THREE-DIMENSIONAL SIMULATION ON RACING CAR REAR WING WITH AND WITHOUT DRAG REDUCTION SYSTEM

NUR AIMAN SYAHMI BIN NAZIFUNNUR

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

THREE-DIMENSIONAL SIMULATION ON RACING CAR REAR WING WITH AND WITHOUT DRAG REDUCTION SYSTEM

NUR AIMAN SYAHMI BIN NAZIFUNNUR

This report is submitted
In fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering (Thermal Fluid)

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

MAY 2017

DECLARATION

I declare that this project entitled "Three-dimensional simulation on racing car rear wing with and without drag reduction system" is the result of my own work except as cited in the references.

Signature :...................................

Name :...................................

Date :...................................

SUPERVISOR'S DECLARATION

I have checked this report and the report can now be submitted to JK-PSM to be delivered back to supervisor and to the second examiner.

Signature :...................................

Name of Supervisor :...................................

Date :...................................

DEDICATION

I dedicate this project to my beloved mother, Mashita Binti Radzi and my beloved father, Nazifunnur Bin Md Ishak who giving their hard time to see their son success.

ABSTRACT

Drag reduction system (DRS) is commonly used is racing car event. DRS is installed on their rear wing for extra speed to overtaking their opponent. The main purpose of this study is to study the coefficient of drag acting on the DRS whether high drag coefficient can be achieved during overtaking process. The result will be compared with experimental result to check whether the result is difference or not. The analysis includes the meshing of rear wing and pressure distribution. This study was conducted by studying the drag and lift coefficient test at every 4° interval from 0° to 16°. The highest drag and lift are -10 and -2.5.

ABSTRAK

Sistem pengurangan seretan (DRS) biasanya digunakan adalah perlumbaan acara kereta. DRS dipasang pada sayap belakang mereka untuk kelajuan tambahan untuk memotong lawan. Tujuan utama kajian ini adalah untuk mengkaji pekali bertindak seretan pada DRS sama ada pekali seretan tinggi boleh dicapai semasa proses memotong. Hasilnya akan dibandingkan dengan keputusan eksperimen untuk memeriksa sama ada hasilnya adalah perbezaan atau tidak. analisis termasuk bersirat sayap belakang dan taburan tekanan. Kajian ini dijalankan dengan mengkaji drag dan lif ujian pekali di setiap 4 ° selang dari 0 ° hingga 16 °. Seretan tertinggi dan lif adalah -10 dan -2.5.

ACKNOWLEDGEMENT

First and foremost, I would like to praise Allah for His guidance and blessing, Alhamdulillah. Together with His blessing, I have been able to perform this PSM report within the time.

My deepest gratitude goes to Mr. Shamsul Bahri for giving me an opportunity to perform this final year project under his supervision with the title of Three-dimensional simulation on racing car rear wing with and without drag reduction system. Many thanks for his encouragement and guidance.

A high appreciation to my beloved mother and father for the moral support in completing this degree. Last but not least, thanks to all my friends whose share up some knowledge and supportive. Thank you very much.

CONTENT

LIST OF FIGURES

LIST OF ABBEREVATIONS

DRS	Drag Reduction System
AOA	angle of attack
CFD	Computational Fluid Dynamics

LIST OF SYMBOLS

C_D	=	Drag Coefficient
P	=	Pressure
ρ	=	density
V	=	velocity
β	=	yaw crosswind

CHAPTER 1

INTRODUCTION

1.1 Background

Aerodynamic drag of racing cars has perhaps received utmost attention over last five decades in experimental and applied field of fluid dynamics. Several researchers and authors have labelled altered forms of drag, potential reasons behind them and several ways of reducing the drag. (Katz's et al, 1995) work was fully devoted for the racing car aerodynamics and he described the different feature of car design starting from the first-generation automobiles to most recent models, but no mathematical or investigational process was explained to measure the drag. Computational analysis to reduce the drag is performed by (Barbut et al, 2011) (Rouméas et al, 2009) on road vehicle and by (Guilmineau et al, 2008) on the simplified car body (Ahmed body et al, 2009).

Drag Reduction System (DRS) is a new technology in F1 and is a form of driver-adjustable bodywork aimed at reducing aerodynamic drag in demand to increase top speed and help overtaking in motor racing. It is an adjustable rear wing of the car, which moves in reaction to driver commands. DRS often comes with conditions, such as the pursuing car need be within a second (when both cars cross the detection point) for DRS to be activated. DRS was introduced in Formula One in 2011. The use of the DRS is a concession to the rule prohibiting any moving parts whose major purpose is to affect the aerodynamics of the car. This sort of overtaking is brought about by a speed difference: the car behind going satisfactorily faster than the car in front to make a pass. The higher the speed difference, the easier the overtaking. As racing car cars are typically very closely matched on performance and braking distances are comparatively short related to other methods of racing, overtaking often involves a great deal of skill, commitment and courage. An innovation that makes the driver's job slightly easier is the DRS overtaking aid. Within designated DRS initiation zones, a driver within one second of an opponent car may initiate his DRS. This alters the angle of the rear wing flap, decreasing drag and thereby given that a temporary speed advantage. To ensure that overtaking is not too easy, the distance and position of DRS zones are cautiously controlled. Outside of the DRS zones, drivers can use several other methods to try to get past an opponent.

One is to exploit an aerodynamic 'tow' from the car in front. This is achieved by moving into an opponent's slipstream – a compact of low-pressure air behind a car through which the following driver can move more freely and gain a small speed lead. However, whilst advantageous on straights, this aerodynamic effect is problematic in corners as the reduced airflow acting on the wings of the second car will radically decrease aerodynamic downforce, and hence grip, meaning that the car behind will be forced to drop back, or to pick a different cornering line in 'clean air'. If a driver can't thorough a pass on a straight, he could designate to overtake into a corner under braking. This requires skill from the overtaking driver, not only is he probable to have had to move off line on to a more slippery part of the track, but he must also predict how late he can leave his braking. Get it wrong and he could overshoot the corner, spin off or worse, which is make unintentional contact with the car he's trying to overtake. As you might expect, tyre grip often plays a vital part in these positions with a driver on newer tyres having an advantage. Similarly, a driver on new tyres will stance a better chance of surpassing the car in front in the traction zone out of a corner, particularly if he's set up the change in progress by winning a different racing line into the corner.

1.2 Problem Statement

The F1 racing cars need speed to be overtaking the other opponent. As racing car cars are typically very closely matched on performance and braking distances are relatively short related to other forms of racing, overtaking often needs a great deal of skill. We need to increase the speed so it's easier to overtaking. By using different angle of attack, the speed may increase or decrease. So, we need to conduct a simulation to check whether the angle of attack is suitable or not for racing car.

1.3 OBJECTIVE

The objectives of this project are as follows:

1. To determine and compare the drag coefficient with and without Drag Reduction System (DRS) on racing car rear wing.
2. To provide a complete analysis on the structural of DRS on racing car rear wing.
3. To compare simulation result with experimental data.

1.4 Scope of Project

The scopes of this project are:

1. Result of simulation using with and without DRS on racing car rear wing only presented in this report.
2. Racing car rear wing is simulated only for the side part of unit due to DRS part take effect.
3. To perform a simulation using software ANSYS based on the racing car rear wing to reduce the drag coefficient, C_D.

CHAPTER 2

LITERATURE REVIEW

2.1 Aerodynamic Design Affect the Drag Force

There are differences between aerodynamic on simple vehicle and racing vehicle like their design because the design affect the vehicle aerodynamic and difference drag force. Large turbulent wakes are formed at the rear and in many cases, contain longitudinal trailing vortices (Hucho, Wolf-Heinirch, 1987). On normal car concept of using aerodynamic isn't use much rather than used on racing car because below speeds of 80kmph (50mph), the aerodynamic drag is very less. According to the drag history of car we can see multiple of design affect the C_D.

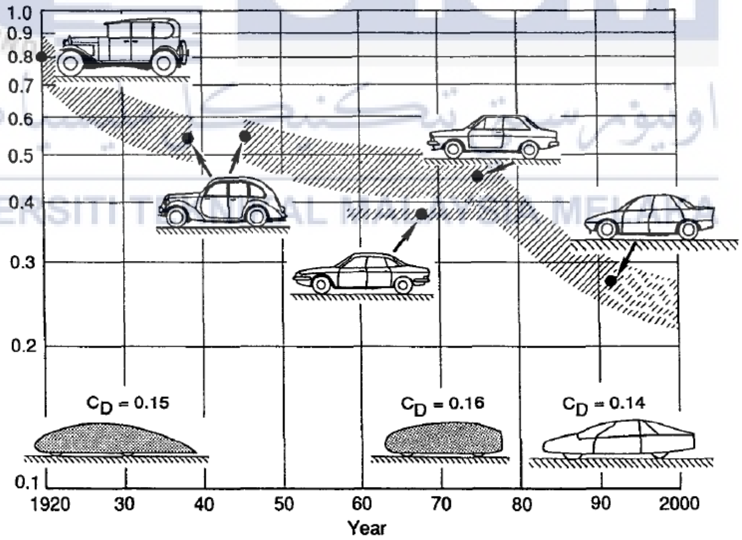


Figure 2.1: The drag history of cars. Using a logarithmic scale for drag emphasizes how difficult it is to achieve very low drag values. Research has been far ahead of what has been realized in production.

The drag can be reduced by streamlining the vehicle so it can have smaller lift. The designed multi winged race car can move easily through the air because of the streamlining is important for aerodynamic downforce (Katz, Joseph, 1995). The streamlining on the racing vehicle design is more than on simple vehicle because to reduce drag and lift to create more speed and more stable. Within couple of years in market, drag will be main thing that will be considered for some specialty vehicle. Drag coefficients less than 0.20 are feasible. However, the development process will be of increased difficulty because there appears to be significant interaction between the local flows in different areas of a car body at these low drag levels (Hucho, Wolf-Heinirch, 1987). Those solid connections the middle of the stream fields of the car's fore-body and back end. Those low drags of a long-tail model might have been supported just when those stream around the fore-body might have been great appended. Those drag enhanced essentially the point when the stream differentiated toward the soak windscreen. (Hucho, Wolf-Heinirch, 1987).

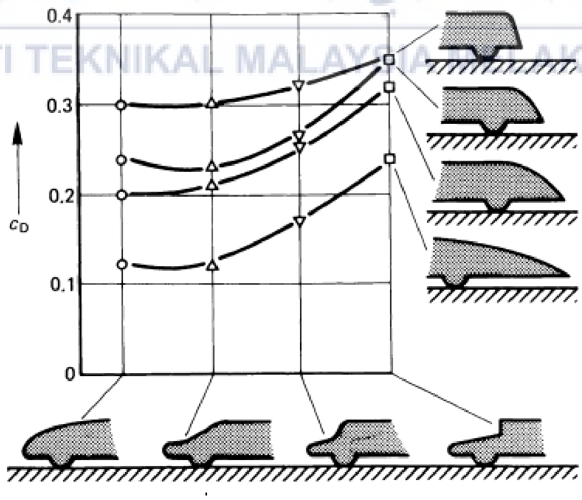


Figure 2.2: influence of main body parameters on the drag of a car and their interactions

The reduction of the drag coefficient from $C_D \approx 0.8$ for cars in the 1920s to an average value of 0.45 for the cars of the 1960s and 1970s occurred in two stages. First stage is on period between the two world wars, the car was designed with rounded bode while sustaining significant characteristics like projecting fenders and headlight will lowering the C_D approximately 0.55, the frontal area also been decrease because to reduce total aerodynamic drag. The second stage is reduction of drag is achieved with the introduction of body design like notchback, fastback and squareback. By using the design with the fenders and headlight in a closed body shape will significantly improve the flow of air around the vehicle, with this design drag coefficient were achieved of 0.4 to 0.5 but depend on the detailed design of it vehicle. This scatter range of drag coefficient has remained unchanged since about 1960. However, it is hard to decide whether the reduction in drag resulted from the impact of aerodynamics, from styling or from more advanced manufacturing techniques (Hucho, Wolf-Heinirch, 1987).

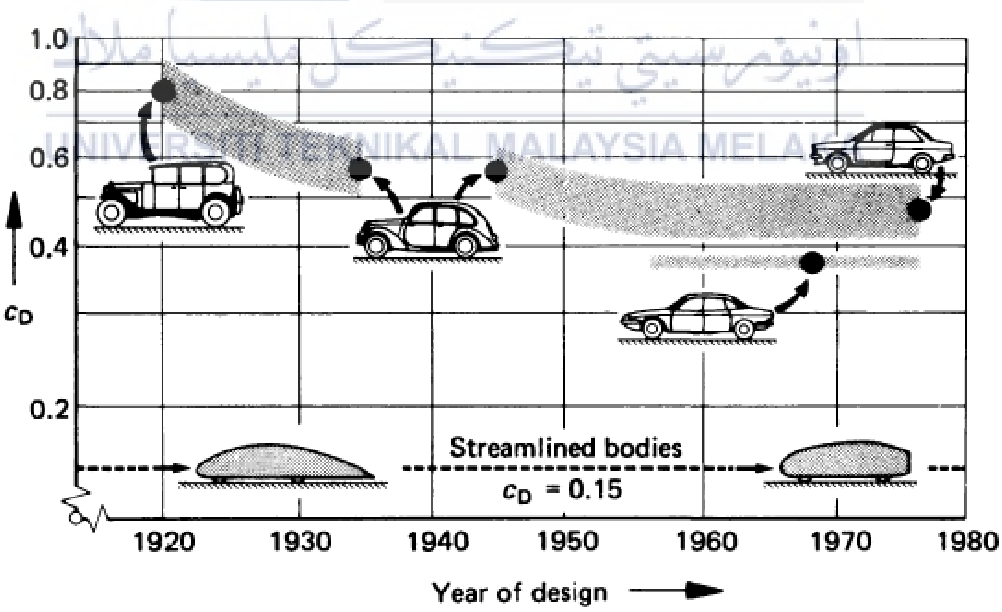


Figure 2.3: Trend in aerodynamic drag coefficient C_D against time, from 1920 to the mid-seventies

The effectiveness of vehicle performance is clearly important for racing car to reduce drag and to increase the downforce (Katz, Joseph, 1995).

2.2 Wing Design on Racing Cars

The basic design used on racing car wing is aerofoil shape because aerofoil shape has better aerodynamic other than shape. Wings in ground effect possess many aerodynamic features of both useful and vital importance. In general, the lift and drag forces of a wing will considerably change near the ground. When an airfoil moves near the ground, flow around the airfoil is viscous and has many viscous interfaces with the ground. In the analysis of ground effect on the aerodynamic properties of the airfoils, the boundary layer on the airfoil must be considered. In contrast, estimate of position of the onset of the transition phenomenon, as an example of boundary layer characteristics, is also necessary in order to predict the drag because the skin friction related to a laminar boundary layer is lower than that of a turbulent one (Venkatesan et al, 2014). The plane design also use a same design of aerofoil and method use in race car wing. Aerodynamic for plane is for to standard aerodynamic practices, the pressure coefficient of the subplots is presented with the negative axis upward. The area trapped by the upper and lower-surface pressure-distribution curves is equivalent to the local sectional lift coefficient (Vassberg et al, 2002).

