NUMERICAL SIMULATION AND WIND TUNNEL MEASUREMENTS OF LATERAL AERODYNAMIC CHARACTERISTICS ON SIMPLIFIED AUTOMOTIVE MODEL

MUHAMMAD RIZA ABD RAHMAN

A thesis submitted in fulfillment of the
requirement for the award of the degree of
Master of Mechanical Engineering

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

December 2010

A thesis submitted in fulfillment of the
requirements for the award of the degree of
Master of Mechanical Engineering

Dengan nama Allah yang Maha Pengasih lagi Maha Penyayang..

Teristimewa buat ibu dan ayah yang tersayang, Zainab Mustafa dan Abd Rahman Arifin seluruh keluarga dan sahabat...
Terima kasih atas sokongan kalian sehingga aku mampu mencapai hingga ke tahap ini. Hanya Allah s.w.t sahajalah mampu membalas jasa kalian..

Amin..

Motivator terbaik adalah diri sendiri...

ACKNOWLEDGEMENT

All praise to Allah S.W.T, the Almighty God and the Lord of the Universe, the Merciful and Gracious. Salam to our beloved prophet, Nabi Muhammad s.a.w for His mercy has given me the strength, blessing and time to complete this project.

Words cannot express my thankfulness to my supervisors PM. Dr. Ir. Shuhaimi Mansor who guided me through the whole study with his knowledge and great patience. His excitement and curiosity toward science inspired me a lot. His encouragement has been with me all the time during the years of my study. Without his endless support and guidance, this thesis would not have been very well presented as for now.

I extend my gratitude to En. Iskandar Ishak and Yaheyia Aldreany, who shared their experience and knowledge in simulation and wind tunnel test analysis. I express my deep sense of gratitude and indebtedness to our engineer, Abd Basid Abd Rahman and all staff of Aeronautic laboratory especially Airi Ali for their guidance, advices and motivation while completing this project.

Last but not least, the biggest appreciation to my parents and family and also to everyone for their precious contribution as being supportive for all the time.

Thank you…

ABSTRACT

Computational Fluid Dynamic (CFD) has become an important tool to solve various engineering problems related to aerodynamics. One such growing interest in CFD is to correlate results between CFD and wind tunnel tests. The accuracy of CFD has improved considerably over the years but still large errors are present and lateral aerodynamic characteristics such as drag, side force and yaw moment due to yaw angle are often poorly predicted especially on bluff body shapes. Due to this, comparison between CFD and wind tunnel measurements has become more on demand. The main goal of this research is to investigate the capability of CFD to determine aerodynamic characteristics of simple automotive type bodies and its effect on crosswind stability. An investigation was performed both experimentally and computationally to analyze the main characteristics of flow past a 1:6 scale wind tunnel model of a simplified automotive body shape with different rear slant angles. The investigations were focused on the prediction and measurement of drag, side force, yawing moment and flow characteristics around the model in Reynolds number range of 1.29×10^6 to 2.14×10^6 at various yaw angles. The wind tunnel measurements were performed to provide aerodynamic data on vehicle stability and also to build a database for validating the numerical simulation model. The CFD solver FLUENT 6.3 was used to simulate incompressible three dimensional flow with the standard k-ε turbulent models. The result of the wind tunnel tests and the numerical simulations were found to be in good agreement. The results show that the rear slant angles have significant effect on aerodynamics lateral derivatives.

