A STUDY OF DRAG FORCE ON DIFFERENT TYPE OF AIRFOIL IN A SUBSONIC WIND TUNNEL

MUHAMAD FARUQI BIN ZAHARI

BACHELOR OF MECHATRONIC ENGINEERING

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

DECLARATION OF THESIS AND COPYRIGHT

Author's full name : ______________________

Date of birth : ______________________

Title : ______________________

____

____

Academic Session : ____________

I declare that this thesis is classified as :

☐ **CONFIDENTIAL** (Contains confidential information under the Official Secret Act 1972)*

☐ **RESTRICTED** (Contains restricted information as specified by the organization where research was done)*

☐ **OPEN ACCESS** I agree that my thesis to be published as online open access (Full text)

I acknowledge that Universiti Malaysia Pahang reserve the right as follows:

1. The Thesis is the Property of University Malaysia Pahang
2. The Library of University Malaysia Pahang has the right to make copies for the purpose of research only.
3. The Library has the right to make copies of the thesis for academic exchange.

Certified By:

______________________	______________________
(Student's Signature)	(Signature of Supervisor)
______________________	______________________
New IC / Passport Number Date :	Name of Supervisor Date :

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechatronic Engineering.

Signature: ……………………………

Name of Supervisor: MR SHAIFUL HAKIM MOHAMAD NOOR

Position: LECTURER

Date: 25 JUNE 2013

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature: ……………………………

Name: MUHAMAD FARUQI BIN ZAHARI

ID Number: FB09012

Date: 25 JUNE 2013

Dedicated to my parents

Zahari Bin Ismail

Rahana Binti Che Soh

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Mr. Shaiful Hakim Bin Mohamed Noor for his invaluable guidance, continuous encouragement and constant support in making this study possible. I really appreciate his guidance from the initial to the final level that enabled me to develop an understanding of this research thoroughly. Without his advice and assistance it would be a lot tougher to completion. I also sincerely thanks for the time spent proofreading and correcting my mistakes.

I also would like to express very special thanks to Assoc. Prof. Dr. Abdul Aziz Bin Jaafar for his suggestions and co-operation especially in Computational Fluid Dynamic study. My sincere thanks go to all lecturers and members of the staff of the Manufacturing Engineering Department, University Malaysia Pahang (UMP), who helped me in many ways and made my education journey at UMP pleasant and unforgettable. Many thanks go to fellow friends for their excellent co-operation, inspirations and supports during this study. This four year experience with all you guys will be remembered as important memory for me to face the new chapter of life as an engineer.

I acknowledge my sincere indebtedness and gratitude to my parents, Zahari Bin Ismail and Rahana Binti Che Soh for their love, dream and sacrifice throughout my life. I am really thankful for their sacrifice, patience, and understanding that were inevitable to make this work possible. Their sacrifice had inspired me from the day I learned how to read and write until what I have become now. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to achieve my dreams.

Lastly I would like to thanks any person which contributes to my final year project directly or indirectly. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

There had been a number of researches that investigated on drag force of airfoil in subsonic wind tunnel. This study was then conducted in order to identify drag force in subsonic wind tunnel using Computational Fluid Dynamics (CFD) software. Specifically, this research aimed to identify the drag force on ten different types NACA airfoil. NACA 0006, NACA 0009, NACA 0015, NACA 1408, NACA 1410, NACA 2408, NACA 2418, NACA 4424, NACA 6409 and NACA 6412 airfoil geometry profiles and its coordinates were generated from a NACA 4 Digits Series Generator and then designed by using ANSYS (Fluent) geometry and computed using ANSYS 14.0 software , these Computational Fluid Dynamic was used to simulate the external flow analysis on the airfoils and then the prediction of the drag force validate its simulation result with experimental result of drag force in subsonic wind tunnel. Nonetheless, the boundary condition was operated at a nominal velocity 17 m/s during the coefficient measurements, a Reynolds's number of about 1,163,798. The airfoil, with a one meter chord, was analyzed at zero degree angles of attack. Moreover, the result show NACA 4424 that have highest drag force and NACA 0006 have the lowest drag force at same boundary condition. Furthermore, the drag was generated was too small to make any significant change to the airfoil performance.

ABSTRAK

Terdapat beberapa kajian yang dijalankan untuk mengesan daya seretan terhadap bentuk aerofoil dalam terowong udara pada kelajuan subsonik. Kajian ini dijalankan untuk mengenal pasti daya seretan di dalam terowong udara pada kelajuan subsonik menggunakan perisian Pengiraan Cecair Dinamik (CFD). Khususnya, kajian ini bertujuan untuk mengenal pasti daya seretan terhadap sepuluh jenis NACA aerofoil. NACA 0006, NACA 0009, NACA 0015, NACA 1408, NACA 1410, NACA 2408, NACA 2418, NACA 4424, NACA 6409 dan NACA 6412, profil geometri aerofoil dan koordinat yang dijana dari NACA 4 Digit Generator dan kemudian direka dengan menggunakan ANSYS (Fluent) geometri dan menggunakan perisian ANSYS 14.0 telah digunakan untuk mensimulasikan analisis aliran ke atas aerofoil dan kemudian andaian daya seretan hasil simulasi disahkan dengan hasil eksperimen daya seretan di dalam terowong udara pada kelajuan subsonik. Walau bagaimanapun, andaian keadaan sempadan yang telah dijalankan pada kelajuan nominal 17 m / s semasa pemalar seretan, manakala nombor Reynolds ini kira-kira 1,163,798. Aerofoil ukuran satu meter pada tunjang, aerofoil telah dianalisis pada sudut sifar bagi sudut serangan. Selain itu, hasil menunjukkan NACA 4424 yang mempunyai daya seretan tertinggi dan NACA 0006 mempunyai daya seretan terendah pada keadaan sempadan yang sama. Tambahan pula, daya seretan dihasilkanoleh aerofoil terlalu kecil untuk membuat apa-apa perubahan terhadap prestasi aerofoil.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

Re	Reynolds's Number
F_L	Lift Force
ρ	Fluid Density
C_L	Lift Coefficients
V	Fluid Velocity
A	Airfoil Area
C_D	Drag Coefficients
F_D	Force Coefficients
kT	Turbulence Kinetic Energy
x	Specific Dissipation Rate
kL	Laminar Kinetic Energy
C_P	Pressure Coefficients

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamic
NACA	National Advisory Committee of Aeronautics
CAE	Computer- Aided Engineering
CAD	Computer- Aided Design

Chapter 1

INTRODUCTION

1.1 Overview

This chapter is discussed about the project background, the problem of the project, the objectives of the project and project scope.

