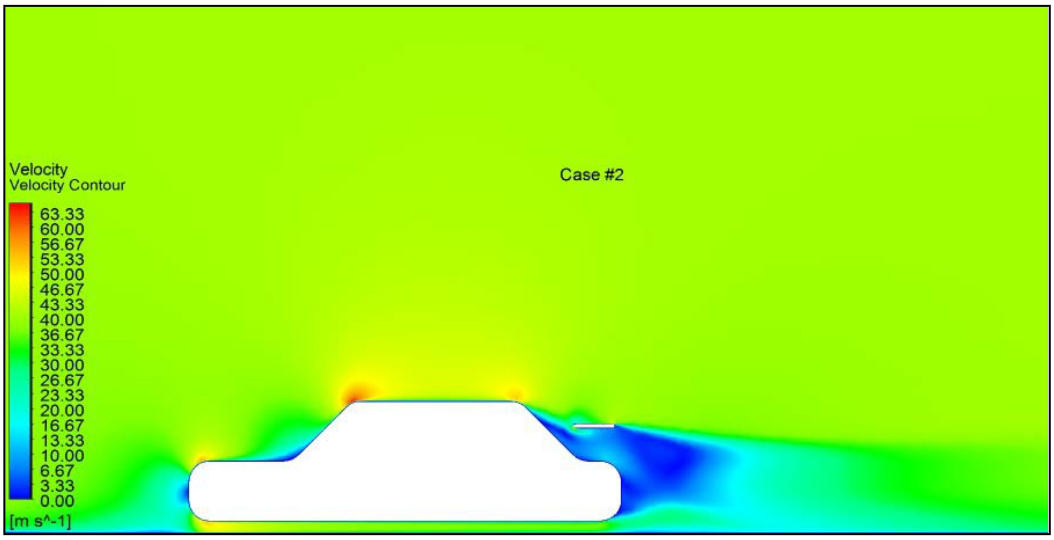


Figure 4-12: Distribution of velocity on case #2

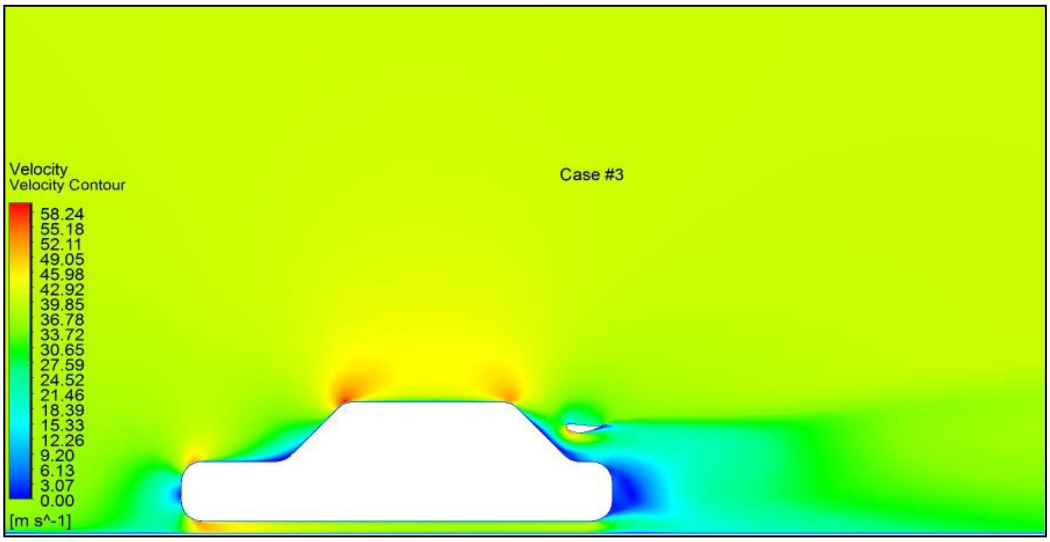


Figure 4-13: Distribution of velocity on case #3

Figure 4-14: Distribution of velocity on case #4

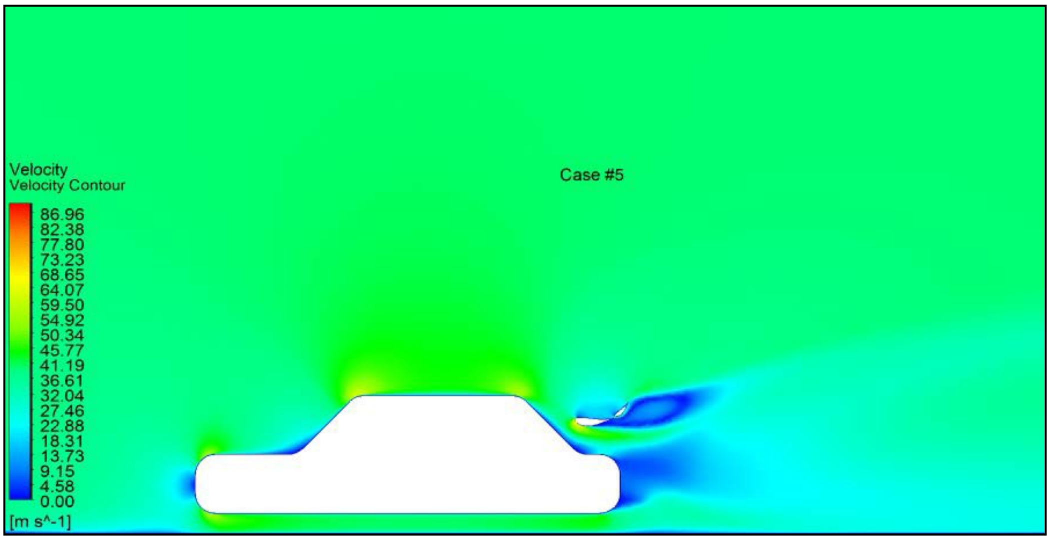


Figure 4-15: Distribution of velocity on case #5

4.1.3 Comparison of rear-spoiler design

In this section, the computational results of the following cases are presented and discussed.

Case 1: results for the base model

Case 2: results for the base model with attached spoiler 1

Case 3: results for the base model with attached spoiler 2

Case 4: results for the base model with attached spoiler 3

Case 5: results for the base model with attached spoiler 4

All the results are obtained with the same meshing resolution and setup, the standard realizable k-epsilon model and also the same boundary conditions. Drag and lift coefficient for all the three cases are presented in Table 2. The reduction rate of the drag and lift coefficients did not depend on the mesh resolution, turbulence model and air domain size.

Table 4-1: Drag and lift coefficient percentage change on different design of rear spoiler

Simulation	C_D	Percentage change	C_L	Percentage change
Case 1	0.19230	-	0.03957	-
Case 2	0.24361	+26.68%	0.03902	-1.39%
Case 3	0.28283	+47.07%	-0.13464	-440%
Case 4	0.28307	+47.20%	-0.16254	-510.77%
Case 5	0.48246	+150.89%	-0.74901	-1992.87%

4.2 Simulation Results of Benchmark #1, Benchmark #2 and Benchmark #3; Examine Grid Convergence

In this section, the computational results of the following benchmarks are presented and discussed.

Benchmark 1: results for the base model with coarse meshing

Benchmark 2: results for the base model with medium meshing

Benchmark 3: results for the base model with fine meshing

Every benchmarks are tested and simulated in ANSYS-Fluent® to obtain graphical results that show the aerodynamic physical properties that affect the model. The colored figures define the aerodynamic physical properties at an instant position.

4.2.1 Convergence history

In this study, the convergence criteria were having default residual values that is below $1x10^{-3}$. The residuals convergence plot history for all three cases are given in Figure 4-16, Figure 4-17 and Figure 4-18.