Fig. 4 shows a sample velocity plot for the rear wing with 0˚ angle of attack and indicates, in general, the higher velocity magnitudes on the bottom surface of the airfoil, as expected due to the inverted setting of these airfoils. Figs. 5 and 6 depict a sample of the variation of the velocity and pressure distribution at 0˚ angle of attack (AOA), respectively. For the inverted airfoils, the low-weight suction territory is on the lower surface of the airfoil, with the positive weight side being on the upper surface of the airfoil (Kieffer et al., 2006)

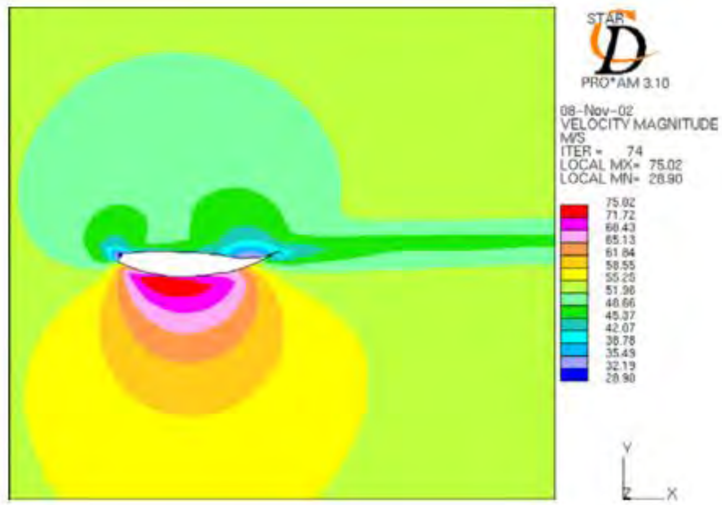


Figure 2.4: rear wing with 0˚ angle of attack

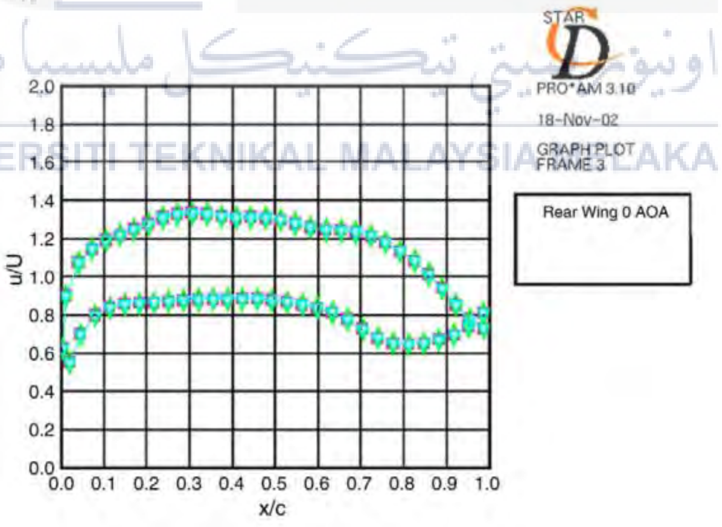


Figure 2.5: Rear wing with 0˚ AOA—velocity distribution.

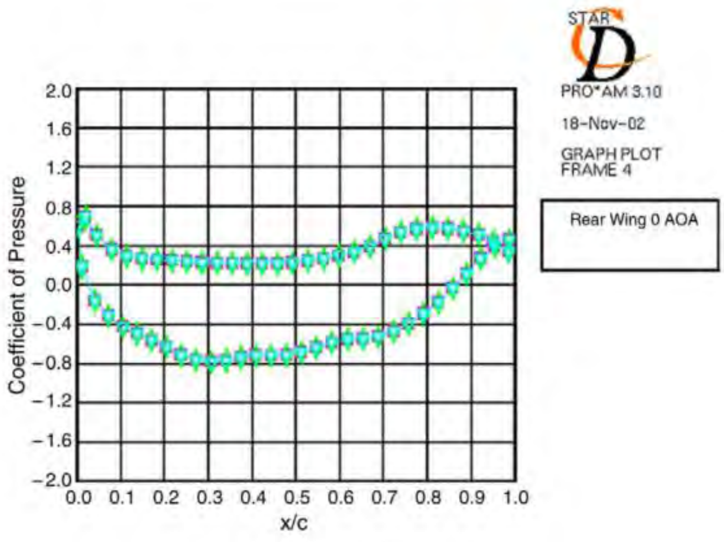


Figure 2.6: Rear wing with 0˚ AOA—pressure distribution.

On other journal review, there are some wing design on racing car F1 which is quite same with my scope. Which is analysis is carried out for F1 car with spoiler for unstructured to 0° and 5° drift angles of spoiler. In this analysis, the journalist used AKN k-ε turbulence model. The normal and axial forces are obtained from the existing analysis. Pressure force and shear force increase gradually with speed due to this drag and normal forces are increasing for 0° and 5° spoiler drift angles. As you can see for this F1 car at 0° spoilers gives less drag then 5° spoilers (Muthuvel et al, 2014).

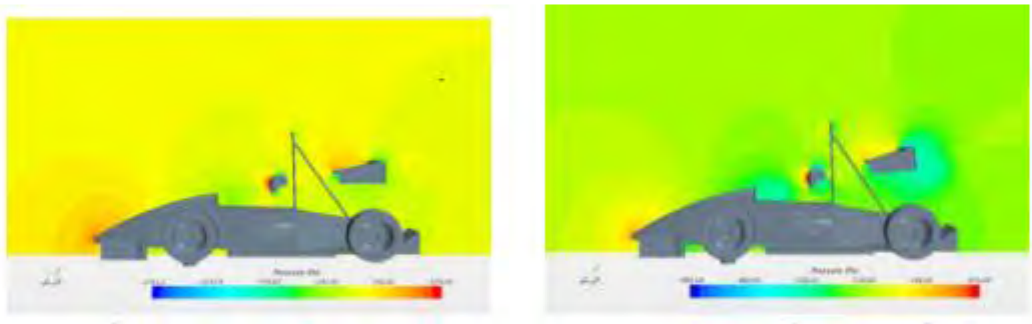


Figure 2.7: Pressure Contours on the Body of 0° and 5° spoiler

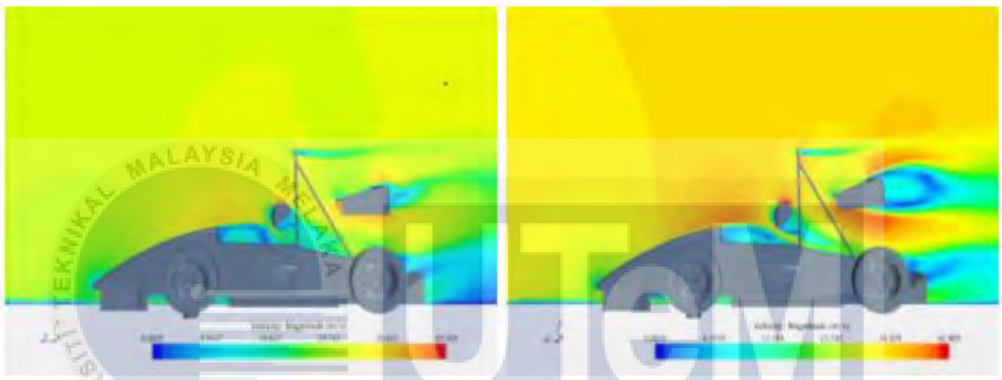


Figure 2.8: Velocity Contours on the Body of 0° and 5° spoiler

2.3 Wake on Racing Cars

The length of the vehicle, the kinematic viscosity and the speed of the vehicle will affect the Reynolds number. There is other significant phenomenon, which affects the flow of the car and the performance of the vehicle. This phenomenon is frequently known as 'Wake' of the vehicle. When air flow over the vehicle, it leaves a large low-pressure turbulent region behind the vehicle known as the wake. This wake contributes to the development of pressure drag, which is eventually reduces the vehicle performance. (Breu et al, 2008). In other journal about this phenomenon is when applying boundary layer suction to race car front wings, it is also important to keep in mind the so-called wake bursting effect. Wake bursting is a phenomenon which only occurs on highly loaded multi-element airfoils. The airfoil can stall without any separation on the wing surface because of flow reversal in the center of the wake of the main element flows from the low pressure at the trailing edge of the main element into a region of high adverse pressure gradient. This happens when the wake of the main element flows from the low pressure at the trailing edge of the main element into a region of high adverse pressure gradient. The displacement thickness of the wake will result with pressure recovery, which in turn suppresses the upper surface pressure distribution on the following elements. The surface pressure distribution on the following element will increasing of angle of attack. In this journal mention that wake bursting will reduces the circulation of airfoil system which is causes stall (Ragavan et al, 2014).

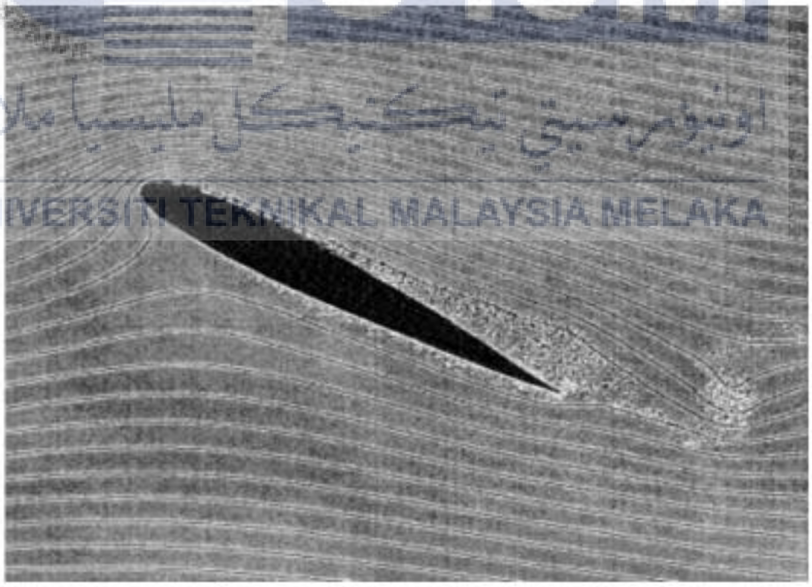


Figure 2.9: Boundary Layer Suction applied to Airfoil in Race Cars.

In the early review, Duncan (1990) focused the impact of back spoiler edges and demonstrated that back downforce increments with bigger edges (measured from the level plane). The effect of a 60-deg rear spoiler was about $\Delta C_L \sim -0.20$. In his second review, Duncan (1994), notwithstanding the investigation of back spoilers, likewise examines the condition when one vehicle is floating behind the other. One of his fascinating results is the reduction in the drag of both vehicles. The trailing vehicle profits by the vast wake of the front auto while the front auto picks up (regarding lessened drag) because of the higher base weight (Katz et al, 2006). When the gap becomes very small. However, viscous effects such as confluent boundary layers and wakes tend to reduce the lift generated, resulting in a value of gap below which there is a decrease in the wing CL. This study again emphasizes the importance of performing viscous computations in designing two-element airfoils for high-lift (Gopalarathnam et al, 1997)

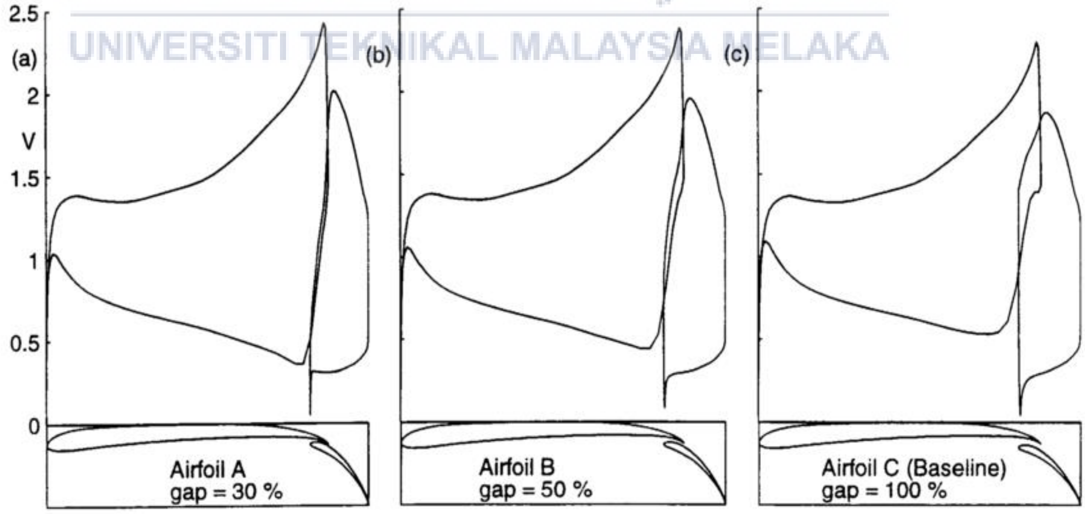


Figure 2.10: The airfoils resulting from (a) gap= 30%, (b) gap= 50% and (c) gap= 100% of the gap of the baseline airfoil.

Other than that, the wake of a Formula 1 car is dominated by the counter-rotating vortex pair originating at the rear wing, which is further energized by the up wash from the rear diffuser (Newbon et al, 2015).

2.4 Overtaking Vehicle

This is one of reason the DRS is widely used on these racing car is to overtaking other racing car. There is some journal can verify that to overtaking is by decreasing the C_D so it can increase the velocity. It can be assumed that the difference in pressure coefficient (C_p) around the overtaking model in isolation is autonomous of the model velocity. Since the pressure coefficient at a position s is defined by $C_p(s) = (P(s)-P_\infty/ (1/2)\rho V^2)$ any increase in the free stream velocity (V) results in an increase in the size of the static pressure (P(s)). Therefore, as the overall velocity of the overtaking vehicle is increased (i.e. V_r is increased) the magnitudes of the static pressures around the model are also increased (Corin, et al., 2008).

