DRAG FORCE STUDY OF VEHICLES IN PLATOON ARRANGEMENT USING COMPUTATIONAL FLUID DYNAMICS (CFD)

MUHAMMAD NAJHI BIN MOHD FAUZI

A report submitted in partial fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2021

“I hereby declare that I have read through this report entitle “Drag Force Study of Vehicles in Platoon Arrangement Using Computational Fluid Dynamics (CFD)” and found that it has comply the partial fulfilment for awarding the degree of Bachelor of Mechanical Engineering with honours”

Signature : ....

Name of Supervisor : Ir. Dr. Fudhail Bin Abdul Munir.

Date : 29/6/2021

I declare that this report entitle “Drag Force Study of Vehicles in Platoon Arrangement Using Computational Fluid Dynamics (CFD)” is the result of my own research except as cited in the references. The report has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : ……………………………………

Name : Muhammad Najhi Bin Mohd Fauzi

Date : 29/6/2021

To the God, messenger of Him, my beloved parents, family and special friends.

Acknowledgement

In preparing this report, I was in contact with many people, researchers, academicians and practitioners. They have contributed towards my understanding and thought. In particular, I wish to express my sincere appreciation to my supervisor, Ir. Dr. Fudhail Bin Abdul Munir, for encouragement, guidance critics and friendship. I am also very thankful to Seminar Panel 1, Prof Madya Dr. Musthafah Bin Mohd Tahir and my Seminar Panel 2, Dr Adnan Bin Roseli for their critics, advices and motivation. Without their continued support and interest, this project would not have been same as presented here.

My fellow postgraduate students should also be recognised for their support. My sincere appreciation also extends to all my colleagues and others who have provided assistance at various occasions. Their views and tips are useful indeed. Unfortunately, it is not possible to list all of them in this limited space. I am grateful to all my family members.

Abstract

The cause of drag force vehicle accounts high fuel losses. "Fuel consumption is adversely affected due to aerodynamic forces (i.e., drag force). About 30 % to 50 % of fuel energy is lost due to aerodynamic force" (Sivaraj, 2018). We need to accept that every single car experience so high drag force that acting on its. Therefore, Platoon principle allows for a significant decrease in fuel consumption at highway speeds by reducing the aerodynamic burden on vehicles. In this project, three cars were arranged back-to-back with difference distance between each car and was simulated using ANSYS software. The distance 1.5m, 2.0m and 2.5m to be suited to the size of Ahmed model car as this project using Ahmed model for the simulation. This project discovers the effect of distance between the cars with the drag coefficient acted on the car. So, 9 cases arrangement with difference distance were studied. At the end of the study, we can conclude that lowest drag force experienced by the all three cars are at case number 3, where the distance between car 1 and 2 (X_1) is 1.5m and the distance between car 2 and 3 (X_2) is 2.5m. Drag coefficient (C_D) experienced by car 1 is only 0.1472, car 2 is 0.1857 meanwhile car 3 is 0.2177. The average drag coefficient from the case 3 (the lowest) drag coefficient recorded in this simulation is 0.1835, therefore from the standard drag coefficient of single car (0.25), we can say that it reduced about 26.8%. Therefore, it can be said that the aerodynamic study of car in platoon arrangement give a positive impact on reducing the drag coefficient of car hence contribute to fuel saving.

Abstrak

Daya tarik pada kenderaan menyebabkan kehilangan bahan bakar yang tinggi. "Penggunaan bahan bakar terjejas teruk kerana daya aerodinamik (iaitu daya tarik). Kira-kira 30% hingga 50% tenaga bahan bakar hilang kerana daya aerodinamik "(Sivaraj, 2018). Kita perlu menerima bahawa setiap kereta mengalami daya tarikan yang begitu tinggi sehingga menaikkan penggunaan bahan bakar yang tinggi. Oleh itu, susunan kenderaan secara "platoon" memungkinkan penurunan penggunaan bahan bakar dengan mengurangkan daya aerodinamik pada kenderaan. Dalam projek ini, tiga kereta disusun secara selari dengan jarak perbezaan antara setiap kereta dan disimulasikan menggunakan perisian ANSYS. Jarak 1.5m, 2.0m dan 2.5m sesuai dengan saiz model kereta Ahmed kerana projek ini menggunakan model kereta Ahmed untuk simulasi. Projek ini menemukan pebezaan jarak antara kereta bertindak sejajar dengan daya tarikan pada kereta. Oleh itu, kajian kes 9 dengan perbezaan jarak dikaji. Pada akhir kajian, kita dapat menyimpulkan bahawa daya tarikan (Drag force) terendah yang dialami oleh ketiga-tiga kereta adalah pada kes 3, di mana jarak antara kereta 1 dan 2 (X_1) adalah 1.5m dan jarak antara kereta 2 dan 3 (X_2) ialah 2.5m. Daya tarikan yang dialami oleh kereta 1 hanya 0.1472, kereta 2 ialah 0.1857 sementara kereta 3 adalah 0.2177. Purata daya tarikan daripada kes 3 (kes yang paling terendah) yang direkodkan dalam simulasi projek ini adalah 0.1835, jadi jika mengambil kira daya tarikan purata bagi sebuah kereta iaitu 0.25, dapat dilihat daya tarikan menurun sebanyak 26.8%. Oleh itu, dapat dikatakan bahawa susunan kereta secara "platoon" ini memberikan kesan positif dalam menurunkan daya tarikan sesebuah kereta seterusnya menjimatkan pengunaan bahan bakar.

Table of Content

List of tables

List of figures

CHAPTER 1

CHAPTER 2

CHAPTER 3

CHAPTER 4

List of abbreviations

PATH	Partners for Advanced Transit and Highways
ITS	Intelligent Transportation System
CFD	Computational Fluid Dynamics
Re	Reynolds number
PDE	Partial differential equations
C_D	Drag Coefficient
C_L	Lift Coefficient
P	Pressure
V	Velocity
μ	Viscosity
ρ	Density
FL	Lift Force
d	Diameter

List of Appendices

Chapter 1: Introduction

1.1 Background

With the growing number of cars on the road today, the Platoon idea was adopted to enhance traffic quality. According to Pedro [1], “a platoon includes the coordination of a local vehicle group that helps to improve road ability”. Meanwhile the main cause we need to apply the platoon formation because of the drag force acting on the car that can increase the fuel consumption of a vehicle. Also, according to Volvo Frasher[2], “the force acting against a car by the air it moves is a function of: Cd x Frontal Area x Density of Air x Speed Squared Speed clearly is an important part of the equation. At stop-and-go speeds, drag isn't a big deal, but the faster you go, the more it matters. At 70 mph, you've got four times the force working against your vehicle that you have at 35 mph.”. So, we need to accept that every single car experience so high drag force that acting on its. Therefore, Platoon principle allows for a significant decrease in fuel consumption at highway speeds by reducing the aerodynamic burden on vehicles. It is therefore important to investigate the effect of the arrangement of car on the reduction of drag and lift forces, with a view to more efficient fuel consumption.

Different institutions have published experiments on the impact of distance on the aerodynamic powers of platoon vehicles. A road trial conducted using heavy trucks fitted with ITS by California Partners for Advanced Transit and Highways (PATH) has shown technological viability for driving two trucks with a 3 m gap and three trucks with a 4 m gap. The findings showed direct fuel savings of 5% for the main truck and 10% to 15% for surrounding vehicles. Due to decreased distances between cars, effective energy saving found from the 'PATH' project was correlated with aerodynamic drag reduction.

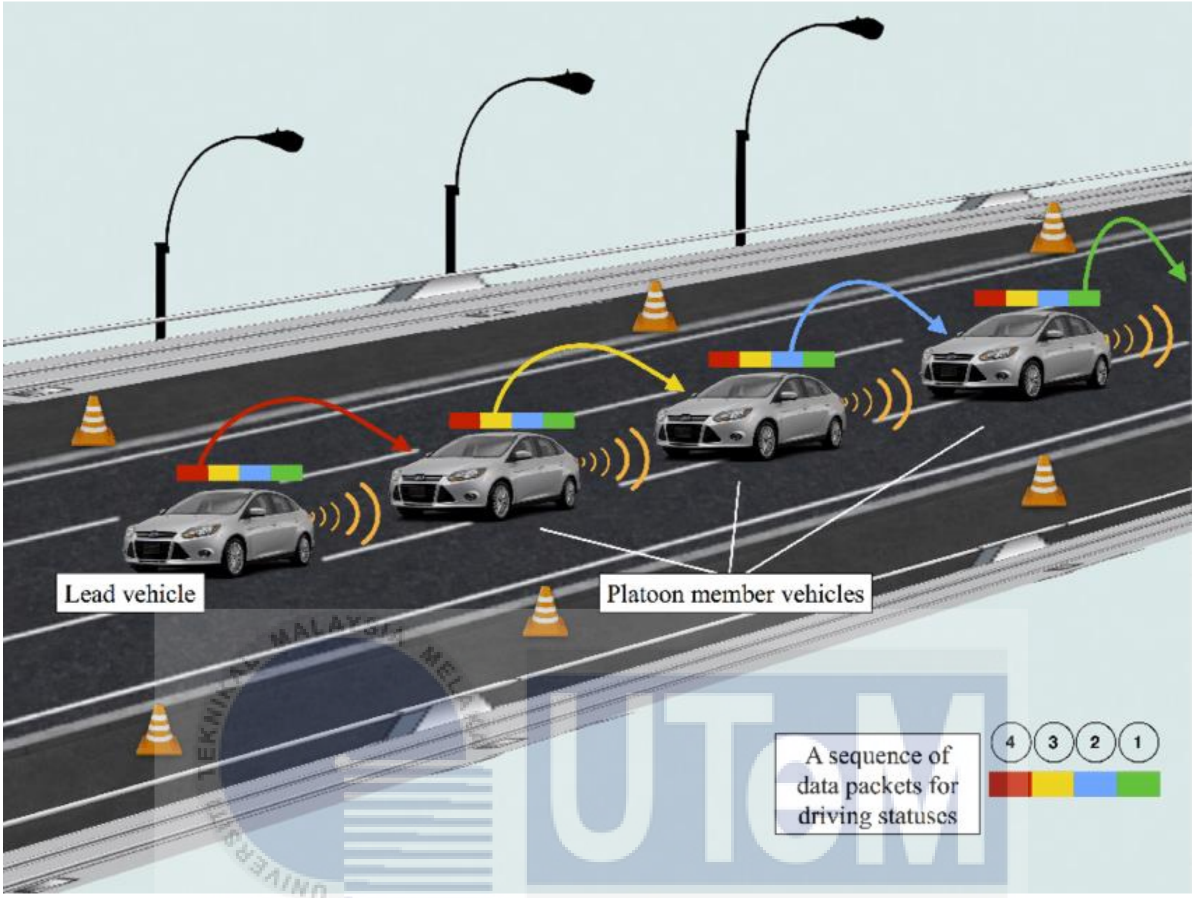


Figure 1.1: Car in platoon arrangement (Li & et al, 2018)

With today's high-speed vehicles being a widespread occurrence, reducing the lift coefficient to boost road stability is no longer a problem for race cars alone. A previous study found that because of uncertainty and traffic situations, platoons with a large number of vehicles could not be practicable and as a result, we restrict our focus to a three-vehicle platoon. Therefore, there will be two arrangements that will studied in this project which is a single arrangement of car and three-car platoon arrangement in order to compare the drag force and the fuel consumption between these two arrangements.

Computational Fluid Dynamics (CFD) seems to be the best tools in order to achieve the results for this project. CFD is use to predicting chemical reactions, heat transfer, fluid flow, mass transfer, and related phenomena by solving the mathematical models that govern these processes using a numerical process. The advantages of using CFD is the development cost can be reduced as physical experiments and test to get the data could be so expensive. It also quick and can be executed in a short period of time and we can test with different design and variations. In terms of simulation results from the three baseline models, ANSYS Fluent is conclusively recommended for CFD modeling of complicated indoor fluid environment compared with Star-CCM+ and IESVE Microflo (Ning Li, 2015). Therefore, Ansys will be used in this project to achieve the objective.

1.2 Problem statement

The cause of force aerodynamics vehicle accounts high fuel losses. “Fuel consumption is adversely affected due to aerodynamic forces (i.e., drag force). About 30 % to 50 % of fuel energy is lost due to aerodynamic force” (Sivaraj, 2018) [3]. Therefore, something need to be done to make sure these fuel losses could be reduced.

1.3 Objective

1. To study the aerodynamic pattern of car in platoon and how it can affect the reduction of drag force.
2. To study how the reduction of drag force can affect the reduction of fuel consumption.

1.4 Scope of project

The scopes of this projects are:

1. Vehicle body model using Ahmed model 1984.
2. Software utilise Ansys Fluent 2019 R3.
3. Comparison between single and three car arrangement.
4. Comparison of the drag force between subsequent arrangement.

1.5 General Methodology

The actions that need to be carried out to achieve the objectives in this project are listed below.

1. Literature review

   Journals, articles, or any materials regarding the project will be reviewed.

2. Simulation

   Simulation of the Computational Fluid Dynamics (CFD) will be made to study the aerodynamic behaviour of the car pattern.

3. Measurement

   Measure the drag force acting on the car for both car arrangements and the fuel consumption.

4. Analysis and proposed solution

   Analysis will be presented based on the results from the Computational Fluid Dynamics (CFD) simulation. Solutions will be proposed based on the analysis.

5. Report writing

   A report on this study will be written at the end of the project.

Chapter 2: Literature Review

2.1 Introduction

Nowadays, the increasing of car in the highway cause bad effect in traffic. Therefore, platoon arrangement model was invented to overcome this problem. According to Horowitz, a platoon involves organizing a vehicle group that is nearby and helps to increase road capacity (H. A. Rahman et & al, 2020). Platoon is the arrangement of car basically 2 and above. Traditionally, such platooning was primarily regarded as means of reducing the air drag acting on the vehicles, and thus fuel consumption, but there are also other benefits, like facilitating a higher level of automation. There has been much work done on controlling the vehicles inside a platoon (M. Cicic, 2019). Therefore, we can see that, by platooning it can reduce the air drag on the vehicles.

Figure 2.1: Platoon of vehicles in an automated highway system.

2.2 Types of fluid over the vehicle body

2.2.1 Streamline flows

Streamline flows in fluids is defined as the flow in which the fluids flow in parallel layers such that there is no disruption or intermixing of the layers and at a given point, the velocity of each fluid particle passing by remains constant with time.

