UNIVERSITI TEKNIKAL MALAYSIA MELAKA

# NUMERICAL STUDY OF AIRFLOW PERFORMANCE AT ENGINE RADIATOR VENT ON COMMERCIAL BUS (PIONEER COACH BUILDER)

This report is submitted in accordance with the requirement of the Universiti Teknikal Malaysia Melaka (UTeM) for the Bachelor of Mechanical Engineering Technology (Automotive Technology) with Honours.

by


**MOHAMAD AZARUL BIN ROSLI**

**B071510718**

**940305025859**

FACULTY OF MECHANICAL AND MANUFACTURING ENGINEERING TECHNOLOGY


2018

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

**BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA**

Tajuk: Numerical Study of Airflow Performance at Engine Radiator Vent on Commercial Bus (Pioneer Coach Builder)

Sesi Pengajian: 2018/2019

Saya **MOHAMAD AZARUL BIN ROSLI** mengaku membenarkan Laporan PSM ini disimpan di Perpustakaan Universiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Laporan PSM adalah hak milik Universiti Teknikal Malaysia Melaka dan penulis.
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja dengan izin penulis.
3. Perpustakaan dibenarkan membuat salinan laporan PSM ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (X)

| | | |
|---|---|---|
| ☐ | SULIT* | Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia sebagaimana yang termaktub dalam AKTA RAHSIA RASMI 1972. |
| ☐ | TERHAD* | Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan. |
| ☒ | TIDAK TERHAD | |

Yang benar,

_______________________

MOHAMAD AZARUL BIN ROSLI
268 LORONG 5 TAMAN AMAN
09600 LUNAS KEDAH

Tarikh:

Disahkan oleh penyelia:

_______________________

MOHD SUFFIAN BIN AB RAZAK

Cop Rasmi Penyelia

Tarikh:

*Jika Laporan PSM ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh laporan PSM ini perlu dikelaskan sebagai SULIT atau TERHAD.

## DECLARATION

I hereby, declared this report entitled Numerical Study of Airflow Performance at Engine Radiator Vent on Commercial Bus (Pioneer Coach Builder) is the results of my own research except as cited in references.

Signature:

Author's Name : MOHAMAD AZARUL BIN ROSLI

Date:

## APPROVAL

This report is submitted to the Faculty of Mechanical and Manufacturing Engineering Technology of Universiti Teknikal Malaysia Melaka (UTeM) as 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:

……………………………………………….

MOHD SUFFIAN BIN AB RAZAK

# ABSTRAK

Sistem penyejukan adalah salah satu sistem enjin penting. Ia juga meningkatkan pemindahan haba dan ekonomi bahan api yang menunjukkan prestasi enjin. Kebanyakan enjin pembakaran dalaman adalah cecair yang disejukkan dengan menggunakan penyejuk cecair atau udara yang melalui penukar haba (radiator) yang disejukkan oleh udara. Pengalir radiator enjin, penukar haba yang menghilangkan haba daripada penyejuk enjin yang mengalir melaluinya. Walaubagaimanapun, gril yang dibentuk daripada lubang radiator telah menghasilkan halaju aliran masuk yang berbeza yang memainkan peranan penting dalam laluan aliran udara, prestasi aerodinamik dan penyejukan bas. Ia adalah disebabkan oleh reka bentuk jeriji pada bolong radiator. Oleh itu, matlamat kajian ini adalah untuk mengubahsuai ujian fizikal dengan menggunakan perisian Computer-Aided Design (CAD) dan Computational Fluid Dynamics (CFD) untuk menganalisis kesan prestasi aliran udara pada gril pada lubang radiator enjin di sisi bas dan mengesahkan hasil simulasi dengan menggunakan perisian Altair Hyperworks. Terdapat 3 reka bentuk gril yang dicipta daripada tanda aras yang merupakan bahagian-bahagian 14, 8 sisi dan reka bentuk mendatar. Data hasil simulasi itu dibandingkan bersama. Keputusan yang diharapkan akan dibincangkan dari segi reka bentuk gril dan prestasi penyejukan yang lebih baik. Oleh itu, reka bentuk terbaik ialah gril sisi 8 menghadap ke arah sebelah kanan kerana ia mempunyai nilai magnitud halaju tertinggi dan keadaan suhu enjin terendah berbanding dengan reka bentuk gril lain.

