AERODYNAMIC INVESTIGATION OF A COMMERCIAL VEHICLE USING CFD

MOHD FADHLAN BIN ZAHARI

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

“I hereby verify that I have read this report and I find it sufficient in term of quality and scope to be awarded with the Bachelor Degree in Mechanical Engineering “

Signature : …………………………………

Supervisor Name : …………………………………

Date : …………………………………

AERODYNAMIC INVESTIGATION OF A COMMERCIAL VEHICLE USING CFD

MOND FADHLAN BIN ZAHARI

This report is submitted to Faculty of Mechanical Engineering in partial fulfill of the requirement of the award of Bachelor's Degree of Mechanical Engineering (Automotive)

Faculty of Mechanical Engineering
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

May 2008

"I hereby to declare that the work is my own except for summaries and quotations which have been duly acknowledge"

Signature : ………………………………….

Author : MOHD FADHLAN BIN ZAHARI

Date : 8th may 2008

DEDICATION

To him who is our source of grace, our source of commitment, and our source of knowledge,
And,
To her, whose love is a source of joy.

ACKNOWLEDGMENT

All praises to the Almighty Allah, for giving me the strength, patience and guidance throughout the process of completing this investigation. I am grateful to have the morally and physically support from many people throughout completing this study. For this opportunity, I would love to thank whose are either directly or indirectly involved during the process of this research is conducted.

Most immediately, I would like to express my very special gratitude and appreciation to my supervisor, Mr. Ahmad Kamal Bin Mat Yamin for his valuable suggestion, comments and advice in every stage of this project. All the discussion makes me understand every detailed in this road vehicle aerodynamic analysis. My token of appreciation also goes to all the lecturers from Faculty of Mechanical Engineering especially from Automotive Department for being very nice and making this study easier to be completed.

Special Thanks should also be forwarded to my all my beloved family members, especially my Papa, Tn. Hj. Zahari Ab. Rahman, and my Mummy, Pn. Hjh. Madihah Haji Salleh, who never failed to be there for their support, prayer and love. Last but not least, I would like to thank all my friends, especially Roslina Aida Bt. Rahimi and all my classmates. Not to forget to all Universiti Teknikal Malaysia Melaka staff that gives cooperation on this project. Thanks you very much.

ABSTRACT

Air poses high aerodynamic resistances over a commercial vehicle due to their box-looked shape and large surface area. However, there is a room to reduce the air resistance so that the fuel consumption and stability of the vehicle can be enhanced. The Computational Fluid Dynamic (CFD) appears to be a good approach since it can serve as a virtual wind tunnel and can give more detail information rather than physical wind tunnel. Hence this project revealed the external flow analysis over a commercial vehicle by means of CFD method to gain further detail aerodynamic characteristics. The one-equation Spalart Allmaras model was chosen as it is widely used in automotive industry and can give appropriate results as other 2-equation model. A commercial vehicle model with scaled 1:87 was aerodynamically tested in a low speed wind tunnel to verify numerical model. The results showed that the pressure drag is dominant rather than skin drag by number of ratio 0.949. Besides, the locations of the separations along the model were manipulated to be visualized and it gave good agreement to experimental.

ABSTRAK

Udara memainkan peranan yang tinngi dalam halangan aerodinamik terhadap kenderaan komersil disebabkan oleh badannya yang bentuk kekotak dan mempunyai kawasan permukaan yang luas. Walaubagaimanapun, masih lagi ada ruang untuk mengurangkan halangan udara ini supaya jumlah pengunaan bahan api dan kestabilan pada kenderaan komersil dapat ditingkatkan. Perisian Dinamik Bendalir Berbantukan Komputer (CFD) menampilkan pendekatan yang baik kerana ia mampu bekerja sebagai terowong angin maya dan mampu untuk memberikan infomasi yang lebih terperinci berbanding terowong angin fizikal. Oleh sebab itu, projek ini mendedahkan analisis aliran luar pada kenderaan komersil dengan mengunakan kaedah CFD bagi mendapatkan cirri-ciri aerodinamik secara terperinci. Model satu-persamaan Spalart Allmaras telah di pilih kerana ia digunakan dengan meluas di dalam industri automotif dan dapat memberikan keputusan yang berpadanan dengan model dua-persamaan yang lain. Satu model kenderaan komersil dengan skala 1:87 telah diuji secara aerodinamiknya di dalam terowong angin berkelajuan rendah untuk dibandingkan dengan model berangka. Keputusan menunjukkan seretan tekanan adalah dominan berbanding seretan permukaan dengan nilai nisbah nya ialah 0.949. Disamping itu, lokasi perpecahan aliran disepanjang model dapat ditonjolkan untuk dilihat and ia dapat memberikan padanan terhadap kaedah eksperimen.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