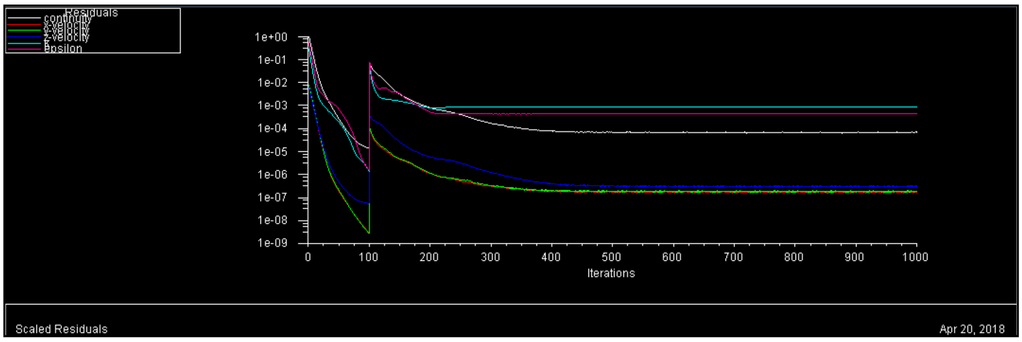


Figure 4-16: The scaled residuals convergence history for benchmark #1

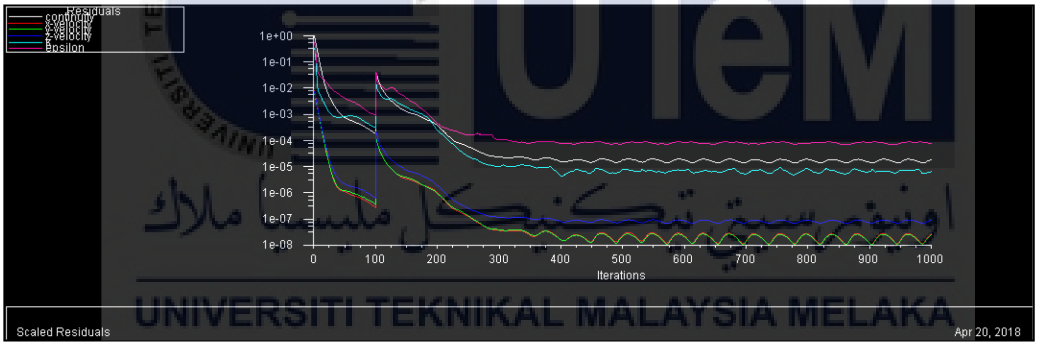


Figure 4-17: The scaled residuals convergence history for benchmark #2

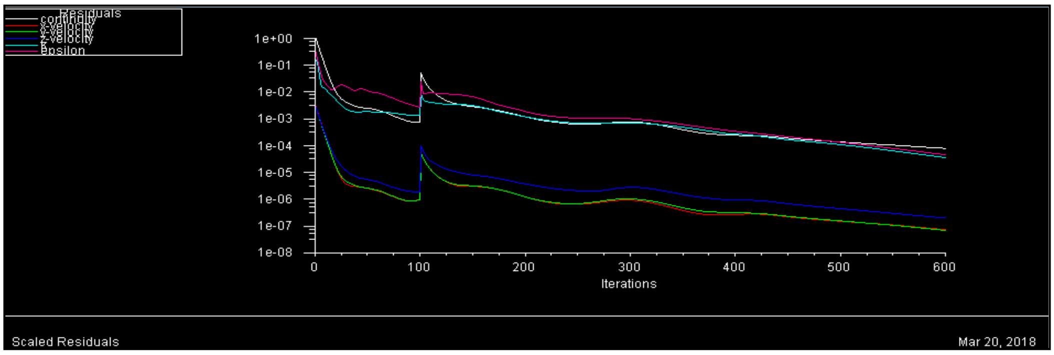


Figure 4-18: The scaled residuals convergence history for benchmark #3

As shown in the Figure 4-16, Figure 4-17 and Figure 4-18, the residuals graphs are jumped off at 101st iteration. The phenomena are the effect of the scheme changing from the first order upwind to second order upwind in order to converge faster. The graphs above clearly shown that the benchmark #1 and benchmark #2 residual graphs refused to converge starting form 300th iterations. In the other hand, the benchmark #3 residual graph shows that it converges at approximately 600th iterations.