# ABSTRACT

A cooling system is one of the vital engine systems. It additionally increases heat transfer and fuel economy which indicates the performance of an engine. Most internal combustion engines are fluid cooled using either liquid or air coolant that runs through a heat exchanger (radiator) cooled by air: the engine radiator vent, a heat exchanger that removes heat from engine coolant that flows through it. However, the grille shaped of the radiator vent had resulted in a different inflow velocity that plays a crucial role in the airflow path, aerodynamic and cooling performance of the bus. It is due to the grille design on the radiator vent. Therefore, the aim of this study is to remodel the physical testing using Computer-Aided Design (CAD) and Computational Fluid Dynamics (CFD) software to analyze the airflow performance effect of grille shape on the engine radiator vent at the side of the bus and verifying the simulation results by using Altair HyperWorks software. There are 3 grille designs created from the benchmarks which are the14 sides, the 8 sides, and the horizontal design. The data that we simulate from the simulation are compared together. The expected results are discussed in term of grille design and better cooling performance. Thus the best design was the 8 sides grille facing the right side direction because it has the highest velocity magnitude value and the lowest engine temperature condition compared to the other grille designs.

# DEDICATION

I dedicate this report to my beloved guardians Mr. Rosli Bin Ahmad and Mrs. Noorizan Binti Mansor. Not forgetting to my supervisor Mr. Mohd Suffian bin Ab Razak and also my automotive studio analysis laboratory technician, Mr. Azrul that gives me a chance to use the laboratory under his supervision. I also want to dedicate this dissertation to my lecturer Mr. Mohd Faruq Bin Abdul Latif who is willing to teach me and assist me to complete this research. I also would like to dedicate to all my friends for giving the assistant, support and underpins in completing this study.

# ACKNOWLEDGEMENTS

Praise to Allah S.W.T The Almighty God and peace be upon Muhammad Rasulullah S.A.W with all His guidance, so I can smoothly complete the study of "Numerical Study of Airflow Performance at Engine Radiator Vent on Commercial Bus (Pioneer Coach Builder)" as the requirement for the Bachelor of Mechanical Engineering Technology (Automotive Technology) with Honours, Faculty of Mechanical and Manufacturing Engineering Technology.

First of all, I would like to thanks and appreciation my supervisor Mr. Mohd Suffian bin Ab Razak for his guidance to finish this final year project. Special thanks also dedicated to Mr. Mohd Faruq bin Abdul Latif for helping me during the progression of this study and encouragement towards the completion of this study.

From the bottom of my heart, my deepest gratitude is expressed to my beloved father Mr. Rosli Bin Ahmad and my beloved mother Mrs. Noorizan Binti Mansor and all my friends

# TABLE OF CONTENTS

# LIST OF TABLES



# LIST OF FIGURES

# LIST OF SYMBOLS

| | | |
|---|---|---|
| D, d | - | Diameter |
| m | - | Mass |
| N | - | Rotational velocity |
| P | - | Pressure |
| Re | - | Reynolds number |
| V | - | Velocity |
| w | - | Angular velocity |
| x | - | Displacement |
| q | - | Angle |
| H | - | Height |
| L | - | Length |
| W | - | Width |
| E | - | Length Of Back Slant |
| R, r | - | Radius Of The Roundness |

# LIST OF ABBREVIATIONS

| | |
|---|---|
| PCB | Pioneer Coach Builder |
| CFD | Computational Fluid Dynamics |
| CAD | Computer-Aided Design |
| CD | Drag Coefficient |
| CL | Coefficient of Lift |
| CO2 | Carbon dioxide |
| AIS | Air Intake System |
| FD | Finite Differences |
| FV | Finite Volume |
| FE | Finite Element |
| DNS | Direct Solution Of The Navier–Stokes Equations |
| LES | Large-Eddy Simulation |
| PC | Personal Computer |
| Re | Reynolds Number |
| CAE | Computer-Aided Engineering |

# CHAPTER 1

# INTRODUCTION

## 1.1. Background

Common form of a bus is the single-decker rigid bus, or with larger loads carried via double-decker buses and articulated buses. Coaches are used for more extended distance services. Bus production has become more and more globalized with the identical design appearing around the sector. Engine cooling system plays a vital role in improving the vehicle fuel economy and to meet the stringent emission norms apart from maintaining the operating temperature of the engine. The cooling system is tremendously important for vehicles as it keeps the engine running at operating temperature. The design of radiator grille is the major contributor to the engine cooling system. There are several types of grilled design that bus manufacturer has been used in recent years such as the perforated grille, cross-hatching grille, honeycomb grille, vertical grille, and horizontal grille. Bus intake grille usually located at the back side along with the engine compartment. Computational fluid dynamic (CFD) is utilized in automotive engineering for more flexibility, getting more insight data, reducing costs and decreasing engineering lead time. The benefit of a CFD approach is that it only requires the surface and fluid data that can be used to diagnose performance issues and investigate the impact of design modifications towards the airflow around engine system. This method will be applied in this project in designing a suitable grille shape on the side panel of the Pioneer Coach Builder (PCB) bus. CFD is used to analyze the design effectively and to get the numerical data with a purpose to gain better airflow performance on the side of the bus. A CFD simulation is used to simulate and analyze the altered grill shape design for the PCB bus drawing model. 3D model drawing is generated, and the model can be tested, grille position and grille shape will be redesign to improve and allow maximum cooling performance by using Altair HyperWorks software.