2.2.2 Stagnation regions

In fluid dynamics, a stagnation point is a point in a flow field where the local velocity of the fluid is zero. Stagnation points exist at the surface of objects in the flow field, where the fluid is brought to rest by the object (Wikipedia, 2020). The Bernoulli equation shows that the static pressure is highest when the velocity is zero and hence static pressure is at its maximum value at stagnation points. This static pressure is called the stagnation pressure.

2.2.3 Separation bubbles

As cited from barnard 1996, when the air touches the surface of the car at some points the air doesn't perform the streamline flows it detaches. So the separation bubbles are formed in the area between the air flows separates and then reattaches. (S. Hassan, 2014).

2.2.4 Reynolds Number

The Reynolds number is the ratio of inertial forces to viscous forces. The Reynolds number is a dimensionless number used to categorize the fluids systems in which the effect of viscosity is important in controlling the velocities or the flow pattern of a fluid. Mathematically, the Reynolds number, NRe, is defined as

$$Re = \frac{\rho u l}{\mu}$$

where

ρ = density

v = velocity

d = diameter

μ = viscosity

The Reynolds number is used to determine whether a fluid is in laminar or turbulent flow. Based on the API 13D recommendations, it is assumed that a Reynolds number less than or equal to 2100 indicates laminar flow, and a Reynolds number greater than 2100 indicates turbulent flow (Rehm et &al, 2008)

2.2.5 Turbulence

Turbulence is an irregular motion of the air resulting from eddies and vertical currents. It may be as insignificant as a few annoying bumps or severe enough to momentarily throw an airplane out of control or to cause structural damage. Turbulence is associated with fronts, wind shear, thunderstorms, etc.

2.2.6 Three-dimensional flow

The road vehicle mostly has the three-dimensional flow patterns over them. Which mostly includes the circular patches, eddies and the swirling which form the three-dimensional flow patterns.

2.3 Aerodynamics Forces

2.3.1 Drag Force

A drag force is the resistance force caused by the motion of a body through a fluid, such as water or air. A drag force acts opposite to the direction of the oncoming flow velocity. This is the relative velocity between the body and the fluid.

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

Where:

C_D is the drag coefficient, which can vary along with the speed of the body. But typical values range from 0.4 to 1.0 for different fluids (such as air and water)

ρ is the density of the fluid through which the body is moving

v is the speed of the body relative to the fluid

A is the projected cross-sectional area of the body perpendicular to the flow direction (that is, perpendicular to v.

2.3.2 Lift Force

Lift is the force that directly opposes the weight of an airplane and holds the airplane in the air. Lift is generated by every part of the airplane, but most of the lift on a normal airliner is generated by the wings. Lift is a mechanical aerodynamic force produced by the motion of the airplane through the air. Because lift is a force, it is a vector quantity, having both a magnitude and a direction associated with it. Lift acts through the centre of pressure of the object and is directed perpendicular to the flow direction. Lift Force is calculated as:

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

Where;

FL = Aerodynamic Lift Force, N

ρ = air density, Kg/m3

V = velocity, m/sec

A = Frontal Area, m2

CL = Lift Coefficient

2.3.3 Laminar and turbulent flow of boundary layers.

Laminar flow or stream flow in pipes (or tubes) happens as the fluid runs in parallel layers, without interference between the layers. At low speeds, the fluid continues to flow without side mixing, and neighbouring layers slip past each other like playing cards. There are no trans-currents perpendicular to the direction of flow, not eddies or swirls of fluids. In the laminar flow, the motion of the fluid ions is very orderly, with all particles flowing in straight lines parallel to the pipe walls. Any lateral mixing (mixing at right angles to the flow direction) happens through the diffusion action between the layers of the liquid. Diffusion mixing can be gradual, but if the tube pipe diameter is limited, this diffusive mixing can be very important.

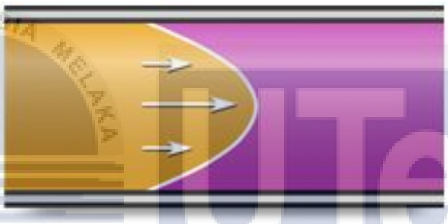


Figure 2. 2: Laminar Flow

Turbulent flow is a regime of flow characterized by turbulent changes in properties. In space and time, this entails rapid variation of pressure and flow rate. The fluid no longer travels in layers in addition to laminar flow, and mixing through the tube is highly efficient. Flows greater than 4000 are normally (but not necessarily) turbulent at Reynolds numbers, whereas those below 2300 at low Reynolds numbers generally remain laminar. Flow in the 2300 to 4000 Reynolds range, known as transition.

Figure 2.3: Turbulent Flow

2.4 Ahmed Model

As for this project, we will use Ahmed model to be use in the CFD simulation. Figure 2.4 shows the Ahmed model dimension that will be used. For the designing of this model, solid works was used. The dimensions of the Ahmed model are as follows.

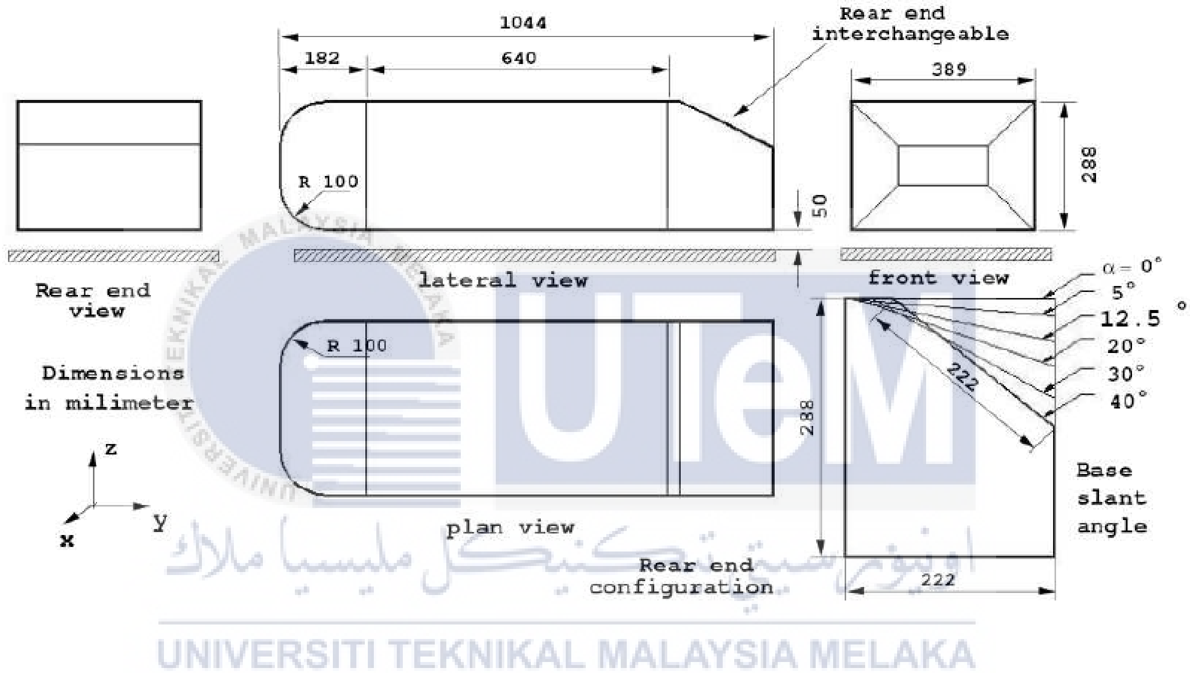


Figure 2. 4 Ahmed Model

2.5 Aerodynamics Principles

2.5.1 Navier-stroke equations

The Navier-stroke equations will describe the relationship between the friction, viscous, and momentum forces in a fluid flow. Air is known as a Newtnian fluid and the Navier-Strokes equations direct its motion as follows: (E.L, A.P, H, & T, 2013, pp. 113-115)

$$\frac{\partial}{\partial t}(\rho\, U) + \nabla.(\rho\, U'U') = \rho F' - \nabla p + \mu \nabla^2 U' + \frac{\mu}{3}\nabla \quad (\qquad \text{(Equation 5)}$$

From the above equation number (5)

ρ = density

μ = viscosity

U'= Velocity

P = Pressure

F'= Body force (Mostly gravity)

The density is essentially constant in low-speed aerodynamic applications (less than 300 mph-133 m/s), giving rise to conditions of incompressibility. By the following equation, this can be expressed:

$$\nabla. U' = \frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} + \frac{\partial W}{\partial Z} = 0 \dots\dots\dots\dots\dots\dots \text{(Equation 6)}$$

So on the basis of that the equation number (1) can be written as:

$$\rho U'.\nabla U' = \rho F' - \nabla P + \mu \nabla^2 U' \;,\;\; \nabla. U' = 0 \;\dots\dots\dots\dots\dots\dots\dots \text{(Equation 7)}$$

2.5.2 Bernoulli equation:

The Bernoulli equation is the most significant equation in the field of aerodynamics. The values of velocities and the pressure differential between the two points in the flow are compared using this equation.

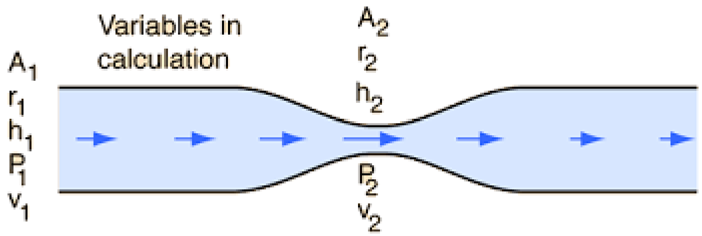


Figure 2. 5: Bernoulli flow

$P_1 + \frac{1}{2}\rho V_1^2 + \rho g h_1 = P_2 + \frac{1}{2}\rho V_2^2 + \rho g h_2$ … … … … … (Equation 8)

So

P = pressure (Pa)

V = velocity of the fluid (ms^{-1}) h = height (m) g = gravitational acceleration (ms^{-2}) ρ = fluid density (kgm^{-3})

So on the usual car speed the air density is constant which is at the usual car speed for example below than (300mph- 133ms^{-1}). The equation number (8) can be written as follows:

$\frac{P_1}{\rho} + \frac{V_1^2}{2} = \frac{P_2}{\rho} + \frac{V_2^2}{2}$ … … … … … … … … … … … ….(Equation 9)

So as a result

$\frac{P}{\rho} + \frac{V^2}{2} = Constant$ … … … … … … … … ..(Equation 10)

The equation number (10) explains the pressure difference on the two points of the fluid.

2.6 Platooning of vehicle from other researches.

2.6.1 Platoon can reduce drag force

Many researches have been made and it shows that by platooning it can reduce the drag force of subsequent car. The aerodynamic drag coefficient of the platooning vehicle is less than that of a single vehicle. When the distance between the vehicles is within the range of 0–0.5 L, the aerodynamic drag coefficient of front vehicle and the middle vehicle is more affected than that of the rear vehicle, and the aerodynamic drag coefficient decreases obviously. However, when the distance between the vehicles is more than 0.5 L, the aerodynamic drag of the rear vehicle begins to be affected more than that of the front vehicle and the middle vehicle. In particular, the aerodynamic drag of the front vehicle is close to the aerodynamic resistance of a single vehicle. (Z. F. Yang et & l, 2019).

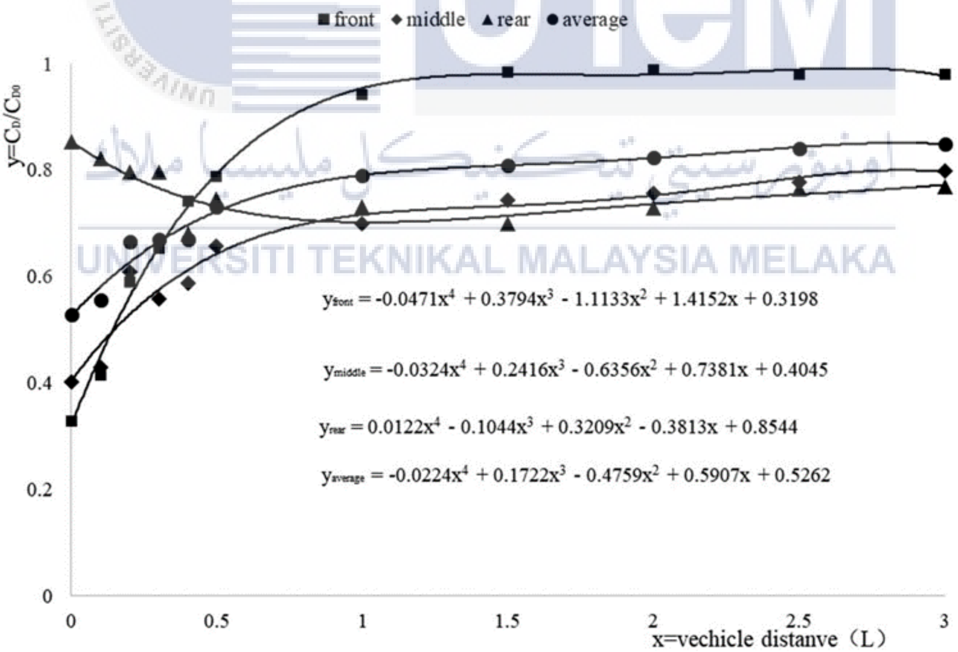


Figure 2. 6: Drag force of car in platoon (Z.F. Yang et & l, 2019)

2.6.2 Numerical analysis of drag force

The average drag coefficient of the vehicles in the platoon is defined as follows (W. Gao et & al, 2020) :

$$C_{Daverage} = \frac{1}{n}\sum_{i=l}^{n}\frac{C_{Di}}{C_{D\infty}}$$

Where $C_{Daverage}$ is the average drag coefficient of the platoon, C_{Di} is the drag coefficient of the ith vehicle in the platoon, $C_{D\infty}$ is the drag coefficient of the corresponding single vehicle, i is the ith vehicle in the platoon, and n is the total number of vehicles in the platoon.

