UNIVERSITI TEKNIKAL MALAYSIA MELAKA

NOVEL CONCEPTUAL DESIGN AND SIMULATION OF REAR PRESSURE DIFFERENT AIR FLOW CORRECTION DEVICE USING COMPUTATIONAL FLUID DYNAMIC (CFD) SIMULATION

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 FARIDHWAN BIN SHOFRI

B071210030

930121-02-5353

FACULTY OF ENGINEERING TECHNOLOGY

2016

NOVEL CONCEPTUAL DESIGN AND SIMULATION OF REAR PRESSURE DIFFERENT AIR FLOW CORRECTION DEVICE USING COMPUTATIONAL FLUID DYNAMIC (CFD) SIMULATION

MUHAMMAD FARIDHWAN BIN SHOFRI

A thesis submitted in fulfilment of the requirements for the degree of Mechanical Engineering Technology (Automotive Technology) with Honours

Faculty of Engineering Technology

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2016

DECLARATION

I hereby, declared this report entitled “Novel Conceptual Design and Simulation of Pressure Different Air Flow Correction Device Using Computational Fluid Dynamic (CFD) Simulation” 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 Engineering Technology Mechanical Engineering Technology (Automotive Technology) with Honours. The member of the supervisory is as follow:

………………………………

(Project Supervisor)

ABSTRAK

Pengurangan seretan merupakan salah satu isu yang paling signifikan dalam industri automotif. Seret tekanan menyumbang lebih daripada 80% daripada jumlah kesuluruhan seretan dan ianya dipengaruhi oleh geometri kenderaan yang disebabkan oleh pemisahan lapisan sempadan dan pembentukan pusaran di belakang kenderaan. Seret disebabkan oleh tekanan yang berbeza di bahagian belakang dan hadapan kenderaan merupakan isu utama untuk semua segment kereta terutamanya segmen kereta yang berbadan legap kerana ia merupakan penyumbang terbesar kepada penggunaan bahan api. Sedutan atau hembusan udara di bahagian belakang badan legap boleh mengubah corak aliran udara, sekali gus dapat mengurangkan kawasan pusaran udara lalu menyebabkan peningkatan tekanan di belakang dan mengurangkan perbezaan tekanan antara depan dan belakang badan tersebut. Objektif kajian ini adalah untuk mereka bentuk konsep sistem pengurangan seretan yang mengubah pengagihan tekanan dan mengurangkan pekali seseret (C_D) pada badan legap iaitu Ahmed body. Reka bentuk dan ujian peranti pembetulan aliran udara ini dibantu oleh bantuan komputer rekaan bentuk (CAD) dan pengiraan dinamik bendalir berkomputer (CFD). Konsep peranti pembetulan aliran udara menggunakan sedutan atau hembusan di belakang badan legap ini diharapkan dapat mengubah corak aliran udara sekaligus mengubah pengagihan tekanan dan mengurangkan pekali seseret (C_D). Oleh itu, dengan penuh harapan positif, mudah-mudahan kaedah ini dapat mengurangkan seretan dan peranti ini diguna secara meluas digunakan untuk segmen kereta berbadan legap di seluruh dunia dan perubahan trend kajian ke arah konsep ini.

ABSTRACT

Drag reduction is one of the most significant issues within the automotive industry. Pressure drag contribute more than 80% of the total drag and it is highly dependent on vehicle geometry due to boundary layer separation and formation of wake region behind the vehicle. Drag caused by pressure different at the rear and front of the vehicle is a major issue for all car segments especially bluff body cars segments because it contribute largest fuel consumption. Air suction or blown at the rear of bluff body can alter the flow pattern of the air hence, shrink the wake region which resulting in increase of rear pressure and reduce the pressure different between front and rear of the body. The objective of this study are to design a concept of drag reduction system that can change the pressure distribution and reduce drag coefficient (C_D) of bluff body which is Ahmed body. The design and testing of the airflow correction device will be aided by Computer-aided Design (CAD) and computational fluid dynamic (CFD). The concept of the rear air flow correction device hopefully will alter the flow pattern which change the pressure distribution and reduce drag coefficient (C_D). Hence, with full of positive expectation, hopefully this method of reducing drag by using rear suction or blown air flow correction device will be broadly used for bluff body car segments for entire world and the trend change toward further study of this concept.

DEDICATION

This thesis is dedicated to my treasured father Mr. Shofri bin Abdullah, my beloved mother Mrs. Badariah binti Mohamed Kari and my benevolent brother Muhammad Firdaus bin Shofri.