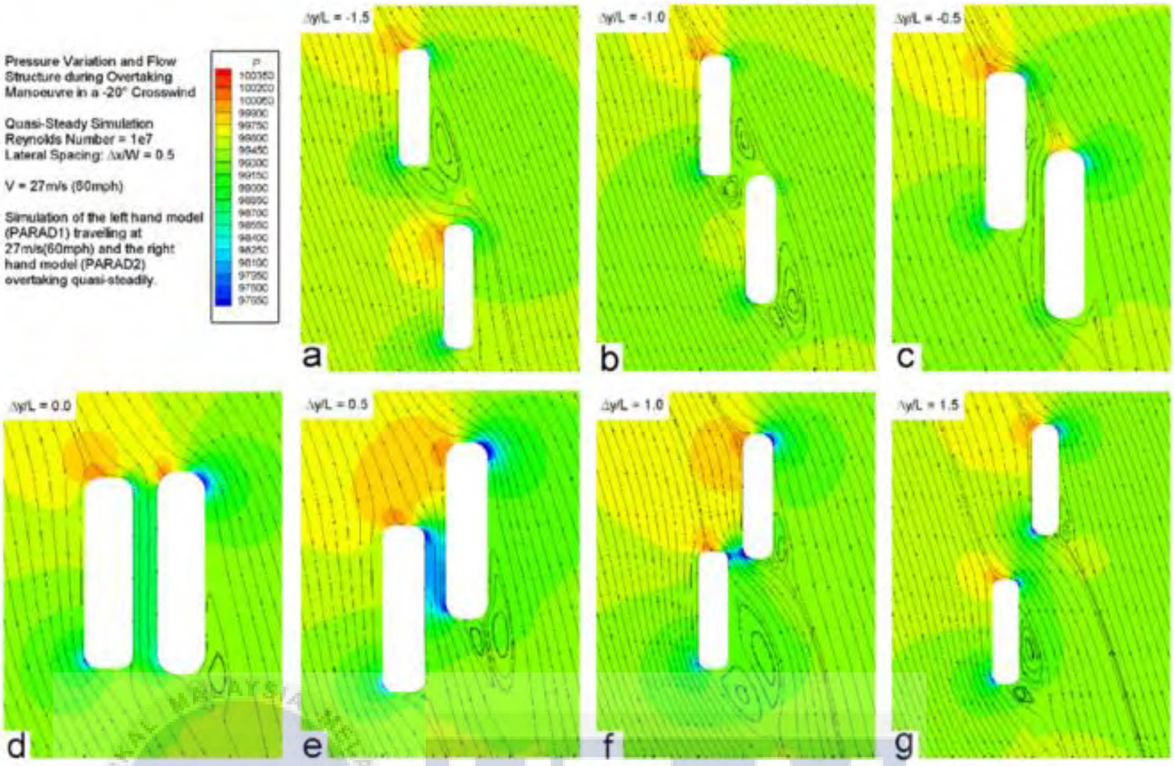


Figure 2.11: Pressure fields and flow streamlines for the quasi-steady analysis of overtaking manoeuvre in a β =-20° yaw crosswind.

Besides that, the loss of downforce happen because of the frontal wing is subjected to variable during race conditions. For example, during overtaking, downforce will lose at frontal wing due to slip stream behind another car. With boundary layer suction, this problem of low energy airflow can be solved, which makes it easier to overtake. The ability to decrease the downforce on high speed parts of the racetrack is the second advantage, and consequently also the induced drag, by blowing a small amount of air into the boundary layer. This way, the location of separation can be moved as the designer wishes. As a consequence, boundary layer suction contributes to better racing results (Ragavan et al, 2014). The aerodynamic interaction between two or more vehicles can adjust vehicle balance and directly affects all safety aspects. Stock cars (e.g., NASCAR), for example, are racing and drafting closely and aerodynamic effects on behaviour are important. Also, both aerodynamic drag and balance change when the vehicles change positions overtaking will be difficult (Katz, Joseph, 2006). On other journal said the stability is also important during overtaking so to increase down force of car when travelling at high speed so it can improve the stability of the vehicle, spoilers/wings are employed but the main disadvantage of having a spoiler/wing, however, is that it will increases the total drag and the rolling resistance of the vehicle. Drag forces are dependent on the velocity of the vehicle. The higher the velocity facing the vehicle, the more aerodynamic forces are applied on the vehicle and hence, the more power the vehicle needs to overcome these frictional forces (Mazyan et al, 2013).

2.5 Verification of Aerodynamic of Racing Cars

2.5.1 **Computational Fluid Dynamics (CFD)**

Using CFD will save your time to do the experiment and also save the cost of making the model. CFD complements experimental and analytical approaches by providing an alternative cost effective means of simulating actual fluid flows. Particularly, CFD significantly reduces lead times and costs in designs and production compared to experimental-based approach and offers the capability to explain a range of complicated flow problems where the analytical approach is lacking (Breu et al, 2008). Besides that, the selection of candidate designs is being made from the computer screen, and aerodynamic data (e.g. C_D) could contribute to these pre-hardware decisions, but only CFD is able to generate results on time (Hucho, Wolf-Heinirch, 1987). The focus is on high reliability road vehicle simulations, but with as short as possible turnaround time as requirement for aerodynamic optimization and innovation at lower development cost. The airflow is modelled using different commercial CFD packages, i.e. Ansys Fluent, CFX, Open FOAM and Power FLOW. Furthermore, references for geometry research, grid and case set-up are given (Muthuvel et al, 2014). The physical characteristics of the fluid motion can typically be described through fundamental mathematical equations, usually in partial differential form, which govern a process of interest and are often called governing equations in CFD. Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu has discussed how to solve mathematical equations with using CFD:

"In order to solve mathematical equations, computer scientists convert them by using high-level computer programming languages into computer programs or software packages. The computational part simply means the study of the fluid flow through numerical simulations, which involves employing computer programs or software packages performed on high- speed digital computers to attain the numerical solutions. Another question arises "Do we actually require the expertise of three specific people from each discipline -fluids engineering, mathematics, and computer science- to come together for the development of CFD programs or even to conduct CFD simulations?" The answer is obviously no, and more likely it is expected that this field demands a person who will proficiently obtain some subsets of the knowledge from each discipline." (Breu et al, 2008).

There are many CFD technique that can be use on to find fluid dynamics and heat transfer of the problem. Large Eddy Simulation (LES) is a CFD technique where large flow structures are directly computed from Navier Stokes equations and only the structures slighter than the computational cells are modelled. Meanwhile the size of turbulent vortices decreases with increasing Reynolds number, LES is performed at adequate Reynolds numbers so that most of the turbulent vortices can be directly resolved rather than modelled. Krajnovi et al. (2004) performed LES on 250 Ahmed model for medium and fine grids. These studies were performed at low Reynolds number (2×105) to facilitate the use of LES (Singh et al, 2014).

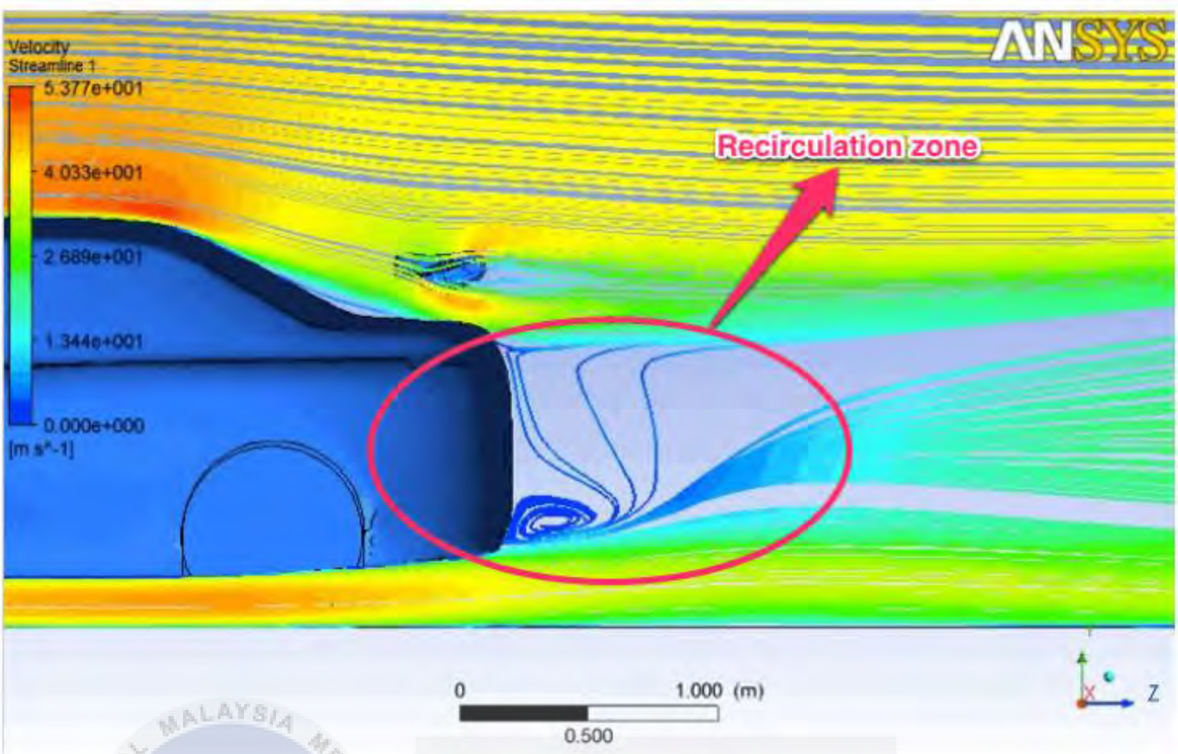


Figure 2.12: Velocity streamlines of flow in the symmetry plane.

2.5.2 **Wind Tunnel**

All the preceding deliberations are based on the statement of an idealized wind, i.e. steady, with a spatially uniform velocity. This is never the case. Natural wind has a boundary layer character. When its velocity profile is combined vector ally with a vehicle's forward speed, a skewed oncoming relative-wind profile is generated (Figure 2.15, Hucho 1974) which cannot be reproduced either in a wind tunnel or by side-wind facility at a test track (Hucho, Wolf-Heinirch, 1987).

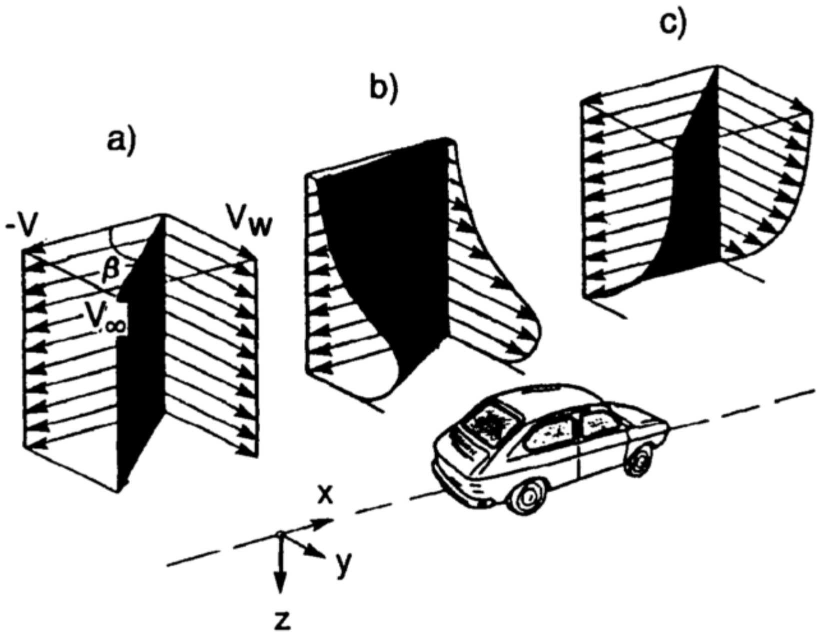


Figure 2.13: Wind profiles: (a) in the wind tunnel, (b) on a crosswind track, and (c) in natural wind

Generally, the wind burrow testing of cars began with little scale models. Scales like 1:4 or 1:5 were favoured in Europe, the bigger 3/8-scale in the U.S. The benefits of little scale testing are that the models are less expensive than full-scale ones, are anything but difficult to handle, and can be immediately changed. Moreover, just little wind passages are required, and these are more by and large accessible and can be leased at direct cost.

For two reasons, little scale testing was in the end just seldom utilized. At to start with, this was because test comes about because of fractional scale models all the time did not repeat full-scale values with the exactness required. This inadequacy was somewhat because of an absence of geometric similitude in the models, and mostly to the erratic impacts of Reynolds number. In any case, geometric likeness is not a basic issue yet rather a matter of the ability and care of the model creator. Likewise, a

Reynolds-number hole on the request of two can be spanned by falsely expanding the turbulence level of the wind burrow. Consequently, small-scale testing has again come into favour and is used by some car manufacturers with great success.

The second and significantly more grounded complaint to little scale testing is non-streamlined in nature. Vehicle outside plan is done in full scale, since shapes in little scale can't be sufficiently evaluated stylishly. In this way, a full-scale show dependably exists, and in the event, that it is based on a practical undercarriage, for example, the one from the first model year, it can likewise be utilized as the wind-burrow demonstrate (Hucho, Wolf-Heinirch, 1987).

To find the drag coefficient of a vehicle, a wind tunnel test is applied on the vehicle in which inlet airflow is blown on the vehicle and the total difference in pressure in front and behind the vehicle is measured. The difference in the pressure is a proportional to the drag coefficient. However, time-varying forces are acting on the vehicles, where a part of these time varying forces and moments is the effect of overtaking of the vehicles and cross winds. These forces need to be studied as they also influence the vehicle's performance and stability (Mazyan, Walid Ibrahim, 2013).

The other thing that we can use virtual wind tunnel to find our drag and lift. A virtual air-box has been created around the 3D CAD model (Figure 2.16), which represents the wind tunnel in the real life. Since we are more concerned in the rear side of vehicle, which is where the "wake of vehicle" phenomenon occurs, more space has been left in the rear side of the vehicle model to capture the flow behaviour mostly behind the vehicle (Breu et al, 2008)

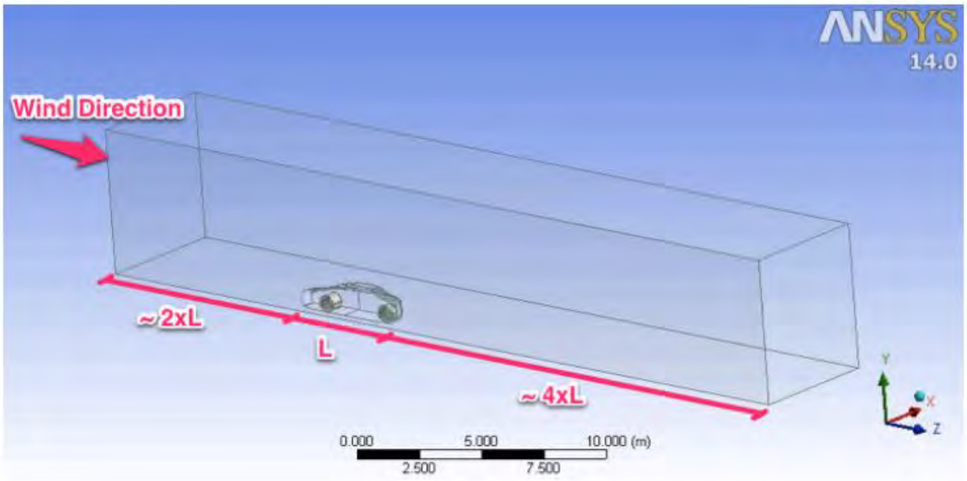


Figure 2.14: Virtual wind tunnel and the vehicle orientation.

2.5.3 Track Test

A few challenges natural to wind tunnel burrow testing are just non-existent in full-scale streamlined testing on the race track. Moving wheels, moving ground, the right Reynolds number, and wind burrow blockage revision are altogether settled and there is no compelling reason to assemble a costly, littler scale demonstrate. Obviously, a vehicle must exist, the climate must participate, and the cost of leasing a track and instrumenting a moving vehicle must not resentful the financial plan. Because of the previously mentioned focal points, and disregarding the uncontrolled climate and cost issues, this type of streamlined testing has enhanced extensively as of late. One of the most punctual types of testing was the drift down test to decide the drag of a vehicle. Disregarding variety in air conditions and irregularities in tire moving resistance, sensible incremental information can be gotten, as talked about by Crewe et al. (1996). With the progress in PC and sensor innovation, before the end of the 1990s the alluring powers, minutes, or weights could be measured and transmitted by means of remote correspondence at a sensible cost. Sensors to quantify suspension uprooting, different anxiety/strains, drive shaft torques, weights, temperatures, and so forth are accessible off the racks. Information obtaining frameworks (see Petrone et al. 2002) can quickly break down burdens and give data, for example, temperature or weight drop over the cooling framework, downforce, and drag of different parts (counting wings and wheels). Indeed, even stream representations can be led by introducing smaller than normal cameras at different areas to give data on stream division, vortex trails, or impromptu distribution in the cooling framework. Regardless of the innovation turning out to be exceptionally powerful and moderate, race track leasing is still very costly, and to spare cost in many types of hustling the coordinators essentially restrict the quantity of track test days and some even deny utilizing telemetry (for building purposes) amid the race. Since this device developed just as of late and because of the aggressive way of the game, just constrained data was accounted for on it in the open writing (Katz, Joseph, 2006).