To evaluate the drag reduction characteristics of vehicle platoons, the average drag reduction rate of a vehicle platoon is defined as follows:

$$D_R = \frac{C_{D\infty} - C_{Daverage}}{C_{D\infty}} = 1 - \frac{1}{n}\sum_{i=l}^{n}\frac{C_{Di}}{C_{D\infty}}$$

2.6.3 Platoon can reduce fuel consumption.

The effect of vehicle separation distance on fuel consumption was investigated for the two trailer configurations (standard and aerodynamic), for which the fuel-savings measurements for the individual vehicles are presented in Figure 3.1. For this vehicle speed of 105 km/h (65 mph), the corresponding time-gap axis is shows on the upper edge of the plot (B. R. McAuliffe et.al, 2017)

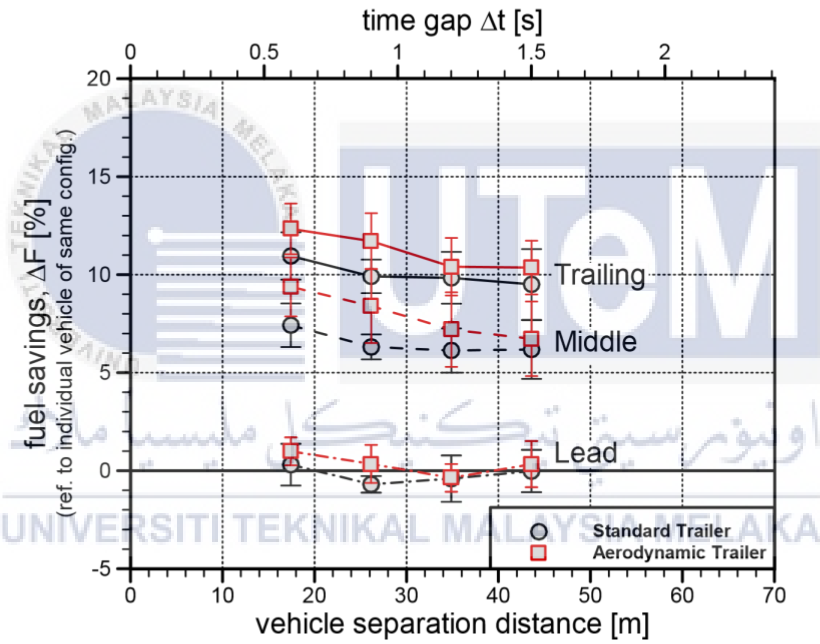


Figure 2. 7: Variation fuel-savings with separation distance

At the shorter separation distances tested, a decrease in fuel savings was observed with increasing distance. Beyond about 22 m for the standard trailer, for which the platoon averaged fuel savings was measured to be 5.2%, no significant change in fuel savings was observed. For the aerodynamic trailer configuration, no significant change was observed beyond 34 m for which the platoon-averaged fuel-savings was measured to be 5.7%. Also, the lead vehicle showed no significant fuel savings for the tested separation distances of 26 m and greater. At the shortest separation distance tested (17 m), a small fuel savings on the order of 1% was observed for some of the test conditions *(B. R. McAuliffe et.al, 2017)*

In the other research, 10% reduction in aerodynamic drag coefficient CD, this gives a 4% reduction in both fuel consumption and in CO2 emissions at a steady 80 km/h (50 mph) and at 112 km/h (70 mph), a 10% reduction in CD gives a 6% reduction in both fuel consumption and CO2 emissions. (G. L. Good, 2018)

2.7 CFD simulation

The word CFD is stand for Computational Fluid Dynamics. Computational fluid dynamics (CFD) is a science that, with the help of digital computers, produces quantitative predictions of fluid-flow phenomena based on the conservation laws (conservation of mass, momentum, and energy) governing fluid motion (Howard H. Hu, 2012). In 70's there was 1st time that the 2D simulation were used to solve the basic equations which is only apply to airplanes. (Nasira, et al., 2012).

Fluids such as (gas and fluids) measure their flux by partial differential equations, which represent the right to conserve weight, momentum and energy.'(CFD) computational fluid dynamics is the art of replacing such PDE system by the set of algebraic equations which can be solved using digital computers.' (Huerta, 2003). Qualitative and quantities fluid flow provided by the (CFD) computational fluid dynamics can predict from the flowing means:

- CFD software's like Ansys Fluent, Solid works, NX idea, solver, post processing utilities.
- Numerical methods

• Mathematical modelling

One of the main benefits of using the CFD technology is that the architects get a CFD technology the ability to build buildings in a protected environment. They are able to evaluate after creating stress, stress on the basic section that will spare a lot of their time and energies. This even saves lots of money, otherwise it was not possible in the old days. Automotive engineers can boost the aerodynamics of vehicles that can overcome several Complex equations, such as the Bernoulli equation and Navier-stroke equations. CFD-CFD it saves a lot of time to try out the real technologies very economically. Checking of the wind tunnel, which is very time consuming and expensive.

In choosing software for CFD, there are many software that offer CFD simulation such as solidwork, Ansys Fluent and Star-CCM+. After making a survey, Ansys Fluent is the best software. As cited from (Ning Li, 2015), for analyzing complicated indoor fluid environment by numerical simulation, ANSYS Fluent is recommended for its' best performance. So I choose Ansys software to do the simulation.

Chapter 3: Methodology

3.1 Introduction

This project will be conducting the CFD simulation of ahmed model that have been design using Solidwork. The CFD simulation will conducted using Ansys Fluent. The car model will be arranged in platoon arrangement and we will use 3 model of ahmed car.

The vehicles in the road commonly defined as bluff-bodies. Normally, aerodynamics researches use the ahmed model to study the basis of aerodynamics acting on the road vehicles. Watkins and Vino, 2008 stated that Ahmed model is very good to study the air flow over the vehicle body. Ahmed model was invented in 1984. Below are the ahmed model.

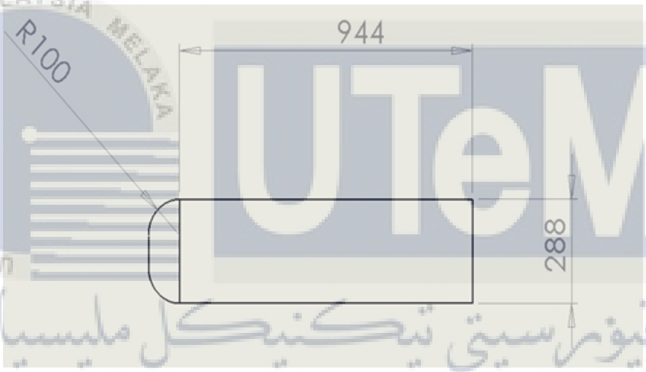


Figure 3. 1: Side dimension (mm)

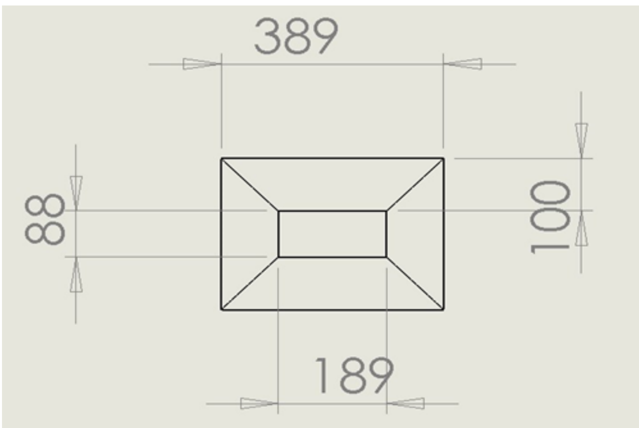


Figure 3. 2: Front dimension (mm)

3.2 Initial Method

In the initial stage the Ahmed model was completed in the CAD software Solid works as the body and the dimensions were discussed earlier. The second and the most important step is to start the Ansys Fluent simulation. It can be done by going to add in and by marking the Ansys Fluent.

The CFD simulation software chosen was Ansys Fluent 2019 R3. The main purpose for choosing this program was it has the inside flow simulation to perform the CFD technique and find the fundamentals of the basic flow other the Ahmed model. On the other hand, this software was faster to perform the CFD flow simulation tests on the Ahmed model and even better for the designing. This software is very user friendly and gives lots options like flow trajectories and the pressure surfaces and the play the flow animation which was done.

3.3 CFD simulation set up

3.3.1 Define the problems:

The 1st and the most starting of the step is to define the problem and then start to analysis that how this can be solved by using the CFD flow simulation software. The objective of the analysis is defined. So the objective of this project is to study the aerodynamic force of vehicle in platoon arrangement. So this project will be conducted using 3d model as representative for bluff body of car. Next is to create the flow domain. This defines the flow around the 3d model and which area the flow is restricted to. The viscous flow will be determined. Inlet and outlet air flow will be determined.

3.3.2 3D modelling.

As stated before, the 3d modelling will be using the ahmed model body. Figure 5 below showing ahmed body that are invented on 1984.

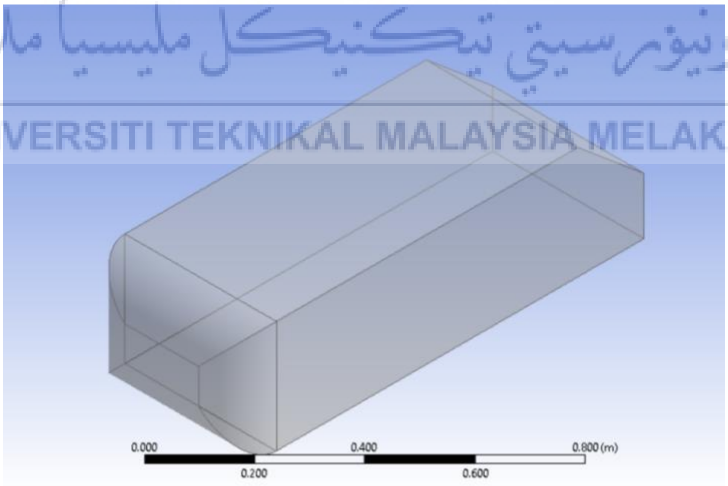


Figure 3. 3: Ahmed Car model

3.3.3 Flow domain.

The area around the 3d model is the restricted area, which is known as the computational domain. This is the boundary under which the flow simulation will take place. The general figure 23 shows the basis of the computational domain. This is the first step where only one car was placed for the simulation.

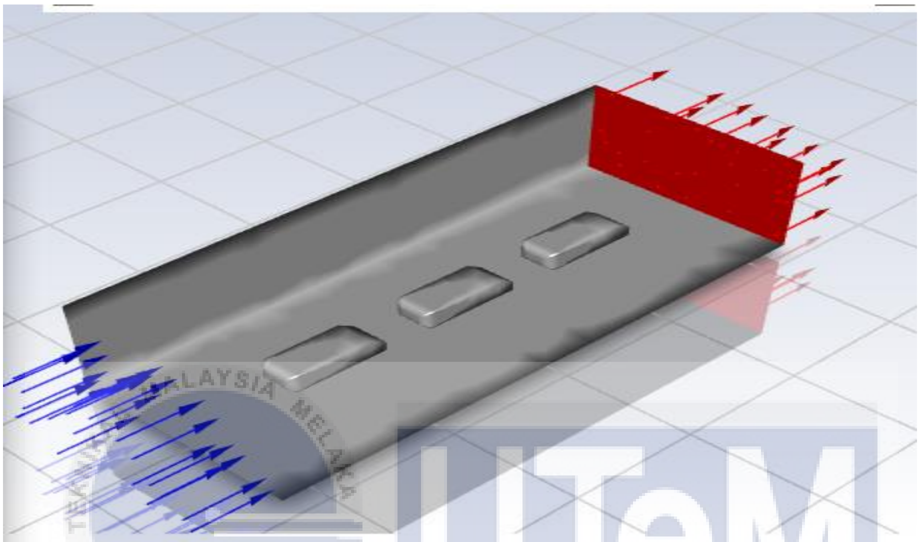


Figure 3. 4: The flow of the inlet and the outlet

After that, 3 ahmed model was arranged to be in platoon arrangement. Figure 2 below shows the symmetric view of the car arrangement after modeling and the boundary setting applied for simulation purposes.

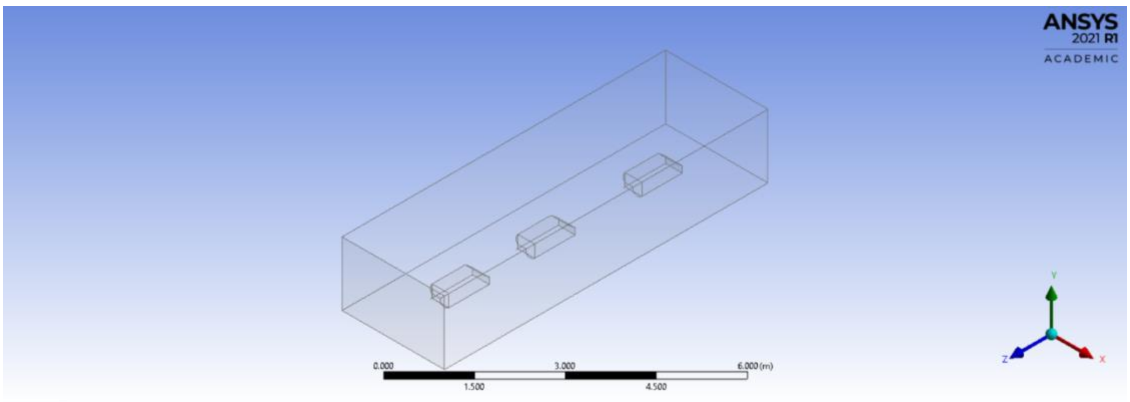


Figure 3. 5: Symmetric view of the car arrangement after modeling.

3.3.5 Creating mesh

In order to obtain accurate and time-saving simulation results, mesh independence studies were first conducted involving comparisons with experimental results by Ahmed et al. for the case of single car and for two car platoon. In the first case involving a single car, it was found that a mesh with a total 230818 elements was sufficient to get results approximating the experimental value.