ABSTRAK

Simulasi Dinamik bendalir berkomputer (CFD) telah menjadi satu alat yang penting dalam menyelesaikan pelbagai permasalahan kejuruteraan yang berkaitan dengan aerodinamik. Antara penggunaan yang semakin meluas pada masa kini adalah mencari perhubungan antara keputusan yang didapati dari CFD dengan ujian terowong angin. Ketepatan CFD semakin baik dari tahun ke tahun tetapi masih terdapat lagi ralat yang besar wujud dan pekali-pekali cirian aerodinamik seperti daya seret, daya sisi dan momen rewang terhadap sudut rewang biasanya kurang tepat terutama bagi bentuk jasad tubir. Oleh kerana itu, perbandingan antara CFD dan ujian terowong angin amat diperlukan. Matlamat utama kajian ini adalah untuk menyelidik kebolehan CFD dalam menentukan ciri-ciri aerodinamik dan kestabilan angin lintang ke atas badan automotif yang dipermudah. Kajian dilakukan secara ujikaji dan simulasi berkomputer bagi menganalisis ciri-ciri utama aliran yang melepasi model badan automotif yang dipermudah berskala 1:6 yang mempunyai sudut belakang yang berbeza-beza. Kajian memfokuskan kepada jangkaan dan pengukuran daya seret, daya sisi, momen rewang dan ciri-ciri aliran udara di sekeliling badan dalam julat nombor Reynolds 1.29×10^6 hingga 2.14×10^6 pada sudut rewang yang berlainan. Pengujian terowong angin dijalankan bagi mendapatkan data aerodinamik bagi kestabilan kenderaan dan juga digunakan untuk mengesahkan simulasi yang dibuat ke atas model. FLUENT 6.3 menggunakan model gelora k-ε dalam simulasi aliran tiga dimensi tak termampat. Keputusan yang diperolehi menunjukkan kaitan yang baik antara pengujian terowong angin dan simulasi. Keputusan kajian ini juga menunjukkan sudut belakang memberikan kesan yang jelas signifikan ciri-ciri aerodinamik.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

A	- model frontal area	m^2
As	- model side area	m^2
Cd	- aerodynamic drag coefficient	
cg	- centre of gravity	
Cn	- aerodynamic yaw moment coefficient	
cp	- centre of pressure	
e_0	- distance center of aerodynamic to center wheel base	m
e_s	- distance center of aerodynamic to center wheel base	m
Cy_β	- aerodynamic side force derivative	rad^{-1}
Cn_β	- aerodynamic yaw moment derivative	rad^{-1}
Cy	- aerodynamic side force coefficient	
ε	- dissipation rate	
I_{zz}	- model rig yaw moment of inertia	$kg.m^2$
k	- kinetic energy	
ℓ	- model characteristic length	m
ℓcp	- distance between *cp* and *cg*	m
ℓwb	- wheel base length	m
l_F	- distance between front axle to *cg*	m
ℓr	- distance between rear axle to *cg*	m
m	- mass of the model	kg
N_a	- aerodynamic yaw moment	Nm
Re	- Reynolds Number	
N_f	- yaw moment fluctuation	Nm
u, v, w	- forward, lateral and vertical speed	$m.s^{-1}$
V	- wind tunnel velocity	$m.s^{-1}$
V_x	- wind tunnel axial velocity	$m.s^{-1}$
V_y	- wind tunnel lateral velocity	$m.s^{-1}$
V_z	- wind tunnel vertical velocity	$m.s^{-1}$
V_f	- lateral velocity fluctuation	$m.s^{-1}$
V_w	- crosswind velocity	$m.s^{-1}$
β	- model yaw angle	deg
β_w	- relative crosswind angle	deg
ρ	- air density	$kg.m^{-3}$
θ	- model angle of rotation	deg
ϕ	- phase angle	deg
ψ	- crosswind angle with respect to vehicle forward speed	deg

CHAPTER 1

INTRODUCTION

1.1 Introduction

Currently, a new environment exist in the industry that want to produce a new design or model in short period and want to reduce cost. One of the best solution for this problem is by using computational fluid dynamic (CFD) simulation. Therefore, computational fluid dynamic is becoming more important and in high demands. Computational fluid dynamic and experiments have their own strengths and limitations. CFD can provide a very detailed view of the flow field, generating velocities, pressure and densities at every point in the domain where it would be very expensive to measure experimentally. However, calculations always approximate the flow in some way, either by solving a simplified equation or by introducing approximations through the numerical method itself. Besides that, the wind tunnel test has the advantage of dealing with a real fluid and measuring the correct physics, though usually not at perfect real conditions (Reynolds number differences) or the right geometry (because of model support interference or wall effects). It often provides good measures of integrated flow properties such as total forces and moments acting on a body.

