CFD SIMULATION AND VALIDATION OF AERODYNAMIC OVER AHMED MODEL

ROSLINA AIDA BT. RAHIMI

This report is submitted as partial fulfillment of the requirement for the Degree of Bachelor of Mechanical Engineering (Thermal Fluid)

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

MAY 2008

CFD SIMULATION AND VALIDATION OF AERODYNAMIC
OVER AHMED MODEL

ROSLINA AIDA BT. RAHIMI

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

I / We* admit that had read this dissertation and in my / our*opinion this dissertation is satisfactory in the aspect of scope and quality for the bestowal of Bachelor of Mechanical Engineering(Thermal Fluid)

Signature : ……………………..
Name of 1st Supervisor :……………………...
Date :………………………

Signature : ………………………
Name of 2nd Supervisor :……………………....
Date :………………………

“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 : ………………………………….

ACKNOWLEDGEMENT

All praises be to God, The Most Gracious, The Most Merciful for His Guidance and Blessing.

Then goes my deepest appreciation to my first supervisor Mr. Ahmad Kamal who have been very committed in giving me the best guidance in preparing this dissertation through out the first and second semester of my final year. Not to forget my family that gave me endlessly aspiration and strength.

Endless gratitude goes to another lecturer that directly and indirectly involved in the preparation of my dissertation, and also technicians En. Razmi,and En. Kamarudin and few others that helped much through completion of this dissertation .

Last but not least, bundle of thanks to my friend, Mohd Fadhlan that always being with me through out the laboratories sessions and others that had been supporting me directly and indirectly completing the study for this dissertation .

ABSTRACT

Nowadays, computational Fluid Dynamics (CFD) method is widely used as a tool to virtually analyze the external flow problem. However, when it come to vehicle aerodynamics , the visibility of the CFD approach as a virtual wind tunnel to substitute physical wind tunnel has not yet been recognized. Hence, this project revealed the capability of a CFD tool to predict the aerodynamic characteristic of a vehicle. In this project, a simplified vehicle model called Ahmed model was chosen. The result is validated as C_D's value for CFD is 0.50 while for experiment wind tunnel is 0.54. It had proven that Spalart-Allmaras able to validate data between numerical and experimental method with only 7.4% different between both method used. Thus, the capability and consistency of the Spalart-Allmaras equation proof its potential in developing automotive industries.

ABSTRAK

Sejak kebelakangan ini, CFD telah digunakan secara meluas sebagai alat untuk menganalisa secara maya masalah aliran udara. Walaubagaimanapun, di dalam bidang aerodinamik kenderaan, kebolehan pendekatan CFD di dalam menggantikan kajian terowong udara maya seoah-olah sama seperti kajian sebenar terowong udara masih belum lagi mendapat perhatian. Oleh itu, projek ini dibuat bagi mendedahkan kebolehan CFD untuk mendapatkan gambaran karektor aerodinamik bagi kenderaan. Sebuah model ringkas kenderaan telah dipilih iaitu model Ahmed. Berdasarkan keputusan yang diperolehi, nilai pekali geseran bagi kaedah CFD adalah 0.50 manakala nilai pekali geseran bagi kaedah eksperimen terowong udara adalah 0.54. Ini telah membuktikan kebolehan kebolehan Spalart-Allmaras dalam mengesahkan nilai data antara kedua-dua kaedah yang digunakan dengan hanya menunjukkan perbezaan peratusan sebanyak 7.4%. Dengan itu, kebolehan dan keberkesanan Spalart-Allmaras ini telah membuktikan potensinya di dalam pembangunan industri automotif.

.

CONTENTS

LIST OF TABLE

LIST OF FIGURES

LIST OF ABBREVIATION

CFD = Computational Fluid Dynamics

CHAPTER 1

INTRODUCTION

1.1 Background research

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.(Heinz Heisler, 2002) Currently, study of the aerodynamics is the main concern for the vehicle manufacturers in fulfilling the customer needs. People nowadays are always searching for a vehicle that is designed with low fuel consumption over mileage used with the outstanding vehicle performance, less wind noise occurrence, good development of road holding and can constantly maintain the stability of the vehicle on the move. (Heinz Heisler, 2002)

Due to that, there are studies made specifically to focus on road vehicle aerodynamics which is called automotive aerodynamics. Automotive aerodynamics usually focus on the performance of the vehicle in reducing drag, decreasing of wind noise level and more importantly, putting off the undesired lift forces at high speeds. Studies of the automotive aerodynamics were based on two methods which are computer modeling and simulation together with wind tunnel testing.