If the volume regulation around the vehicle, i.e. the number of 538398 components, is not used in the car, the drag coefficient produced varies considerably from the value of the experimental data and an error of 17.96%. The cumulative mesh is increased to 724974 elements as volume control is added throughout the vehicle. With an error of just 6.98%, the drag coefficient reaches experimental significance. This error is caused by a finer mesh detail in the wake region of the rear automobile which affects the drag coefficient value. The mesh is then lifted to 914797 components to optimize the mesh around the car further. However, the drag value obtained does not vary substantially from the previous mesh setup. The mesh model

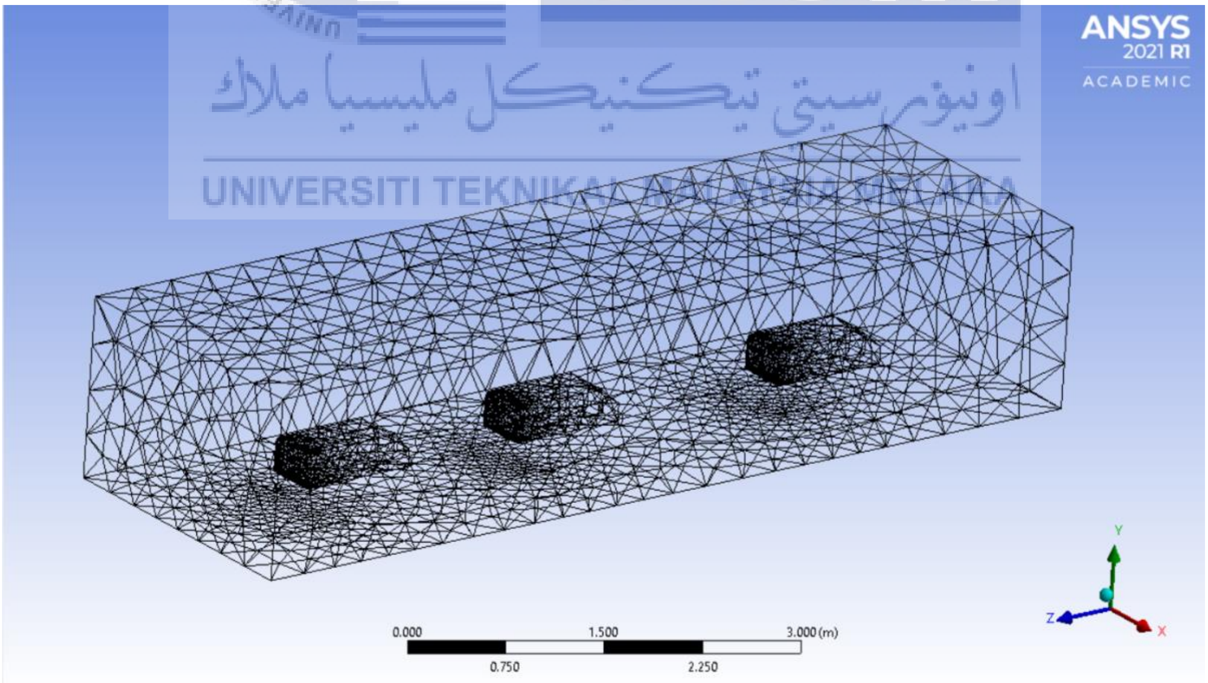


Figure 3. 6: the mesh of the car in platoon arrangement

that generates 712998 components will, however, use CFD to replicate the actual three platoon for the purposes of precision and time-saving simulation.

3.3.6 Establishing the Goals

After all the measures have been done, objectives such as lift and drag are added, which will the essence of the simulation of flow over the body was described. All the conclusions will be based on the goals offered. By solving the objectives, parameters such as cut plot, surface plot and for every pressure plot or velocity, the flow trajectories may be studied. For this one, for this plan consideration is given to speeds such as lift and drag.

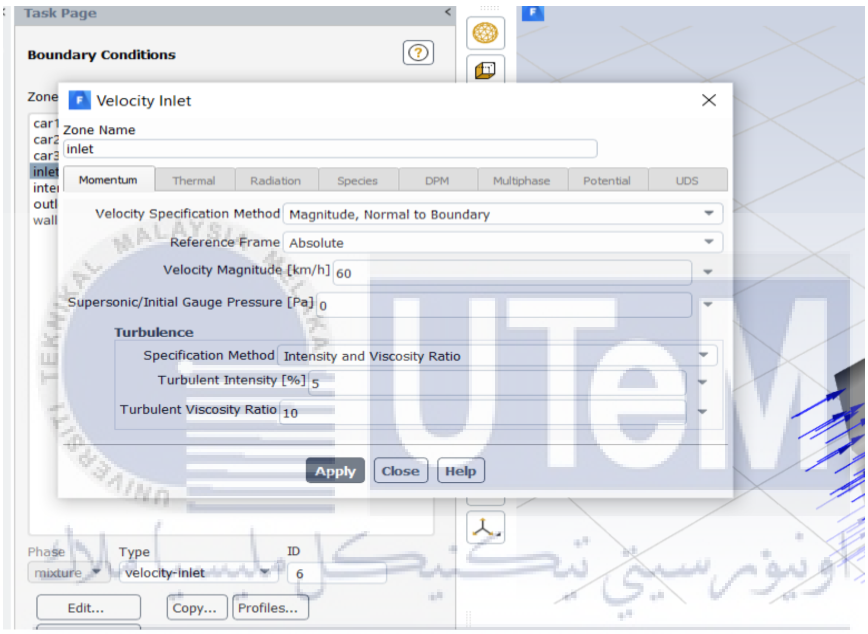


Figure 3. 7: Velocity inlet was set to 60km/h

In the boundary conditions, velocity of the air inlet was set to be 60 km/h as a simulation purpose. It is said to be the average velocity for the simulation purposes. Velocity specification method was set to magnitude, normal to boundary. The initial gauge pressure was set to be 0 Pa

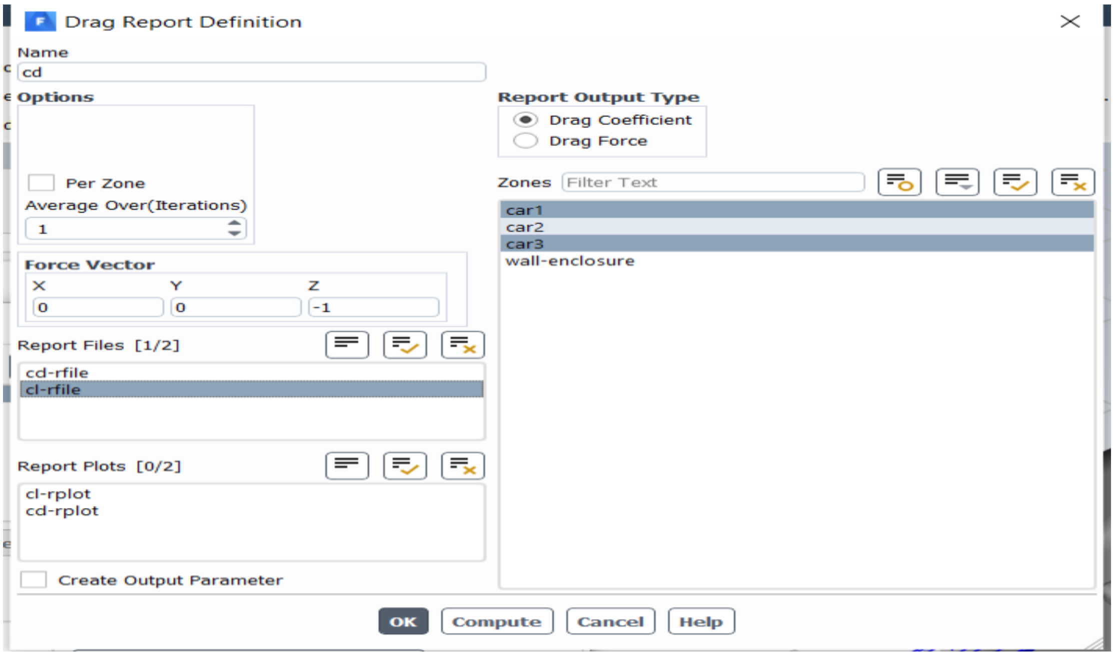


Figure 3. 8: The Drag Report Definition

Report definitions was set to calculate the drag coefficient, therefore report output will come out with the drag coefficient report after the simulation has been completed. The zones that had been picked up was on the car 1, car 2, car 3 and the force vector was set at Z-axis as the inlet velocity is coming thru the Z-axis.

3.3.7 Performing the Simulation

The program is ready to administer the tests after completing the entire phase. It is supposed to be make sure that there are ample spaces on the hard disk to store the data. Often the app presents any mistake that you need to search for the mistake and solve the problem in order to run the simulation again.

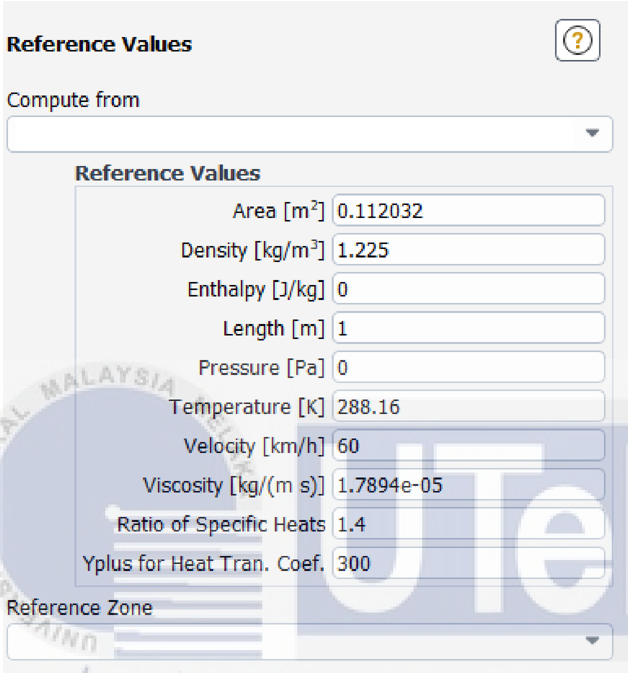


Figure 3. 9: The reference values table.

These are the reference values that had been set. The area is 0.112032 m^2 had been calculated for the surface of car on the Z-axis where the air inlet is coming from. Also, the velocity we can see is 60km/h that had been stated before.

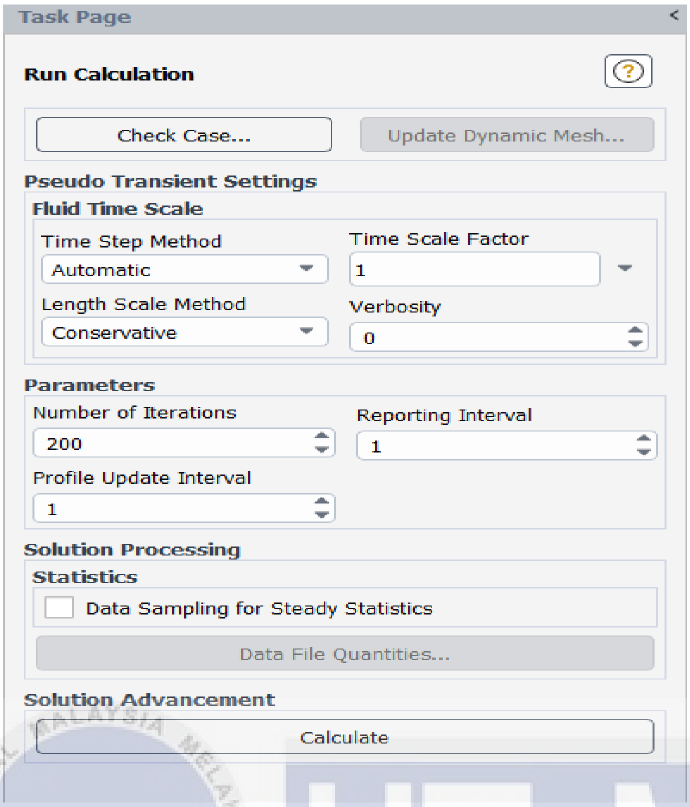


Figure 3.10: Running Calculation

After all the parameters had been set, we need to run the calculation. The number of iterations was set to 200 iterations.

3.3.8 Results of the simulation

The findings are available for viewing after the simulation is done. There are plenty of options to the reports are reviewed and recorded. The findings may be derived in the context of the term documents and plots represent the lift coefficient and the lift coefficient. A drag of each specimen.

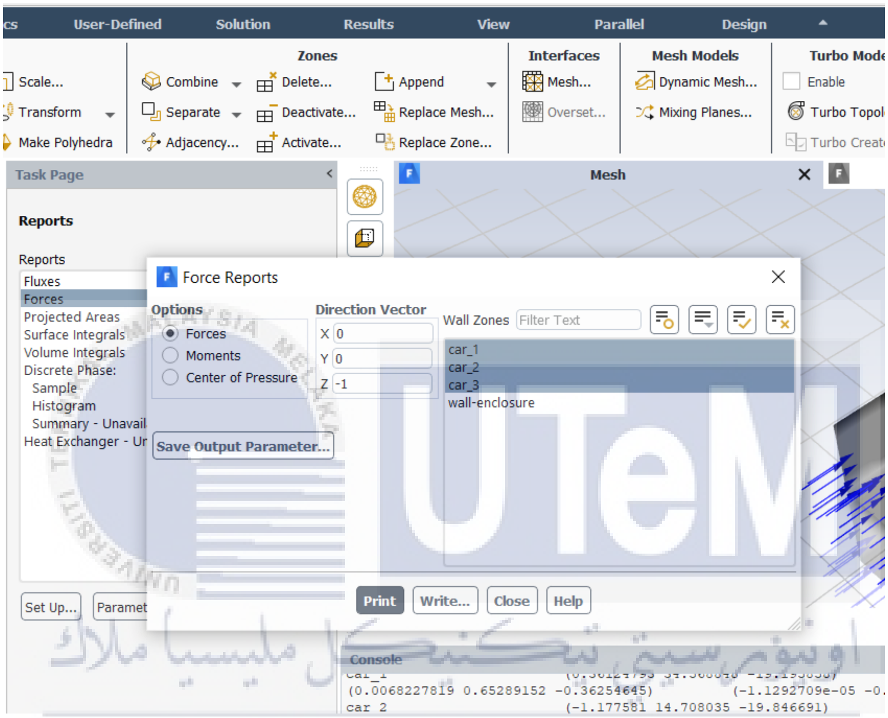


Figure 3.11: The force reports.

The force reports able us to clarify which car that we want the value of drag force and from which direction vector. Then, we can print the result to get the outcome.

Chapter 4: Result and Analysis

4.1 Introduction

This chapter explain the result and analysis of the simulation of the platoon car to find the aerodynamic force using ANSYS software. This chapter also review the result with the other previous research that had been conduct the most likely project. Three cars had been arranged to 9 cases in order to find the best platoon arrangement. This chapter also explain why the arrangement got least or highest drag force.

4.2 Results from the simulation.

In this simulation, nine (9) cases of car distance were studied for the three-car platoon. This involving combination of distances between 1.5*m*, 2.0m and 2.5*m*. X_1 represents the distance between Car 1 and Car 2 while the X_2 represents the distance between Car 2 and Car 3. The figures below are some graph from the simulation and the drag force that obtain in ANSYS.

