I hereby declare that I have read this thesis and in my opinion this thesis is sufficient in terms of the scope and quality for the award of the degree of Bachelor Degree of Mechanical Engineering (Thermal Fluid)

Signature	: ....
Name of Supervisor	: EN. SHAMSUL BAHARI
Date	: 08-05-07

I declare that this thesis entitled "Flow Investigation around Immersed Bodies in Subsonic Wind Tunnel" is the result of my own research except as cited in the references.

Signature : ....

Name : MASTURA MAAROF

Date : 08-05-07

FLOW INVESTIGATION AROUND IMMERSED BODIES IN SUBSONIC WIND TUNNEL

MASTURA MAAROF

This report is submitted to Mechanical Engineering Faculty in partial fulfillment of the requirements for the award of the Degree of Bachelor Degree of Mechanical Engineering (Thermal Fluid)

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

April 2007

ACKNOWLEDGMENT

Alhamdulillah, Syukur to Allah saw for blessing me.

First and foremost, I would like to express my heartfelt gratitude and appreciation to my supervisor Mr. Shamsul Bahari Bin Azraai, who give me good guidance all throughout the preparation of this thesis. Thanks for his time to help and guide me from the beginning until the completion of this thesis. He was very helpful and tirelessly extended his help when I needed the most. I owe him so much for his advice, patience, encouragement, ideas and suggestions throughout this thesis.

I also like to thank to all lectures and technicians of Mechanical Faculty, who had contributed directly or indirectly to ensure the success of this thesis. To my beloved family and friends (FKM UTeM and FKM UiTM), whose name was not mentioned here, thank you for their ideas and moral support.

Without helps from this people, this thesis cannot be successfully completed. All your kindness will be remembered and appreciated. Thank you very much.

ABSTRACT

This thesis focuses on investigation and discussion of the characteristics of fluid flow around bluff bodies in which the simulations are done using Computational Fluid Dynamic and experimentally in subsonic wind tunnel.

Various models have been tested to developed understanding and proper modeling techniques for the flow around such bodies. Throughout this thesis, the Large Eddy Simulation has been validated through comparative study with experiment work.

This research section concentrates on modeling flow characteristics around bluff bodies to investigate the impact of fluid flow on them. This aids in the understanding of a more complex flow in engineering applications.

The major part of the work discusses on the drag force of flow around simple circular cylinder and flat plate. From the analysis, the results state that the high velocities produced large drag forces. Both of CFD computations and experiments have been performed in the different range of Reynolds numbers between 50000 to 200000 that tested at the different shape of the bodies.

Finally, the conclusions outline the achievements and findings of the work done in this thesis and give recommendations for further research on the topic.

ABSTRAK

Tesis ini memfokuskan kepada kajian dan perbincangan tentang ciri-ciri aliran bendalir di sekeliling objek-objek tumpul dimana simulasi telah dijalankan dengan menggunakan kaedah CFD dan secara experimentnya di dalam terowong angin subsonik.

Beberapa jenis model terlah di uji untuk penghasilan pemahaman dan teknik permodelan yang lengkap untuk aliran disekeliling objek-objek tersebut. Secara keseluruhannya, LES telah disahkan melalui perbandingan kaedah experimen.

Penyelidikan ini tertumpu pada ciri-ciri aliran disekitar objek untuk mengkaji kesan-kesan aliran bendalir ke atas objek-objek itu. Ianya bertujuan untuk memberi pemahaman terutamanya mengenai aliran dalam aplikasi kejuruteraan.

Sebahagian besar daripada projek ini membincangkan daya seretan aliran disekeliling silinder bulat dan plat rata yang ringkas. Daripada analisis, keputusan menunjukkan halaju yang tinggi menghasilkan daya seretan yang besar. Kedua-dua kaedah iaitu simulasi dan experimen dihasilkan dengan berpandukan julat nombor Reynolds antara 50 000 hingga 200 000 yang telah di uji keatas beberapa bentuk objek yang berbeza-beza.

Kesimpulannya, beberapa pencapaian dan penemuan telah diperolehi dan disertakan dengan beberapa cadangan untuk kajian yang seterusnya.

CONTENTS

LIST OF FIGURES

LIST OF FIGURES

LIST OF TABLES

LIST OF GRAPH

LIST OF SYMBOLS

SYMBOL	DEFINATION
D	Drag Force, N
L	Lift Force, N
ρ	Density
U	Velocity, m/s
V_∞	Inlet Velocity, m/s
A	Area of plate, m^2
α	Angle of attack, ^{0}
C_L	Coefficient of Lift
C_D	Coefficient of Drag
Re	Reynolds Number

LIST OF ABBREVIATIONS

SYMBOL	DEFINITION
CFD	Computational Fluid Dynamic
LES	Large Eddy Simulation
3-D	Three Dimensional
2-D	Two Dimensional
RANS	Reynolds Average Narier Strokes
DES	Detach Eddy Simulation
HAVC	Heating Ventilation Air-Conditioning

CHAPTER 1

INTRODUCTION

1.1 Research Background

Introductory Concepts

1.1.1 Flow past Immersed Bodies

Flows past immersed bodies, or external flows, are flows of fluids surrounding objects with solid walls, while, ideally, subject to no outer boundary, except at particularly infinite distance from the object. Examples are flows around ground structure or land, marine and air vehicles. In contrast, internal flows are those in which the following fluid in the surrounded by solid walls, such as flows though pipes, ducts and channels or within fluid machinery.