## 1.2. Problem Statement

Engine cooling system plays a vital role in improving the vehicle fuel economy and meeting the stringent emission norms apart from maintaining the operating temperature of the engine. The airflow through vehicle subsystems like the grille, bumper, hood-latch baffles, the heat exchangers, the fan and shroud is called as front-end flow (Baskar & Rajaraman, 2015). One of the factors determines the heat-transfer efficiency of the system is the inflow air velocity which is controlled by the grille shape (Hutacharern & Ridluan, 2017). Moreover, the heat exchanger (radiator) position, performance, and its material play a crucial role in offering resistance to the airflow path (Baskar & Rajaraman, 2015). Typically in the past, vehicle thermal management system capabilities have been studied through experimental testing (Tojcic, 2017).

Intake air is generally controlled by a grille which is carefully designed to direct the air into the engine chamber because the shape of the grille has much effect in the aerodynamic and cooling performance of the vehicle (Hutacharern & Ridluan, 2017). The result has shown that grille blade setting angle has a significant effect on the velocity of air flowing through the engine bay (Hutacharern & Ridluan, 2017). Nowadays, the louvered fins are favorable for the air side of a radiator. This kind of fin enormously increase the heat transfer coefficient and keep a low-pressure drop on the air side, especially at high Reynolds number of the air (Tojcic, 2017). With improved heat exchanger performance there is a scope to minimize the requirement of the airflow. The space between the radiator and engine block could be adjusted automatically to increase heat exchangers evacuation capacity in situations throughout engine overheat condition. It was observed that an increase in heat transfer performance of radiator about 18% at fan low velocity and 7% at high velocity (Khaled et al. 2012).

Furthermore, the shielding methods were vertical, horizontal, from both sides and side toward to the center. Based on the observation the best methods of shielding would be a horizontal method followed by the vertical method. More uniform airflow distribution obtained in a horizontal method compared to others (Jama et al. 2004). (Regin et al.) CFD use to study the effect of different front end opening area temperatures monitored with thermocouples. The results show that the CFD model can be used for the development of cooling modules for new vehicles and new designs (Lin, 2014).

In this project, the effect of grilled shape on the engine radiator vent at the side of the bus will be studied. Thus for optimum design of a cooling system, subjects to be studied to ensure satisfactory performance are heat exchanger and airflow (Masatoshi et al. 1993). Optimize the cooling flow entry into the engine compartment, especially in the area of an active grille shutter system(Baskar & Rajaraman, 2015). More specifically, the goal is to gain a better understanding of the cooling airflow and its interaction with the various cooling modules when heat transfer effects are considered (Tojcic, 2017). In the nutshells, the Pioneer Company will be leading the bus industry globally in modeling techniques for vehicle airflow ventilation simulations as investigating the effects of heat transfer and compressibility on the cooling airflow toward non-commercial buses.

## 1.3. Objectives

1. To construct the grille design and engine compartment of the Pioneer Coach Builder Bus. (Dragoes 3)
2. To simulate the benchmark models using Altair HyperWorks.

3. To analyze the airflow performance effect of grilled shape on the engine radiator vent at the side of the bus for a better cooling system.

## 1.4. Project Scope

The primary purpose of this project is to study the airflow performance at engine radiator vent on the side of an actual bus. The engine radiator vent on the side of the bus will be covered in this project. The interior and engine modification will be not included because only the grille shape of the bus will affect the airflow performance and will increase the cooling system. The bus is targeted as the primary target for this project, and CAD drawing of the bus is needed. The CAD generation of the commercial bus is specifically only at Pioneer Coach Builder company.

Furthermore, the airflow performance only applies on grilled engine radiator at the side of the Pioneer Coach Builder bus. The grilled shape design depends on the findings and only the optimize design airflow will be used. The 3D data of the bus with optimize the grilled shape for the best airflow performance which is from CAD drawing by simulating using CFD to get the cooling system efficiency. The simulation uses a variation of speed to get a different set of data. Moreover, the airflow performance result of Pioneer Coach Builder bus can be calculated. Thus, in this project, the study of the airflow performance at engine radiator vent on Pioneer Coach Builder bus is limited to the numerical method.