NUMBER **TITLE** **PAGE**

CHAPTER 1

INTRODUCTION

1.1 Research Background

Aerodynamics is the study of a solid body moving through the atmosphere and the interaction which takes place between the body surfaces and the surrounding air with varying relative speeds and wind direction. The study on road vehicle aerodynamic is quite different from the aircraft aerodynamic study. There are no simple formulate that used in earlier day for determining the drag and lift forces of an aircraft that equivalent to calculate drag and lift of a road vehicle. The aerodynamics of road vehicle does not lend itself readily to mathematical analysis and there are no straightforward methods to predicting the air flow around the vehicle shape.

The aerodynamic of road vehicle research is very important due to promote economic fuel consumption, optimum performance also to improve stability and handling in term of safety. The constant air flow are needed for better fuel economy, greater vehicle performance, reduction in wind noise level and improved road holding and stability for a vehicle on the move.

The size and shape of the commercial vehicle are the major things to consider due to this zero-yaw aerodynamic characteristic investigation. The contributions of this study are primarily deal with zero-yaw flow fields. However, the non-zero yaw flows are still important with respect to real world fuel economical and stability, but in keeping the study objectives of symposium, the investigation coverage has been restricted to an in-depth analysis of zero-yaw flow. The study will include the aerodynamic characteristic for truck container body under zero-yaw flow such as drag and lift force.

A modern Class 7-8 tractor-trailer can weigh up to 8 tons and has a wind-averaged drag coefficient around C_D=0.60 to 0.80. The drag coefficient is defined as the drag/ (dynamic pressure x projected area).

Table 1.1: Approximate ranges of drag coefficient values for various type of vehicle.

Source: Barnard, R.H. [2001]

Vehicle type	Pre-1970	Current	Probable near-future minimum
Medium-sized cars	0.4–0.55	0.28–0.4	0.25
Light vans	0.4–0.6	0.35–0.5	0.3
Buses	0.5–0.9	0.4–0.8	0.3
Large articulated trucks	0.7–0.95	0.55–0.8	0.4
Box truck and drawbar trailer	0.75–1.0	0.7–0.9	0.5

Aerodynamic characteristic usually not obvious at low vehicle speed but the magnitude of air resistance becomes considerable with rising speed. The higher the speed the more energy consumed in overcoming aerodynamic drag. The common highway speed today is at 110 km/h, to overcoming aerodynamic drag represents about 65% of the total energy expenditure for a typical heavy truck vehicle. Reduced fuel consumption for heavy vehicles can be achieved by altering truck shapes to decrease the aerodynamic resistance (drag). It is conceivable that present day truck drag coefficients might be reduced by as much as 50%.

It is estimated that in the year 2012, Class 8 trucks will travel 60 billion highway miles per year. The 60 billion highway miles is predicted by applying a 30% growth factor to the figure of 48 billion miles obtained from the FHWA annual vehicle-travel estimates for 1992 [1]. For a typical Class 8 tractor-trailer powered by a modern, turbocharged diesel engine operating at a fixed specific fuel consumption, bsfc=0.34 pounds/HP-hr., reducing the drag coefficient from 0.6 to 0.3 would result in a total yearly savings of 4 billion gallons of diesel fuel for travel at a present day speed of 70 miles per hour. The mileage improvement is from 5.0 miles per gallon to 7.7 miles per gallon - a 50% savings. (For travel at 60 miles per hour, the equivalent numbers would be 3 billion gallons of diesel fuel saved, and a mileage improvement from 6.1 miles per gallon to 8.7 miles per gallons.)

References: Highway Statistics 1992, p 207, US Government Printing Office, SSOP, Washington DC 20402-9328.

