UNIVERSITI TEKNIKAL MALAYSIA MELAKA

THE OPTIMIZATION OF PRESSURE DIFFERENT REAR SUCTION AIRFLOW CORRECTION DEVICE BASED ON THE RATIO INLET OVER OUTLET IN MOTIVATION OF AERODYNAMIC DRAG REDUCTION

This report submitted in accordance with requirement of the Universiti Teknikal Malaysia Melaka (UTEM) for the Bachelor's Degree in Mechanical Engineering Technology (Automotive Technology) with Honours

By

MUHAMMAD AIMAN AFIQ BIN MOHD JAILANI

B071310844

940617 – 05 – 5213

FACULTY OF ENGINEERING TECHNOLOGY

2016

DECLARATION

I hereby, declared this report entitled "The Optimization Of Pressure Different Rear Suction Airflow Correction Device Based On The Ratio Inlet Over Outlet In Motivation Of Aerodynamic Drag Reduction" is the results of my own research except as cited in references.

Signature	:	…………………………………………
Author's name	:	…………………………………………
Date	:	…………………………………………

APPROVAL

This report is submitted to the Faculty of Engineering Technology of UTeM as a partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering Technology (Automotive Technology) with Honours. The member of the supervisory is as follow:

………………………

(Project Supervisor)

ABSTRAK

Salah satu bahagian yang paling penting dalam aerodinamik kenderaan adalah pengurangan seretan. Tekanan seretan menyumbang sehingga 80% daripada jumlah seretan dan ia secara langsung berkaitan dengan geometri bentuk kenderaan. Drag yang dicipta oleh tekanan yang berbeza di bahagian hadapan dan belakang kenderaan adalah isu utama dalam semua segmen kenderaan termasuk segmen badan besar kerana segmen ini menggunakan sejumlah besar bahan api. Sedutan aliran udara di bahagian belakang kenderaan itu boleh mengurangkan kawasan bangun di mana tekanan di rantau ini meningkat. kesan Venturi adalah dicadangkan dari prinsip Bernoulli. Pembukaan kecil di kawasan masuk halaju tinggi tekanan rendah dan pada kadar hasil di kawasan keluar halaju rendah tekanan tinggi. Sedutan ke dalam salur masuk mengurangkan kawasan bangun di belakang model manakala di bahagian salur keluar udara yang meniup keluar dari bentuk badan model menyebabkan tekanan udara venturi meningkat pada rantau bangun. Objektif kajian ini untuk melaksanakan bentuk venturi di bahagian belakang badan yang boleh mengubah tekanan yang berbeza dan mengurangkan pekali seretan (Cd) badan besar yang digunakan kes ini ialah badan Ahmed yang diubahsuai. Reka bentuk dan simulasi model akan dibantu dengan menggunakan reka bentuk bantuan komputer (CAD) dan perisian dinamik bendalir pengiraan (CFD). Kemudian, penggunaan bahan api juga dikira untuk mengetahui perbezaan peratusan yang boleh didapatkan dengan melaksanakan reka bentuk ini kepada badan besar dan boleh mengubah trend pada rekabentuk badan besar dalam dunia sebenar.

ABSTRACT

One of the most significant part in aerodynamic of vehicle is drag reduction. Pressure drag contribute up to 80% of the total drag and it is related directly to geometry shape of vehicle. Drag that created by pressure different at the front and rear of vehicle are the major issue in all segment of vehicle including bluff body segment as this segment consume large number of fuel. Suction the airflow at the rear of the vehicle can reduce the wake region where the pressure at the region is increased. Venturi effect is proposed from Bernoulli principle. The small opening at the inlet yield high velocity low pressure and at the outlet yield low velocity high pressure. The suction into the inlet reduce the wake region behind the model while at the outlet the air is blow out from the venturi shape causing air pressure to increase at the wake region. The objective of this study to apply the venturi shape at the rear of the body that can change the pressure different and reduce drag coefficient (Cd) of bluff body which in this case use modified Ahmed body. The design and simulation of the model will be aided by using Computer-aided Design (CAD) and computational fluid dynamic (CFD) software. Then, the fuel consumption also calculated in order to find out the percentage difference that can be get by implementing this design to bluff body and can change the trend on designing the bluff body in the real world.