4.2.2 Comparison of CD and CL value

In this section, the computational results of the benchmarks #1, benchmark #2 and benchmark #3 are presented and discussed. All the results are obtained with the resolution and setup as shown in Table 3-1, Table 3-2, Table 3-3 and Table 3-4. Drag and lift coefficient for all the three benchmarks are presented in Table 4-2. The change of the drag and lift coefficients mainly depend on the mesh resolution of the model.

Table 4-2: Drag and lift coefficient on different meshing resolution

Model	Mesh	Cell	C_D	C_L
Benchmarks #1	Coarse	81,470	0.18885	-0.27512
Benchmarks #2	Medium	91,397	0.27776	0.14078
Benchmarks #3	Fine + volume box	1,830,143	0.19230	0.03957

4.3 Simulation Results of benchmark #4, benchmark #5 and benchmark #6; Examine model uncertainties

In this section, the uncertainty in the turbulence models are examined by simulating three simulations with three turbulence models. The output of each turbulence models is then examined. The turbulence models used are k-ε, k-ω and Spalart-Allmaras. The base model was used as geometry with a same meshing size for each simulation.

In this section, the computational results of the following benchmarks are presented and discussed.

Benchmark 4: results for the base model (SST k--ω)

Benchmark 5: results for the base model (Spalart-Allmaras)

Benchmark 6: results for the base model (Realizable k-ε)

4.3.1 Convergence history

In this study, the convergence criteria were having default residual values that is below $1x10^{-3}$. The residuals convergence plot history for all five cases are given in Figure 4-19, Figure 4-20 and Figure 4-21.