ACKNOWLEDGEMENT

Bismillahirrahmanirrahim. In the name of Allah the Beneficent, the Merciful.

First and foremost, special thanks to my benevolent parents Mr. Shofri Abdullah and Mrs. Badariah Mohamed Kari for their love and support throughout my life. Not to forget, my elder brother Muhammad Firdaus Shofri who always support me, take care of me motivated me and inspire me. My small humble family have granted me strength, passion, and sprits to perform at my best and pursuit my visions and goals.

I would like to acknowledge my supervisor, Engr. Mohd Faruq bin Abdul Latif, for trust and confidence granted to me for conducting this plus his comprehensive guidance and support throughout this study. I would also like to thank my co. supervisor, Engr. Mohd Suffian bin Ab Razak. His comments and lessons were very beneficial in order for me to complete this thesis and conduct this study. I learned from his insight a lot. I was grateful for the discussion and clarification made with him. Also, I would like to thank my thesis examiners Engr. Luqman Hakim bin Hamzah and Engr. Saiful Naim bin Sulaiman for their time and effort to examine my thesis and evaluate my presentation.

To all my friends, thank you for your understanding and encouragement. Our friendship makes our life wonderful. Although your name are not mentioned here, but you are always on my heart.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLE

LIST OF ABBREVIATIONS, SYMBOLS AND NOMENCLATURE

A	-	Area
B	-	Base (width)
BC	-	Before Christ
C	-	Specific heat
CAD	-	Computer-aided Design
CAE	-	Computer-aided engineering
CATIA	-	Computer Aided Three-dimensional Interactive Application
C_D	-	Coefficient of drag
CFD	-	Computational fluid dynamic
C_L	-	Coefficient of lift
C_p	-	Coefficient of pressure
C_{DU}	-	Underbody drag coefficient
D	-	aerodynamic loads
D_{ext}	-	Drag due to the airflow on external upper surfaces of the vehicle
DNS	-	Direct Numerical Simulation
dm	-	mass of fluid
D_{ub}	-	Drag due to the vehicle under the body flow
$d\,x$	-	Element area
$d\,y$	-	Force on a side due to pressure
dv	-	elemental volume
f	-	Particle distribution function
FEM	-	Finite element method
FVM	-	Finite volume method
H	-	Height
k	-	Boltzmann constant
k	-	Kinetic energy turbulence per unit mass
$k-\omega$	-	k–omega turbulence model
$k-\varepsilon$	-	K-epsilon turbulence model
K_D	-	Coefficient that represent the underbody drag to total drag ratio

KE	-	Kinetic energy
K_Q	-	Coefficient that express the contribution of the flow rate of underbody on flow rate of total body
L	-	Length
L	-	Lift load
LBM	-	Lattice Boltzmann method
LES	-	Large Eddy Simulation
m	-	mass
n_i	-	Equilibrium value for the population of particles
n_i^{EQ}	-	Local equilibrium value for the population of particles in the direction
p	-	pressure
Δp_{tot}	-	The total pressure variation
$p_{dyn\infty}$	-	Dynamic pressure reference
Q_{ext}	-	Volume of air that enveloping the external upper surfaces of the vehicle
Q_{ub}	-	Volume of air that enveloping the body flow of the vehicle
RANS	-	Equations such as Reynolds Averaged Navier Stokes
SST	-	Shear Stress Transport
T	-	Temperature
t	-	Time
u	-	Initial velocity
$\vec{u}$	-	Particle velocity
V	-	Volume
v	-	Velocity
v_i	-	i^{th} component of the averaged air velocity
v_j	-	j^{th} component of the averaged air velocity
v'_i, v'_j	-	Fluctuating velocity parts
VWT	-	Virtual Wind Tunnel
$\overline{v'_i v'_j}$	-	Reynolds stress turbulent tensor
v_∞	-	Free stream velocity
w	-	Velocity in opposite direction (-v)
x	-	Direction of x-axis
x_i, x_j, x_k	-	Cartesian coordinates
y	-	Direction of y-axis
y+	-	Boundary layer theory in defining the law of the wall

α	-	Slant angle
Ω	-	Collision operator
ρ	-	density
ρ_∞	-	Density of air free stream
ζ_d	-	Aerodynamic resistance coefficient of diffuser.
ζ_i	-	Aerodynamic resistance coefficient of the inlet section
ζ_m	-	Aerodynamic resistance coefficient of middle section
ζ_{ub}	-	Equivalent aerodynamic resistance coefficient of the nozzle
τ	-	Relaxation time
μ_t	-	Viscosity of turbulent
μ_∞	-	Dynamic viscosity of air free stream
δ_{ij}	-	Kronecker delta

CHAPTER 1
INTRODUCTION

1.1 Background

Reducing fuel consumption and emissions is one of the most important environmental issues within the automotive industry. With the continuous increment of fuel price, automotive companies have to compete with each other and constantly developing methods to improve fuel efficiency. The fuel consumption reduction can be achieved by improving efficiency of the engine, reducing drag pressure resistance and improving aerodynamics (Håkansson & Lenngren, 2010; Wojciak, 2012).