1.2 Project Background

1.2.1 Wind Tunnel

A wind tunnel is a tool that used for aerodynamic test in order to study the effects of linear flow of air moving past a solid objects. The theory of the operation was first proposed as mean of studying vehicle in free flight. Then, the wind tunnel was reversing the usual paradigm instead of the air's standing still and the model moving with speed through it, the same effect would be obtained if we reversed as the model is standing still and the air will move at speed past the model.

Nowadays, that direct investigation with complex equipment and special measuring techniques are used in different types of models, and for testing separate elements of these machines. There are several types of wind tunnel in order to produce flows of air that simulate the natural flow that occur outside the laboratory.

The wind tunnel types depend on the flight speed and have been divided into five ranges which is low subsonic speeds, high subsonic speeds, transonic speeds, supersonic speeds and Hypersonic speeds. Apparently, low speed wind tunnels are one type of tools in the teaching of aerodynamics or fluid mechanic. The basic test methods for large and small wind tunnel are similar which is for measuring forces, pressures and speeds. While, the main method that used in wind tunnel research which determine the success of aerodynamics and its application in technology is the testing in wind tunnel.

1.2.2 Load Balance

The aerodynamic forces and moments acting on models tested in wind tunnels can be determined indirectly by measuring the pressures at many points of the model surface. A more accurate and reliable method is the direct measurement of the forces and moments with the aid of wind-tunnel balances.

Furthermore, the main characteristic of wind tunnel balances is the number of measured components. It depends on the considered situation and this number can vary from 1 to 6 of measured components. Wind tunnel balances can be divided into two types which is balances located outside the model and test section and balances located inside the model or its supports.

For the first type balances design which is external of the model and test section, total aerodynamic forces are resolved into component with aid of various mechanism such as spring, string, weight scale and etc. This type of balances will be called mechanical balances. While, for the second type of balances which is internal design wind tunnel balances will become possible with the development of the strain gages measurement methods, this method was used during the past two decades.

Sensitivity and accuracy of the wind tunnel balances depend on the design of lever or hinges or rods. So, there are several design requirements that need to look up which is small friction during measurement displacement, high sensitivity, high accuracy of the transmission ratios of the system, rigidity of all levers or hinges or rods which is necessary for minimum distortion of the system under the action of the loads and others than mentioned. In addition, main characteristics of the wind tunnel balances are their load capacities, accuracy and how fast responses were deliver.

1.2.3 Computational Fluid Dynamics

To assist development of aerodynamic on NACA 4 digit airfoils now days, we use some software package of Computational Fluid Dynamics (CFD). The main advantage is that the results are obtained without construction of the required prototype. The major concern over a software simulation is the validity of its results. The accuracy of the obtained results cannot be guaranteed for a given study. Hence, before analyzing the results obtained from the CFD simulations, a validation study has to be carried out in order to know the specific parameters and conditions under which the software yields the most accurate results when compared to a set of established data. (Henrik D, 2008)

1.3 Problem Statement

Wind tunnel is an equipment that important in order to measured aerodynamic force that will occur in real flight. Based on that, we have design the airplane that will overcome the aerodynamic force which can crushed the airplane. So, the force balance is important to design for force been measured.

Todays, in Faculty of Manufacturing had built the wind tunnel but still lack of the force balance which is to measured aerodynamics force in a wind tunnel. So, we have been decided to simulate 10 different type of NACA airfoil first, in order to have simulation data before design and fabricate the force balance in order to complete the construction of a wind tunnel in Faculty of Manufacturing at University Malaysia Pahang.

1.4 Objective of the project

The main purpose of this project is to investigate the drag force on different type of NACA airfoil using Computational Fluid Dynamics Software. Besides that, this project will identify the relationship between type NACA airfoil and drag force in Computational Fluid Dynamics Software.

1.5 Scope of the project

The scope of the project covered the geometry of ten different type of 4 digit NACA airfoil based on 4 digit NACA generator to generated geometry data and analyze it using commercial software, ANSYS. Data validation was compulsory in order to validate data that compared with simulation and experimental data of NACA 0015 by consideration of Re = 232940 @ 2.33×10^5 and 17m/s as inlet velocity. After simulation data was validated, continue on others nine types of NACA airfoils and find the drag force on them. This research using 17 m/s as the velocity design and Re = 11.64×10^5 for ten different geometry of NACA 4 digit airfoils.

Chapter 2

Literature Review

2.1 Overview

The purpose of this chapter is to provide a review of past research efforts related to measure the aerodynamic forces. It also includes the important component in the Computational Fluid Dynamics. From the related journal and article, the idea in simulate the aerodynamics force in wind tunnel is developed.

2.2 Aerodynamic force

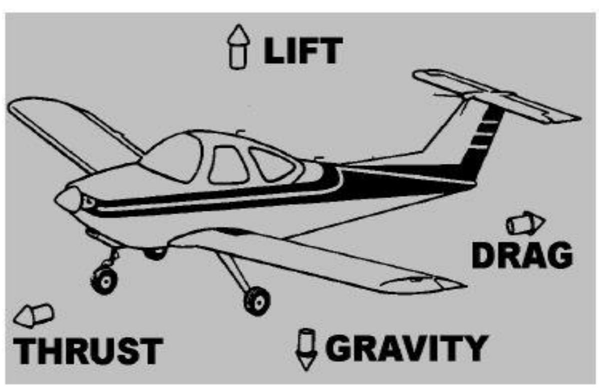

Figure 2.1: Forces acting on the plane

Source: (*http://www.allstar.fiu.edu/aerojava/flight11.htm*)

An airplane in flight is the centre of a continuous tug of war between the four forces which is lift, gravity force or weight, thrust, and drag. Lift and Drag forces are considered aerodynamic forces because they exist due to the movement of the aircraft through the air.

2.2.1 Drag force

Drag is the aerodynamic force that opposes a model motion through the air. It is a mechanical force that is by the contact of a solid body with a fluid (liquid or gas). It is not force generated by gravitational effect or electromagnetic field effect where the object can affect another object without being in physical contact. If there is no fluid, the drag force does not occur because of the drag to be generated, the model must have physical contact with the fluid. Furthermore, drag generated by the difference velocity between two objects, which is it will make no difference if the moving object past static fluid or the moving fluid oast the static object. If there is no motion, there is no drag because drag acts in the direction that opposes the motion. So, it must be a motion for both of the object and the fluid. (Mehrdad, 2001)

In addition, in the wind tunnel there a concept which is moving fluid past through static object, as mentioned before, if there are no motion for fluid and model there are no drag. But, in the wind tunnel we considered the lift force that acts perpendicular in motion will generate the motion of objects and will generate the drag force for the model.