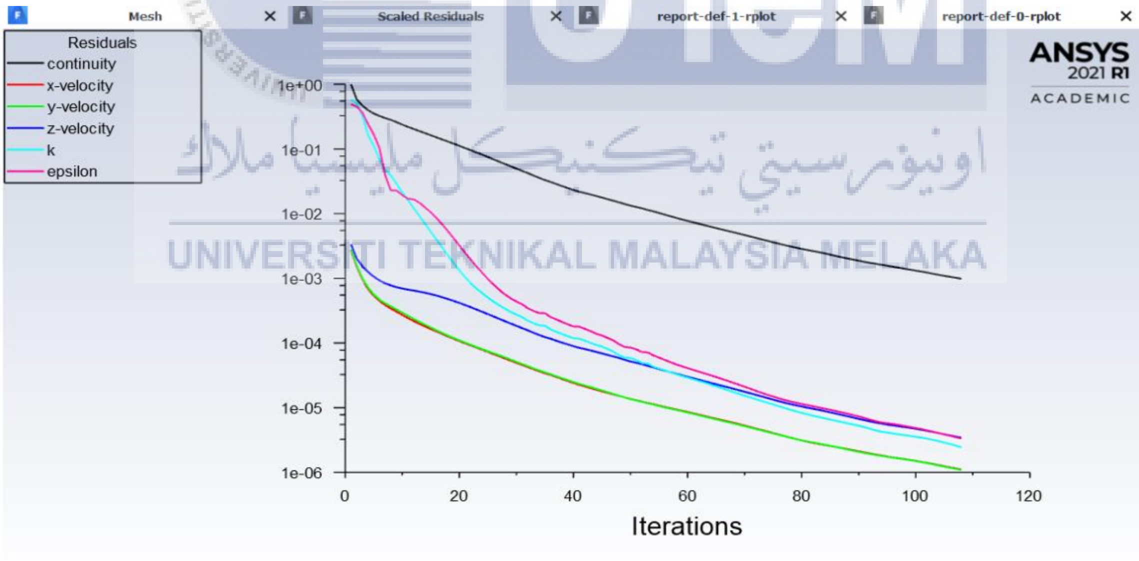


Figure 4.1: Scaled Residuals graph

This graph shows the scaled residual where it is one of the most fundamental measures of an iterative solution's convergence, as it directly quantifies the error in the solution of the system of equations. In a CFD analysis, the residual measures the local imbalance of a conserved variable in each control volume.

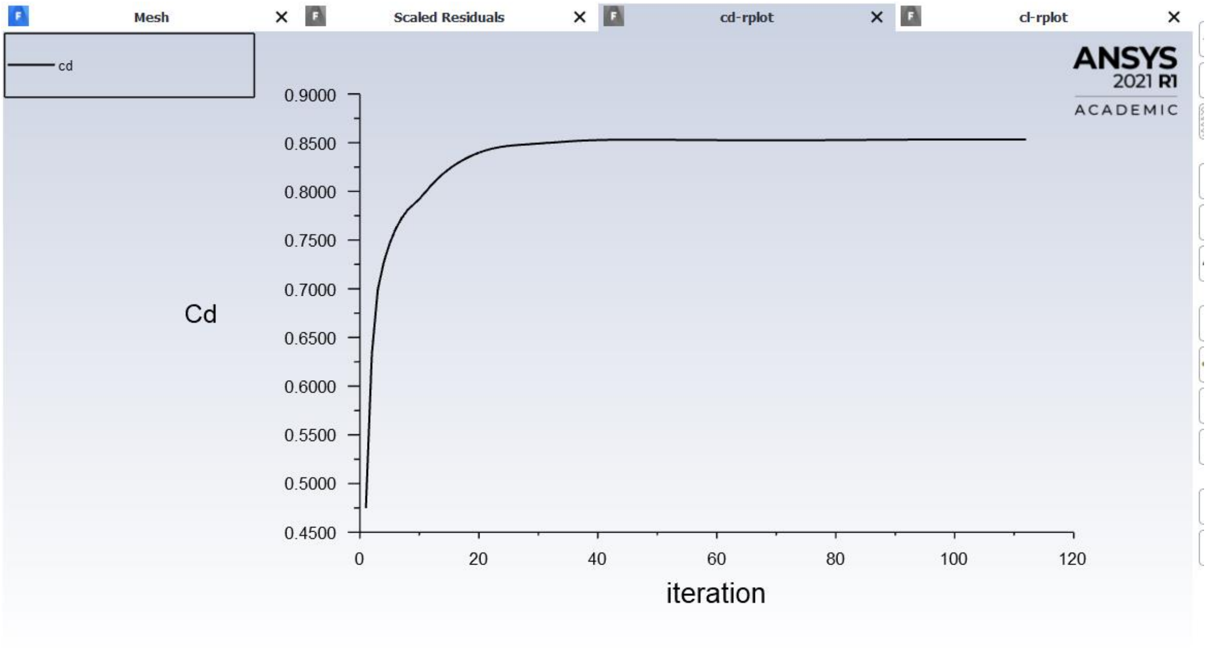
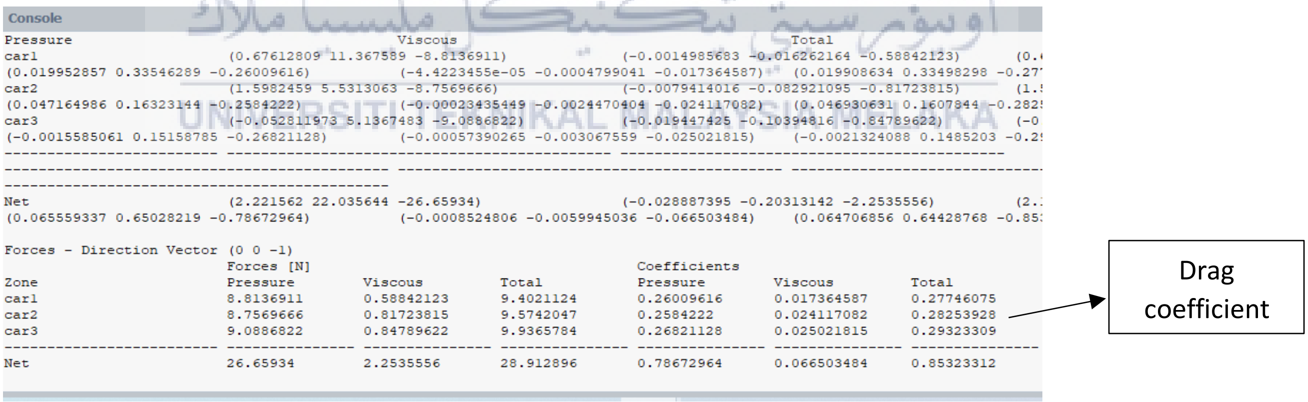


Figure 4.2: Graph of drag coefficient.

This graph shows the drag coefficient versus the iteration. So here we can monitor the drag coefficient of the car during the simulation.



```
Console
Pressure                                        Viscous                                        Total
car1                      (0.67612809 11.367589 -8.8136911)         (-0.0014985683 -0.016262164 -0.58842123)         (0.
(0.019952857 0.33546289 -0.26009616)          (-4.4223455e-05 -0.0004799041 -0.017364587)   (0.019908634 0.33498298 -0.27
car2                      (1.5982459 5.5313063 -8.7569666)          (-0.0079414016 -0.082921095 -0.81723815)         (1.
(0.047164986 0.16323144 -0.2584222)           (-0.00023435449 -0.0024470404 -0.024117082)   (0.046930631 0.1607844 -0.282
car3                      (-0.052811973 5.1367483 -9.0886822)       (-0.019447425 -0.10394816 -0.84789622)           (-0
(-0.0015585061 0.15158785 -0.26821128)        (-0.00057390265 -0.003067559 -0.025021815)    (-0.0021324088 0.1485203 -0.2
------------------------- --------------------------------------------- ---------------------------------------------
--------------------------------------------- --------------------------------------------- ---------------------------
---------------------------------------------
Net                       (2.221562 22.035644 -26.65934)            (-0.028887395 -0.20313142 -2.2535556)            (2.
(0.065559337 0.65028219 -0.78672964)          (-0.0008524806 -0.0059945036 -0.066503484)    (0.064706856 0.64428768 -0.85

Forces - Direction Vector (0 0 -1)
                          Forces [N]                                     Coefficients
Zone                      Pressure        Viscous         Total          Pressure        Viscous         Total
car1                      8.8136911       0.58842123      9.4021124      0.26009616      0.017364587     0.27746075
car2                      8.7569666       0.81723815      9.5742047      0.2584222       0.024117082     0.28253928
car3                      9.0886822       0.84789622      9.9365784      0.26821128      0.025021815     0.29323309
------------------------- --------------- --------------- -------------- --------------- --------------- --------------
Net                       26.65934        2.2535556       28.912896      0.78672964      0.066503484     0.85323312
```


Figure 4.3: Results of drag coefficient from simulation

As we choose to have drag coefficient during setting up the parameters, here we can observe the results of drag coefficient of each car

4.3 Drag Coefficient of each Car

Table 4.1: Drag Coefficient (Cd) for Car 1

Case	Car Distance 1&2 X_1 (m)	Car Distance 2&3 X_2 (m)	Drag Coefficient
1	1.5	1.5	0.2296
2	1.5	2.0	0.2187
3	1.5	2.5	0.1472
4	2.0	1.5	0.2693
5	2.0	2.0	0.2647
6	2.0	2.5	0.3149
7	2.5	1.5	0.3191
8	2.5	2.0	0.3149
9	2.5	2.5	0.3351

Table 4.2: Drag Coefficient (Cd) for Car 2

Case	Car Distance 1&2 X_1 (m)	Car Distance 2&3 X_2 (m)	Drag Coefficient
1	1.5	1.5	0.2266
2	1.5	2.0	0.2986
3	1.5	2.5	0.1857
4	2.0	1.5	0.2143
5	2.0	2.0	0.2721
6	2.0	2.5	0.2838
7	2.5	1.5	0.2180
8	2.5	2.0	0.2838
9	2.5	2.5	0.2889

Table 4.3: Drag Coefficient (Cd) for Car 3

Case	Car Distance 1&2 X_1 (m)	Car Distance 2&3 X_2 (m)	Drag Coefficient
1	1.5	1.5	0.3352
2	1.5	2.0	0.2857
3	1.5	2.5	0.2177
4	2.0	1.5	0.3345
5	2.0	2.0	0.3039
6	2.0	2.5	0.2802
7	2.5	1.5	0.2850
8	2.5	2.0	0.2802
9	2.5	2.5	0.3394

4.4 Drag coefficient of all three cars.

Table 4.4: Table 4 shows the Drag Coefficient (Cd) for all three cars.

Case	Car distance 1&2 X_1 (m)	Car distance 2&3 X_2 (m)	Drag Coefficient		
			Car 1	**Car 2**	**Car 3**
1	1.5	1.5	0.2296	0.2266	0.3352
2	1.5	2.0	0.2187	0.2986	0.2857
3	1.5	2.5	0.1472	0.1857	0.2177
4	2.0	1.5	0.2693	0.2143	0.3345
5	2.0	2.0	0.2647	0.2721	0.3039
6	2.0	2.5	0.1881	0.2053	0.1817
7	2.5	1.5	0.3191	0.2180	0.2850
8	2.5	2.0	0.3149	0.2838	0.2802
9	2.5	2.5	0.3351	0.2889	0.3394

4.5 Discussion

The average modern automobile achieves a drag coefficient of between 0.25 and 0.3. Sport utility vehicles (SUVs), with their typically boxy shapes, typically achieve a Cd=0.35–0.45. The drag coefficient of a vehicle is affected by the shape of body of the vehicle (Wikipedia, 2021). From the results that we obtain from the simulation, it shows a positive value that a slightly close to the standard drag coefficient of a single car. Also, from the previous research study there also found that the value of drag coefficient of car in platoon arrangement changing simultaneously when the distance between car is set to certain value.

H.A Rahman et.al, 2019 said that "for the first car, the lowest drag coefficient is when the distance between cars 2 and 3 is 0.5*L* (i.e. C_D = 0.1217). For the second car, the lowest drag coefficient is also when X_2 = 0.5*L* between cars 2 and 3, where C_D = 0.2004. While for the third car, the lowest drag coefficient is when X_2 = 1.5*L* between cars 2 and 3 (i.e. C_D = 0.2630). Overall for this case, the lowest drag coefficient value is on the first car i.e. when X_1 = 0.5*L* between cars 1 and 2, and X_2 = 0.5*L* between cars 2 and 3, where C_D = 0.1217." So here we can see that the value of drag coefficient change when the distance of the platoon arrangement is changed.

Here in this discussion, we will analyses for all cases for the drag coefficient and three cases where we set the X_1 is constant and X_2 where change to 1.5m, 2.0m and 2.5m. This is because we can analyze which distance shows the lowest drag coefficient in each particular distance.

4.5.1 Analysis of all cases

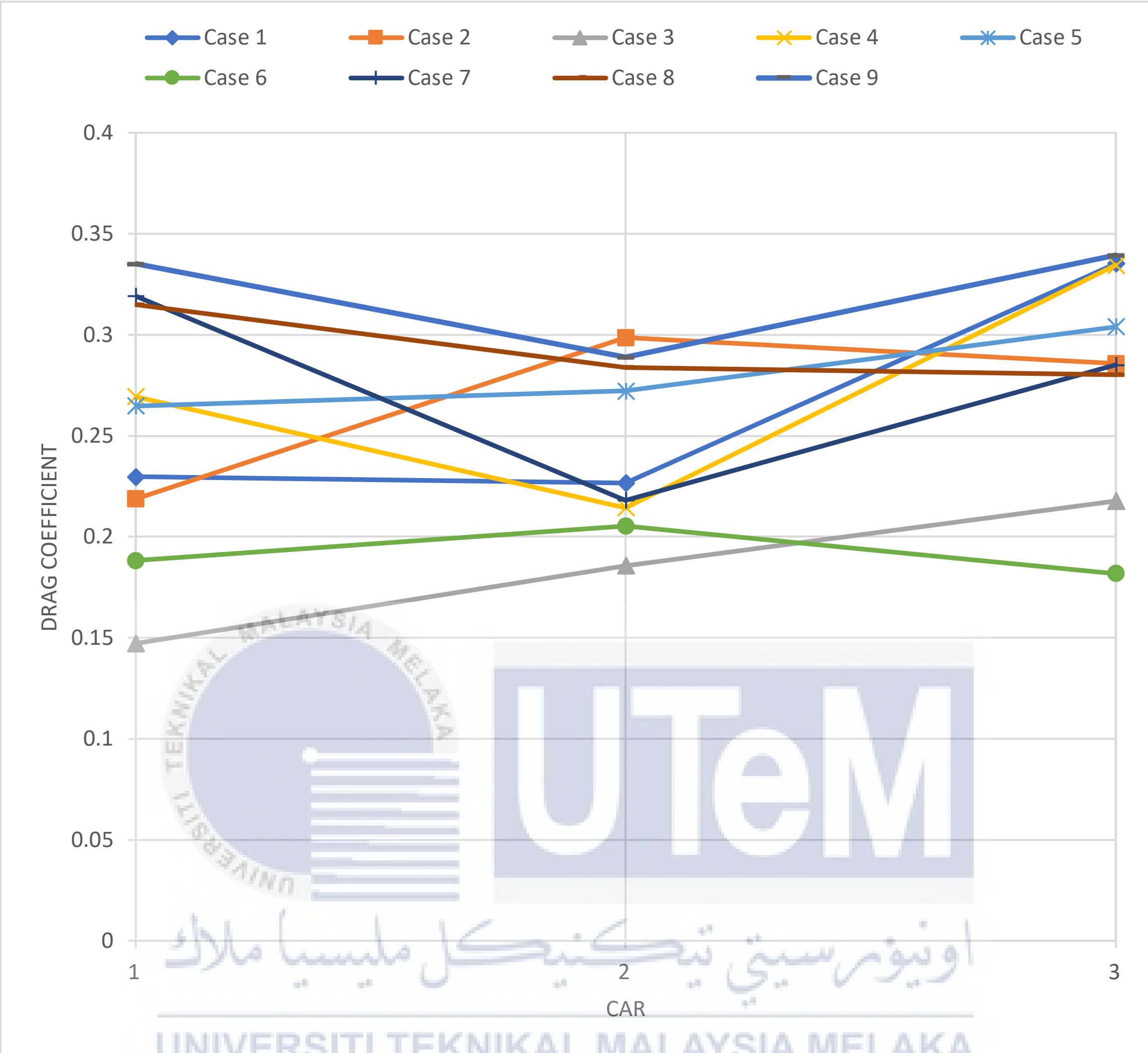


Figure 4.4: Graph of drag coefficient versus cars for all cases.