DEDICATION

This project is dedicated for my lovely parents for their enthusiastic caring and love throughout my life, my lovely siblings and also my friends for their supports and encouragement.

ACKNOWLEDGEMENT

I dedicate this project to my family and all of my friends. Enormous gratitude to my supervisor, Mr. Muhd. Faruq Bin Abdul Latif for all the guidance in order to complete this project. The supervision and support that he gave me vey help me in term of progression and smoothness of completing the project. It one of the main factor for this project to succeed.

To my lovely parent, Mr. Mohd Jailani Bin Ali and Mrs Zalina Binti Jamal thank you so much for always supportive and become my motivator to finish this project. Not to forget, special thanks to all of my friends, who have supported me throughout the process of this project. All the encouragement that had been given will always bear in my mind to keep moving forward for next step in my life. Thank you.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS, SYMBOLS AND NOMENCLATURE

A	-	Area
B	-	Base (width)
CAD	-	Computer Aided Design
CAE	-	Computer-aided Engineering
CATIA	-	Computer Aided Three-Dimensional Interactive Application
C_D	-	Drag Coefficient
CFD	-	Computational Fluid Dynamic
C_L	-	Lift coefficient
C_P	-	Pressure coefficient
DNS	-	Direct Numerical Simulation
FEM	-	Finite element method
H	-	Height
K	-	Boltzman Constant
L	-	Length
LBM	-	Lattice Boltzman Method
LES	-	Large eddy simulation
m	-	mass
p	-	Pressure
RANS	-	Reynolds Averaged Navier Stokes
T	-	Temperature
t	-	Time
V	-	Volume
v	-	Velocity
VWT	-	Virtual wind tunnel
$\boldsymbol{\rho}$	-	Density

CHAPTER 1

INTRODUCTION

1.0 Introduction

Aerodynamic drag of road vehicle is important to be control as it become problem in vehicle aerodynamics. It is given attention full attention in recent decade because of high cost of fuel and to reduce environmental burden cause by fuel consumption from vehicle (Hanfeng et al. 2016).

Then, overcoming aerodynamic drag in truck and bus are needed because this kind of vehicle travel for a long time to deliver goods or transport people. This vehicle also have large frontal area and bluff shape which are aerodynamically inefficient and cause up to 65% of fuel to overcome the drag. Aerodynamic drag force on a body is cause by the vehicle body and its surface area. At the rear side of the vehicle, air flow passing the moving vehicle is separate which cause pressure drop. The separation then can be categorized into two main factor that is inability of flow to past sharp corner and lack of energetic flow. In bluff body aerodynamic, profile drag caused up to 90% of total drag while the remaining is due to skin friction drag (Altaf, Omar, and Asrar 2014)

Computational fluid dynamic (CFD) simulation development has been greatly develop in recent years. It is important element in vehicle aerodynamics analysis especially in design stage because do not required high cost. Computational fluid dynamic (CFD) has great advantage to approach the surface and fluid data that will be use to analyse aerodynamic problems and assess the impact of air correction device that will be applied. The data obtained then will be used to find the complex interactions between surface pressure distributions, overall force, flow features such as vortex and wake and also show how the geometry change give impact to aerodynamic.