An object immersed in a flowing fluid will generally experience a force, whose magnitude and direction depend on several parameters. The source of this force is the fiction between the fluid and the solid wall. An entirely immersed solid object, heaving a constant relative velocity with respect to an unbounded, incompressible, frictionless, will experience a zero force, although not necessarily a zero moment.

In general, the flow pattern around bodies depends on the place of observation, while the various flow parameters, such as velocity and pressure, vary along any direction. This is the case of three dimensional flows. However, if the object is assumed to extend to infinity o two opposite sides and has a uniform cross section normal to certain axis and if the fluid motion is such that the flow pattern and all flow parameters are independent of position along that axis, the flow is call two-dimensional or planar. Obviously, the total force on 2-D immersed objects would be infinite and so it becomes necessary to refer to force to a unit span.

For this case study, the consideration is more to flows past a bluff body which have been studied extensively because of their many practical applications and very importance in engineering fields. It also call external flow where so effective and safe design of machine components, aircraft sectors, automotive bodies, missiles, propeller, turbines, compressors, ships, trains and structures can benefit from a general understanding of this flows. Also the air past buildings and bridges are other examples of external fluid flows.

There are various phenomena associated with flow around bluff body is one of the most important of these is the force acting on the body due to the fluid. The forces namely as a drag forces, lift forces, shear force and also pressure force. The drag, that component of the force parallel to the direction of and resisting the motion, is of concern in most flows. The component of force normal to the direction of motion, the lift, is also of obvious importance in many flows. In certain applications, it needed to increase the lift force of the body and the same time needed to reducing the drag force the acting the bodies.

1.1.2 Drag Force

Drag is the force exerted on a body moving in a medium depends in a complex way upon the velocity of the body relative to the medium, the viscosity and density of the medium, the shape of the body and the roughness of it surface. The shape the object determines the amount of drag which is produced as shown in Figure 1.1. Objects that are streamlines produce the least amount of drag, while, for bluff or blunt objects, produce high amount of drag. The direction of the drag force is always opposite the direction of the body's velocity.

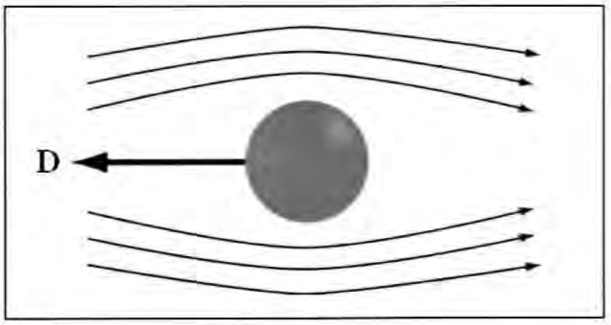

Figure 1.1: Drag Notation

There are three kinds of drag. First is friction drag. It occurs next to the surface of an object. It affects all vehicles. The second type is form drag that has eddies which air is flowing past an objects breaks away from the surface to form litter swirling pockets of air. These take energy from the object and slow it down. It occurs with non-streamlined objects. The third kind of drag is induced drag. This type of drag affects only airplanes.

The most common method of mathematically modeling the drag force is the equation,

Drag Force;

$$D = \frac{1}{2}\rho U^2 A \quad (1.1)$$

Drag Coefficient;

$$C_D = \frac{(D/A)}{\frac{1}{2}U^2\rho} \quad (1.2)$$

D = Drag Force. SI: N

C_D = Drag Coefficient. SI: Dimensionless

A = Cross-sectional area perpendicular to the flow. SI: m^2

ρ = Density of the medium. SI: kg/m^3

V or U = Velocity of the body relative to the medium. SI: m/s

The drag coefficient C_D is not constant. C_D depends upon the velocity of the body, viscosity of the medium, the shape of the body, and the roughness of the body's surface.

1.1.3 Lift Force

The concept of lift force on the immersed bodies is generated any time there is flow asymmetry about a plane parallel to the flow direction. Example of lift generation mechanisms included; a symmetric object with its axis inclined with respect to the flow direction (Figure 1.2), any asymmetric object at any arbitrary orientation, any object in shear flow, any object in density, stratified flow and a rotating object.