From the graph above, the car that oppose lowest drag coefficient differ by the distance set for each case. The lowest Cd for Car 1 and Car 2 is at Case 3 meanwhile for Car 3 is at case 6. For case 3, the distance between car 1 and 2 is 1.5m and the distance between car 2 and 3 is 2.5m. In general, the variations in minimum values for each case are due to differences in vehicle distance for each case studied. Furthermore, evidence reveals that when the three cars are separated by the same size, the drag coefficient increases from the first to the second to the third car.

4.5.2 Analysis of three cases car distance.

Graphs for both drag coefficient against car distance were plotted to obtain a clearer and more detailed view of how drag coefficient vary by car distance. This part of the results will be divided into three main cases. The first case is when the distance between X_1 is set at 1.5*m*. The second case when the distance of X_1 is 2.0*m* and the third case when the distance of X_1 is set at 2.5*m*. For all three cases, the value of X_2 was varied between 1.5*m*, 2.0*m* and 2.5*m*, while drag coefficient values were compared. Figures 8 and 9 below respectively show graph of drag coefficients against car distance, for the first case.

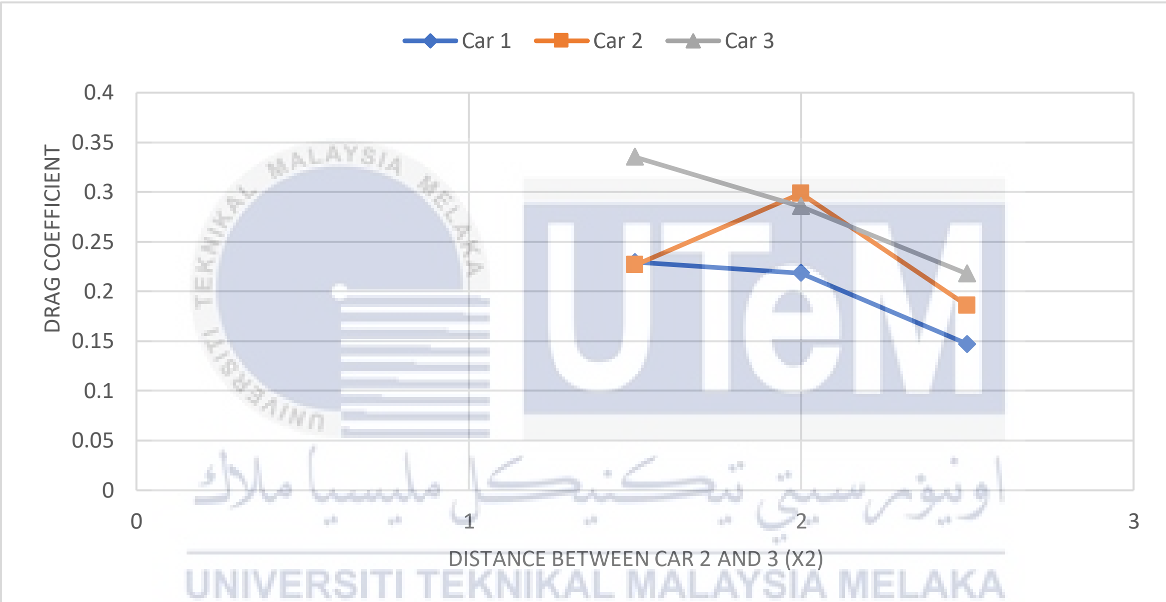


Figure 4.5: Graph of drag coefficient against car distance (Case 1-3).

X_1 = 1.5m is constant, X_2 = 1.5m, 2.0m and 2.5m.

Based on the graph above, when X_1 is set to 1.5m, the drag coefficient for car 1 is decreasing from distance 1.5m(0.2296) to 2.0m(0.2187) and 2.5m(0.1472). This also happen to car 3 where the drag coefficient is decreasing when the distance of the second car to third car is going further. But for car 2 when X_2 is 2.0m the drag coefficient is slightly high. Therefore from this result we can conclude that if X_1 is set to 1.5m the best car distance between car 2 and car 3 is X_2= 2.5m, as the drag coefficient is the lowest.

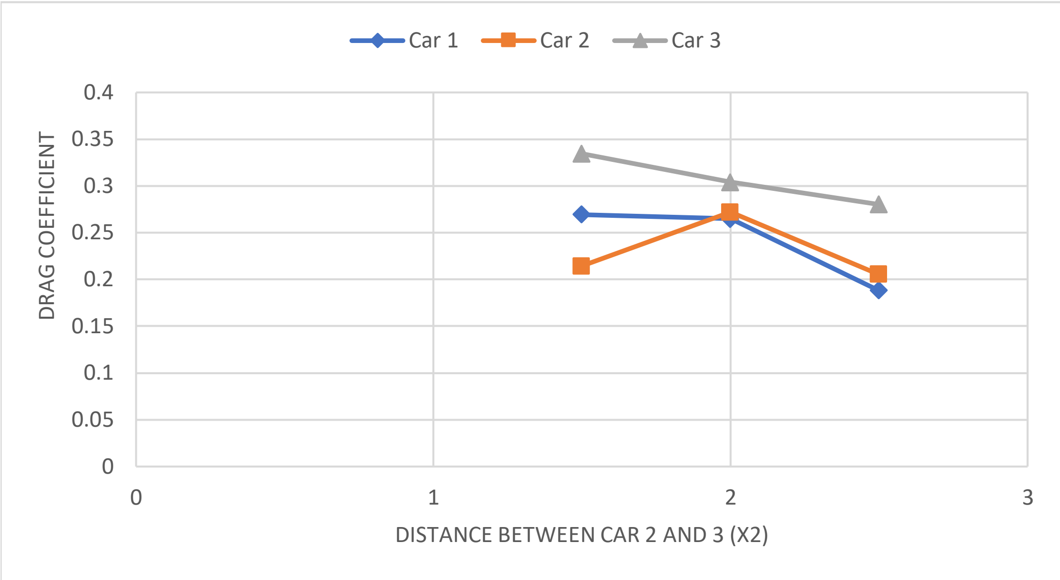


Figure 4.6: The drag coefficient graph against the car distance (Case 4-6). X_1 = 2.0m is set as constant, X_2 = 1.5m, 2.0m and 2.5m.

Meanwhile if X_1 is set to 2.0m, drag coefficient experience by car 1 decreases from X_2 = 1.5m(0.2693) to 2.0m(0.2647) and 2.5m(0.1881). From the graph also show car 3 experience the same pattern of drag coefficient. This pattern is slightly the same when X1 is constant at 1.5m. Therefore, we can conclude that if X_1 is 2.0m, the best X_2 distance is 2.5m as the drag coefficient for car 1, 2 and 3 is the lowest; car 1(0.1881), car 2(0.2053) and car 3(0.1817).

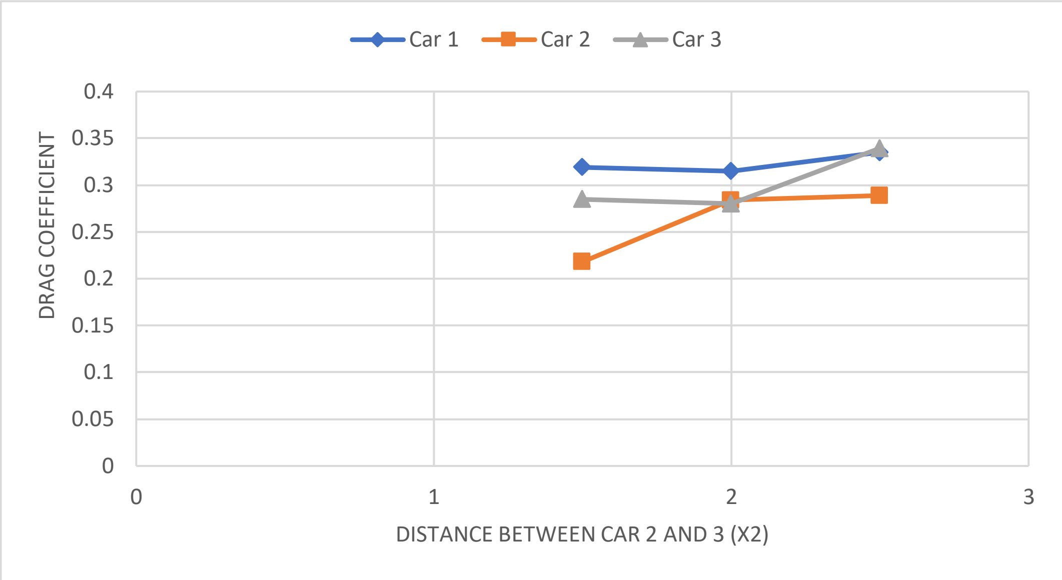


Figure 4.7: The graph of drag coefficient against car distance (Case 7-9).

X_1 = 2.5m is set as constant, X_2 = 1.5m, 2.0m and 2.5m.

For this case we can see that all three cars 1, 2 and 3 experienced increasing of drag coefficient from X_2 =1.5 to 2.0m to 2.5m. This is more likely because when the distance between the car is futher, the force acting towards each car is also higher. Meanwhile the drag coefficient value for the first car is approaching the single car drag coefficient which is higher as the distance between car 1 and 2 is much more futher. This results proves what we assuming in the very first we starting this project as when there is no platoon arrangement of car the drag coefficient is higher rather than drag coefficient of car in platoon arrangement.

4.5.3 Fluid Flow Characterization

Based on the figure 4.8, it shows car velocity streamline that can be related with the drag coefficient that acted upon the car. This figure shows case no 3 where it obtain the lowest drag coefficient for car no 1. The velocity streamline shows the vortex is weaker and could not be formed comapred to behind the second car. Thus ir reduces the kinteic energy behind the 1st car resulting low velocity. This will increase the rear pressure of first car hence the drag coefficient of the first car is reduces.

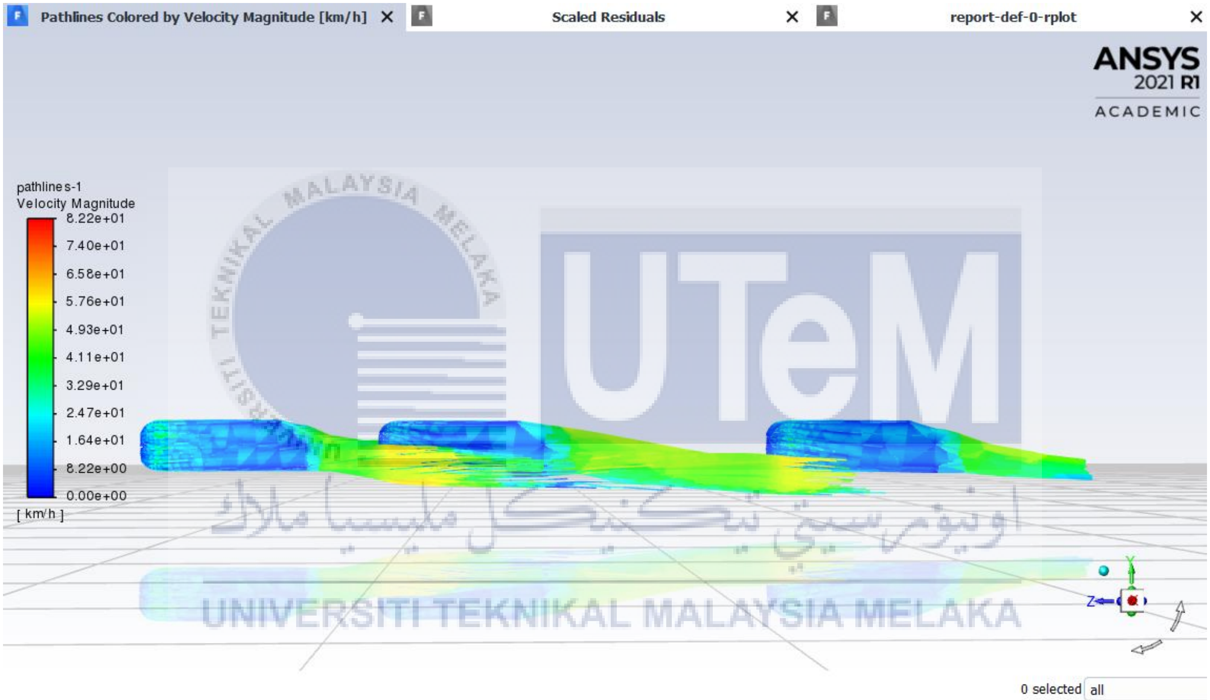


Figure 4.8: Car velocity streamline (X_1 = 1.5m, X_2 = 2.5m).