1.1 Problem Statement

One of the most significant effect in aerodynamic is drag coefficient. According to Mohamed-Kassim & Filippone (2010), pressure drag in large vehicle are contributed by the base wake, crossflow effect inside tractor and complex underbody structure. Then, aerodynamic drag bring significant effect to fuel consumption especially on bas and truck because of their large frontal area and bluff shape that are aerodynamically inefficient (Altaf, Omar, and Asrar 2014). Pressure forces generated by the vortices on the back wall of body and the strength depends on the size and trajectory of the vortices as they moving away from the body.(C. H. Bruneau et al. 2014)

Several attempts have been made to reduce the drag coefficient at the rear bluff body such as use of elliptically shaped flap (Alamaan A. et al., 2014), reducing the rear slant angle (Tural T. et al., 2014), and control the flow by using steady blowing and synthetic jets (Wenshi C. et al.,2015).

However, only little attempt have been made that specifically study about the pressure different at rear suction air flow correction base. Thus, this study will enhanced the method to control the drag coefficient by finding the optimum ratio of inlet over outlet at the rear body of vehicle in order to get the least drag coefficient.

The effect of venturi is fully utilised to reduce the pressure different rear suction air flow correction base. This study will be carry out using Computer-aided Design (CAD) and Computational Fluid Dynamic (CFD) because referring to previous research by Khalighi et al. (2012) as this method improve the capabilities of code, reduce cost of technology and cut the cost of maintain the experiment facilities.. The design of the bluff body will be created using CAD while analysing the test are using CFD.

Hopefully by introducing this idea can help to reduce drag coefficient at the rear body significantly. Thus, can reduce fuel consumption in vehicle as it contribute large effect in aerodynamic drag. When less fuel is use it will help to decrease air pollution in the world that cause by carbon monoxide and nitrogen oxide released from the vehicle. Beside, positive encouragement are given so that this method will be used widely especially in automotive sector and set a new trend of reducing drag in bluff body vehicle.

1.2 Objective

1. To apply venturi shape at the rear of bluff body in order to reduce overall drag coefficient.
2. To calculate the fuel consumption after drag reduction method is apply.

1.3 Scope

The main scope of this paper will use bluff- body that can be example as bus or large truck in real world. In this case, Ahmed Body is taken as it is generic shape for bus. Then, there are several type of external aerodynamic involved in bluff-body such as skin friction, vortex and boundary separation. The boundary separation is taken into more deep study by using the effect of venturi. The shape of venture based on inlet over outlet is fully optimized to find the best ratio that give lowest drag coefficient (C_D). In order to conduct the analysis, there are two method commonly use which is using wind tunnel and simulation. For this case, simulation method are chosen to investigate the drag coefficient. Computer Aided Design (CAD) and Computational Fluid Dynamic (CFD) software are used to create the venturi at rear of Ahmed body and perform the analysis of aerodynamic specific on the boundary separation that contribute to drag coefficient. Then the common CFD setting such as air viscosity, temperature, humidity and velocity are follow the standard in Malaysia. This study will show the pressure different at the back of Ahmed Body after modification is made by adding venturi shape.

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

Literature review purpose in this study is to review other research related to the study conducted in order to obtain the idea and concept. Literature review also use in this study to obtain problem statement and to get best and suitable methodology.

The literature review flow in this study can be summarized in form of k-chart in Figure 2.1. It start with introduction in vehicle that consist three type vehicle exist in this world that is aerospace, sea and land. Then it is followed by the overview of land vehicle that divide into two which are personal and public use. In public use, train and bus are the vehicle that are commonly use.

Then, brief overview on bus aerodynamic is provided based on internal and external aerodynamic. The study then converged to type of external aerodynamic. Besides that, study on simulation method which are wind tunnel and simulation are also being reviewed in order to get the factual content.

Methodologies are studied in order to get the best option to conduct this study. The methodologies then is considered and studied for this study are physical wind tunnel and simulation. Judgement on the method that will be used are also stated.