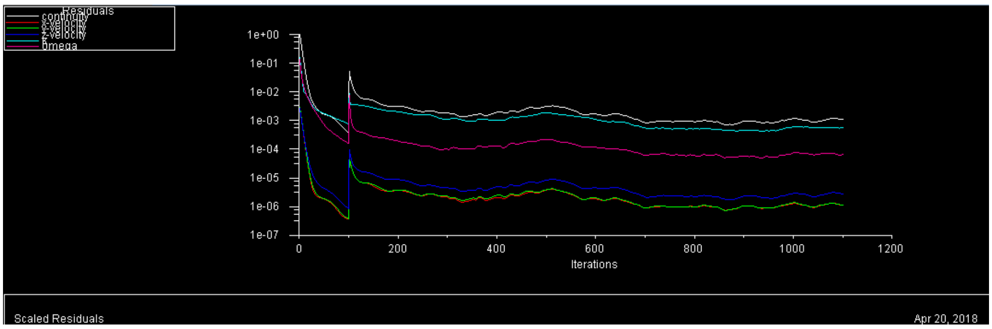


Figure 4-19: The scaled residuals convergence history for benchmark #4

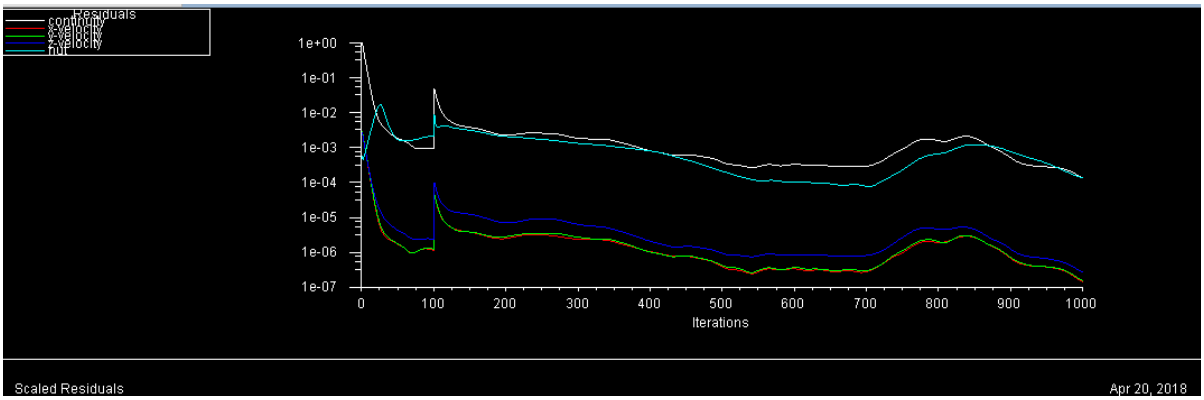


Figure 4-20: The scaled residuals convergence history for benchmark #5

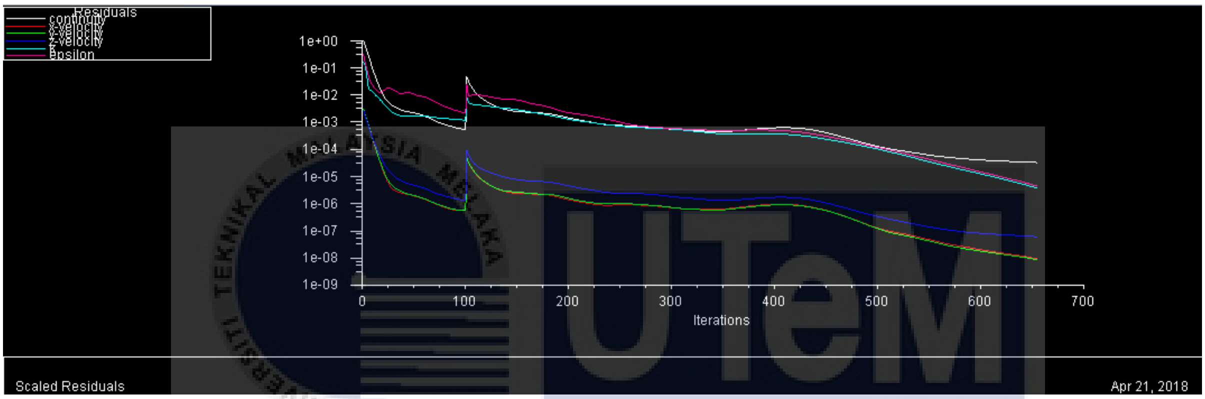


Figure 4-21: The scaled residuals convergence history for benchmark #6

As shown in the Figure 4-19, Figure 4-20 and Figure 4-21, the residuals graphs are jumped off at 101st iteration. The phenomena are the effect of the scheme changing from the first order upwind to second order upwind in order to converge faster. However, the residual graphs of benchmark #4 and benchmark #5 refused to converge. Besides, it is also observed that the time consumption for every iteration in benchmark #4 and benchmark #5 has been increased comparing to the benchmark #6 that used k-ε turbulence model.

4.3.2 Comparison of C_D and C_L value

In this section, the computational results of the benchmarks #4, benchmark #5 and benchmark #6 are presented and discussed. All the results are obtained with the resolution and setup as shown in Table 3-1, Table 3-2, Table 3-3 and Table 3-4. Drag and lift coefficient for all the three benchmarks are presented in Table 4-3. The change of the drag and lift coefficients is not depending on the mesh resolution of the model.