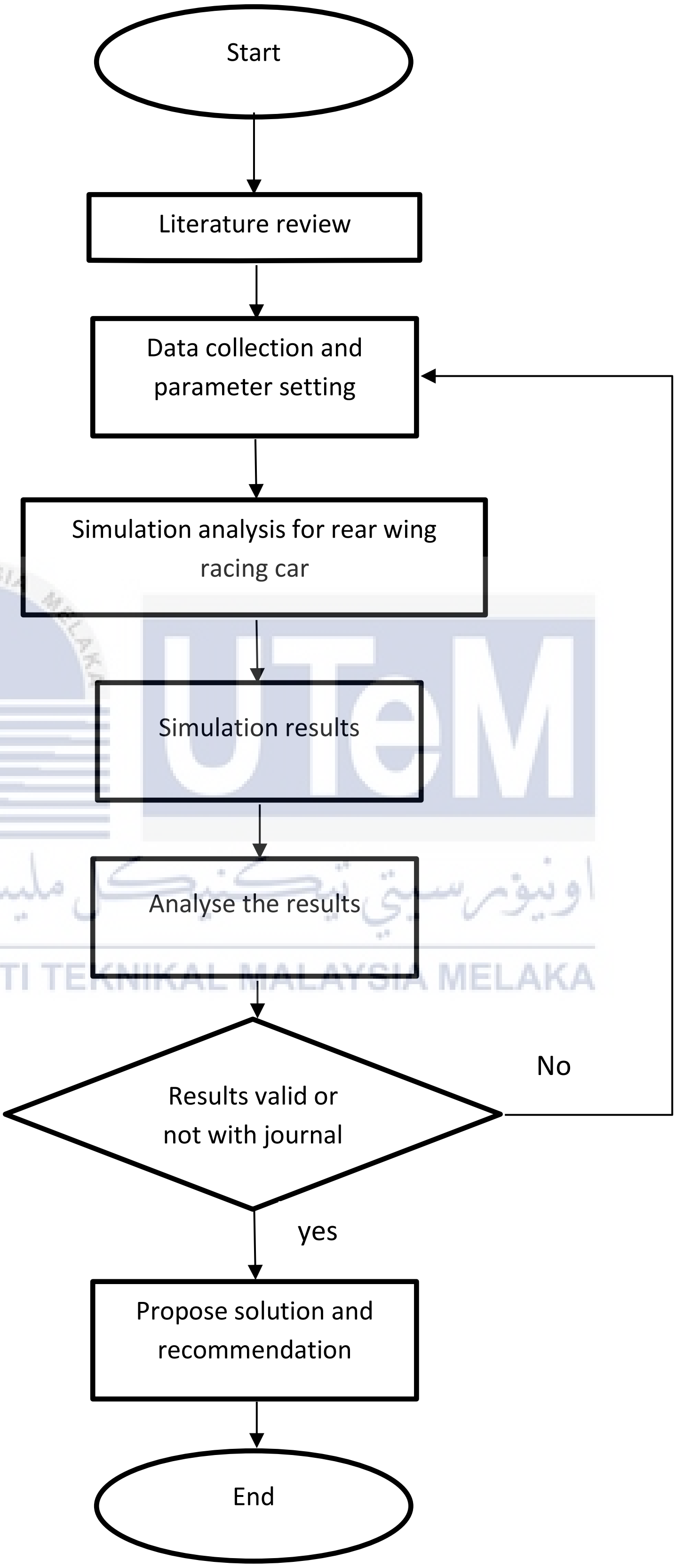
CHAPTER 3

METHODOLOGY

3.1 Introduction

This chapter describes the methodology used in this project to obtain data input for coefficient of drag, lift and the angle of attack that will effect of racing car to overtaking. This project starts by studying and used the commercialize software called ANSYS workbench 17.1, a computational fluid dynamics and to study the drag reduction system on racing car to obtain correct and valid measurement data. After mastering using the ANSYS software, the rear wing of racing car is chosen as a criterion of this project. Suitable rear wing for our DRS have its angle of attack that will produce less C_D and lift. To find the suitable DRS design for overtaking other racing car will be determined based on the journal reading.

3.2 Methodology Flowchart



3.2 Research Method

The information of racing car on different of angle of attack and the shape of aerodynamic for rear wing racing car that taking effect in DRS had been gathered and referred based on the journal, internet sources and lecture to get the general idea and planning for the project. All information will be gathered based on the title of the project and literature review will be made based on the journal reading to get better understanding for the topics. The apparatus that will be used in this project will me ANSYS for simulation and Solidwork software for 3D drawing of racing car. The drawing will be transferred into ANSYS for CFD simulation

3.3 Simulation

3.3.1 Making model using SolidWork

The dimension of rear wing design is shown on the figure 3.1 until 3.4. The units are in inch. For the figure 3.1 will be extrude for 50 inch for the exact size of rear wing of racing car.

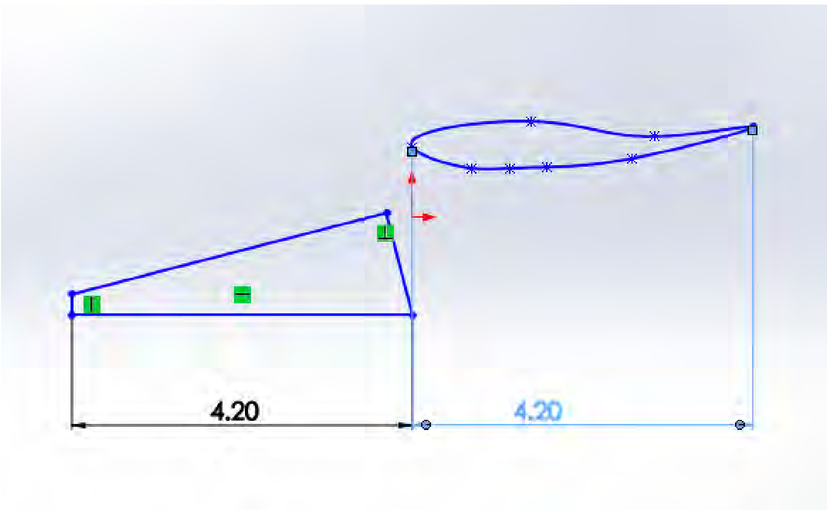


Figure 3.1: dimension of aerofoil of rear wing

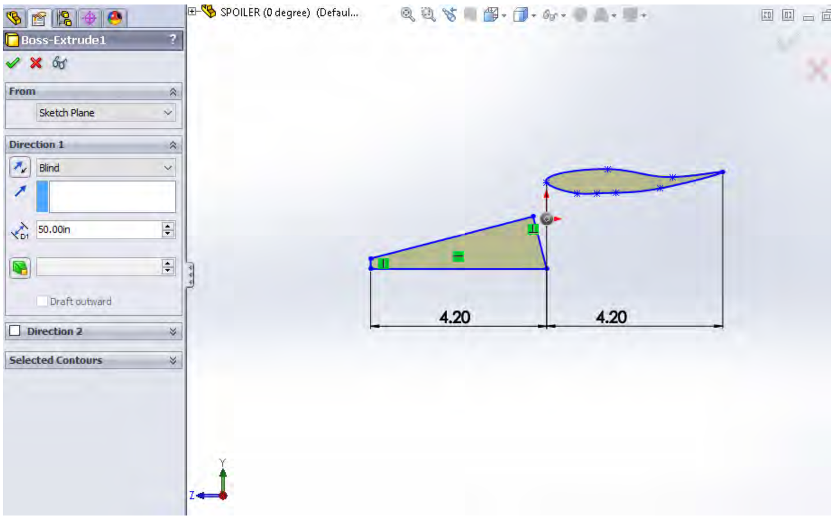


Figure 3.2: extrude size of aerofoil of rear wing

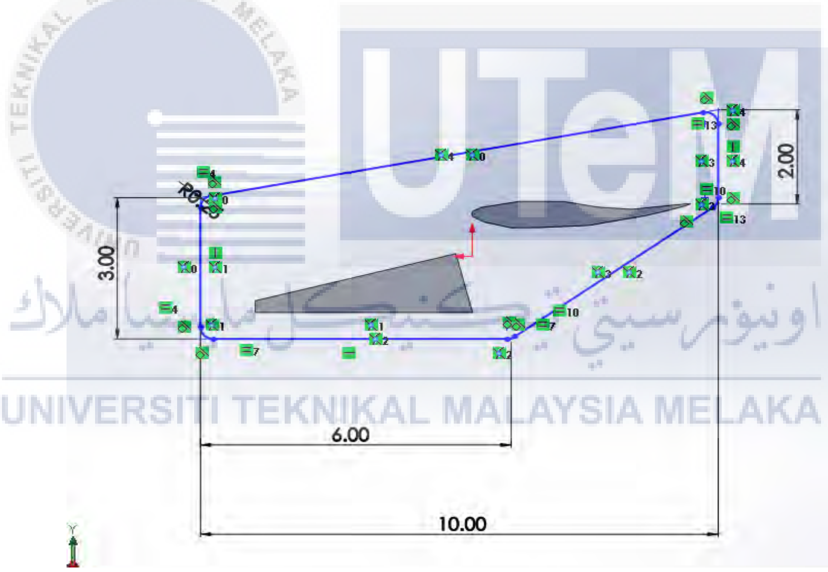


figure 3.3: dimension for side rear wing

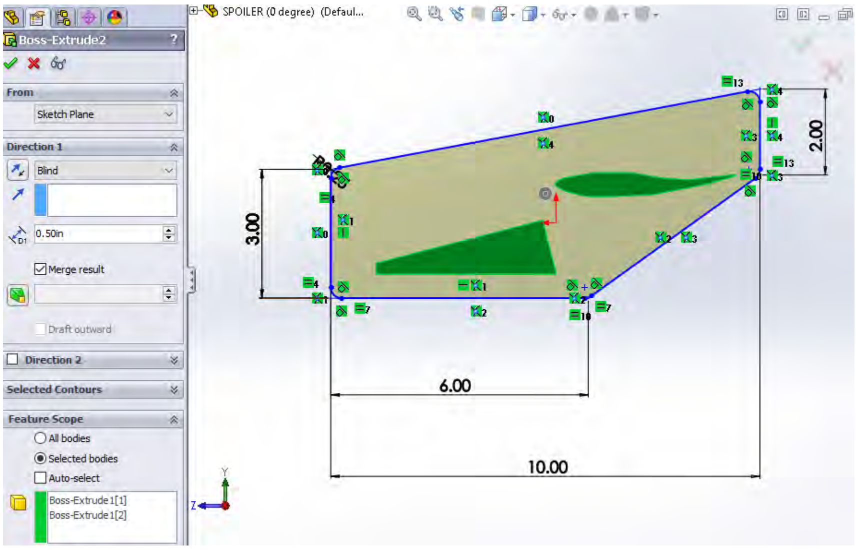


Figure 3.4: extrudes size for side rear wing

After that the model will be scale down to 1:10 size because of limitation of node for ANSYS.

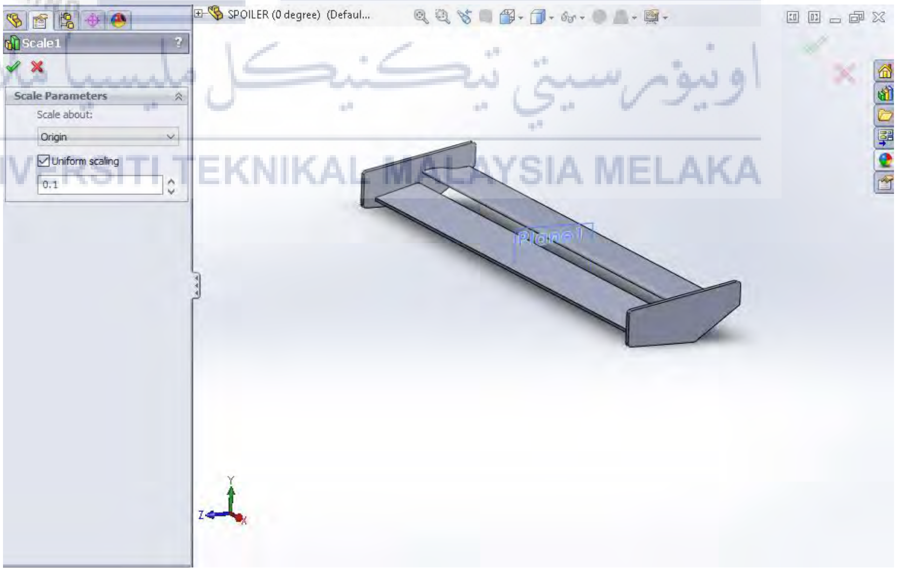


Figure 3.5: scaling on rear wing.

3.3.2 Importing Drawing into Workbench

The aerofoil profile of racing car rear wing had been drawn by using Solidwork software and then will be imported to the ANSYS workbench. The size of racing car rear wing will be scale down size to 1:10 for the simulation because of the limited nodes for student for using ANSYS software. First the enclosure was added to create the surrounding parameter like in wind tunnel.

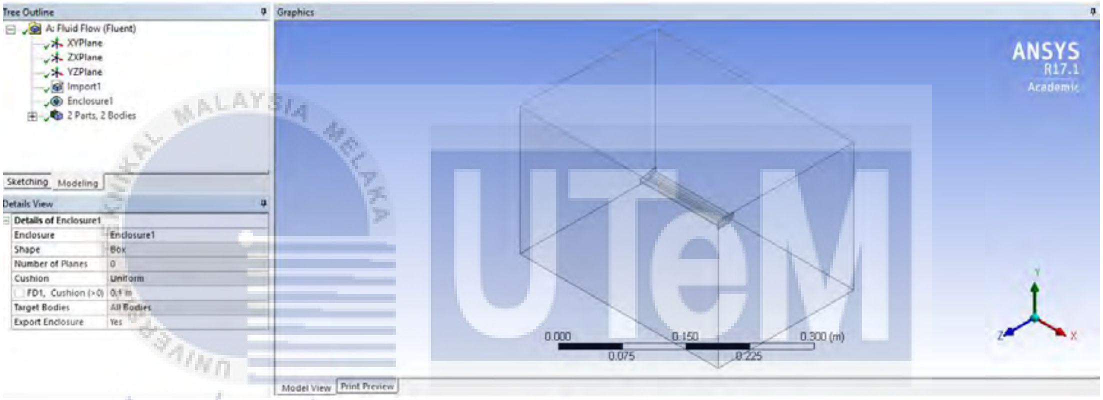


Figure 3.6: enclosure

Next rename the part for the meshing later.