1.2 Objective

The objective of this research is to carry out CFD simulations of aerodynamics over an articulated truck model and validate with the experiment data to gain further understanding in aerodynamic characteristics.

1.3 Research Scope

i. Develop a CFD model and perform zero –yaw characteristic study of aerodynamics of a truck container.

ii. Carry out wind tunnel tests on the truck container model to validate the CFD simulation.

1.4 Research Approach

This research used the Mercedes-Benz Actros truck container model from Welly Die Casting Fty Ltd. The main thing of this research is to make the numerical model to test using Computational Fluid Dynamics (CFD). The results will be validating with the experimental result in order to understand further the aerodynamic characteristics over the commercial vehicle. The research will conducted using GAMBIT 2.2.30 and FLUENT 6.2.16 software, where GAMBIT as a modeler and FLUENT as for simulate. The research also used the wind tunnel facility to investigate the aerodynamic characteristics of the truck container model. The wind tunnel that used in this investigation is from the model MP 130D subsonic tunnels (Essom Company Limited). There is only one things to test that will be execute by using wind tunnel, which is the drag force measurement. The two load component balances are used to measured the drag force. The model is attached with the centre of the test section and a horizontal plate. This plate is needed to create the ground effect during the testing.

1.5 Problem Statement

To design a new vehicle model, the designers have to focus and consider two main characteristics when they designing a vehicle model. The two important things are, first, the segmentation of the model and second, the aerodynamic characteristic. The segmentation can be defined as hatchback, sedan, SUV, MPV, semi trailer van, truck, tanker and flatbeds or any segment related. However, the aerodynamic characteristic only can be determined through experimental or computational method. For this research both method are used.

1. For the first stage of styling process, any designer cannot predict either the model having a good aerodynamic characteristic or not. For this case, the truck container Mercedes-Benz Actros is the

main subject of this research. As a solution, a wind tunnel testing and CFD simulation will be used to measure the aerodynamic characteristic.

2. Since not much people understand the aerodynamic characteristic for the commercial vehicle especially for articulated one. Through this study, it will define how the aerodynamic characteristics of this articulated truck in order to further understanding.

1.6 Limitations

Limitation is problems that occur during completing this investigation and we cannot avoid it. There are a few limitations during the wind tunnel experimental test and listed as follows:

1. The maximum speed can be used of the wind tunnel approximately 30m/s ≈ 108km/h because it is a subsonic downstream fan type.
2. There are no smoke provide to observe the streamline. As a solution, wool turf is use to observe the flow over the whole body.
3. Since the wind tunnel test section is a 1ft x 1ft, the wind tunnel size is not suitable for scale model within the range 1:1 to 1:10 due to blockage factor. The maximum scale model can be used for road vehicle is 1:20.

1.7 Significant of Research

The significance of this research is important to anyone who read this paper, especially for researcher in vehicle aerodynamic or more specific for articulated commercial vehicle aerodynamic. Where as the result of this paper can be useful as one of the reference for any individual or manufacturer to design the road vehicle. This paper emphasizes more on how to implement an aerodynamic experiment and gathered qualitative and quantitative values.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Literature review is about to understand the theory of the research. The literature review is very important part to understand first in any research. Since we know the theory of the research, we can proceed it easily. It is will be easy to determine the way of the research will be implementing and also the related theory that has used in previous research. The information about the research can be found through the journal, internet, thesis and reference books. In this research case, all the information and theory that related to the road vehicle aerodynamic or heavy road vehicle aerodynamic are needed. In this chapter will discus about previous research, fundamental of Vehicle Aerodynamic Force and Moment, theory of road vehicle aerodynamic, theory of wind tunnel method.

2.2 Previous Research

In previous research, there are many research related to heavy vehicle or commercial vehicle aerodynamic. The research title Advanced Aerodynamic Devices to

Improve the Performance, Economics, Handling and Safety of Heavy Vehicles by Robert, J. E [2001], The Research is being conducted at the Georgia Tech Research Institute (GTRI) to develop advanced aerodynamic devices to improve the performance, economics, stability, handling and safety of operation of Heavy Vehicles by using previously developed and flight-tested pneumatic (blown) aircraft technology. Recent wind-tunnel investigations of a generic Heavy Vehicle model with blowing slots on both the leading and trailing edges of the trailer have been conducted under contract to the DOE Office of Heavy Vehicle Technologies. These experimental results show overall aerodynamic drag reductions on the Pneumatic Heavy Vehicle of 50% using only 1 psig blowing pressure in the plenums, and over 80% drag reductions if additional blowing air were available. Additionally, an increase in drag force for braking was confirmed by blowing different slots.