2.3 Background of airfoils

In the late 1800's, the development of airfoil sections had been started. Early, it was known that flat plates would produce lift when set at an angle of attack. However, some suspected that shapes with curvature are more closely resembled bird wings would produce more lift or do so more efficiently. In 1884, a series of airfoils shapes was tested in one of the earliest wind tunnels which the artificial currents of air produced from a steam jet in a wooden trunk or conduit by H.F. Philips.

According to Octave Chanute in 1893, the difference between success and failure of a proposed flying machine will depend upon the sustaining effect between a plane surface and a curve to get a maximum of lift. Furthermore, Otto Lilienthal have same opinions with Octave Chanute, he was carefully measuring the shapes of bird wings and tested the airfoils as shown in Figure 2.2 on a 7m diameter whirling machine. According to Lilienthal, the main factor of successful flight was wing curvature or camber after he experimented the different of nose radii and thickness distributions.

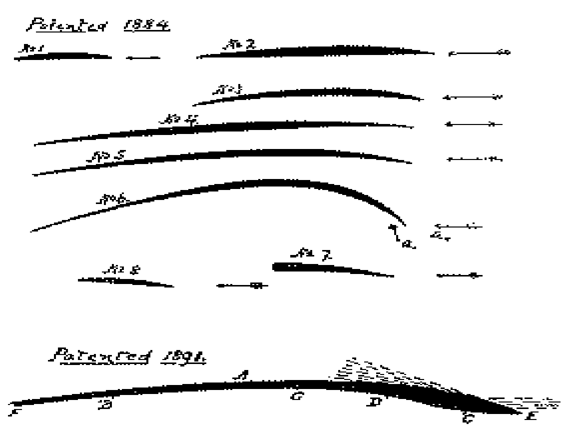

Figure 2.2: Phillips patented Aerocurves

Source: (http://adg.stanford.edu/aa241/airfoils/airfoilhistory.html)

After that, the Wright Brothers closely resembled Lilienthal's sections which is thin and highly cambered in design on the airfoils. Early tests of airfoil sections thin sections was better than thick sections because at time the test was conduct at an extremely low Reynolds number. As result, the first airplane were biplanes because of wrong belief that the efficient airfoils had to be thin and high cambered.

Over a decade, the use of thin and high cambered was gradually decreased based on the trial and error. In the early 1920, NACA started test the section using thick and high cambered based on successful of experiments by Clark Y and Gottingen as the basic for a family of sections.

2.4 Airfoils Nomenclature

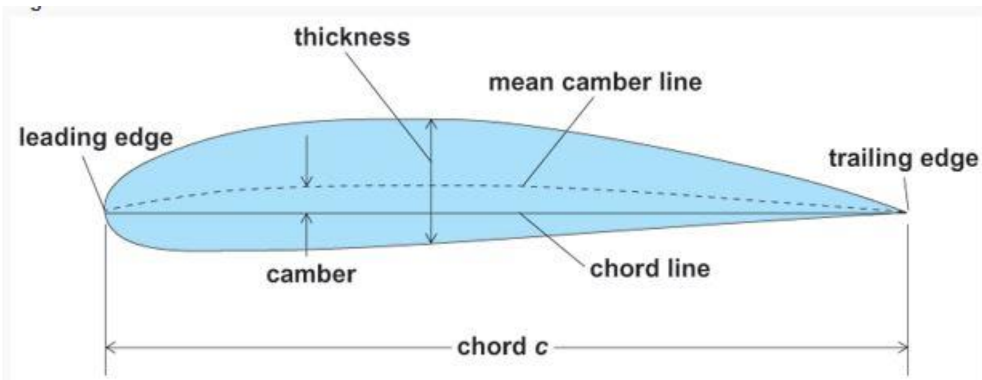

Figure 2.3: Airfoil nomenclature

(Source:http://www.accessscience.com/search.aspx?topic=ENG:AERO:AENAUT&term=Airfoil)

A characteristic shape with a rounded leading edge and then followed by a sharp trailing edge which the subsonic flight airfoils characteristic with asymmetric camber. As shown in figure 2.3, the locus of points halfway between the upper and lower surfaced is known as mean camber line and measured perpendicular to the mean camber line. While, leading and trailing edge are forward and backward point of the mean chamber line respectively. The chord line of the airfoil is a straight line that connecting leading and trailing edge. The camber is the maximum distance between camber line and chord line which measured perpendicular to the chord line. The thickness of an airfoil is the distance between the upper and bottom surfaces was also measured perpendicular to the chord line.

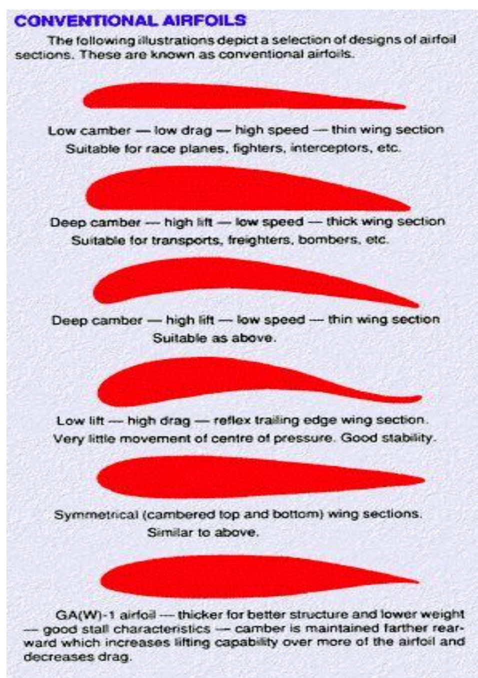

Figure 2.4: Different types of airfoil

Source: (http://www.allstar.fiu.edu/aerojava/flight11.htm)

There are several type of airfoil that designed before National Advisory Committee for Aeronautics (NACA) developed NACA airfoil series which generated using analytical equations that describe the chamber (curvature) of the mean-line (geometric centerline) of the airfoil section as well as the section's thickness distribution along the length of the airfoil.