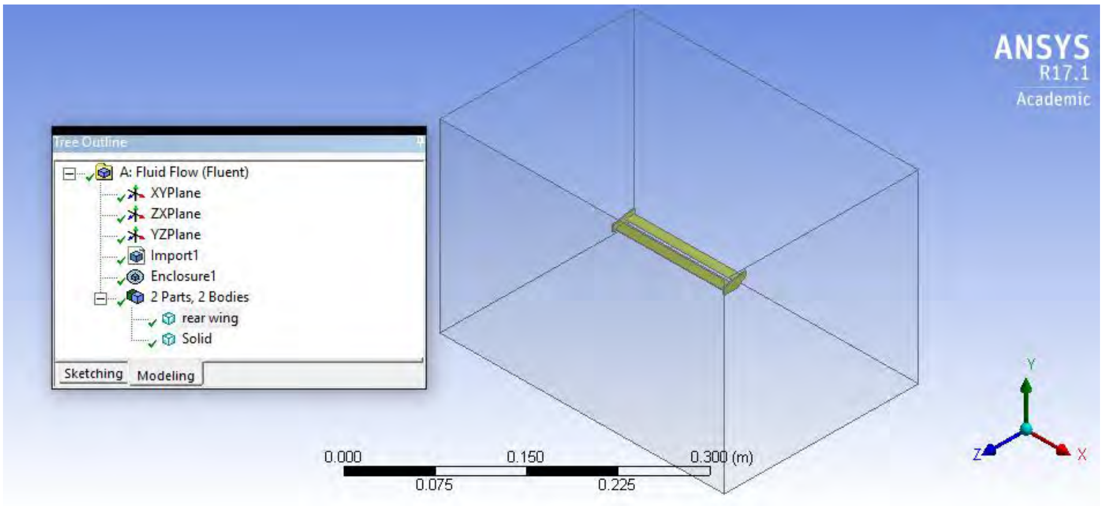


Figure 3.7: rear wing part

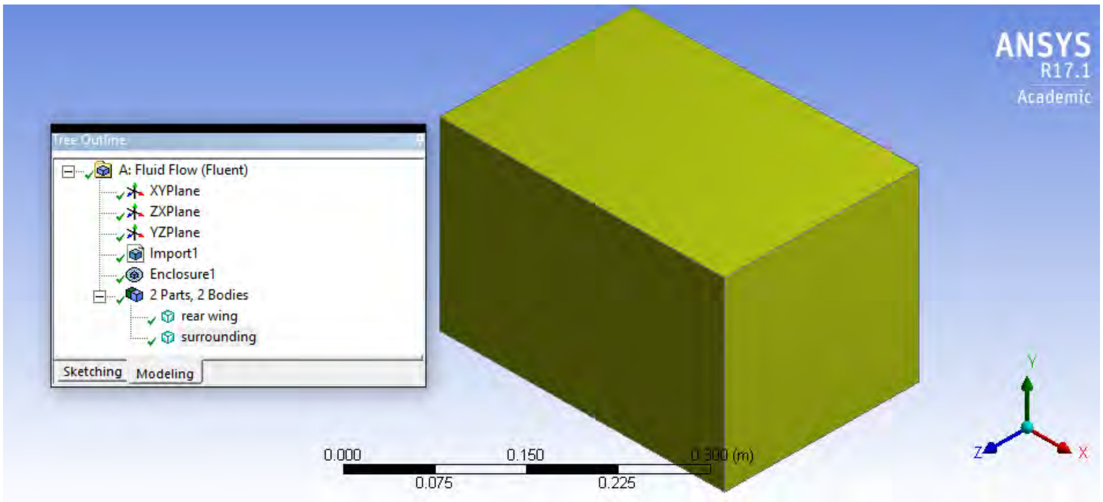


Figure 3.8: surrounding part

After renaming the part, Boolean is added to the simulation for the rear wing and surrounding will combine but still a separate part. The operation that will be use is 'subtract' operation. The target bodies will be surrounding and the tool bodies will be rear wing part.

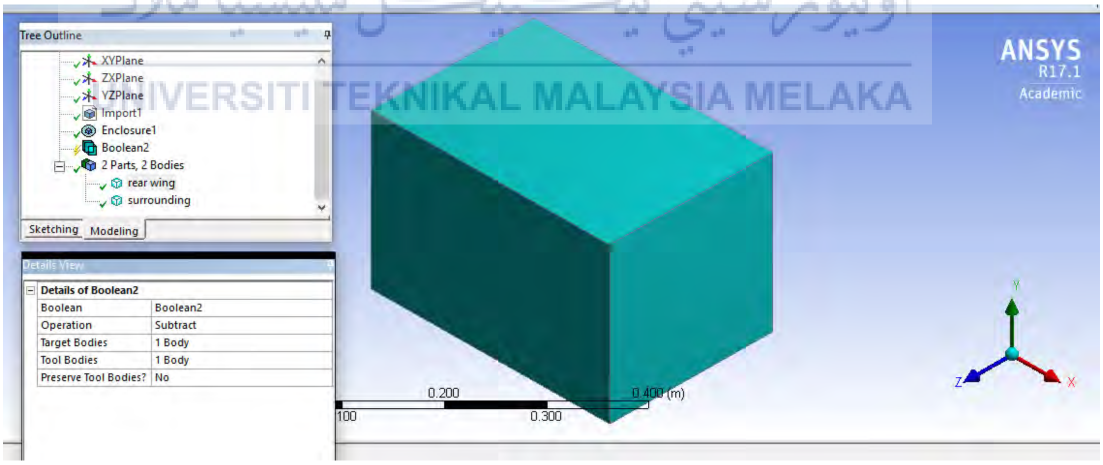


Figure 3.9: Boolean

3.3.3 Meshing

Meshing is very important in the CFD simulation, to run the simulation, the meshing need to be done first. First, open the meshing modular and the geometry will be imported to the meshing modular. The default will be generated at the surface on the geometry. The sizing for default mesh is course. There is other refinement on the mesh will be used in this project which is coarse, fine and medium. This different refinement will show more detail of mesh which is increasing of the nodes used in simulation. First named the selection face for inlet and outlet.

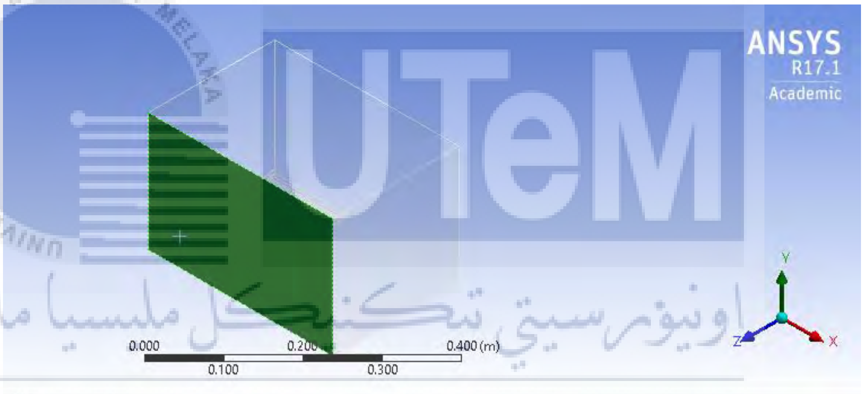


Figure 3.10: inlet

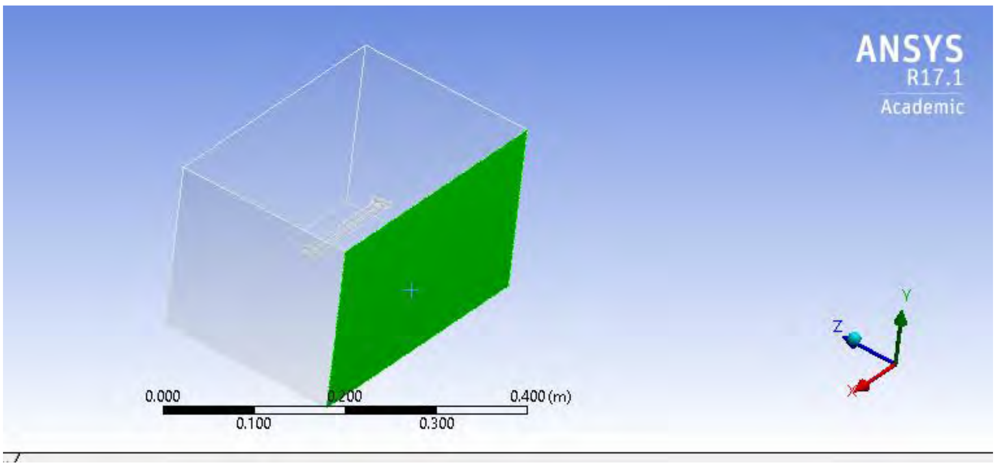
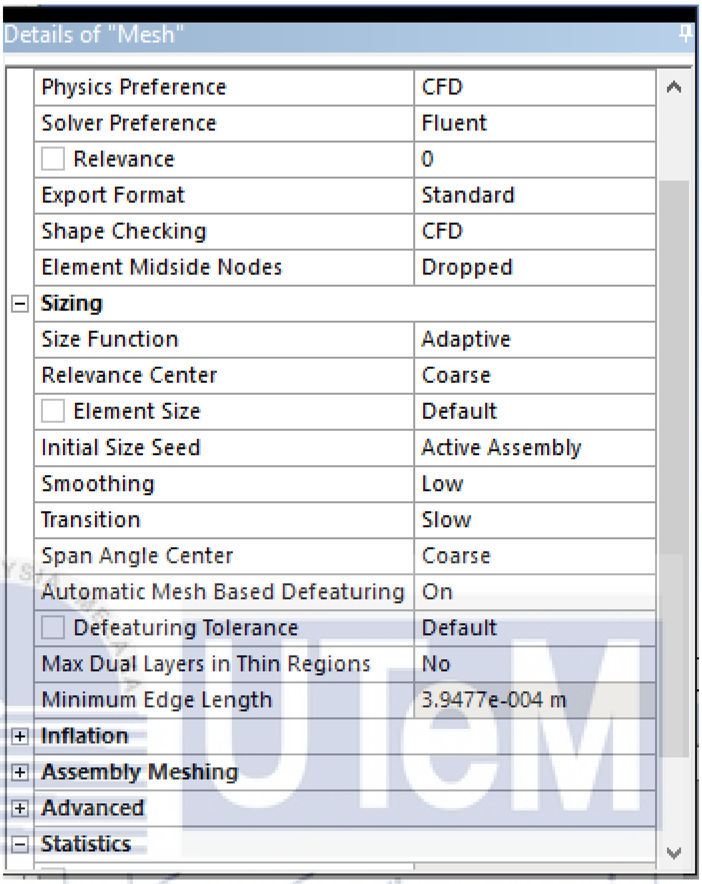


Figure 3.11: outlet



Details of "Mesh"

Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Export Format	Standard
Shape Checking	CFD
Element Midside Nodes	Dropped
Sizing	
Size Function	Adaptive
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Low
Transition	Slow
Span Angle Center	Coarse
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Max Dual Layers in Thin Regions	No
Minimum Edge Length	3.9477e-004 m
Inflation	
Assembly Meshing	
Advanced	
Statistics	

Figure 3.12: detailed of mesh

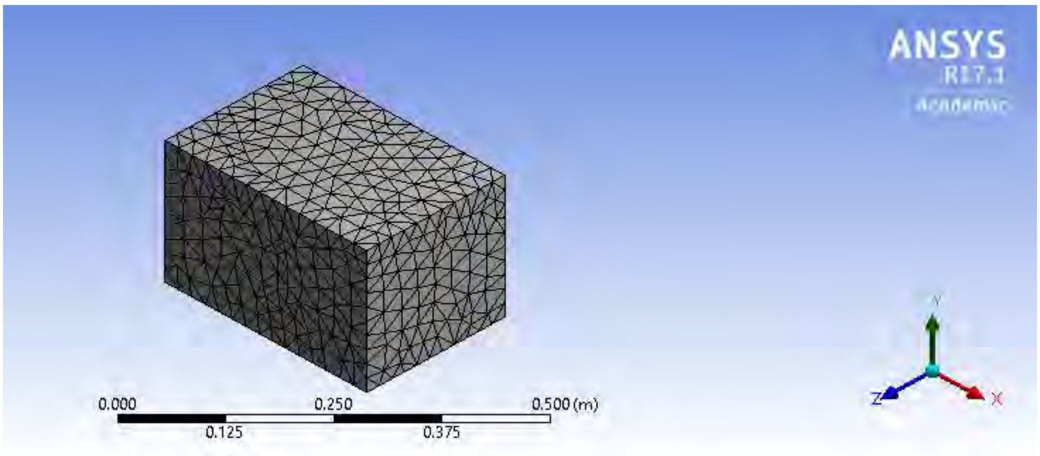


Figure 3.13: mesh complete

3.3.4 Setup for simulation

First check the mesh in general because the quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. The attributes associated with mesh quality are node point distribution, smoothness, and skewness.

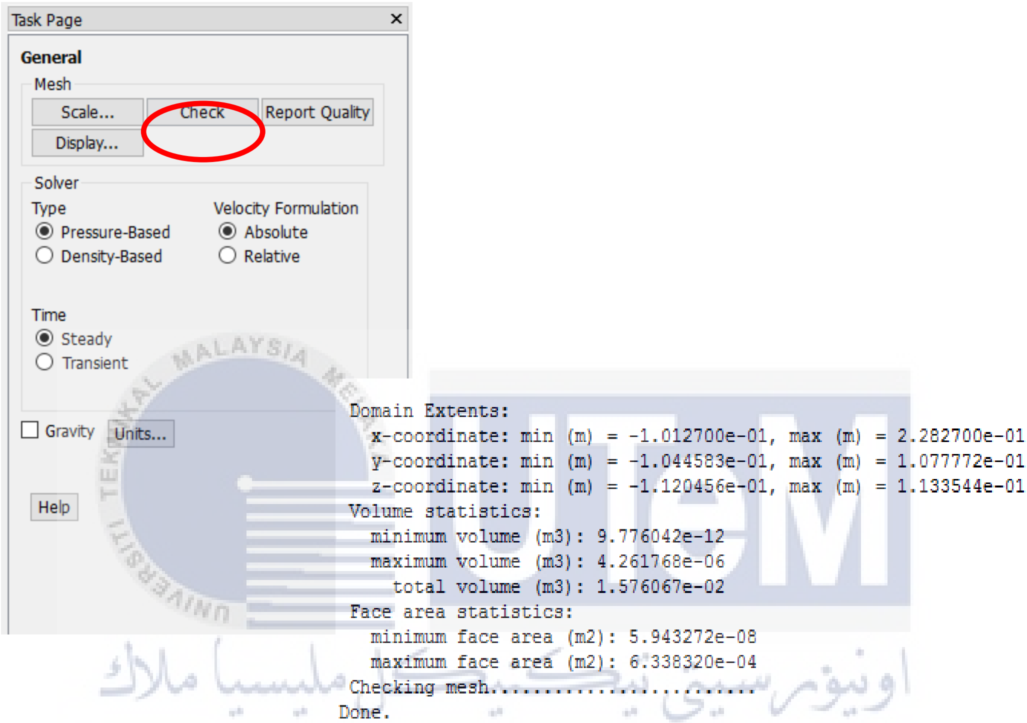


Figure 3.14: checking mesh

Next choose the model for the simulation. In this case choose viscous model of k-epsilon.