The aerodynamic characteristics of passenger cars have been a fruitful area of research for several decades, and continue to this day. However, it is well appreciated that there still remains much more things to be learned in this area, and for that purpose, further research is required to understand the complex aerodynamic and flow around the model.

Crosswind stability is an important area of study in vehicle aerodynamic design since it leads to safety issues. The main concern in aerodynamic design for years has been concentrated on reducing the drag for fuel efficiency. Later on, it was found that the streamlined vehicle shapes are sensitive to crosswind disturbance. The styling trend towards rounder shapes especially at the rear of the vehicles and a continuing reduction in aerodynamic drags are suspected to contribute to the crosswind sensitivity (Howell, 1993).

Crosswind sensitivity was one of the major concerns in the design stage process. But this area is still not fully understood. In practical, this behavior sometime will be happen after production. Previously, CFD is used to predict the aerodynamic loads and flow characteristics around the model only but now this research also to predict and see the aerodynamic derivatives.

The ability of computational fluid dynamic (CFD) to predict critical flow characteristics has always been questionable. The accuracy of CFD, has improved considerably over the years but still large errors are present and vehicle parameters such as drag and lift are often poorly predicted. Due to this, comparison between computational fluid dynamic and wind tunnel testing has become demanding. The main goal of this research is to investigate the capability of CFD to determine aerodynamic characteristics on simple automotive type bodies and its effect on crosswind sensitivity. Numerical analysis using CFD modeling and simulation will be compared with experimental results in the wind tunnel.

1.2 Problem Statement

Currently, during the design of a new model both wind tunnel test and computational fluid dynamic will be used. In real application wind tunnel test will consume more cost and time and have limitation in data requirement. To overcome this problem all designer try to change to simulation but the confident level of simulation prediction is still not too accurate compare to wind tunnel test results. In current practice CFD has been used to predict aerodynamic loads and flow field around the model. However, there are few researchers focus on aerodynamic derivatives (side force and moment derivatives) which is very important to estimate the stability of model. The stability of the model play the important role to make sure the shape of the vehicle can be optimized.

1.3 Research Objective

1. To investigate the capability of CFD to determine lateral aerodynamic characteristics on simple automotive type body.

2. To determine the aerodynamic derivative characteristics of a bluff body with various rear slant angles.

1.4 Scope of Work

The current research work is limited to the following:

1. Computational Fluid Dynamic (CFD) simulation using FLUENT 6.3 and wind tunnel test on a bluff body

2. The study is based on a Davis model with different rear slant angles (0^0, 10^0, 20^0, 30^0 and 40^0).

3. Air velocity between 30 to 50 m/s which corresponds to a range of Reynolds number based on model length between 1.29×10^6 and 2.49×10^6.

4. The yaw range was between -16^0 and 16^0 with increment of 2^0.

1.5 Research Methodology

This research comprises of two main parts, Computational Fluid Dynamic (CFD) simulation using Fluent 6.3 and wind tunnel test measurement. In both parts, Davis model with five different rear slant angles (0^0, 10^0, 20^0, 30^0, and 40^0) were test and simulate in various wind speeds ranging from 30 m/s to 50 m/s with interval of 5 m/s. Wind tunnel test has been conducted at Universiti Teknologi Malaysia Low Speed Tunnel (UTM-LST) and forces and moments subjected to the models were measured using six component external balances. In this research, Davis model with rear slant angles of 20^0 become a base model for Validation of simulation and experimental verification with Mansor (2006) works before other rear angles being tested. Both results are then compared to find any correlation between experiment and simulation. The overall flow chart of the research methodology is shown in Figure 1.1.