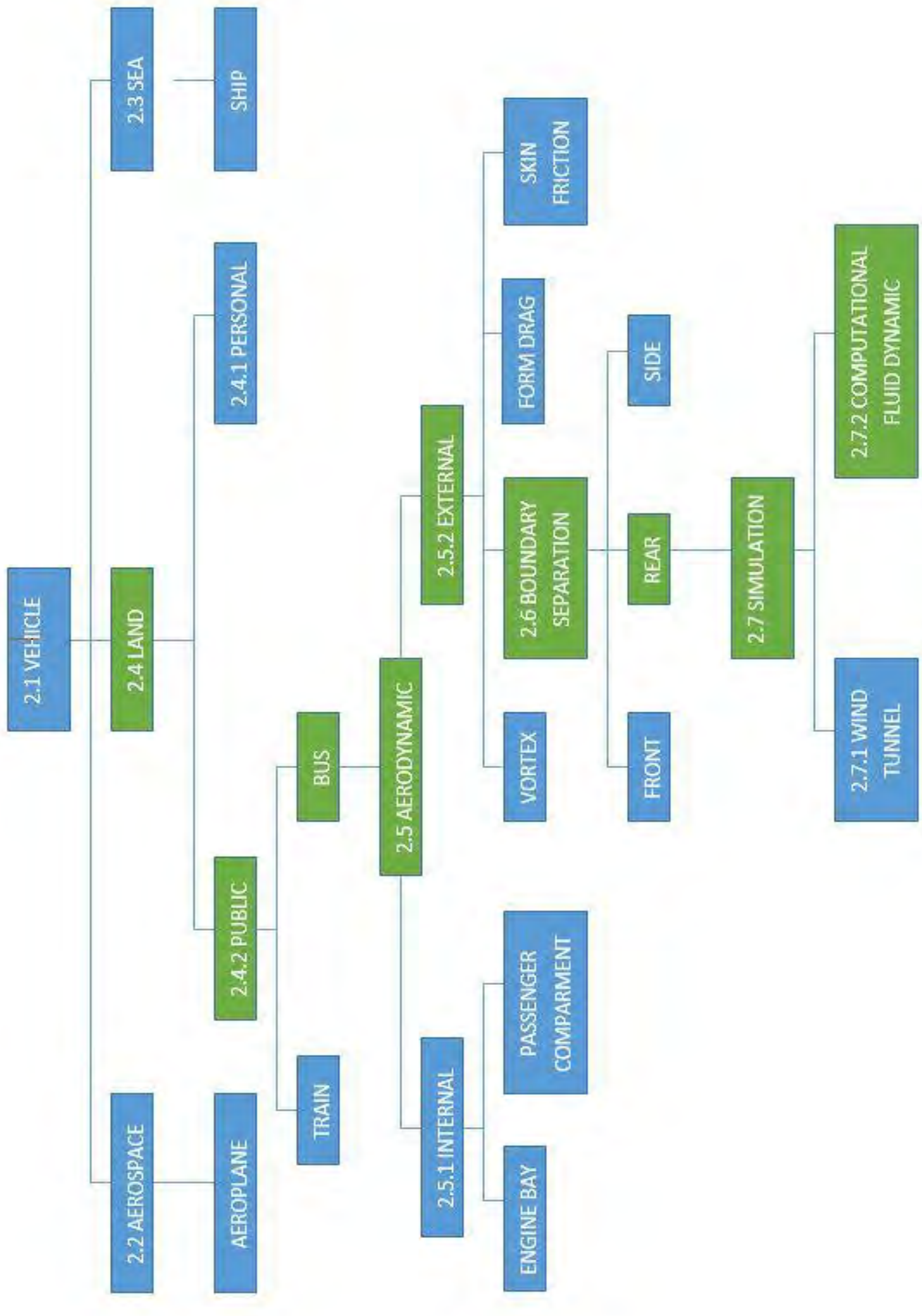

Figure 2.1 : K-chart

2.1 Vehicle

In this modern era, transportation has become the important element in daily life. It can be divided into three type which is sea, aerospace and land. The importance of having multiple type of transportation because each of them operates on different purposes. Besides, the improvement that has been made from time to time make the variation of the vehicle become widen.