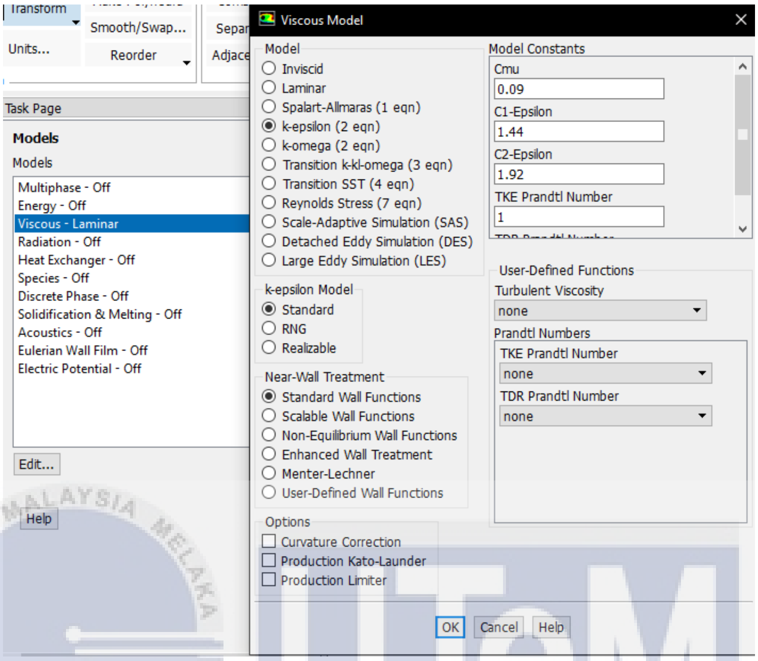


Figure 3.15: viscous model for simulation

After that, boundary condition in the inlet for velocity magnitude should be 120 m/s for the actual racing car speed but in this simulation, use 1:10 scale so the velocity is 1/120 which 12m/s.

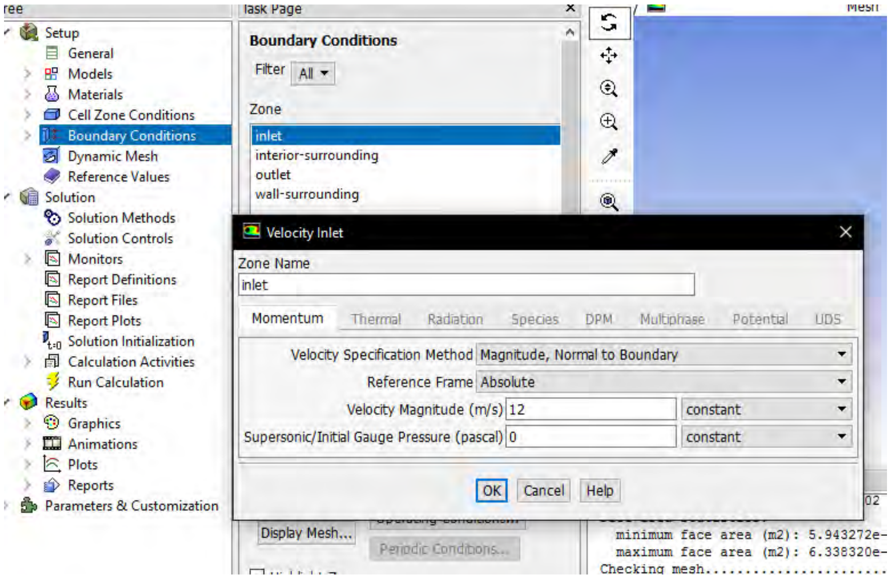


Figure 3.16: boundary condition for inlet

For the solution method, choose second order upwind for turbulent kinetic energy and turbulent dissipation rate because second-order discretization generally yields better accuracy while first-order discretization yields more robust convergence.

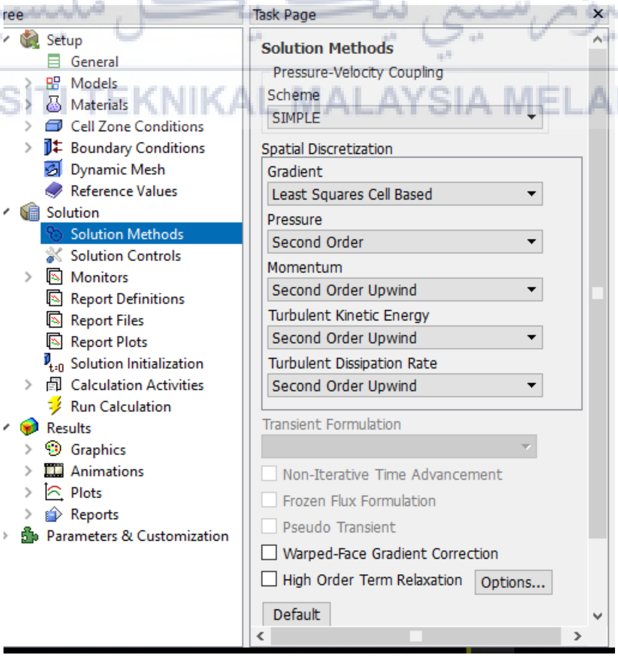


Figure 3.17: solution methods

In the monitor, drag and lift monitor is added.

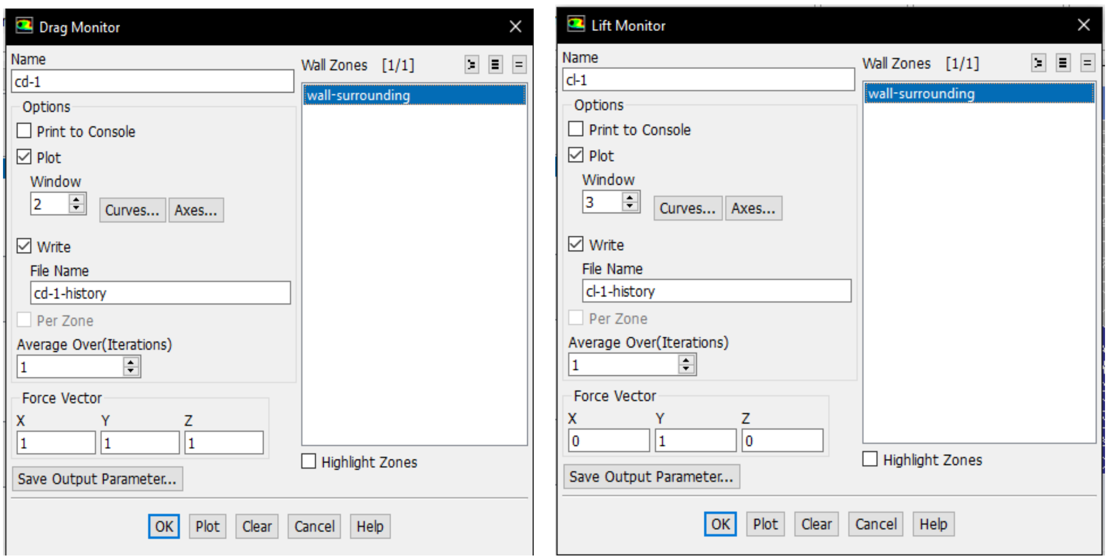


Figure 3.18: drag and lift monitors

After drag and lift monitor are added, compute the standard initialization from inlet.

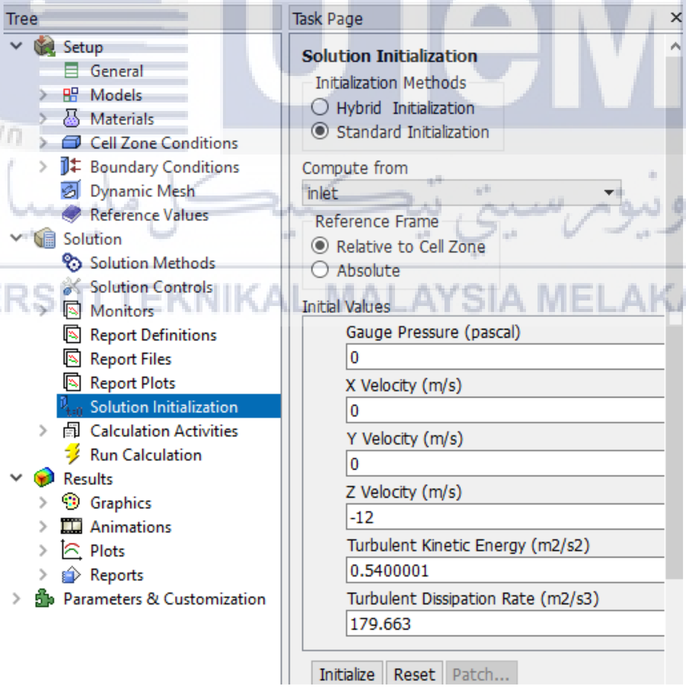


Figure 3.19: solution initialization

Finally, run the calculation to see the result for drag and lift.

CHAPTER 4

DATA AND RESULT

4.1 Degree of Angle of Attack for Rear Wing

The angle of attack will be 0°, 4°, 8°, 12° and 16° because according in journal CFD study of section characteristics of Formula Mazda race car wings. By increasing the angle of attack therefore it would increase the downforce. It was shown that the ground effect has a marked effect on the lift coefficient and that the angle of attack has a significant effect on the lift and drag coefficients, and it was shown that an angle of 12° below the horizontal seems to indicate stalling conditions. (W. Kieffer et al, 2005).

4.2 Drag and Lift Coefficient For 0°, 4°, 8°, 12° and 16°

As I mentioned before, when the angle of attack in increase the downforce also increase. This the result that obtained from the simulation for each degree taken with speed of 12m/s because the velocity is scale down to 1:10 for the simulation.

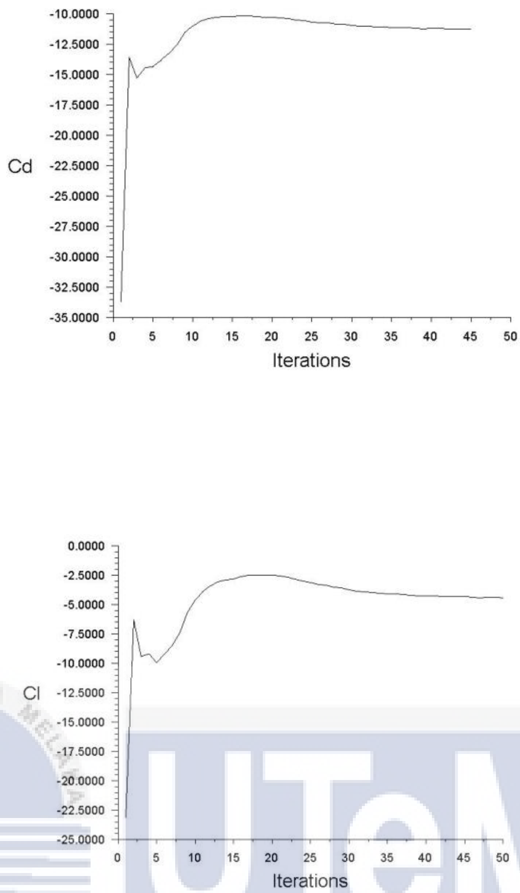
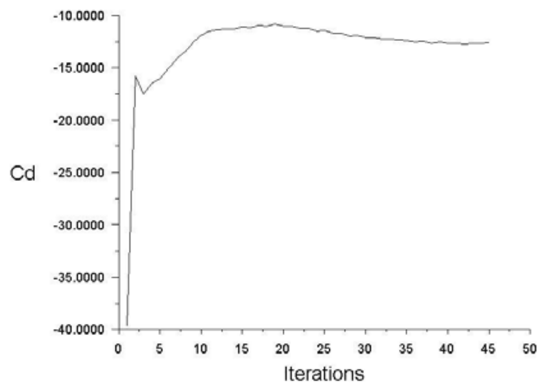


Figure 4.1: drag and lift for 0°

The graph shows the increasing of coefficient due of increasing of iterations. The highest coefficient for drag and lift for 0° is -10 and -2.5.



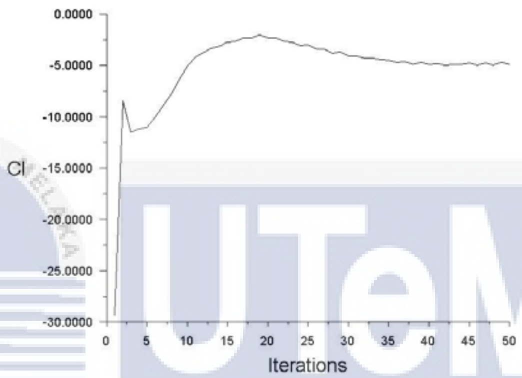


Figure 4.2: drag and lift for 4°

The highest coefficient for drag and lift for 4° is -11 and -2.5

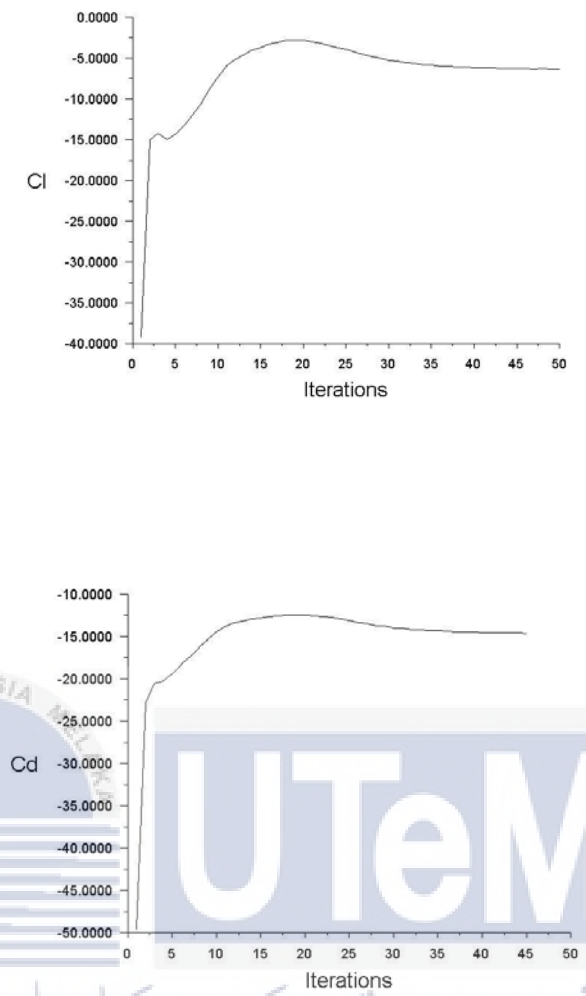
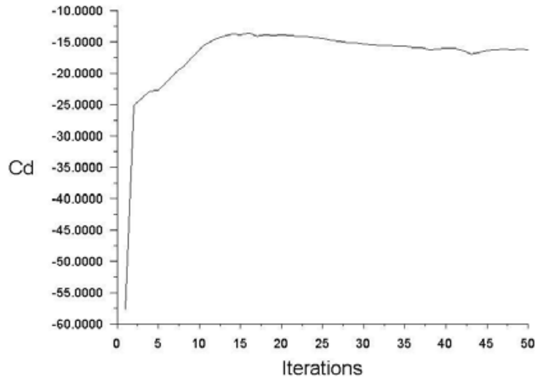