In the early 1930s, NACA was reported the developed first family of NACA airfoils is 4-digit (NACA XXXX) series. To know the airfoil shape was standardize by NACA which is the first digit specifies the maximum camber in percentage of the chord. Second digit indicates the position of the maximum camber in tenths of chord and the last two digit indicates maximum thickness of the airfoil in percentage of chord. As example, the NACA 4412 that has maximum of 4% of chord located at 40% chord back from the leading edge and 12% thick while NACA 0015 is a symmetrical section of 15% thickness.

2.5 Computational Fluid Dynamic

In early1950s, the development of computational fluid dynamics started with the arrival of the digital computer. The basic tools used in the solution of computational dynamics is discretization method which is finite difference, finite element and finite volume methods. However, the solution for computational fluid dynamics problems are Navier- Stokes equations which almost all problems solve using numerical method because Navier-Stokes equations can define any single-phase fluid flow. In addition, the science of replacing the differential equations governing the fluid flow, with a set of algebraic equations which is turn can be solved numerically with a computer to get an approximate solution.

Chapter 3

Methodology

3.1 Overview

Throughout the project, there are few stage needs to be done in order to complete the task. Firstly, Study on aerodynamics force and NACA airfoils in order to compute using CFD. Then, select 10 different geometry of NACA 4 digit airfoil. After that, choose one of ten NACA airfoils to validate simulation data on experimental data using ANSYS 14.0 software. Lastly the analyze simulation data according to its drag coefficient and drag force on airfoil. However, in this report, it will focus only on the simulation data on 10 different geometry of NACA airfoil.

3.2 Aerodynamic Forces

Force balance is an important instrument to measure aerodynamic force in a wind tunnel. First, we need to know about lift force, drag force and the formula that use to compute the forces that involved in a wind tunnel.

3.2.1 Lift force

The lift force is produced by a lower pressure occur on the upper surface of an object compared to the lower surface where the pressure is high and it is causing the object lifted upward. (Mehrdad, 2001). The formula that involved in this project is

$$F_L = \frac{\rho v^2 C_L A}{2}$$

Where,

F_L = Lift Force (in units Newtons, N)

ρ = Fluid density (kg/m^3)

C_L = Lift Coefficient

V = Fluid velocity (m/s)

A = Reference area (m^2) or airfoil area

3.2.2 Drag force

Drag is the aerodynamic force that opposes a model motion through the air. It is a mechanical force that is by the contact of a solid body with a fluid (liquid or gas). It is not force generated by gravitational effect or electromagnetic field effect where the object can affect another object without being in physical contact. If there is no fluid, the drag force does not occur because of the drag to be generated, the model must have physical contact with the fluid. Furthermore, drag generated by the difference velocity between two objects, which is it will make no difference if the moving object past static fluid or the moving fluid past the static object. If there is no motion, there is no drag because drag acts in the direction that opposes the motion. So, it must be a motion for both of the object and the fluid. (Mehrdad, 2001). The formula involved for drag force in a wind tunnel is:

$$F_D = \frac{\rho v^2 C_D A}{2}$$

Where,

F_D = Drag Force (in units Newtons, N)

ρ = Fluid density (kg/m^3)

C_D = Drag Coefficient

V = Fluid velocity (m/s)

A = Reference area (m^2) or airfoil area

3.3 Assumption of Boundary condition

The scope of present computers have limited function in order to complete solution in practical application for basic equations of fluid motions .Thus, mathematical model is required to study for the transition to turbulence mechanism which is part of the turbulence motion. The Navier-Stokes equations express the conservation of mass, momentum and energy which fully described in fluid motion. Clay Mathematics Institute was proposed to solve Navier- Stokes equations problem in theoretical view. While, practical flows in aerospace applications are mostly turbulent which characterized by a large variety of scales all these scales must be captured in the flow computation. Therefore, instability of flow will lead to the transition laminar to the turbulent scale and fully turbulent regime.

For this study, ANSYS Fluent will implement transition-sensitive turbulence model which have three-equation eddy-viscosity type that will solved for turbulent kinetic energy (kT), specific dissipation rate (x), and laminar kinetic energy (k L). According to Mayle and Schulz, 1997, laminar kinetic energy concept was used to represent the energy of velocity fluctuation modes that increase in the boundary layer prior to transition which dramatically different from fully turbulent flow in term of structural and dynamic characteristics.

3.4 Meshing Parameter

The most critical part of engineering simulation is meshing generation which is too many cells meshed solver will have long time in iterate process and if too few will have inaccurate result. ANSYS meshing technology developed to meet this requirement and to obtain right meshing cells on the model in the most automated method possible. There are several mesh type that can be generated using ANSYS meshing technology such as Tetrahedral, Hexahedral, Hexahedral core, Prismatic Inflation Layer, Hexahedral Inflation Layer, Body Fitted Cartesian and Cut cell Cartesian. The different between refined and unrefined meshing parameter can be determined by using ANSYS meshing modeler as shown in Figure 3.2 and Figure 3.3 respectively.

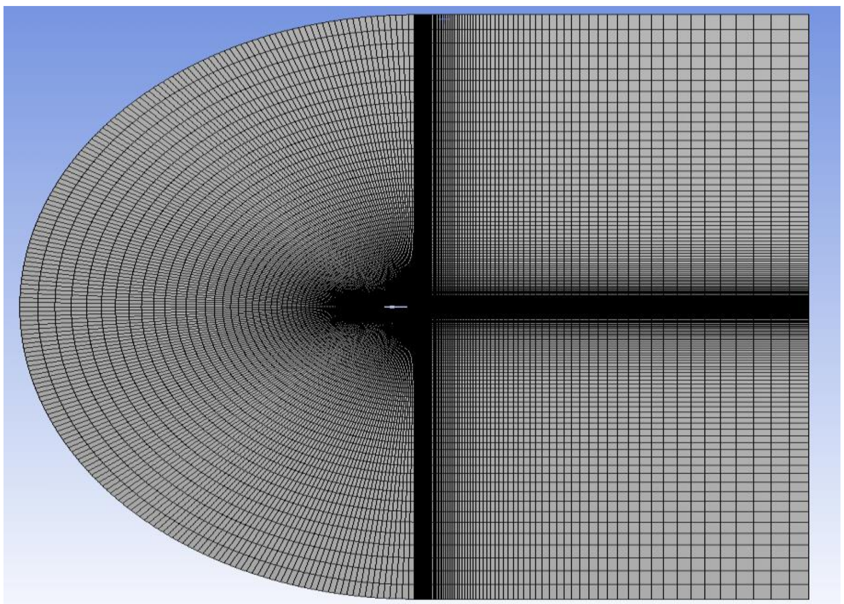

Figure 3.2: Refined Mesh

(Source: https://confluence.cornell.edu/display/SIMULATION/ANSYS+WB+-+Airfoil+-+All+Pages)