Aerodynamic drag consists of two main components that is skin friction drag and pressure drag. Pressure drag contribute more than 80% of the total drag and it is highly dependent on vehicle geometry due to boundary layer separation and formation of wake region behind the vehicle (Sudin & Abdullah, 2014).

Computational fluid dynamic (CFD) simulation has become an important research way especially in vehicles aerodynamic characteristics. Computational fluid dynamic (CFD) used in automotive engineering for programming flexibly, getting more information, cutting costs and reducing engineering time. The advantages of a computational fluid dynamic (CFD) approach is the surface and fluid data that can be used to diagnose aerodynamic performance problems and assess impact of design changes. The data obtained can be used to find the complex interactions between surface pressure distributions, overall force, flow features such as vortices and wakes, and the geometry changes that have aerodynamic impact (Jing, Yun-zhu&Deng-feng,2010).

1.2 Problem Statement

Bluff body car segment such as suburban utility vehicle (SUV), multi-purpose vehicle (MPV), van, hatchback, and station wagon produce high pressure drag especially at the rear of the vehicle. Drag caused by this rear end pressure contribute largest fuel consumption (Baltas & Saridakis, 2009; Gilliéron & Kourta, 2013; Steininger, Vogl, & Zettl, 1996).

researcher discover and study a lot of method to reduce drag caused by pressure different between front and rear of the vehicle including study of rear spoiler by (Sapienza, 2002; Sudin & Abdullah, 2014), study of bumper by (Håkansson & Lenngren, 2010; Huminic, Huminic, & Soica, 2012; Sapienza, 2002; Sudin & Abdullah, 2014), and study of diffuser by(Sudin & Abdullah, 2014; Thuwis, De Breuker, Abdalla, & Gürdal, 2010).

Current research focus on shape minimizing to increase the pressure at the rear of the vehicle hence reduce the pressure different between front and rear of the vehicle(Loth, 2008; Tran-Cong, Gay, & Michaelides, 2004).

But yet, there are no effort putted to study about rear suction air flow correction device in order to increase the back pressure and reduce the pressure different between front and rear of the car.

Hence, this study is conduct to introduce rear suction air flow correction device to increase the back pressure and reduce drag caused by pressure differential.

The introduction of the rear suction air flow correction device hopefully will reduce drag coefficient (C_D). Hence, with full of positive expectation, hopefully this method of reducing drag by using rear suction air flow correction device will be broadly used for bluff body car segments for entire world and the trend change toward further study of this concept.

To induce the pressure at the rear of the bluff body by rear suction air flow correction device, the effect of venturi will be utilised. The study will be conduct by using computational fluid dynamic (CFD) method and Computer-aided Design (CAD). The device will be design by using Computer-aided Design (CAD) and test by using computational fluid dynamic (CFD).

1.3 Objective

1. To design a concept of drag reduction system that can change the pressure distribution of bluff body.
2. To reduce the drag coefficient of bluff body.

1.3 Scope

This project does not consider any costs of either implementation of the devices or production of the devices. Applicability and selection of material are not discussed. The devices are only aerodynamically evaluated and furthermore, no consideration is taken to legislate about the dimensions of the Ahmed body with added devices. Due to time limitation only a restricted number of devices are tested and evaluated to verify their efficiency. Devices optimization of is not included in this study. The devices are added on a standard Ahmed body model. No modifications are done on the Ahmed body. The CFD settings used in this study such as density of air, Specific heat (C), air viscosity, temperature and humidity are referred to the local standard and evaluation of how different settings would affect the result is not included in this project. The velocity of air used are fixed and the different velocity effect on the drag coefficient (C_D) will not be presented. The study will only show the effect of the pressure different rear suction air flow correction device to the drag coefficient (C_D) and the unimplemented pressure different rear pressure air flow correction device Ahmed body to the drag coefficient (C_D). Equations such as Reynolds Averaged Navier Stokes (RANS), Large Eddy Simulation (LES), Direct Numerical Simulation (DNS) and Lattice Boltzmann method (LBM) with $k-\omega$ near-wall region model and $k-\varepsilon$ far field model are use in the study as the numerical method.