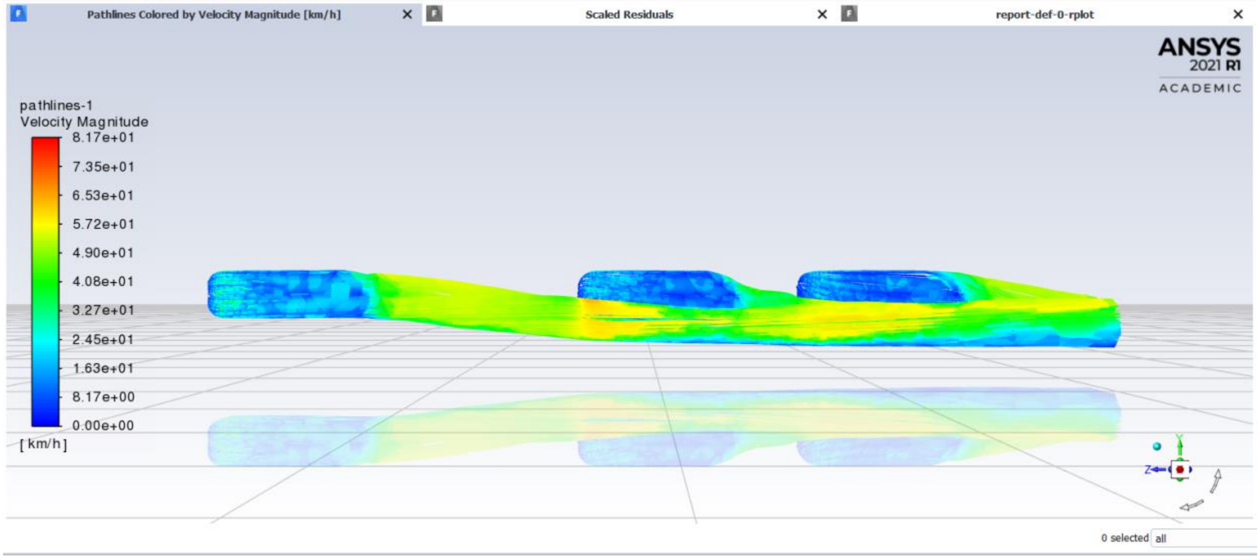


Figure 4.9: Car velocity streamline (X_1 = 2.5m, X_2 = 1.5m).

For the second car that opposess the lowest drag coefficient if we compare with its consequent car 1 and 3 is in case no 7. The Cd of car 2 is 0.2187 meanwhile car 1 and car 3 is 0.3191 and 0.2850. The high drag coefficient of car 1 and car 3 are because there is no effective intereference to disrupt the formation of vortex behind these cars thus it will increase the rotary kinetic energy hence causing the higher drag coefficient. as cited “pressure difference between the front and the rear of the trailing vehicle in the platoon is decreased, which leads to the reduction of the drag coefficient(W. Gao, Z. Deng, Y. Feng, Y. He). Meanwhile for car 2, as the presence of car 3 at the close range which is 1.5m disrupt the production of vortex whirls which reduces the resulting kinetic energy hence reducing the drag coefficient.

4.5.4 Reduction of fuel consumption

From this study, the main objective of conducting this project is to reduce the fuel consumption of car. And finally the results of the aerodynamic force which are drag force give a positive result towards the objective of this project where when car is arrange in platoon arrangement, the drag force acting on the car is observed as it change based on the distance of car 1 to car 2 and to car 3.

This change of drag force give a clear result of what distance should the platoon arrangement should be set to. As we all know when the drag force is reduced, the fuel consumption of the car is also reduced. As cited from ARC website it said that "Aerodynamic drag is proportional to the square of velocity, and hence the power needed to overcome drag is proportional to the cube of velocity. This means that there is a very strong relationship between the speed that a vehicle is travelling and the proportion of the fuel used to overcome drag". Thus our project succesfully determined which platoon arrangement is best to give the lowest drag coefficient of car.

Also as cited from Auto Research Centre website "For passenger cars, this means that aerodynamics accounts for a significantly greater proportion of fuel consumption in the highway cycle than in the city cycle: 50% in the highway and 20% in the city. This means that lowering aerodynamic drag by 10% will increase highway fuel economy by 5% and city fuel economy by 2%." Therefore, we successfully achieved the second objective which is to reduce the assumption of fuel consumption.

Calculation:

Assume, 0.25 as standard drag coefficient (Cd) and

0.1835 as the average drag coefficient from case 3(the lowest case) observed from this simulation.

Difference of drag coefficient:

0.25 – 0.1835 = 0.067

Percentage reduction of drag coefficient

$$\frac{\mathbf{0.067}}{\mathbf{0.25}} \times \mathbf{100} = \mathbf{26.8}\%$$

- So here the percentage reduction of Cd is 26.8%, therefore the fuel consumption will be reduced approximately by 13.4% for highway and 5.36% for city.

Chapter 5: Conclusions and Recommendations

As a conclusion, this project have succefully achieved its objectives which are to study the aerodynamic pattern of car in platoon and how it can affect the reduction of drag force. Also the second objective how the reduced of drag force can contribute to reduction of fuel consumption. The computational fluid dynamics using ANSYS software that are used in this project give a slight correct result as to be compared with previous researches. From this project, it was found that the lowest drag coefficient that impacts on vehicle fuel savings varied depending on car positions. It can be concluded that the for lead car ahead, the lowest drag coefficient may be observed for a distance $X_1 = 1.5m$ and $X_2 = 2.5m$ (where $C_D = 0.1472$). For the middle car, the lowest drag coefficient occurs at the same distance as the lead car which is $X_1 = 1.5m$ and $X_2 = 2.5m$, where C_D is 0.1857. Meanwhile for the last car, the lowest drag coefficient may be seen when $X_1 = 2.0m$ and $X_2 = 2.5m$ (where $C_D = 0.2630$).

Also it can be concluded where the lowest drag force experienced by the all three cars are at case number 3, where the distance between car 1 and 2 (X_1) is 1.5m and the distance between car 2 and 3 is 2.5m. Drag coefficient experienced by car 1 is only 0.1472, car 2 0.1857 meanwhile car 3 is 0.2177. Therefore, it can be said that the decreasing of drag coefficient of a car in platoon arrangement give a positive impact on platoon driving and fuel saving.

In Malaysia, this platooning system are not applied throughly yet, therefore this platooning idea should be implemented in the highway at malaysia as it can reduces the fuel consumption of car. One inisiative that can be applied is the platooning of car or truck that carry goods as an example 3 trucks that carry oil from Terengganu to Melaka, they can implement this platooning position as it can reduces the drag coefficient thus reduces the fuel consumption of the vehicles. But each of the vehicle need to communicate with each other in order to know the upcoming direction of the leading vehicle and also to prevent accident.

References

Pedro, Urbano (2010), Platooning of Autonomous Vehicles with Intervehicle Communications in SUMO Traffic Simulator, 10.1109/ITSC.2010.5625277.

Edmunds, (2009), "Improving Aerodynamics to Boost Fuel Economy", https://www.edmunds.com/fuel-economy/improving-aerodynamics-to-boost-fuel-economy.html

Sivaraj, Parammasivam, Suganya (2018), Reduction of aerodynamic drag force for reducing fuel consumption vehicle using basebleed, 10.29252/jafm.11.06.29115

Watkins, S., & Vino, G. (2008). The effect of vehicle spacing on the aerodynamics of a representative car shape. *Journal of Wind Engineering and Industrial Aerodynamics*, *96*(6–7), 1232–1239. https://doi.org/10.1016/j.jweia.2007.06.042

Ahmed, S. R., Ramm, G., & Faltin, G. (1984). Some salient features of the time-averaged ground vehicle wake. In *SAE Technical Papers*. SAE International. https://doi.org/10.4271/840300

Watkins, S., & Vino, G. (2008). The effect of vehicle spacing on the aerodynamics of a representative car shape. *Journal of Wind Engineering and Industrial Aerodynamics*, *96*(6–7), 1232–1239. https://doi.org/10.1016/j.jweia.2007.06.042

Rehm, B., Consultant, D., Haghshenas, A., Paknejad, A. S., & Schubert, J. (2008). *Situational Problems in MPD. Managed Pressure Drilling, 39–80.* doi:10.1016/b978-1-933762-24-1.50008-5

Zhi-Fa Yang, Shu-Hong Li, Ai-Min Liu, Zhuo Yu, Huan-Jing Zeng & Shi-Wu Li (2019): Simulation study on energy saving of passenger car platoons based on DrivAer model, Energy Sources, Part A: Recovery, Utilization, and Environmental Effects, DOI: 10.1080/15567036.2019.1587050

Wei Gao, Zhaowen Deng, Ying Feng & Yuping He (2020): Numerical Simulation and Analysis of Aerodynamic Characteristics of Road Vehicles in Platoon, DOI: 10.32393/csme.2020.1154

Wikipedia, (2020), " Stagnation Point at aerodynamic car " https://en.wikipedia.org/wiki/Stagnation_point#:~:text=In%20fluid%20dynamics%2C%20a%20stagnation,to%20rest%20by%20the%20object.

Weather, (2020), "Turbulance Flow of air in by ZHU training page", https://www.weather.gov/source/zhu/ZHU_Training_Page/turbulence_stuff/turbulence/turbulence.htm

Brian R. McAuliffe, Mark Croken, Mojtaba Ahmadi-Baloutaki, Arash Raeesi (2017): "Fuel-Economy Testing of a Three-Vehicle Truck Platooning System", 23-25. doi: http://doi.org/10.4224/23001922

Le Good, G., Resnick, M., Boardman, P., Clough, B. et al., "Effects on the Aerodynamic Characteristics of Vehicles in Longitudinal Proximity Due to Changes in Style," SAE Technical Paper 2018-37-0018, 2018, doi:10.4271/2018-37-0018.

Li, K., Ni, W., Tovar, E., & Guizani, M. (2018, January 18). LCD: Low latency command dissemination for a platoon of vehicles. *ArXiv*. arXiv.

Ning Li, (2015), Comparison Between Three Different Cfd Software And Numerical Simulation Of An Ambulance Hall.

E.L, H., A.P, C., H, C. S., & T, V. D. (2013). Navier-Stokes Equations. In H. E.L, C. A.P, C. S. H, & V. D. T, Aerodyanamics for Engineering students (pp. 113-115). Kidlington: Elsevier. FENTON J 1999, Advances in Vehicle Design, ISBN-10: 1860581811

Nasira, R. E., Mohamada, F., Kasirana, R., Adenana, M. S., Mohamed, M. F., Mata, M. H., & Ghania, A. R. (2012). Aerodynamics of ARTeC's PEC 2011 EMo-C Car. Procedia Engineering 41, 1775-1780. National Aeronautics and Space Administration. (2012). Factors That Affect Aerodynamics. Retrieved 03 15, 2014, from Exploration NASA: http://exploration.grc.nasa.gov/education/rocket/factord.html

Howard H. Hu, (2012) Chapter 10- Computational Fluid Dynamics, pages 421-472 Fluid Mechanics (fifth Edition), https://doi.org/10.1016/B978-0-12-382100-3.10010-1

Huerta, J. D. (2003). Finite Element Methods for Flow Problems. Sussex: John and Wiley & Sons.

Hu, J., Bhowmick, P., Arvin, F., Lanzon, A., & Lennox, B. (2020). Cooperative Control of Heterogeneous Connected Vehicle Platoons: An Adaptive Leader-Following Approach. *IEEE Robotics and Automation Letters*, *5*(2), 977–984. https://doi.org/10.1109/LRA.2020.2966412

Bello-Millán, F. J., Mäkelä, T., Parras, L., del Pino, C., & Ferrera, C. (2016). Experimental study on Ahmed's body drag coefficient for different yaw angles. Journal of Wind Engineering and Industrial Aerodynamics, 157, 140–144.

Sivaraj, G., Parammasivam, K. M., & Suganya, G. (2018). Reduction of aerodynamic drag force for reducing fuel consumption in road vehicle using basebleed. Journal of Applied Fluid Mechanics, 11(6), 1489–1495. https://doi.org/10.29252/jafm.11.06.29115

Ebrahim, H. M., Dominy, R. G., & Leung, P. S. (2017). Evaluation of vehicle platooning aerodynamics using bluff body wake generators and CFD. In 2016 International Conference for Students on Applied Engineering, ICSAE 2016 (pp. 218–223). Institute of Electrical and Electronics Engineers Inc. https://doi.org/10.1109/ICSAE.2016.7810191

Ebrahim, Hesham, Dominy, Robert, Martin, Nick (2021). Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering. 1396-1408. 10.1177/095440702096502

Li, Q., Dai, W., Yang, Z., & Jia, Q. (2020). Investigation on aerodynamic characteristics of tailing vehicle hood in a two-vehicle platoon. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, 234(1), 283–299. https://doi.org/10.1177/0954407019857430

Aerodynamic Analysis of Vehicle Using CFD. (2017). International Journal of Recent Trends in Engineering and Research, 3(3), 131–137. https://doi.org/10.23883/ijrter.2017.3056.s0sem

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

Bello-Millán, F. J., Mäkelä, T., Parras, L., del Pino, C., & Ferrera, C. (2016). Experimental study on Ahmed's body drag coefficient for different yaw angles. Journal of Wind Engineering and Industrial Aerodynamics, 157, 140–144. https://doi.org/10.1016/j.jweia.2016.08.005

Blocken, B., Toparlar, Y., van Druenen, T., & Andrianne, T. (2018). Aerodynamic drag in cycling team time trials. Journal of Wind Engineering and Industrial Aerodynamics, 182, 128–145. https://doi.org/10.1016/j.jweia.2018.09.015

Blocken, B., van Druenen, T., Toparlar, Y., Malizia, F., Mannion, P., Andrianne, T., … Diepens, J. (2018). Aerodynamic drag in cycling pelotons: New insights by CFD simulation and wind tunnel testing. Journal of Wind Engineering and Industrial Aerodynamics, 179, 319–337. https://doi.org/10.1016/j.jweia.2018.06.011

Corno, M., Bottelli, S., Tanelli, M., Spelta, C., & Savaresi, S. M. (2014). Active control of aerodynamic surfaces for ride control in sport vehicles. In IFAC Proceedings Volumes (IFAC-PapersOnline) (Vol. 19, pp. 7553–7558). IFAC Secretariat. https://doi.org/10.3182/20140824-6-za-1003.01546

Capone, A., & Romano, G. P. (2019). Investigation on the effect of horizontal and vertical deflectors on the near-wake of a square-back car model. Journal of Wind Engineering and Industrial Aerodynamics, 185, 57–64. https://doi.org/10.1016/j.jweia.2018.12.011

Hobeika, T., & Sebben, S. (2018). CFD investigation on wheel rotation modelling. Journal of Wind Engineering and Industrial Aerodynamics, 174, 241–251. https://doi.org/10.1016/j.jweia.2018.01.005

Luo, S. C., & Chen, Y. S. (2012). Ground effect on flow past a wing with a NACA0015 cross-section. Experimental Thermal and Fluid Science, 40, 18–28. https://doi.org/10.1016/j.expthermflusci.2012.01.014

Kapsalis, P. C. S., Voutsinas, S., & Vlachos, N. S. (2016). Comparing the effect of three transition models on the CFD predictions of a NACA0012 airfoil aerodynamics. Journal of Wind Engineering and Industrial Aerodynamics, 157, 158–170. https://doi.org/10.1016/j.jweia.2016.07.007

Aljure, D. E., Calafell, J., Baez, A., & Oliva, A. (2018). Flow over a realistic car model: Wall modeled large eddy simulations assessment and unsteady effects. Journal of Wind Engineering and Industrial Aerodynamics, 174, 225–240. https://doi.org/10.1016/j.jweia.2017.12.027

Fuerst Pacheco, V., & Alves de Miranda, D. (2019). Aerodynamic Analysis of High Energy Efficiency Vehicles by Computational Fluid Dynamics Simulation. Advanced Engineering Forum, 32, 41–51. https://www.scientific.net/AEF.32.41

Sharma, R. B., & Bansal, R. (2013). CFD simulation for flow over passenger car using tail plates for aerodynamic drag reduction. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), 7(5), 28-35.