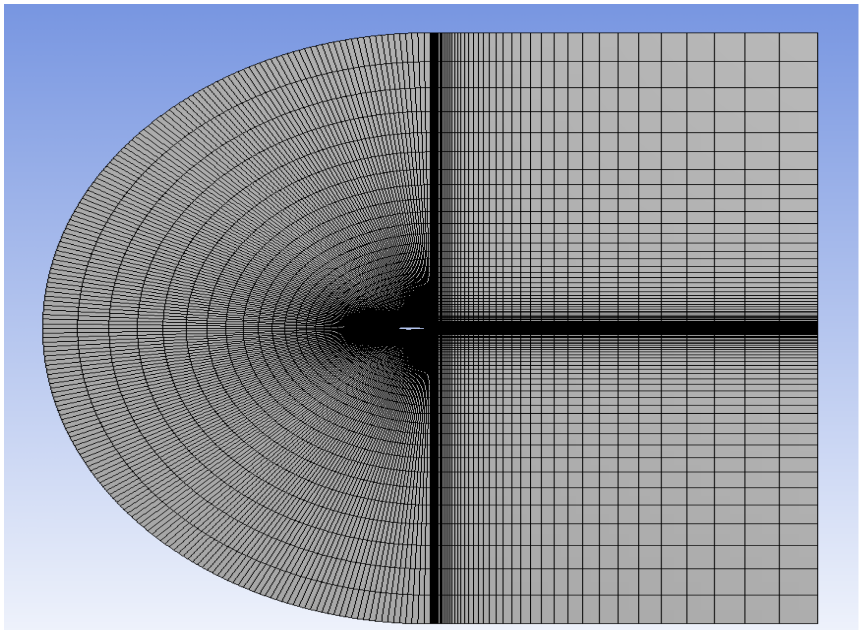

Figure 3.3: Unrefined Mesh

(Source: https://confluence.cornell.edu/display/SIMULATION/ANSYS+WB+-+Airfoil+-+All+Pages)

3.5 Simulation Parameter

There are several steps that involve to setup simulation parameter in ANSYS software. Firstly, we should define models whether to use laminar mathematical models or turbulence model. In this study we used turbulence models because of its stability and most experimental using this model. So, in ANSYS software we used transition k-kl-omega to simulated NACA airfoil in order to determine its drag force and drag coefficients.

Then, we have to choose fluid to simulate NACA airfoil whether air or liquid for fluid flow and type of materials for airfoil. After that, boundary condition at inlet which is velocity parameter will be defined in order determine the Reynolds number by its inlet velocity. Before start the calculation, check on drag monitors and start the initialization of flow field from inlet using boundary condition settings. Lastly, run the simulation in ANSYS Fluent and wait for iteration converged as Figure 3.4 or wait for calculation complete.

```
  iter continuity x-velocity y-velocity         cl         cd     time/iter
   188 1.0888e-06 1.9312e-08 7.2772e-09 1.3215e-01 4.6470e-03  0:00:18  813
   189 1.0695e-06 1.8510e-08 6.9401e-09 1.3215e-01 4.6470e-03  0:00:14  812
!  190 solution is converged
   190 9.9940e-07 1.7769e-08 6.6678e-09 1.3215e-01 4.6469e-03  0:00:11  811
Writing "Y:\AE Research Prof Sankar\FINAL RESEARCH PAPER\Fluent Files\NACA_
    5880 quadrilateral cells, zone  2, binary.
   11621 2D interior faces, zone  3, binary.
      98 2D pressure-outlet faces, zone  4, binary.
     120 2D velocity-inlet faces, zone  5, binary.
      60 2D wall faces, zone  6, binary.
    6019 nodes, binary.
    6019 node flags, binary.
```

Figure 3.4: Solution Converged

(Source: Chandok, Tanveer., 2010)

3.6 Data Validation

Validation is a part of simulation because it is concern about accuracy of conceptual simulation model for the system under study. Besides that, if the simulation data or simulation model was believed in correct programmed, it should analyze it with experimental data in order to validate the simulation data. Furthermore, validation was achieved through between the calibrations of actual system and the iterative process in simulation system which is repeated until the accuracy of simulation model accepted. (Kleijnen, Jack P.C., 1993)

There are several step that involved in calibrate and iterative process method. First, simulate the initial model and compare it with real system. Then, revised the first model and compare with actual system. After that, revised again until the accuracy is accepted which in ANSYS software just refined meshing parameter in order to get accepted value. Lastly, compared the accepted value with actual value which usually convey the data using graph as shown in Figure 3.5. (Kleijnen, Jack P.C.,1993)

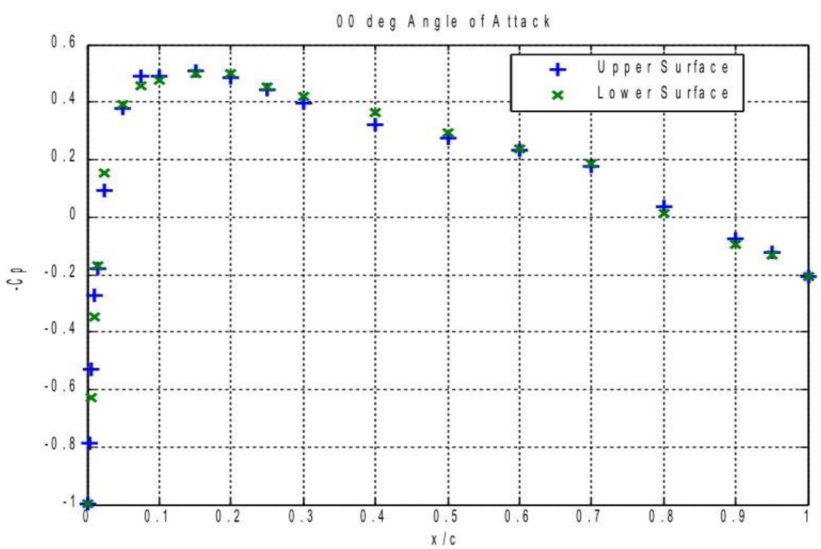

Figure 3.5: Experimental Data

(Source: Steven Miller D., 2008)

3.7 Materials and Tools Used

3.7.1 ANSYS 14.0 Software

ANSYS, Inc. is a computer-aided Engineering (CAE) that offers engineering simulation solution sets which design process most needed. This tool was widely used by industries because it can testing product in simulation state before it becomes physical object. While, this software developed to work as finite element analysis software for structural physics that could simulate in static state, dynamic state and heat transfer problems.