Figure 4.3: drag and lift for 8°

The highest coefficient for drag and lift for 8° is -12.5 and -3



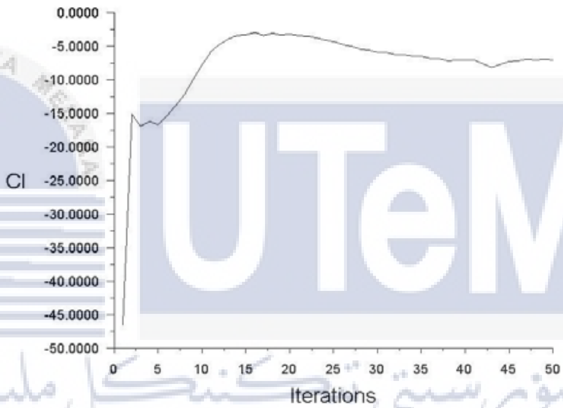
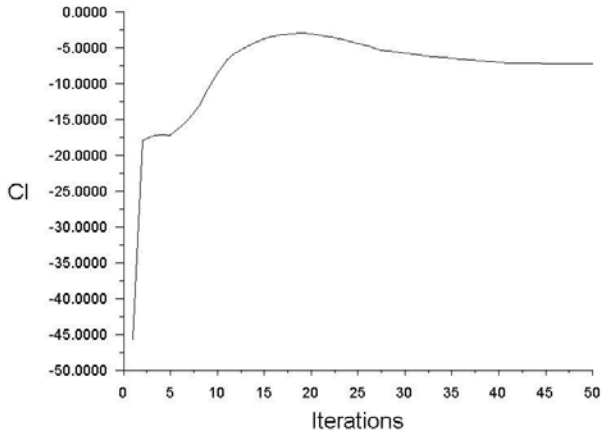


Figure 4.4: drag and lift for 12°

The highest coefficient for drag and lift for 12° is -13.5 and -3.5



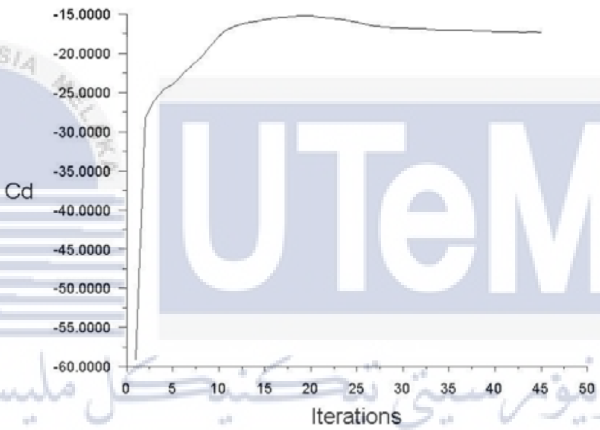


Figure 4.5: drag and lift for 16°

The highest coefficient for drag and lift for 16° is -15.5 and -3.5

4.3 Pressure Plot for Contour, Streamlines and Vector For 0°, 4°, 8°, 12° and 16°

Figure 4.6 until 4.10 shows the profile of pressure distribution in rear wing. Some of the values observes in this figure are negative on the lower surface between 0.005 and -0.0001, which indicate, as expected, the correctly intended wing operation in creating a negative lift from the airfoil geometry so as to help more in keeping the vehicle on the track. The units are in pascal.

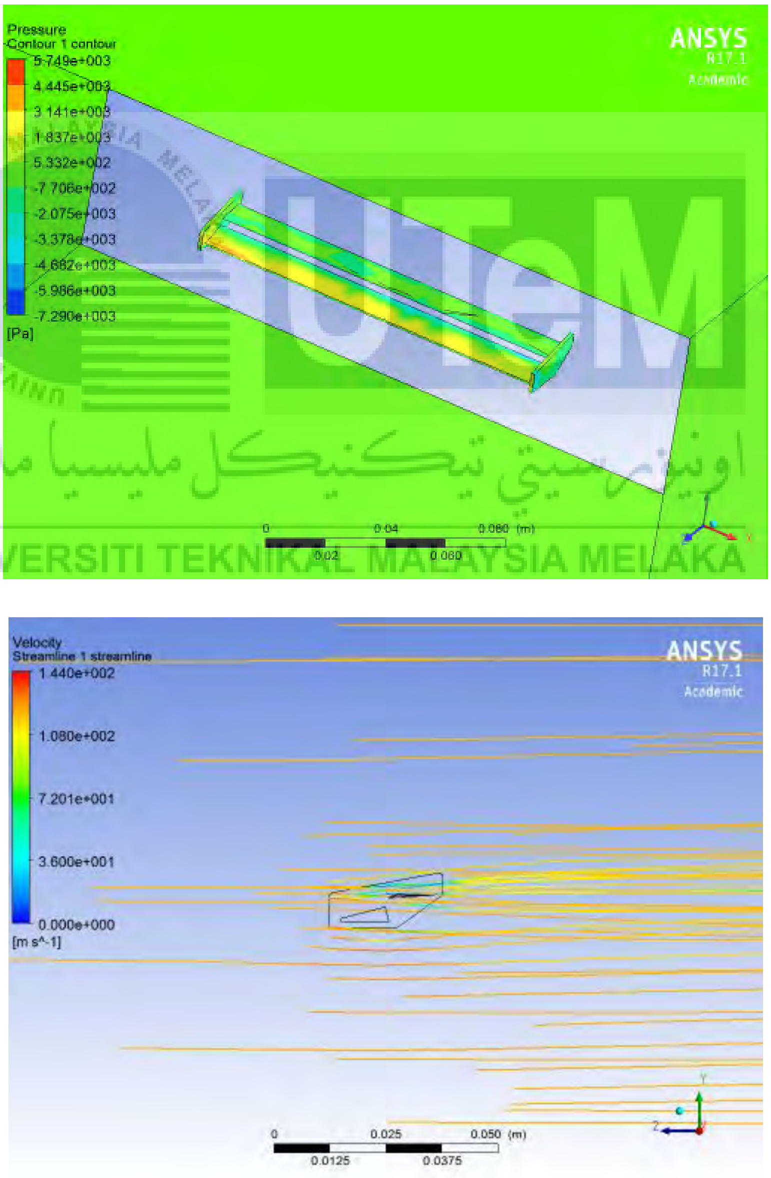


Figure 4.6: profile of pressure distribution for 0°

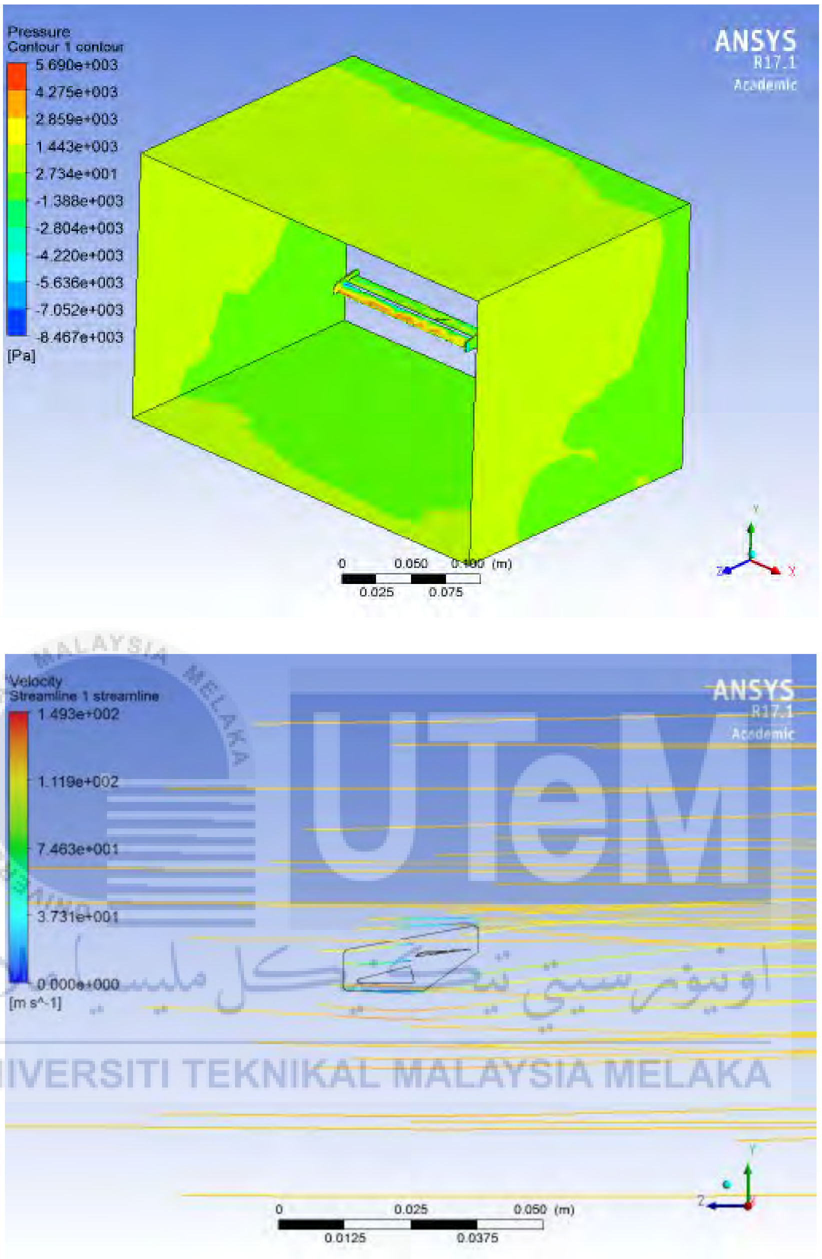


Figure 4.7: profile of pressure distribution for 4°

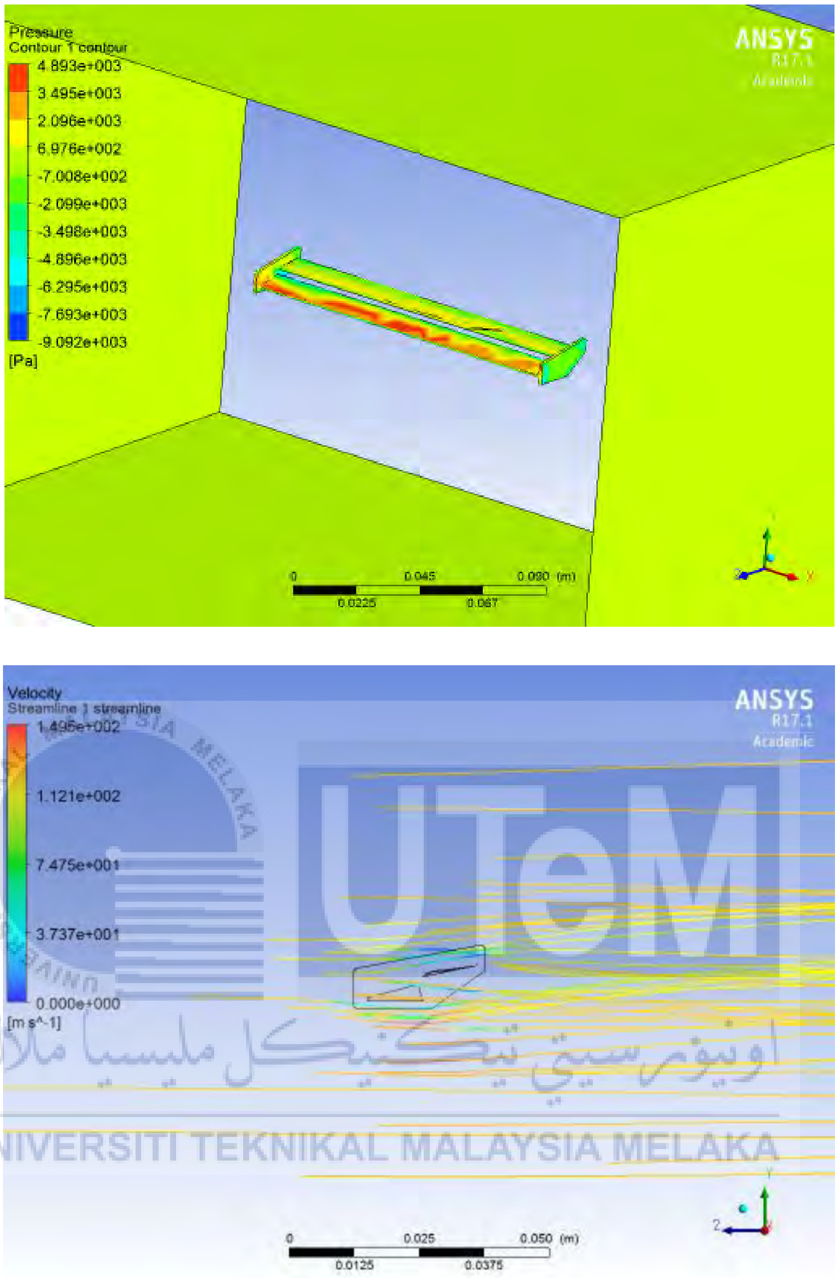
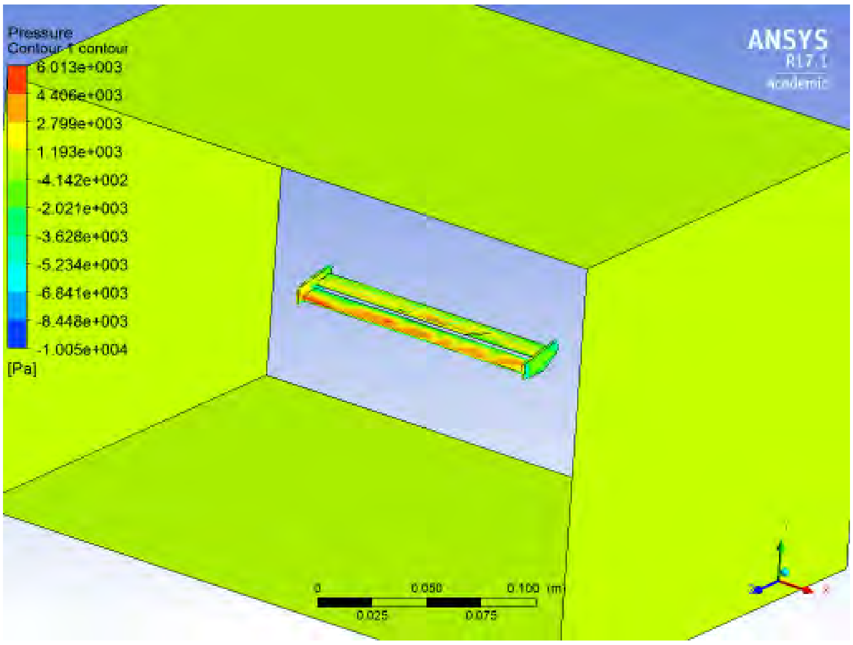