Table 4-3: Drag and lift coefficient on different meshing resolution

Model	Mesh	Cell	Turbulence model	C_D	C_L
Benchmarks #4	Fine	1,421,012	SST k-omega	0.31102	0.20889
Benchmarks #5	Fine	1,421,012	Spalart-Allmaras	0.30864	0.31237
Benchmarks #6	Fine	1,421,012	Realizable k-epsilon	0.24715	0.20701

CHAPTER 5

5 CONCLUSION AND RECOMMENDATION

This chapter will conclude all of the simulation and findings of this study. Recommendation is concluded for further improvement on further work and simulation

5.1 Conclusion

The aerodynamic drag and lift coefficient and flow characteristics of a passenger vehicle with the speed of 40m/s were numerically simulated. Analyzing the benchmark #1, benchmark #2 and benchmark #3 shows that the simulation may results in an inaccurate result as the meshing size is not sufficient enough. It is clearly shown that the higher quality and finer resolution mesh leads to more precise results. The convergence rate also getting better with a finer mesh. Besides, analyzing the benchmark #4, benchmark #5 and benchmark #6, the lack of converged solution and time consuming for completing each iteration, comparing these three benchmarks show that the most suitable turbulence model for external flows around a passenger car body is realizable k-ε.

The results of the simulations with the third spoiler design (case #4, airfoil with end plate) has showed an increment of the drag coefficient from 0.19230 to 0.28307, which is 47.2% of increment. While the lift coefficient value has shown a tremendous decrement from 0.03957 to -0.16254 which is about -510.77% of reduction. By comparing the velocity distribution on the car body, it was found that the recirculation zone above the rear window was almost gone by using spoiler in case 4. The air flow shows a smooth and gentle sloped above the rear window. Higher negative force is more important than having lower drag coefficient in this case as stability and safety of the car is a priority. When comparing the drag and lift coefficient of the second spoiler (case #3, airfoil) with the forth spoiler (case #4, airfoil with endplate), the second spoiler shows a lesser improvement of lift characteristic when compared with the forth spoiler. The addition of the endplate on the airfoil spoiler had prevent the spilling of air from the high pressure to low pressure side of the spoiler and reducing vortices. The endplate forces the airflow to move in one direction than multiple flow thus improving the lift characteristic of the car. Numerical analysis also has showed that the forth spoiler (case #5, double airfoil with end plate) has showed a tremendous improvement from 0.03957 to 0.74901 in the aerodynamic lift characteristic. However, the increment of the drag coefficient on the car from 0.19230 to 0.48246 is unacceptable as it is too much that it will affect the efficiency of the car. This phenomenon may be cause of the big turbulence or recirculation zone at the back of the spoiler.

The best spoiler design chosen in this study is spoiler 3. The results of the simulations with the third spoiler design has showed a little increment of the drag coefficient from 0.19230 to 0.28307. While the lift coefficient value has shown a tremendous and acceptable decrement from 0.03957 to -0.16254. By comparing the velocity distribution on the car body, it was found that the recirculation zone above the rear window was almost gone by using spoiler 3. The air flow separation shows a smooth and gentle sloped above the rear window. The area of the wake is also further at the back of the car. Thus, the drag and lift force act on the car are improved and give better aerodynamic characteristic to the car. The shape of the spoiler 3 that is an airfoil with end plate give the best aerodynamic performance. The end plate helps to channel the airflow towards the back of the spoiler. Without the end plate, the airflow could be much dirtier thus worsen the drag and lift characteristic of the car. Based on Bernoulli’s equation, the slowing moving air exerts higher pressure than faster velocity air. Spoiler 3 has succeeded in decreasing the area of lower velocity air above the rear hood. Hence, the lift coefficient of the car is decreased. Higher negative force is more important than having lower drag coefficient in this case as stability and safety of the car is a priority.