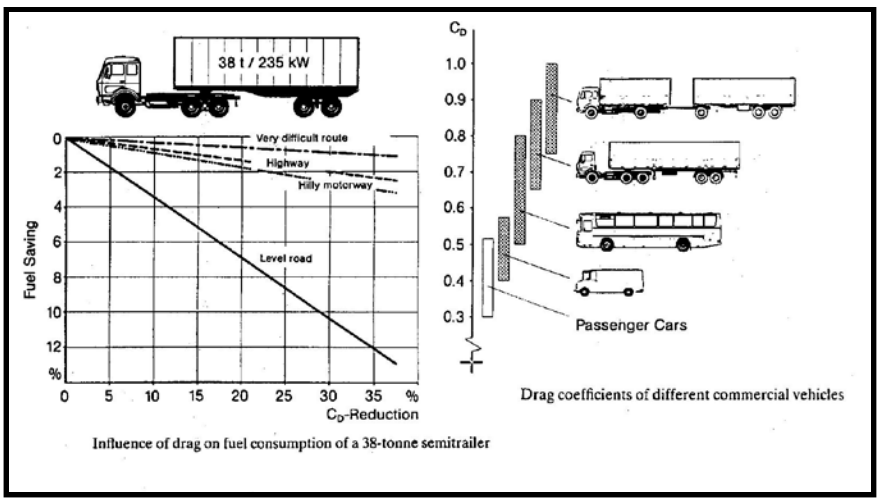

Figure 2.1: Effect Of Drag Coefficient Reduction on Fuel Consumption.
Source: Huncho, W.H. [1990]

Lift coefficient was increased for rolling resistance reduction by blowing only the top slot, while down force was produced for traction increase by blowing only the bottom. Also, side force and yawing moment were generated on either side of the vehicle, and directional stability was restored by blowing the appropriate side slot. These experimental results and the predicted full-scale payoffs are presented in this paper, as is a discussion of additional applications to conventional commercial autos, buses, motor homes, and Sport Utility Vehicles.

In another research, the title Aerodynamic Drag of Heavy Vehicles (Class 7-8): Simulation and Benchmarking by McCallen, R. et al. [2001] describes research and development for reducing the aerodynamic drag of heavy vehicles by demonstrating new approaches for the numerical simulation and analysis of aerodynamic flow. Experimental validation of new computational fluid dynamics methods are also an important part of this approach. Experiments on a model of an integrated tractor-trailer are underway at NASA Ames Research Center and the University of Southern California (USC). Companion computer simulations are being performed by Sandia National Laboratories (SNL), Lawrence Livermore National Laboratory (LLNL), and California Institute of Technology (Caltech) using state-of-the-art techniques.

The research title Vehicle Dynamics Model For Predicting Maximum Truck Acceleration Levels done by Rakha, H. et al. [1999] presents a simple vehicle dynamics model for estimating maximum vehicle acceleration levels based on a vehicle’s tractive effort and aerodynamic, rolling, and grade resistance forces. In addition, typical model input parameters for different vehicle, pavement, and tire characteristics are presented. The model parameters are calibrated/validated against field data that were collected along the Smart Road test facility at Virginia Tech utilizing a truck and trailer for 10 weight-to-power configurations, ranging from 85 kg/kW to 169 kg/kW (140 lb/hp to 280 lb/hp). The model was found to predict vehicle speeds at the conclusion of the travel along the section to within 5 km/h (3.1 mi/h) of field measurements, thus demonstrating the validity and applicability of the model.

The following is an overview of the DOE Heavy Vehicle Aerodynamic Drag Consortium’s accomplishments in the project title DOE’s Effort to Reduce Truck Aerodynamic Drag-Joint Experiments and Computations Lead to Smart Design. The investigation done by McCallen, R.C. et al. [2004] focus areas are Drag reduction devices, Experimental testing, and Computational modeling. This paper submitted for