The ship that firstly operate on the sea was using energy of wave to propel it that has been introduced by Linden. The craft has been said moved around 3 to 4 mile per hour by only using the energy from the wave (Bøckmann and Steen 2013). The shape of the steel plate was design like fish fins because it can act to wave direction whether up and down. The buoyancy of the ship was applied using Archimedes principle. (Chondros 2010)

Sir George Cayley in 1979 invented a vehicle that able to travel in the aerospace. Four elements of aerodynamic has been used in order to make the device fly which are lift, thrust, drag and weight. The device was basically a glider and it contains 2 wings and flap and also it was made from wood to carry one people at one time. (Ackroyd 2011). The Wright brother further the design from Sir George Cayley and invented the first ever engine powered aircraft in 1903. Helicopter was also invented after further development on the aircraft has been made. Bernoulli principle was used in helicopter main rotor as the thrust was gained to produce lift force. Ludwig Prandtl stated that pressure distribution over surface body is changed once the flow separates. Pressure drag was created due to unbalanced force that act in multiple direction. (Jr 2005). The design on the aircraft structure require information on external load that are being applied to the individual external parts of the aircraft. The flight dynamics of the aircraft analysis calculates aerodynamic loads exerting on the cockpits, fuselage and wings and after further analysis is conducted to predict structural response for example deformation, stress distributions and vibrations (Wang 2016).

Public and personal use transportation are two main type in land use. In the past time, people use animal such as cow, horse and camel to travel. Then, in 1879, Karl Benz invented a car and it is driven by internal combustion engine. Aerodynamic in vehicle plays important role in designing a car. It is because drag reduction in vehicle give significant effect to energy consumption which is related to fuel consumption (Je 2011). For public use, train is the main mode of transportation. Aerodynamic in train has been studied in designing the shape of the train's head. For example, high speed train applied the concept of the bullet-shape so that it can travel in maximum speed at 334.7 km/h such as Eurostar that operates from London to Paris (J. Choi and Kim 2014)

2.2 Sea

Study for aerodynamic in ship has been conducted by using horizontal foils for propulsion and motion reduction in waves. By doing that, it will include the no harmonically foil forces of lift and drag because of nonlinear drag and lift coefficient curves (Bøckmann and Steen 2013). Then, wind also give impact to the aerodynamic in ship when the wind hit the ship directly and also the moments act on the above-water part of the ship also will give effect however it is not significant. Aerodynamic force acting on the ship basically will use $\mathbf{V_{WR} = V_W - V_G}$ where $\mathbf{V_G}$ is the ship velocity at static condition, and $\mathbf{V_W}$, speed of the wind. When it is moving, the formula used to calculate the aerodynamic force is $\mathbf{V_A = -V_{WR}}$ (Wn et al. 2015)

For yacht, (Lee et al. 2016) stated that it is important to optimize the aerodynamic performance of the sail because it is related directly to propulsion force which consist of drag and lift force. The propulsion force generated is used to move the yacht forward at desired speed and also keeping it stable between wind and water surface.

2.3 Aerospace

In aeroplane, the wing part play important role in its aerodynamic force. Cross-sectional of the wing is called as aerofoil. The lift force which generate at the wing when specific speed and angle of attack are applied, it will produce greater force than drag. The low pressure acted on upper side of the wing while high pressure produce on lower side of the wing thus this pressure different will produce the lift force. Four type of force give effect at the same time at the aeroplane which is drag, lift, thrust and weight. Drag force is resistance towards air when aeroplane moves through it. It is set to be in stable motion so that it will act to drag force. Air passing by the wing of aeroplane will generate lift force so that it will counter the weight of the aeroplane to rise in the aerospace. When the aeroplane moving in the air, thrust force that make it happens. It is generated by the propeller to move the aeroplane forward. Gravity also act on the aeroplane, thus weight is exerted on the plane. The centre of gravity of the plane is the main point where the weight is acting on (Suresh, Paramaguru, and Ramesh 2014).