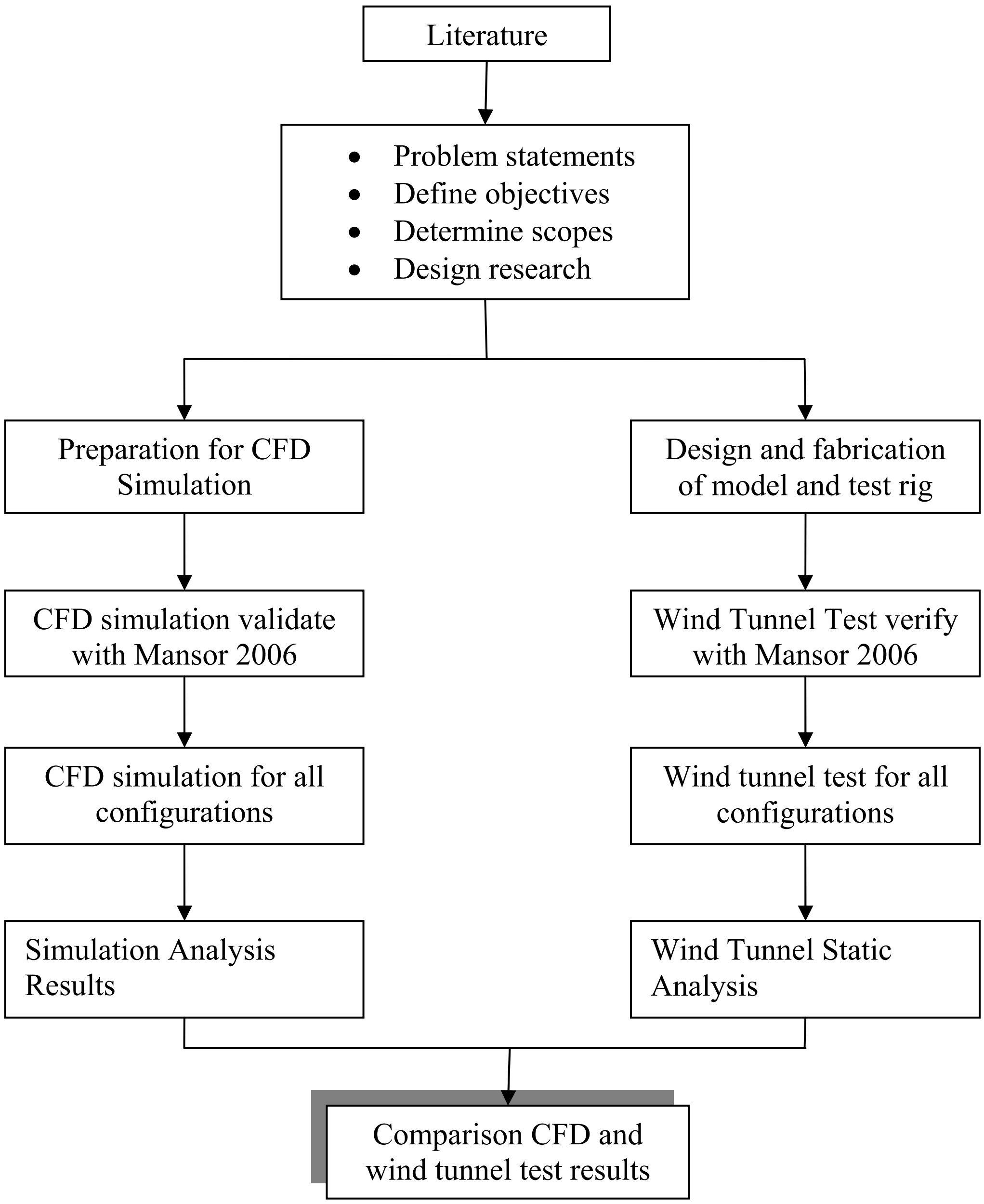

Figure 1.1 Flow chart for research methodology

1.6 Organization of The Thesis

This dissertation is structured in six chapters. The background, a short description of the methodology, motivation and objectives has been presented in this chapter. Chapter 2 is devoted to literature survey, a detailed review of the research work conducted in the area. Chapter 3 briefly discusses the numerical tools, solution procedure and turbulence modeling that are being used in this dissertation. Then, Chapters 4 and 5 investigate bluff body aerodynamics on Davis model. Presented in Chapter 4 are the experimental measurements of drag, side force and yaw moment at various yaw angle from the static wind tunnel tests, and in Chapter 5 the numerical results and their comparison with experimental measurements are provided. Finally, conclusions of the present study and recommendations for future enhancement of the work are given in Chapter 6.