## 1.5. Expected Result

From this project, we can see the difference between the optimization of radiator vent design and the benchmark result. The optimizations of the design have more cooling airflow performance than the benchmark result. Additionally, the fuel consumption will be reduced by optimizing it. The different parameter of the radiator vent will contribute to change the drag coefficient on fuel consumption and most importantly the cooling performance. The result expectation of this project is to improve and increase the cooling airflow performance while maintaining the engine temperature from overheating.

# CHAPTER 2

# LITERATURE REVIEW

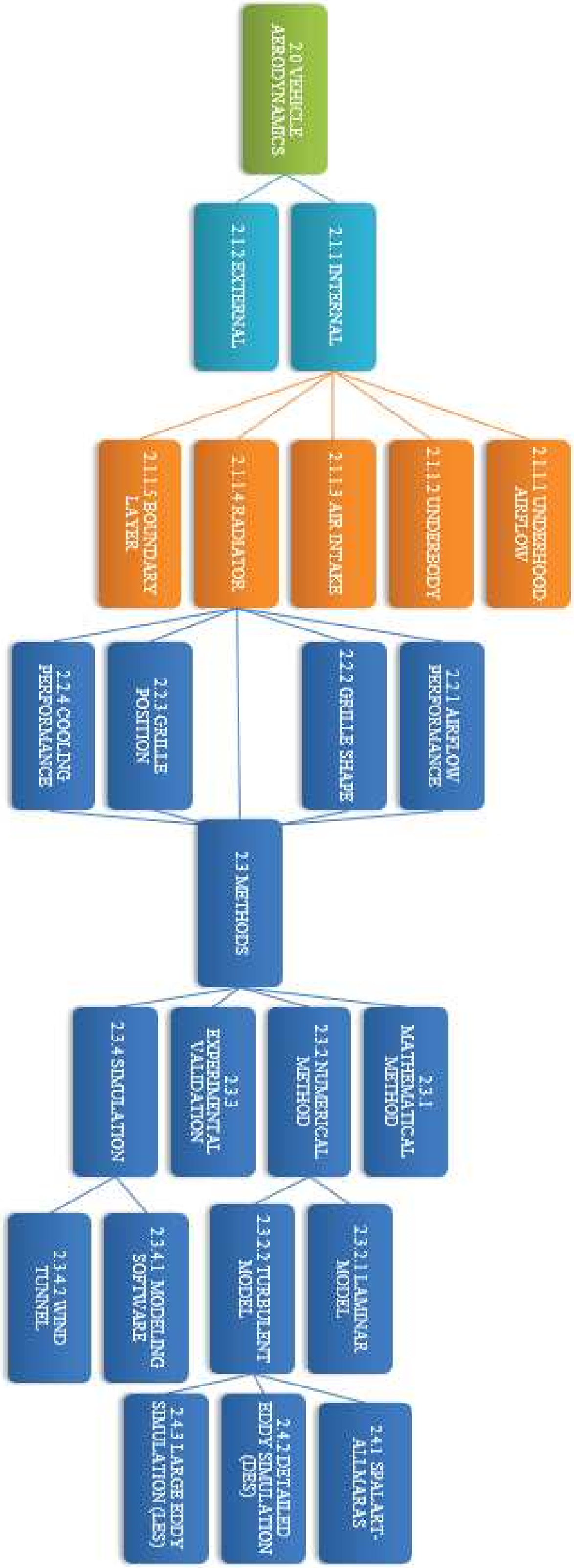
## 2.0 Introduction

Literature review purpose for this study is to review other research related to the study conducted to get the right idea and concept. The literature review also uses in this study to obtain a problem statement and suitable methodology.

The literature review flow of this study was illustrated in the K-Chart. It starts with an introduction which consists of a brief history of vehicle aerodynamics. Then it is followed by the overview of external aerodynamic and internal aerodynamic which consist of a brief of the engine compartment, outer body, airflow intake, airflow radiator, and boundary layer.

The study is then converging to vehicle airflow intake. The fundamental theory was studied thoroughly. The study airflow performance, grille shape, and grille position were reviewed to enhance the factual knowledge.