Figure 4.8: profile of pressure distribution for 8°



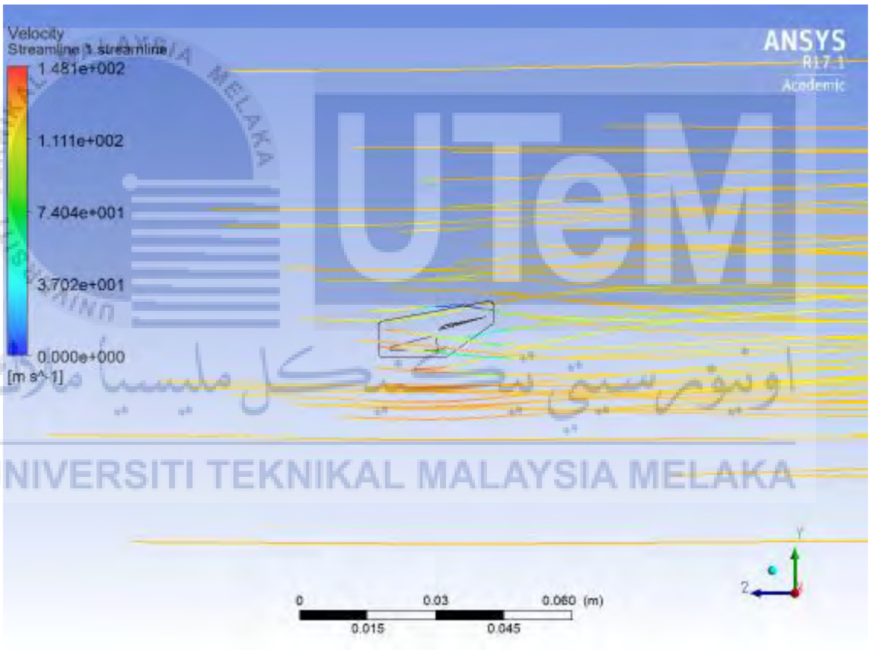


Figure 4.9: profile of pressure distribution for 12°

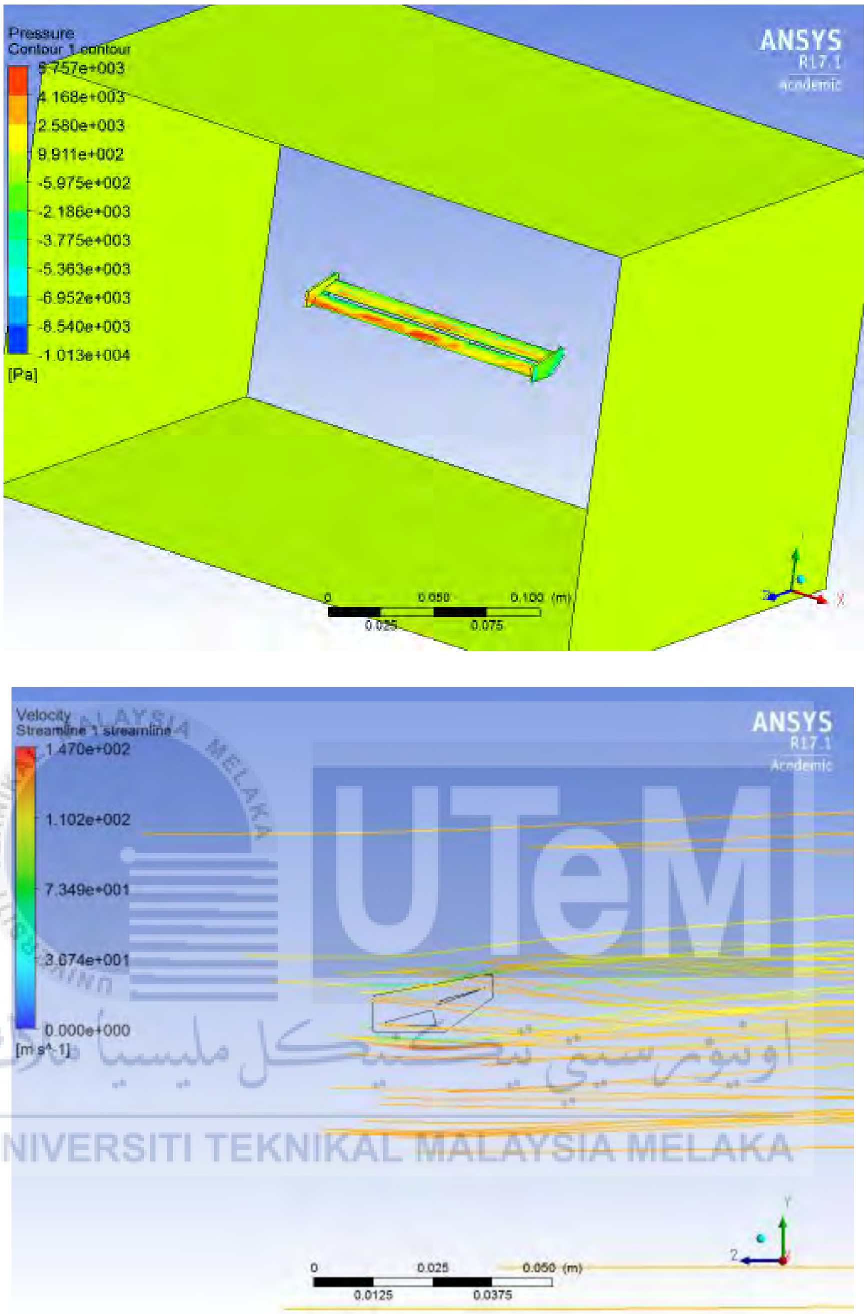


Figure 4.10: profile of pressure distribution for 16°

CHAPTER 5

DISCUSSION AND ANALYSIS

5.1 The Design of Rear Wing

The design made from multiple of aerofoil design so it can thrust or more aerodynamic. Where in journal said the designed multi winged race car can move easily through the air because of the streamlining is important for aerodynamic downforce (Katz, Joseph, 1995). The shaped of rear wing is designed like aeroplane wing shaped but vice versa because need to increase downforce and reduce lift force. Where in journal said the plane design also use a same design of aerofoil and method use in race car wing. Aerodynamic for plane is for to standard aerodynamic practices, the pressure coefficient of the subplots is presented with the negative axis upward. The area trapped by the upper and lower-surface pressure-distribution curves is equivalent to the local sectional lift coefficient (Vassberg et al, 2002).

5.2 Meshing of simulation

The meshing was made but insufficient nodes and had to reduce the nodes. This will give complicated flow problem to compute with different number of nodes. Particularly, CFD significantly reduces lead times and costs in designs and production compared to experimental-based approach and offers the capability to explain a range of complicated flow problems where the analytical approach is lacking (Breu et al, 2008). Krajnovi et al. (2004) performed LES on 250 Ahmed model for medium and fine grids. These studies were performed at low Reynolds number (2×105) to facilitate the use of LES (Singh et al, 2014).

5.3 Viscous Type for simulation

In this study will be use k-epsilon turbulence model because the most common model used in Computational Fluid Dynamic in study to simulate mean flow characteristics for turbulent flow conditions. However, viscous effects such as confluent boundary layers and wakes tend to reduce the lift generated, resulting in a value of gap below which there is a decrease in the wing CL. This study again emphasizes the importance of performing viscous computations in designing two-element airfoils for high-lift (Gopalarathnam et al, 1997)

5.4 Comparison to similar established works

The similar work by (Julianous Ketihus, 2014) from Univesiti Teknikal Malaysia Melaka (UTeM) which is almost the same. His study is about study and development of the drag reduction system for go-kart. Which is his study also affecting the drag and lift force on the go-kart when he uses DRS to reduce the lift and increasing the drag to overtaking. but he need to fabricate the DRS for the go-kart but in this study only do simulation of rear wing racing car to achieve the same objective.

5.5 Comparison to experimental results

The similar work by (Muhammad Izzat, 2017) which is investigation of drag force acting on racing car rear wing with and without drag reduction system. Where his objective to find drag force and this study objective is about drag coefficient. His result quite the same where by increasing the velocity and angle of attack, the drag force increase.

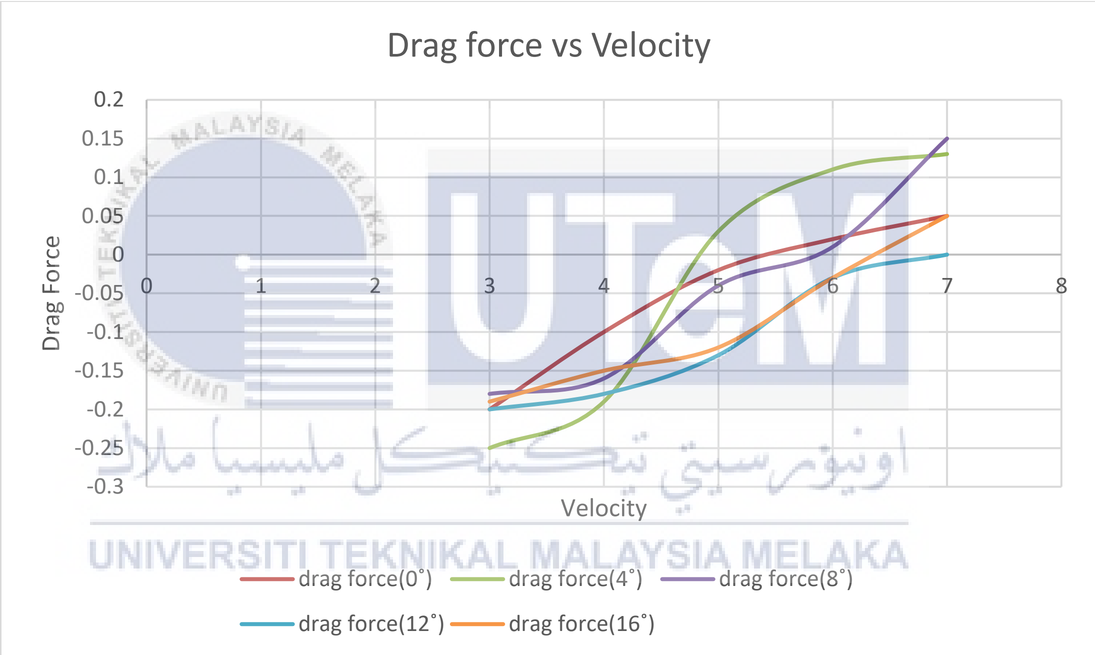


Figure 5.1: velocity effect the drag force for each degree

CHAPTER 6

CONCLUSION AND RECOMMENDATION

6.1 Conclusion

This study is to do investigate the drag coefficient effect the racing car rear wing with and without drag reduction system. Data and result that have been obtained in this study will be compared with analytical result to prove whether using the simulation on ANSYS can get same range of result with the calculation and wind tunnel experiment. This result can be more accurate if the nodes for ANSYS is unlimited to the user so the result can be more promising like study on CFD study of section characteristics of Formula Mazda race car wings by (W. Kieffer, et al, 2005).

The greatest downforce should be occurred in 12° angle of attack and this condition could be considered the stalling condition due to limitation of nodes and have reduce the mesh sizing can cause inaccurate data obtained. When comparison have been made with the experimental result, the reading not so difference because use the same model and wind tunnel also have limitation for space model to be put.

6.2 Recommendation

Recommendation for future work the nodes should be limitless so the data obtained can be more accurate and also the consideration of the thermal temperature gradient around the rear wing racing car and its effects through the air density on these coefficients should be studied.

REFERENCES

Breu, F., Guggenbichler, S., & Wollmann, J. (2008). CFD Study on Aerodynamic Effects of a Rear Wing Spoiler. Vasa, (December). Retrieved from http://medcontent.metapress.com/index/A65RM03P4874243N.pdf

Corin, R. J., He, L., & Dominy, R. G. (2008). A CFD investigation into the transient aerodynamic forces on overtaking road vehicle models. Journal of Wind Engineering and Industrial Aerodynamics, 96(8–9), 1390–1411. https://doi.org/10.1016/j.jweia.2008.03.006

Gohlke, M., Beaudoin, J. F., Amielh, M., & Anselmet, F. (2010). Shape influence on mean forces applied on a ground vehicle under steady cross-wind. Journal of Wind Engineering and Industrial Aerodynamics, 98(8–9), 386–391. https://doi.org/10.1016/j.jweia.2009.12.003

Gopalarathnam, A., Selig, M., & Hsu, F. (1997). Design of high-lift airfoils for low aspect ratio wings with endplates. AIAA Paper. Retrieved from http://arc.aiaa.org/doi/abs/10.2514/6.1997-2232
De Silva, C. W. (Ed.). 2007. *Vibration damping, control, and design.* CRC Press.

Hucho, W.-H. (1987). Aerodynamics of road vehicles, (Figure 1), 577.

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

Katz, J. (1995). Race Car Aerodynamics: Designing for Speed. Bentley Publishers. https://doi.org/800-423-4595

Kieffer, W., Moujaes, S., & Armbya, N. (2006). CFD study of section characteristics of Formula Mazda race car wings. Mathematical and Computer Modelling, 43(11–12), 1275–1287. https://doi.org/10.1016/j.mcm.2005.03.011

Mariani, F., Poggiani, C., Risi, F., & Scappaticci, L. (2015). Formula-SAE racing car: Experimental and numerical analysis of the external aerodynamics. Energy Procedia, 81, 1013–1029. https://doi.org/10.1016/j.egypro.2015.12.111

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

Muthuvel, A., Prakash, N., & John, J. G. (2014). Numerical Simulation of Drag Reduction in Formula One Cars, (March), 28–32.

Newbon, J., Dominy, R., & Sims-Williams, D. (2015). CFD investigation of the effect of the salient flow features in the wake of a generic open-wheel race car. SAE International Journal of Passenger Cars - Mechanical Systems, 8(1), 217–232. https://doi.org/10.4271/2015-01-1539

Publishers, E. S. (1986). Journal of Wind Engineering and Industrial Aerodynamics, 22 (1986) 279--289, 22, 279–289.

Ragavan, T., Palanikumar, S., Anastraj, D., & Arulalagan, R. (2014). Aerodynamic Drag Reduction on Race Cars, 1(4), 99–103.

Rakibul Hassan, S. M., Islam, T., Ali, M., & Islam, M. Q. (2014). Numerical study on aerodynamic drag reduction of racing cars. Procedia Engineering, 90, 308–313. https://doi.org/10.1016/j.proeng.2014.11.854

Road, S., & Kingdom, U. (2015). The Aerodynamic Characteristics of a Fully Deformable Formula One Wind Tunnel Tyre, 44(February). https://doi.org/10.4271/2012-01-1166

Sapienza, L. (2002). Basics of vehicle aerodynamics. Technology.

Singh, J., & Randhawa, J. S. (2014). CFD Analysis of Aerodynamic Drag Reduction of Automobile Car - A Review. International Journal of Science and Research, 3(6), 2012–2014.

Vassberg, J. C., & Jameson, A. (2002). Aerodynamic shape optimization of a reno race plane. International Journal of Vehicle Design. https://doi.org/10.1504/IJVD.2002.001993

Venkatesan, D. V., Shanjay, K. E., H, S. K., Abhilash, N. A., D, A. R., & Kumar, V. R. S. (2014). Studies on Race Car Aerodynamics at Wing in Ground Effect, 8(7), 1169–1174.

Julianous Ketihus, (2014). Study and development of the drag reduction system for go-kart. Faculty of electrical engineering, Universiti Teknikal Malaysia Melaka.