Forrester T. Johnson *, Edward N. Tinoco, N. Jong Yu had affirmed in their journals that the tremendous cost involved in testing had caused the use of CFD and wind tunnel in developing the aerodynamics design. While Gerardo Franck and Jorge D'El´ stated in their journal, that aerodynamics specialists are constantly seeking new ideas and solutions capable of providing a fast and accurate answer to the design targets which including shorten lead time, lower experimental work and lower the costs associated with vehicle design work. One way to obtain this goal is by combining numerical simulation with experimental measurements in wind tunnel tests. However, the current state of the art in the computational fluid dynamics shows that over the last years accurate results for the automotive aerodynamics expectations have appeared. One of the key points in the development of computational codes is its validation with experimental results

In order to prove the validation of results from computed solution and wind tunnel testing, various types of standards have been invented. One of them was Ahmed model which is designed as a simplified car used in automotive industry to investigate the influence of the flow structure on the drag. The Ahmed model was chosen in this project for its geometrical simplicity, and because of the body that retains the main flow features, especially the vortex wake flow where most part of the drag is concentrated and it is a good candidate to be used as a benchmark test (Franck G. and D'El´ J.)

1.2 Problem Statement

This report was made to compare and validate the credibility between numerical calculated data with actual experimental data with a known model. The model chosen to be assessed is the Ahmed model. Previous literature on Ahmed model shows the difficulty of convergence between numerical data and experimental data. The contributing factor for the validation of the numerical vs calculated data has been identified as the Drag Coefficient (CD). This in turn will effect the Reynolds number (Re) which will be used to determine the validity of the data

1.3 Objective

To appraise the capability of CFD method as a virtual wind tunnel.

1.4 Scope

i) develop a numerical CFD model & physical mode of 1:9 scale
ii) zero-yaw characteristic study
iii) carry out wind tunnel tests to validate the CFD simulation
iv) Simulation at Re = 2.30×10^5
v) Turbulence model : Spalart Allmaras

1.5 Summary

This research is mainly focused on study of aerodynamics on Ahmed model. According to Guilmineau E., Ahmed's body is 1044mm long (L), 288mm high and 389mm width. The slant part is 222mm long with 30^o of angle. This geometry is represented in Fig. 2.1. In this research, the Ahmed design shape will be formed in CAD software which then would be used in extension of manufacturing the model by using Rapid Prototyping machine. The Rapid Prototyping complete model will then be put in as the test specimen to be tested in the wind tunnel on several different configurations. The data gained from the testing would be used as the benchmark for the CFD validation purposes. The wind tunnel testing is important as it determines the measure of accuracy of a real test specimen in comparison to the computed modeling solution. In this case, the test model would be experimented in wind tunnel at zero yaw condition with certain velocity that can be provided by wind tunnel utilized. Application of the state-of-the-art CFD software, GAMBIT and FLUENT with application of Spalart-Allmaras viscous model would be used for modeling and analysis of aerodynamics characteristic over the Ahmed structure, similar to the one tested in the wind tunnel. Several major factors have been identified and will be researched in this study. The most significant factor is the flow region which presents the major contribution to a car's drag is the wake flow behind the vehicle. Other than that, there would be an investigation on the location at which the flow separates which determines the size of the separation zone, and consequently the drag force. Then, a more exact simulation of the wake flow and of the separation process will be made and is essential for the accuracy of drag predictions. After the simulation is made and taking into account all the contributing parameters, certain output data will put into related equation to figure out value of coefficient of drag. The value of coefficient of drag gained from simulation will then validate with the value of C_D from the wind tunnel testing result by making the comparison from the established theoretical value of C_D as reference.