5.2 Recommendation

In this study, a general concept of aerodynamic and effect of spoiler on drag and lift coefficient on a sedan passenger car had been studied. As a result, there are a lot of possible future work that can be conducted by referring to this research project as a base to initiate. In future, in order to further study the effect of rear-wing spoiler, there are few ways that can be undertaken such as:

1. Conduct an experiment by undertaking the same boundary and situation conducted and stated in this study.
2. Conduct a simulation on parametric study by using the same condition to study the effect of the spoiler position, spoiler height and clearance and spoiler's angle of attack.
3. Propose a new rear-wing spoiler design that could give better output on the aerodynamic characteristic of sedan passenger car.

REFERENCES

Adam Lohonyai. (2014). To Engineer is Human: Effects of Winter on Fuel Economy. Retrieved December 12, 2017, from http://alohonyai.blogspot.my/2014/11/effects-of-winter-on-fuel-economy.html

Alex Oagana. (2010). The Origins of Streamline Design in Cars - autoevolution. Retrieved December 12, 2017, from https://www.autoevolution.com/news/the-origins-of-streamline-design-in-cars-2998.html

Bansal, R., & Sharma, R. B. (2014). Drag Reduction of Passenger Car Using Add-On Devices. Journal of Aerodynamics, 2014, 1–13. https://doi.org/10.1155/2014/678518

Cakir, M. (2012). CFD study on aerodynamic effects of a rear wing/ spoiler on a passenger vehicle. Retrieved from http://scholarcommons.scu.edu/mech_mstr

Chainani, A., Perera, & Noel. (2008). CFD investigation of airflow on a model radio control race car, 2–4. Retrieved from http://nrl.northumbria.ac.uk/235/

Chao, K., Moises, V. A., Shandas, R., Elkadi, T., Sahn, D. J., & Weintraub, R. (1992). Influence of the Coanda effect on color Doppler jet area and color encoding. In vitro studies using color Doppler flow mapping. Circulation, 85, 333–341. https://doi.org/10.1161/01.CIR.85.1.333

Cheng, S.-Y., & Mansor, S. (2017). Influence of rear-roof spoiler on the aerodynamic performance of hatchback vehicle, 01027. https://doi.org/10.1051/matecconf/20179001027

Colby Mazzuca. (2017). Ahmed Body - SOLIDWORKS - 3D CAD model - GrabCAD. Retrieved December 12, 2017, from https://grabcad.com/library/ahmed-body-4

David Villarreal. (2012). Kammback y por qué son tan “feos” algunos híbridos - Diariomotor. Retrieved December 12, 2017, from https://www.diariomotor.com/2012/10/31/kammback-y-por-que-son-tan-feos-algunos-hibridos/

Dio Rhapsody. (2016). Aerodynamics car 2 - Pro/Engineer Wildfire - 3D CAD model - GrabCAD. Retrieved December 12, 2017, from https://grabcad.com/library/aerodynamics-car-2-1

Dr Mark Jermy and Lindsey Underwood. (2011). Causes of aerodynamic drag — Science Learning Hub. Retrieved December 12, 2017, from https://www.sciencelearn.org.nz/resources/1346-causes-of-aerodynamic-drag

G, S. K., Mukkamala, Y. S., & Kulkarni, P. S. (2014). To improve the aerodynamic performance of a model hatchback car with the addition of a rear roof spoiler, (August).

Glenn Elert. (2017). Aerodynamic Drag – The Physics Hypertextbook. Retrieved December 13, 2017, from https://physics.info/drag/

Hassan, A. (n.d.). Dimensional Analysis and Similitude. Retrieved from http://www.fkm.utm.my/~abuhasan/content/teaching/skmm2313/skmm2313-ch06-textbook.pdf

Hinterberger, C., & Rodi, W. (2004). Ahmed body, 6–7. https://doi.org/8

Hu, X., & Wong, E. T. T. (2011). A numerical study on rear-spoiler of passenger vehicle. International Journal of Mechanical, Aerospace, Industrial, Mechatronica Dn Manufacturing Engineering, 5(9), 636–641. Retrieved from http://waset.org/publications/10504/a-numerical-study-on-rear-spoiler-of-passenger-vehicle

Jomo K. S. (2003). M way : Mahathir’s economic legacy. Forum.