Furthermore, there are several technologies that involve in this software such as Simulation Technology which consist of Structural Mechanics, Multiphysics, Fluid Dynamics, Explicit Dynamics, Electromagnetics, Hydrodynamics (AQWA) and Workflow Technology which consist of ANSYS Workbench Platform, High-Performance Computing, Geometry Interfaces and Simulation Process & Data Management.

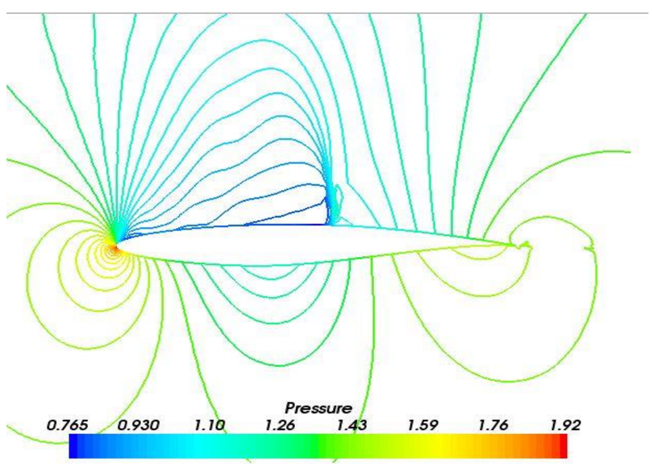

Figure 3.6: Example of ANSYS Fluent Result of Pressure Coefficient

(Source: http://www.cfd-online.com/Wiki/File:Vtc_rae2822_pressure.png)

In this study we used ANSYS Fluent benchmark suite which provides ANSYS Fluent hardware performance data was measured sets of benchmark problems needed to represent typical usage. This benchmark suite contains both pressure-based and density-based implicit solver cases using a variety of cell types and a range of physics. This benchmark had broad physical modeling capabilities that need to model flow, turbulence, heat transfer, and reactions for industrial applications.

3.7.2 NACA 4 Digit Generator

NACA 4 digit airfoil can be determine by its 4 digit as example NACA 2412 which number two is maximum camber divided by 100and become 0.02 or 2% of the chord. Then, number four is the position of maximum camber divided by 10 and become 0.4 or 40% of the chord. Then, number twelve is the thickness divided by hundred and become 0.12 or 12% of the chord. In order to get the geometry of NACA 0006, 0009, 0015, 1408, 1410, 2408, 2418, 4424, 6409 and 6412 we are using NACA 4 digit generator which provide geometry data for NACA 4 digit airfoil. In addition, it is more simple compare to draw by using Computer Aided Design (CAD) software which is more complicated and the data example as shown in Figure 3.7.

Figure 3.7: NACA 4 Digit Generator

(Source: http://airfoiltools.com/airfoil/naca4digit)

3.8 SUMMARY

After all the process completed it will big achievement to have a wind tunnel in Faculty of Manufacturing at University Malaysia Pahang. Every step is crucial to achieve the result, it will give others step a stall if there have some disturbance in process of simulation data for NACA 4 digit airfoil.

CHAPTER 4

RESULT AND DISCUSSION

4.1 Overview

This chapter will discuss about the findings and result on simulation which had explained in previous chapter. In this chapter, it will show result and data that can be used in order to achieve all the objective for this project. Furthermore, this chapter will include of data validation, Simulation result and summary of simulation result in order to compare the result that achieved in ANSYS 14.0 software. Lastly, the data will be discussed in this chapter and will be conclude in next chapter with some recommendation to improve the data given in this chapter.

4.2 Data Validation

Validation or benchmark technique was a good engineering practice of using an analysis technique to test similar configuration as experimental configuration. This validation process is essential to test the process by using numerical analysis on a replica of the experimental time testing. While, performing validation technique, the simulation test will be compared in aspect of pressure coefficient and drag coefficient of the NACA 0015.

While in CFD, this validation process result should be in specific parameter which is boundary condition, turbulence model and meshing strategy was required in order to achieve desired result and the accuracy of the result compared to real experimental result. For current benchmarking process the NACA 0015 geometry model will be simulated. The wind tunnel test was performed by Department of Aerospace Engineering at The Ohio State University. The test result of wind tunnel are compared with the result of the numerical simulation using advance solvers like ANSYS Fluent. The flow model used by wind tunnel test will be used in ANSYS Fluent which is turbulence model.

4.2.1 Experimental Result of NACA 0015

The experimental result was taken from Department of Aerospace Engineering of The Ohio State University wind tunnel test. The result shown in Figure 4.1 which is –Cp against x/c.

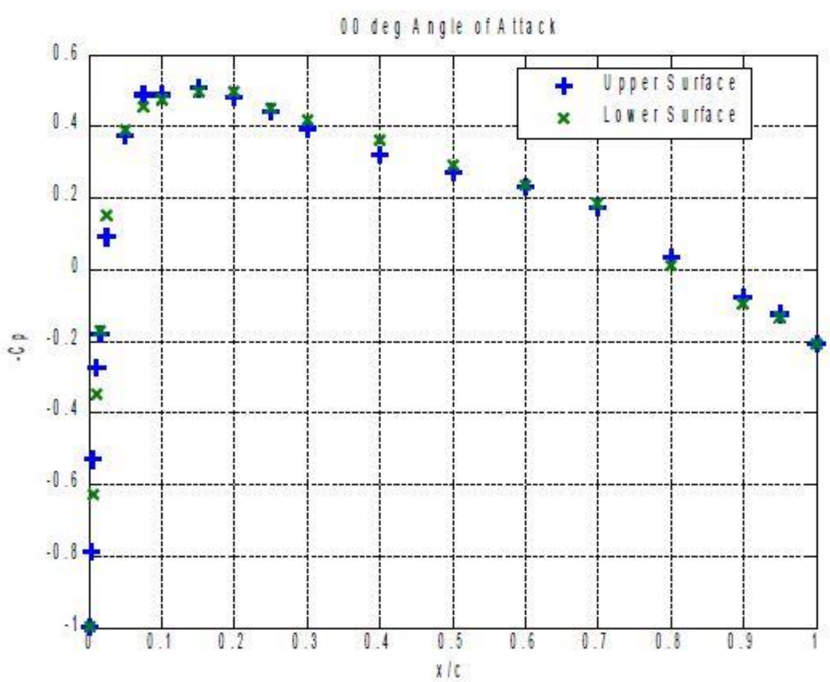

Figure 4.1: Pressure Distribution at 0 degree Angle of Attack

(Source: Miller, Steven D., 2008)

Graph in Figure 4.1 shows the pressure distribution point on the upper and lower surfaces almost match exactly for a 0 degree Angle of Attack. Then, the following Table 4.1 contains all the measured and theoretical values for the lift and drag coefficients of the NACA 0015 according to Wind Tunnel testing.