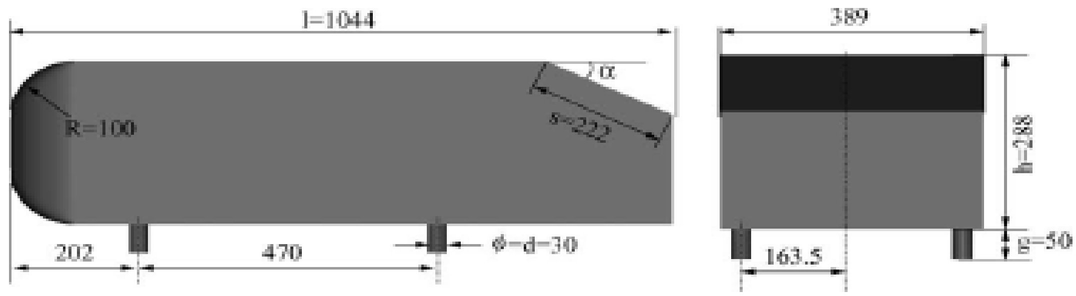

Figure 2.1: Figure of Ahmed model

CHAPTER 2

LITERATURE REVIEW

2.1 Aerodynamics Background History

The history of the word “aerodynamics” actually came from the Greek philosopher Aristotle in 350 B.C. by his observation of fluids and their effect on objects. Aristotle conceived the notion that air has weight and observed that a body moving through a fluid encounters resistance.(www.wikipedia.org)

Archimedes, another Greek philosopher, also has a place in the history of aerodynamics. A hundred years later, in 250 B.C, he presented his law of floating bodies that formed a basic principle of lighter-than-air vehicles and stated that a fluid—either in a liquid or a gaseous form—is continuous, basically restating Aristotle's theory of a hundred years earlier. He comprehended that every point on the surface of a body immersed in a fluid was subject to some force due to the fluid. He stated that, in a fluid, “each part is always pressed by the whole weight of the column perpendicularly above it.” He observed that the pressure exerted on an object immersed in a fluid is directly proportional to its depth in the fluid. A direct proportional relationship means that if one part increases, the other will increase by the same factor. In other words, the deeper the object is in the fluid, the greater the pressure .(www.wikipedia.org)

In 1490, the Italian painter, sculptor, and thinker Leonardo da Vinci began documenting his aerodynamics theories and ideas for flying machines in personal notebooks. He first believed that birds fly by flapping their wings, and thought that this motion would have to occur for manmade aircraft to rise. He later correctly concluded that the flapping of the wings created forward motion which allowed air to pass across the bird's wings to create lift. It was the movement of the wing relative to the air and the resulting reaction that produced the lift necessary to fly. He then noticed that water in a river moved faster at a greater velocity where the river narrowed. In numerical terms, the area of a cross-section of a river multiplied by the velocity of the water flowing through that section equals the same number at any point in the river. This is known as the law of continuity (Area x Velocity = constant or AV = constant). The law of continuity demonstrates the conservation of mass, which is a fundamental principal in modern aerodynamics. He also observed the different ways in which a fluid flowed around an object—called a flow field. (www.wikipedia.org)

Leonardo also stated that the aerodynamics results are the same if an object moves through the fluid at a given velocity or if the fluid flows past the object at rest at the same velocity. This became known as the "wind tunnel principal." For example, the results are the same aerodynamically whether a runner moves at 10 miles per hour in calm air and if the wind is blowing at 10 miles per hour past a stationary person. He also determined that drag on an object is directly proportional to the area of the object. The greater the area of an object, the greater the drag it will produce. Further, Leonardo pointed out the benefits of streamlining as a way to reduce an object's drag. (www.wikipedia.org)

Scientists working in the 17th century contributed several theories relating to drag. The Italian mathematician and inventor Galileo Galilei built on Archimedes' work and discovered that the drag exerted on a body from a moving fluid is directly proportional to the density of the fluid. Density describes the mass of an object per unit volume. A very dense fluid produces more drag on objects passing through it than a less dense fluid. The density of air (a fluid) changes with its distance from the Earth's surface, becoming less dense the farther it is above the Earth's surface and, as such, exerting less pressure. .(www.wikipedia.org)