JONATHAN CROUCH. (2006). Audi 100 (1983 - 1994) used car review | Car review | RAC Drive. Retrieved December 12, 2017, from https://www.rac.co.uk/drive/car-reviews/audi/100/207946/

Joseph F. Alward. (1997). Aerodynamic Lift. Coanda Effect. Bernoulli's Equation. Angle of Attack. Retrieved December 12, 2017, from http://www.aerodynamiclift.com/

Julian Edgar. (2011). AutoSpeed - Car Aerodynamics Have Stalled. Retrieved December 13, 2017, from http://www.autospeed.com/cms/article.html?&title=Car-Aerodynamics-Have-Stalled&A=112241

Katz, J. (2006). Aerodynamics of Race Cars. Annual Review of Fluid Mechanics, 38(1), 27–63. https://doi.org/10.1146/annurev.fluid.38.050304.092016

L.E. Mayfield. (n.d.). The Mayfield Homepage - Coefficient of Drag for Selected Vehicles. Retrieved December 12, 2017, from http://www.mayfco.com/dragcd~1.htm

Laha, D., & Ramezanpour, A. (2AD). Computational aerodynamic analysis of a rear spoiler on a car in two dimensions. Academia.Edu. Retrieved from http://www.academia.edu/download/37629662/1227201_Report.pdf

Lanfrit, M. (2005). Best practice guidelines for handling Automotive External Aerodynamics with FLUENT. Fluent, 2, 1–14.

Li Yanlong Yang Zhigang, Z. C. (2013). Electric Car Styling Design and Aerodynamic Drag Optimization. Industrial Design and Mechanics Power II, 437, 9. https://doi.org/10.4028/www.scientific.net/AMM.437.463

Mattijs Diepraam. (2005). The rear-engined revolution: horses pushing the cart Part 1: The pre-war experiments. Retrieved December 12, 2017, from http://www.forix.com/8w/rear-engines-prewar.html

Mazyan, W. I. (2013). Numerical Simulations of Drag-Reducing Devices for Ground Vehicles by, (January).

Meile, W., Brenn, G., Reppenhagen, A., Lechner, B., & Fuchs, A. (2011). Experiments and numerical simulations on the aerodynamics of the ahmed body. CFD Letters, 3(1), 32–38. https://doi.org/10.1017/CBO9781107415324.004

Michael Ballaban. (2014). How A German Rocket-Powered Monstrosity Somehow Predicted The Future. Retrieved December 12, 2017, from https://jalopnik.com/how-a-rocket-powered-german-monstrosity-somehow-predict-1604025788

Michael Grayen. (2016). What Kind of Spoiler is Right for You? Retrieved December 12, 2017, from https://www.carid.com/articles/what-kind-of-spoiler-is-right-for-you.html

Sandra May. (2015). What Is Aerodynamics? Retrieved from https://www.nasa.gov/audience/forstudents/k-4/stories/nasa-knows/what-is-aerodynamics-k4.html

Syon, P. G. de. (2008). The Teardrop That Fell From The Sky: Paul Jaray and Automotive Aerodynamics, 29, 14–16.

APPENDIX A1

Gantt Chart PSM 1

No.	Task	Week														
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Title project selection															
2	Project research: Journal finding criteria															
3	Journal search															
4	Journal discussion															
5	Progress report															
6	Study about CFD															
7	Report: Chapter 1															
8	Report: Chapter 2															
9	Report: Chapter 3															
10	Presentation															
11	Draft final report PSM 1															

APPENDIX A2

Gantt Chart PSM 1I

No.	Task	Week														
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Chapter 4															
2	Progress report															
3	Chapter 5															
4	Conclusion															
5	Final report PSM 2															
6	Presentation															