CHAPTER 2
LITERATURE REVIEW

2.1 Introduction

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

The literature review flow of this study was illustrate in the k-chart in figure 1. It starts with introduction which consist of brief history of computational fluid dynamic (CFD) and brief history of aerodynamic. Then it is followed by the overview of vehicle aerodynamic and aerodynamic airflow which consist of brief outline of internal and external aerodynamic.

The study then converging to vehicle aerodynamic. Fundamental theory was studied thoroughly. The study converge even more from vehicle to bluff body to Ahmed body. The study of drag reduction, simulation and wind tunnel were reviewed to enhance the factual knowledge.

Some case study were conduct in order to grasp others drag reduction device theory and fundamental. The case studies revised are venturi effect and diffuser by (Huminic et al., 2012), spoiler by (Hu & Wong, 2011), and race car wings by (Yang, Gu, & Li, 2011).

Methodologies were studied to obtain the best way to conduct the experiment. The methodologies considered and studied for this study are physical, wind tunnel, and simulation. Judgement were made based on the comprehensive studies of various methodology stated.

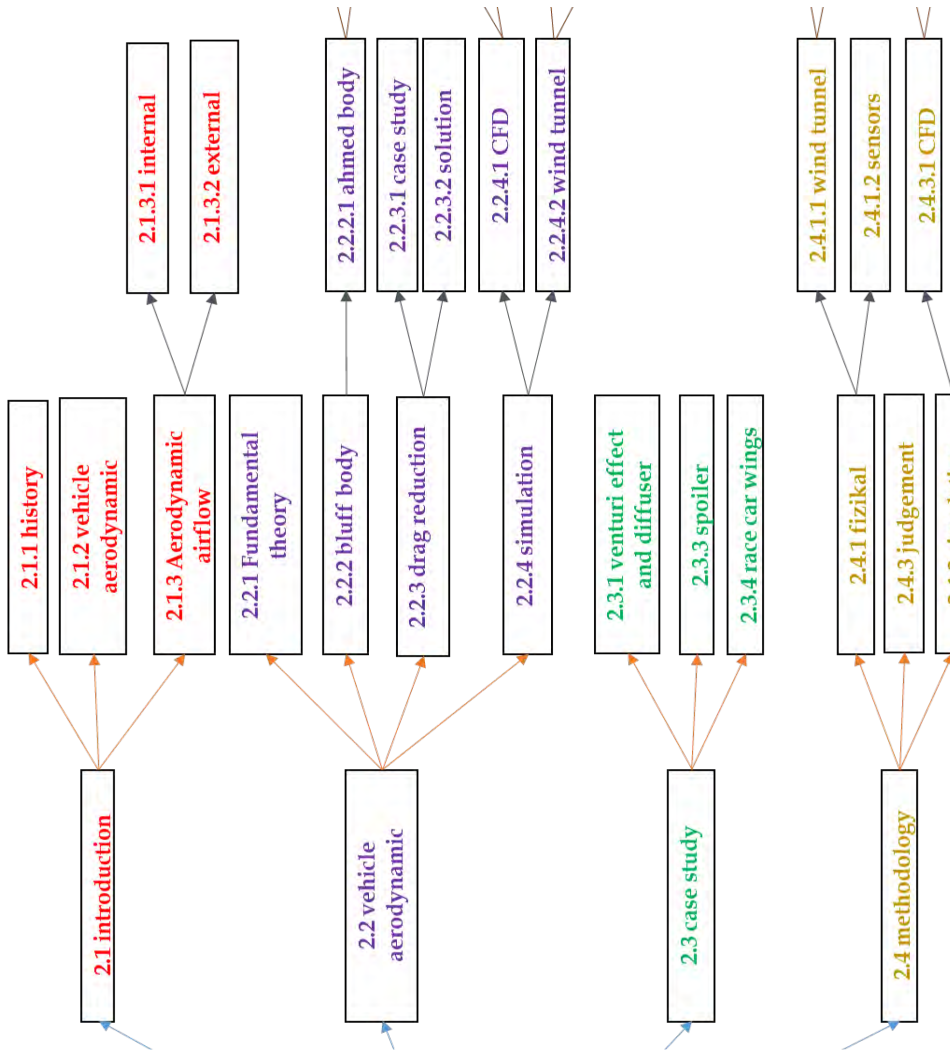

Figure 2.1 K-Chart

2.1.1 History

The functions of complex variables theory was established by Leonhard Euler who had a strong concern in fluid dynamics and related subjects during all his adult life (R. R. Cosner & Roetman, 2000; Darrigol & Frisch, 2008). By the 18th and 19th centuries, Cauchy and Riemann specialised the functions of complex variables theory (R. R. Cosner & Roetman, 2000). By 1752, Cauchy-Riemann equations were first written down by dAlembert in his study of ideal fluids. The further development was predominantly guided by the evaluation of integrals in closed form. The original system of dAlembert is velocity vector components which can be summarised as:

$$\text{Irrotationality} \quad \frac{\partial u}{\partial y} + \frac{dw}{dx} = 0 \tag{1}$$

$$\text{Mass conservation} \quad \frac{du}{dx} - \frac{dw}{dy} = 0 \tag{2}$$

Where

w = -v

v in the direction of y

u in the direction of x

The "irrotationality" refers to the assumption that the flow is an irrotational flow field and "mass conservation" is the mathematical expression of the physical assumption of energy conservation that mass is neither created nor destroyed (R. R. Cosner & Roetman, 2000; Darrigol & Frisch, 2008; Grimberg, Pauls, & Frisch, 2008).

The Cauchy-Riemann Equations discovered in 1900s. The mathematics community had ample confidence in their understanding of the theory of complex analytic functions that they could and did publish books for the technical world. The work of Riemann and followers on conformal mappings with singularities provides a method of solving the Cauchy-Riemann equations in multi-connected domains. The development continue with ideal two-dimensional flow around bodies using the Schwarz-Christoffel formula. Further study realised the necessity for a singular source term (vortex singularity) related to circulation to make the description of two-dimensional, constant-density flow around a figure with a pointed trailing edge physically reasonable (Bergonio, n.d.; Cochran, Kanso, & Krstic, 2009; R. R. Cosner & Roetman, 2000; Costamagna, Elettronica, Via, & Piazza, 1998; Tan, 2013).