Table 4.1: Experimental Result of NACA 0015

Angle of Attack, α	Measured	
	C_D	C_L
0°	0.0101	-0.0044

4.2.2 Simulation Result of NACA 0015

The simulation result was compute using ANSYS 14.0 software. The result shown as Figure 4.2 which Cp against x/c. Graph in Figure 4.2 shows the pressure distribution point between upper and lower surfaces was exactly match for a 0 degree Angle of Attack.

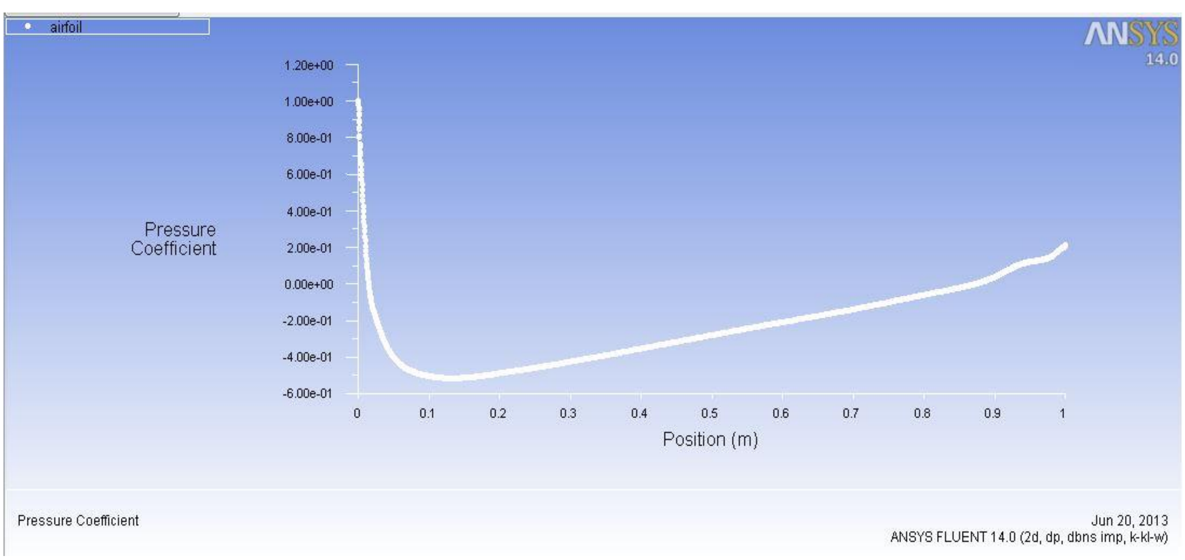

Figure 4.2: Simulation Result of NACA 0015 Airfoil

Furthermore, the drag coefficient of simulation result as shown in Figure 4.3 which is 0.0113. So, drag force in simulation result almost similar to experimental result by difference of 0.0012. The pressure coefficient graph will be compare manually because of unavailable graph data of experimental result, the comparison of graph can be refer at Appendices. The data was validated for further study on ten different geometry of NACA 4 digit airfoils. In addition, Table 4.2 shows the summary of data validation of this study.

```
Forces - Direction Vector (1 0 0)
                          Forces (n)                                   Coefficients
Zone                      Pressure        Viscous         Total           Pressure        Viscous         Total
airfoil                   0.61031412      1.3981764       2.0084906       0.0034478588    0.0078987439    0.011346603
------------------------- --------------- --------------- --------------- --------------- --------------- ---------------
Net                       0.61031412      1.3981764       2.0084906       0.0034478588    0.0078987439    0.011346603
```

Figure 4.3: Forces Report on NACA 0015

Table 4.2: Experimental result and Simulation result

Angle of Attack, α	**Measured**		**Simulation**			
			Unrefined Mesh		**Refined Mesh**	
	C_D	C_L	C_D	C_L	C_D	C_L
0°	0.0101	-0.0044	0.0045	-0.2063	0.0113	-8.05e -05

4.3 Simulation Result

4.3.1 NACA 0006

Simulation result for NACA 0006 was shown in Figure 4.4 which shows the velocity vector that occurred at airfoil surfaces. Velocity was highest at upper and lower surfaces. There are vortex occurred at trailing edge while velocity was hit at leading edge at flow over to upper and lower surfaces and at leading edge where the drag force was determined to big but it depend on how big the leading edge area or vortex that occurred at trailing edge. Velocity inlet is 17m/s but the velocity at airfoil surfaces was 11m/s.

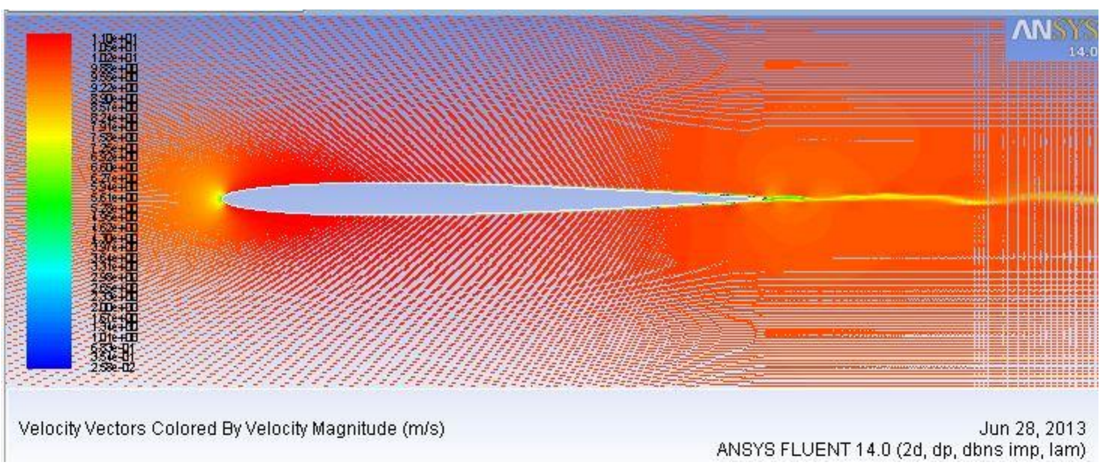

Figure 4.4: Velocity Vectors of NACA 0006 Airfoil

Furthermore, forces report on NACA 0006 was shown in Figure 4.5. Drag forces on NACA 0006 is 0.2762 N while drag coefficients is 0.0045.