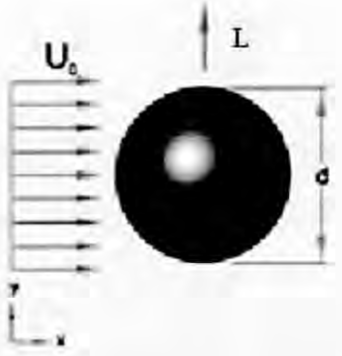

Figure 1.2: Lift Notation

For many practical applications, the computation of lift based on in viscid flow theory is of sufficient accuracy (it is reminded that such approaches are entirely inadequate for the computation of drag). A particularly suitable method for aerodynamic application is the vortex panel method.

The most common method of mathematically modeling the lift force is the equation,

Lift Force;

$$L = \frac{1}{2}\rho U^2 A \tag{1.3}$$

Lift Coefficient;

$$C_L = \frac{(L/A)}{\frac{1}{2}U^2\rho} \tag{1.4}$$

L = Lift Force. SI: N

C_L = Lift Coefficient. SI: Dimensionless

A = Cross-sectional area perpendicular to the flow. SI: m^2

ρ = Density of the medium. SI: kg/m^3

V or U = Velocity of the body relative to the medium. SI: m/s

1.1.3 Reynolds Number

The Reynolds number has been found to be a useful dimensionless number that can characterize the drag coefficient's dependence upon the velocity. It is the ratio of internal forces to viscous forces and consequently it quantifies the relative importance of these two types of forces for given flow conditions. Thus, it is used to identify different flow regimes such as laminar or turbulent flow. Re is very important in fluid dynamics and is used usually along with other dimensionless numbers, to provide a criterion for determining dynamic similitude.

$$\mathrm{Re} = \frac{\rho VL}{\mu} = \frac{VL}{\nu} = \frac{\rho VD}{\mu} \qquad (1.5)$$

Re = Reynolds number. SI: Dimensionless

ρ = Density of the medium. SI: kg/m^3

V = Velocity of the body relative to the medium. SI: m/s

μ = Dynamic Viscosity of the medium. SI: N s/m^2

L or D = Characteristic length of the body along the direction of flow. SI: m

For small values of the Reynolds number, it calls laminar flow, while for the large Re, the flow is turbulent.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

The development of certain project needs to be done systematically and does not exceed the time limit. In order to solve this problem, a concrete planning needs to be done. There are a few things that need to be taken off when doing research to complete this chapter. Literature reviews are important to know what the other researchers achieved during their research on the related investigations. From that, the comparison between experimental and simulation results can be achieve to evaluate a good conclusion. In addition, by choosing the right of methodology, the needs on knowing the right hardware and software requirement and the laboratory setup are also need.

2.2 Fact and Finding

In according to develop and finish this project, the fact that related to this investigation must be searching by internet or books and gather all the information. The best way to get the related information is form previews journal either resources from local or international journal. For my research, the Journal of Fluids Mechanics and Journal of Fluids Engineering or Fluids Flow is needed to find the related research articles. Other than that, the references from similar books are also help to solve the problems and helps complete this project.

2.3 Objective of Research

The objectives of this research are, to determine the lift and drag forces at the different of Reynolds number on immersed bodies with different type of plates. The research included a comparison between the experimental results and simulation results by CFD method.

For laboratory case studies, there are six bodies that used to compute the results from different type of shapes. The shapes included square plate, hemisphere, circular cylinder, and etc. (Refer Chapter 3: Methodology).

The investigation will be interested in the drag (D) of models and also the lift (L), which the two basic aerodynamic forces. Let take flat plate as an example to describe the theoretical of this both forces. These forces are shown in the Figure1.1 for a two-dimensional flow over flat plate heaving theoretically infinite length. The angle formed between plate and the direction of flow (V), is the angle-of-attack (α).

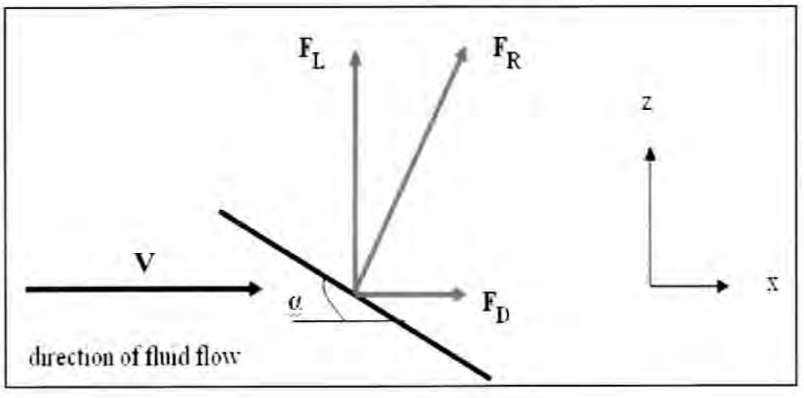

Figure 2.1: Difference between lift (L) and drag (D) versus normal force (N) and axial force (A)