**K Chart**


2.0 VEHICLE AERODYNAMICS
2.1.1 INTERNAL
2.1.2 EXTERNAL
2.1.1.1 UNDERHOOD AIRFLOW
2.1.1.2 UNDERBODY
2.1.1.3 AIR INTAKE
2.1.1.4 RADIATOR
2.1.1.5 BOUNDARY LAYER
2.2.1 AIRFLOW PERFORMANCE
2.2.2 GRILLE SHAPE
2.2.3 GRILLE POSITION
2.2.4 COOLING PERFORMANCE
2.3 METHODS
2.3.1 MATHEMATICAL METHOD
2.3.2 NUMERICAL METHOD
2.3.3 EXPERIMENTAL VALIDATION
2.3.4 SIMULATION
2.3.2.1 LAMINAR MODEL
2.3.2.2 TURBULENT MODEL
2.3.4.1 MODELING SOFTWARE
2.3.4.2 WIND TUNNEL
2.4.1 SPALART-ALLMARAS
2.4.2 DETAILED EDDY SIMULATION (DES)
2.4.3 LARGE EDDY SIMULATION (LES)

## 2.1 Vehicle Aerodynamics

When objects move through the air, forces are generated by the relative motion between air and surfaces of the body, the study of these forces generated by air is called aerodynamics. According to (I. Hucho, 1993), the concept of aerodynamics never been stress on during the replacement of carriage horse to the thermal engine over than 100 years ago. During that time, carriage horse is used to shelter the driver and passenger from wind, rain, and mud. This idea has developed, and the concept of aerodynamics is applied to the road vehicle after the technology of flight had made.

Three principles related to the vehicle aerodynamics which is a flow of air around the vehicle body, flow of air through the body and lastly, flow processes within the machinery for the first two flows related to each other. It is because the flow of air through the engine compartment is dependent on the flow field around the vehicle. These flows are considered together. Besides that, the flow processes related to the engine and transmission but not connected first two flows (Wolf-Heinrich Hucho, 1998).

According to (Wolf-Heinrich Hucho, 1998), vehicle aerodynamics more focuses on the forces and moments which can influence the vehicle performance and stability. By focus on this matter, few aspects can consider such as the windows and lights free of dirt and accumulated rainwater, reduce wind noise, prevent windscreen wipers lifting, cool the engine oil sump and lastly brake.

Vehicle aerodynamics classify by two categories which are internal aerodynamics and external aerodynamics. Internal aerodynamics related to the interior of vehicle motion in term of ventilation, cooling system, and heat exchange. While external aerodynamics more focus on forces that form during high acceleration (Badih A. Jawad and Maria M. Longnecker)

### 2.1.1 External Aerodynamics AirFlow

Internal Vehicle Aerodynamics is an improvement of performance vehicle components which involve in cooling performing due to increased airflow. There are three common areas application of internal vehicle aerodynamics which is ram air system to the intake, ram air to the heat exchanger and lastly ducting to the oil cooler (Badih A. Jawad and Maria M. Longnecker).

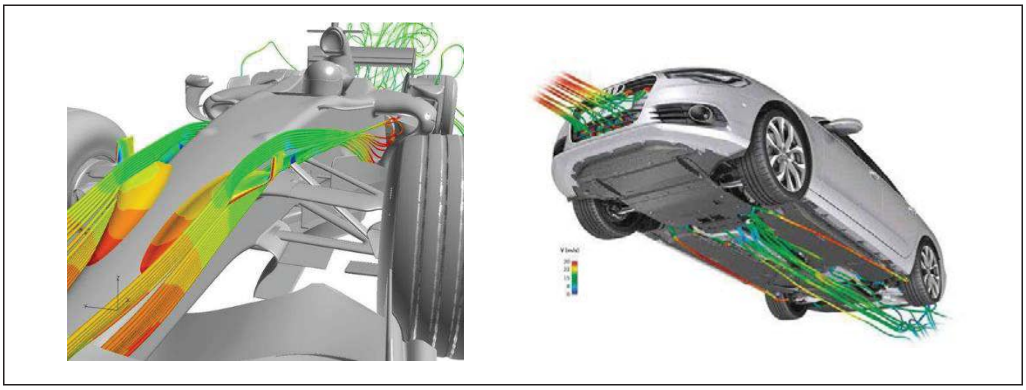


*Figure 2.1 Internal Aerodynamic Airflow (Iru, 2004; James Thompson, 2005)*

### 2.1.2 Internal Aerodynamic Airflow

Internal Vehicle Aerodynamics is an improvement of performance vehicle components which involve in cooling performing due to increased airflow. There are three common areas application of internal vehicle aerodynamics which is ram air system to the intake, ram air to the heat exchanger and lastly ducting to the oil cooler (Badih A. Jawad and Maria M. Longnecker).