Conan, B., Anthoine, J. & Planquart, P. Experimental aerodynamic study of a car-type bluff body. Exp Fluids 50, 1273–1284 (2011). https://doi.org/10.1007/s00348-010-0992-z

Heft, A., Indinger, T., and Adams, N., "Introduction of a New Realistic Generic Car Model for Aerodynamic Investigations," SAE Technical Paper 2012-01-0168, 2012, https://doi.org/10.4271/2012-01-0168.

akirlic, S., Kutej, L., Basara, B., and Tropea, C., "Computational Study of the Aerodynamics of a Realistic Car Model by Means of RANS and Hybrid RANS/LES Approaches," SAE Int. J. Passeng. Cars - Mech. Syst. 7(2):559-574, 2014, https://doi.org/10.4271/2014-01-0594

Simmonds, N., Pitman, J., Tsoutsanis, P., Jenkins, K. W., Gaylard, A., & Jansen, W. (2017). Complete body aerodynamic study of three vehicles. http://dx.doi.org/10.4271/2017-01-1529

Niu, J., Wang, Y., Li, R. et al. Comparison of Aerodynamic Characteristics of High-Speed Train for Different Configurations of Aerodynamic Braking Plates Installed in Inter-Car Gap Region. Flow Turbulence Combust 106, 139–161 (2021). https://doi.org/10.1007/s10494-020-00196-0

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

Altaf, A., Omar, A. A., & Asrar, W. (2014). REVIEW OF PASSIVE DRAG REDUCTION TECHNIQUES FOR BLUFF ROAD VEHICLES. IIUM Engineering Journal, 15(1). https://doi.org/10.31436/iiumej.v15i1.477

Mukut, A. N. M. M. I., & Abedin, M. Z. (2019). Review on Aerodynamic Drag Reduction of Vehicles. International Journal of Engineering Materials and Manufacture, 4(1), 1-14. https://doi.org/10.26776/ijemm.04.01.2019.01

Nuszkowski, J., Smith, H., McKinney, M., McMahan, N., Wilder, B., Boehringer, E., … Parker, K. (2017). Increasing the on-road fuel economy by trailing at a safe distance. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, 231(9), 1303–1311. https://doi:10.1177/0954407017703233

Gnatowska, R., & Sosnowski, M. (2018). The influence of distance between vehicles in platoon on aerodynamic parameters. EPJ Web of Conferences, 180, 02030. https://doi:10.1051/epjconf/201818002030

Appendices

Appendix A: Ansys Report

Title

Generic Report

Date

2021/06/18 19:05:00

Contents

2. Mesh Report

Table 2. Mesh Information for FFF 3

Domain	Nodes	Elements	Tetrahedra	Wedges	Pyramids	Hexahedra	Polyhedra
enclosure	14792	80353	0	0	0	0	80353

Table 3. Mesh Statistics for FFF 3

Domain	Minimum Face Angle	Maximum Face Angle	Maximum Edge Length Ratio	Maximum Element Volume Ratio	Connectivity Range	
enclosure	22.2369 [degree]	123.042 [degree]	2.5237	20.3423	3	40

3. Physics Report

Table 4. Domain Physics for FFF 3

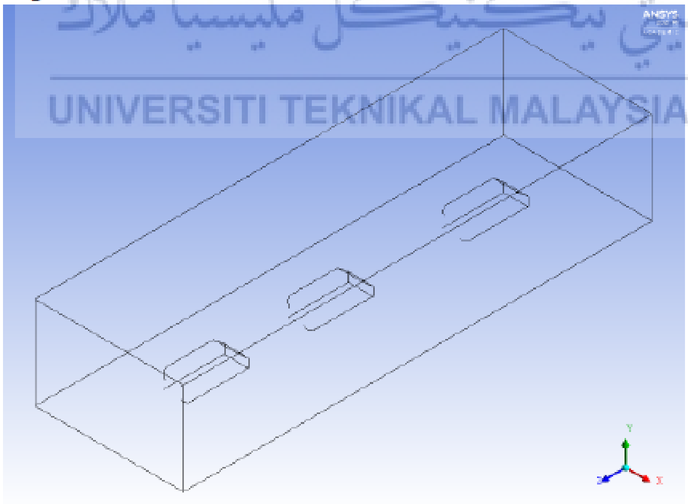
Domain - enclosure	
Type	FLUID
Location	enclosure

Table 5. Boundary Physics for FFF 3

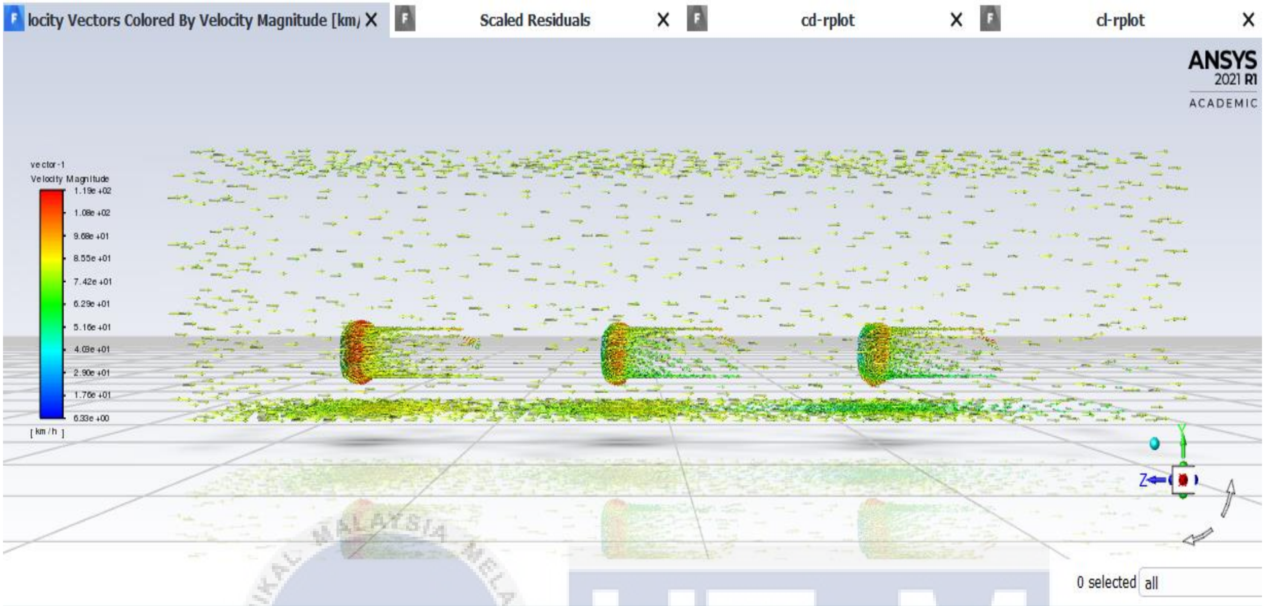
Domain	**Boundaries**	
enclosure	**Boundary - car1**	
	Type	WALL
	Location	car1
	Boundary - car2	
	Type	WALL
	Location	car2
	Boundary - car3	
	Type	WALL
	Location	car3
	Boundary - inlet	
	Type	VELOCITY-INLET
	Location	inlet
	Boundary - outlet	
	Type	PRESSURE-OUTLET
	Location	outlet
	Boundary - wall enclosure	
	Type	WALL
	Location	wall-enclosure

4. Pictures

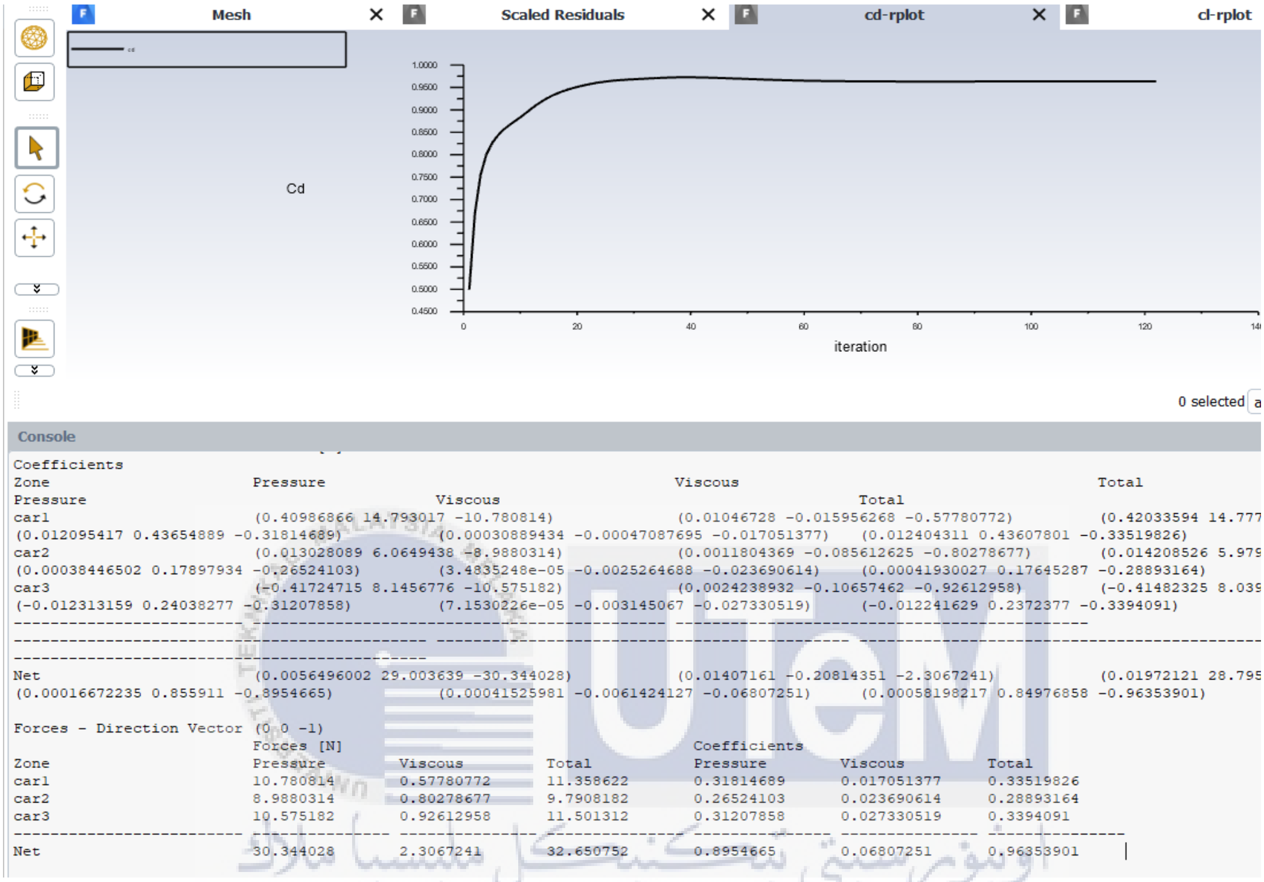
Figure 1. Iso View of Wireframe

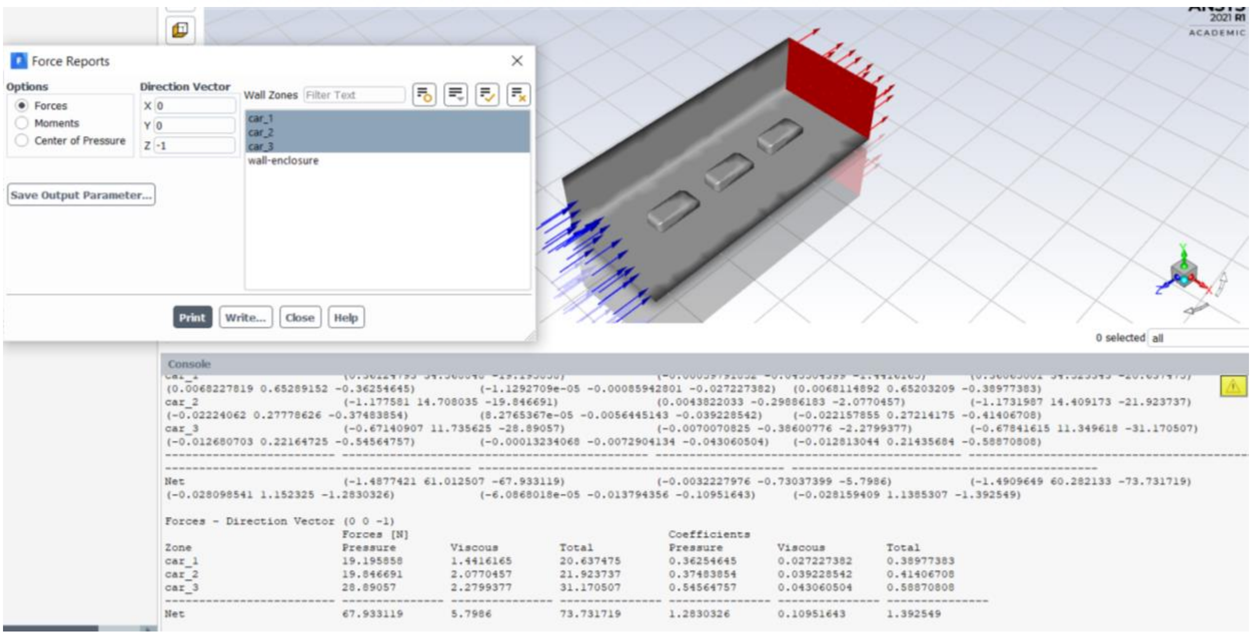


Appendix B: Velocity Vectors from ANSYS Simulation



Appendix C: Force data from ANSYS simulation.





Appendix D: Graph from ANSYS simulation