The design of aeroplane has been widely improvise since its introduction in 1920s. The design of new aeroplane has gone through theoretical research and wind tunnel testing. These new design techniques required advanced design tools and have high capabilities to provide the result. Past researchers activities on aeroplane design aim on drag reduction and they focus on wing and lifting surface design which use aerofoil design. However, in high speed condition, fuselage design is crucial in order to reduce the total drag acting on the body. Then, wing and body are two main element in fuselage aerodynamic. Between two of them, there is a junction that connect the bodies with other aeroplane components. In particular, this junction induces the interactions between the parts, creating a boundary layer causes by a flow phenomenon which is very difficult to describe and simulate (Nicolosi, Della Vecchia, and Corcione 2015).

2.4 Land

After several decade of research, land transportation has become more advance. In modern city, the public and personal vehicle are growing very well. In well develop country, the efficiency of public vehicle service is very reliable. Other than that, using public transportation such as bus and train can help reducing pollution and the user can avoid from traffic jam. Beside, people living in low and middle class income more tend to use public transport because it is more saving than having a personal vehicle (Glaeser, Kahn, and Rappaport 2008).

2.4.1 Personal

In land transportation, personal vehicle is a must for each family. This is because in some place, public transport not provided in their place. Then, the vehicle that operate using fuel can cause pollution and increase the usage of fuel day by day. Hence, further research has been conducted to analyse aerodynamic in car such as reducing the drag and also improve fuel efficiency. The road vehicle drag force impact significantly on the wake flow downstream of the vehicle. Drag force and size of the separation zone are important parameter in designing future car. Angle of rear slanted angle (Tunay, Sahin, and Ozbolat 2014) and elliptically shape flap (Altaf, Omar, and Asrar 2014) are been introduced in order to reduce drag reduction especially at the back on the car. However, each design of car give different result for experimental studies, hence “Ahmed model” , a simplified vehicle design was used to study the aerodynamic force and the 3D wake flow (Hanfeng et al. 2016).

2.4.2 Public

In public transportation, train and bus are commonly studied for it characteristic in aerodynamic. In train, when operate in open field, it has different aerodynamic properties. When entering a tunnel, compression and expansion waves occurs. Then, aerodynamic drag, noise, force and moments are commonly act on the train also increased because of surrounding confinement (Baron, Mossi, and Sibilla 2001). There are several factors on why the expansion waves happen. This has been discussed by (Baron, Mossi, and Sibilla 2001) where they found that the factor are shape of nose and tail, the tunnel length, speed of train, roughness of tunnel wall and ratio of the train cross-section to the tunnel cross-section. The importance of measuring the aerodynamic characteristic is to know the train limits before designing new one (N. Yang et al. 2015).

The design of high speed train especially on its head modelling design has great impact on the aesthetic function and aerodynamic performance. The train's head design must achieved to both artistic and superior aerodynamic requirement which gain support from intersecting subject of industrial design and aerodynamic. Optimal aerodynamic performance can be obtain directly from the train's head which efficiently reduce the influence of aerodynamic phenomena on train operation. Aerodynamic performance of high-speed train directly related to head shape of the train and the whole train aerodynamic performance is effected by the streamlined shape and incline angle. In order to reduce air drag and improve the aerodynamic, researcher studied on shape on high-speed train and optimization of the head shape design is happen (X. Yang, Jin, and Shi 2013).

For bus, it is aerodynamically inefficient due to its large frontal areas and bluff body and because of that it takes about 65% of fuel usage in order to overcome the pressure drag. The drag pressure is contribute by the surface area and vehicle body profile. The square back at the vehicle make the air flowing through it to separate and cause large pressure drop in which form large wake at the back of the body (Altaf, Omar, and Asrar 2014). Beside, pressure drag occur in many place at the bus such as base wakes, complex underbody structures and direct flow exposure on the front.