By 1913, the pressure distribution around an air foil theoretically calculated Blumenthal. This might be considered the foundation of computational methods in air vehicle analysis. Regardless of the limitations of the analysis methods based upon conformal mapping, the development of a dictionary of mappings gave the practicing engineers opportunities to design air foils (Bergonio, n.d.; R. R. Cosner & Roetman, 2000).

With desired properties, the methods continue to be useful today. Discussion of design optimization using inverse complex analytic function theory can be found in quite recent literature (Jameson, 1988).

These developments led to great numbers of calculations generating tables of data and analogous graphs. The massive number of required calculations motivated theoretical studies in the aerodynamics community to develop scaling methods to extend the applicability of the tables. The limitations and difficulties of the analysis trigger stimulus to a parallel development of wind tunnels(R. R. Cosner & Roetman, 2000). Again the limitations of the calculation devices and expenses for construction and maintenance generate numerous further studies dedicated to understanding fluid dynamics in order to make the necessary corrections to the data obtained from the wind tunnels and to improve the wind tunnel designs (R. R. Cosner & Roetman, 2000).

Aerodynamics is studies that concerned with the motion of air. Aerodynamic is one of a gas and fluid dynamics sub-field, and the "aerodynamics" term is generally referring to gas dynamics (Anderson & Jr, 2009; Anderson, 1991).

Early track of aerodynamics fundamental concepts date back to the Archimedes and Aristotle work in the 2nd and 3rd centuries BC, but the efforts to develop a quantitative air flow theory does not rise until the 18th century. When Sir Isaac Newton developed a theory of air resistance in 1726, he became one of the first aerodynamicists in the modern era, which was then verified for low speeds flow. Throughout the 18th and 19th centuries, experiments of air resistance were performed by investigators which aided by the first wind tunnel construction in 1871. By the year of 1738 Daniel Bernoulli described a fundamental relationship between pressure, velocity, and density in his publication, Hydrodynamica. Until now, Bernoulli's principle, are widely use which is a method of lift calculation (Anderson & Jr, 2009; Anderson, 1991).

Throughout the 19th century aerodynamics required to achieve more than just air flight. The concept of the modern fixed-wing aircraft was developed George Cayley in 1799, this event lead to identification of four fundamental forces of flight which are drag, thrust, lift, and weight. The reasonable development predictions of the