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.

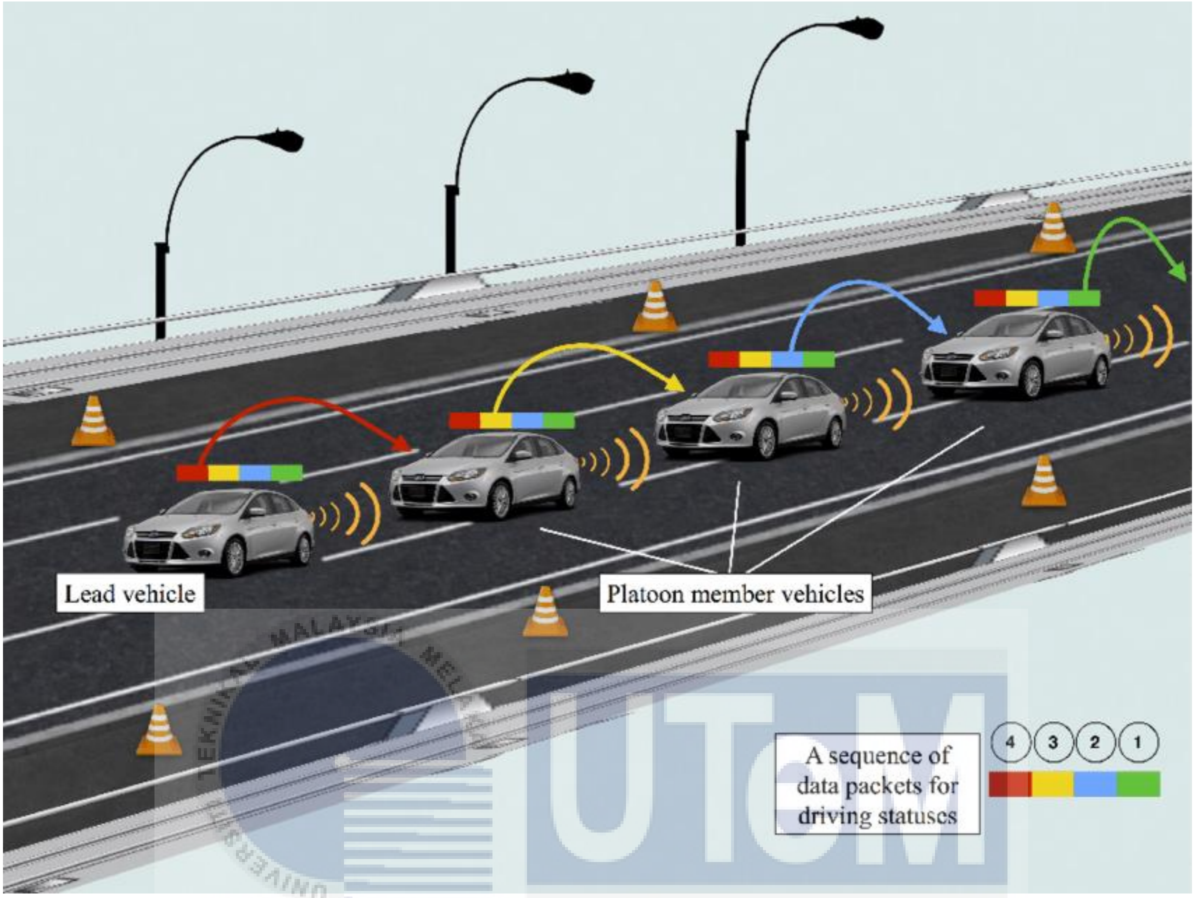

Figure 1.1: Car in platoon arrangement (Li & et al, 2018)

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