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	0.10351062	0.17268349	0.27619411	0.0016899693	0.0028193222	0.0045092916
Net	0.10351062	0.17268349	0.27619411	0.0016899693	0.0028193222	0.0045092916

Figure 4.5: Forces Report on NACA 0006 airfoil

4.3.2 NACA 0009

Simulation result for NACA 0009 was shown in Figure 4.6 which shows the velocity vector that occurred at airfoil surfaces. Velocities were highest at upper and lower a surface which is 19.5 m/s while inlet velocity was 17 m/s.

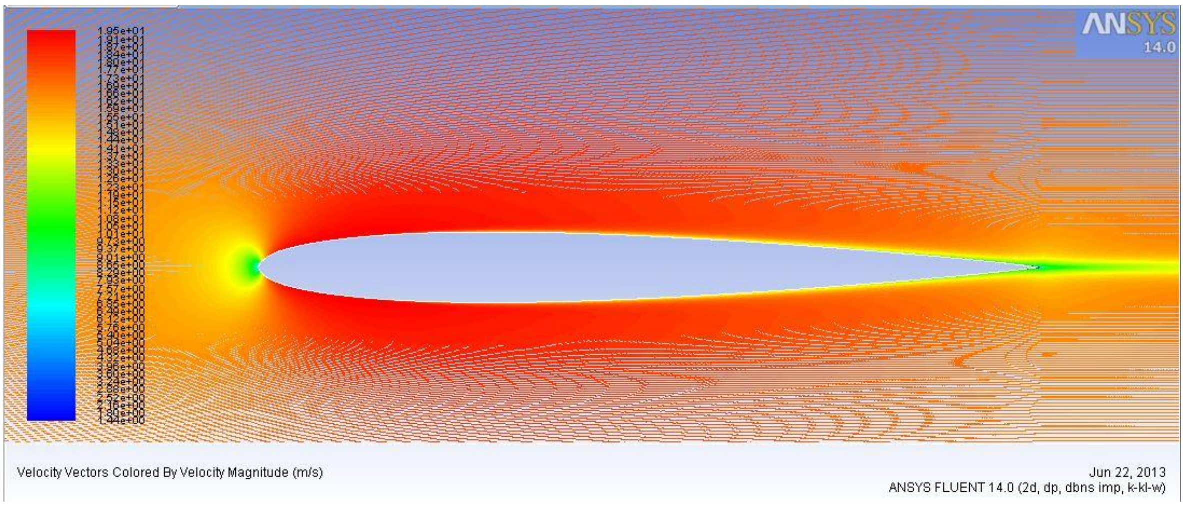

Figure 4.6: Velocity Vector of NACA 0009

Furthermore, forces report on NACA 0009 was shown in Figure 4.7. Drag forces on NACA 000 is 1.7878 N while drag coefficients is 0.0101.

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	0.38234012	1.4054279	1.787768	0.0021599611	0.0079397099	0.010099671
Net	0.38234012	1.4054279	1.787768	0.0021599611	0.0079397099	0.010099671

Figure 4.7: Forces report on NACA 0009 airfoil

4.3.3 NACA 0015

Simulation result for NACA 0015 was shown in Figure 4.8 which shows the velocity vector that occurred at airfoil surfaces. Velocities were highest at upper and lower a surface which is 20.9 m/s while inlet velocity was 17 m/s.

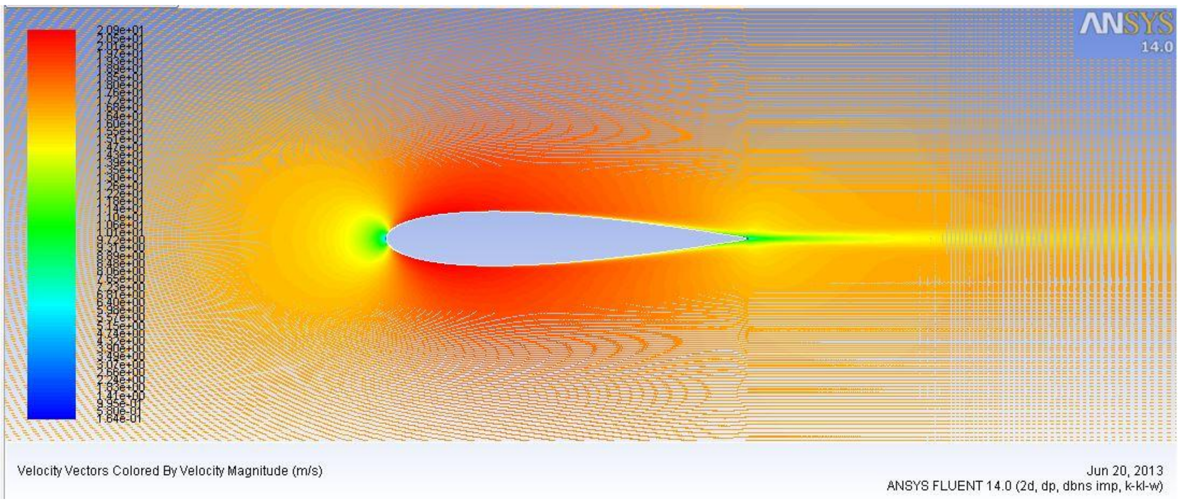

Figure 4.8: Velocity Vectors of NACA 0015

Furthermore, forces report on NACA 0015 was shown in Figure 4.9. Drag forces on NACA 0015 is 2.0085 N while drag coefficients is 0.0113.

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	0.61031412	1.3981764	2.0084906	0.0034478588	0.0078987439	0.011346603
Net	0.61031412	1.3981764	2.0084906	0.0034478588	0.0078987439	0.011346603

Figure 4.9: Forces report on NACA 0015

4.3.4 NACA 1408

Simulation result for NACA 1408 was shown in Figure 4.10 which shows the velocity vector that occurred at airfoil surfaces. Velocities were highest at upper and lower a surface which is 19.6 m/s while inlet velocity was 17 m/s.

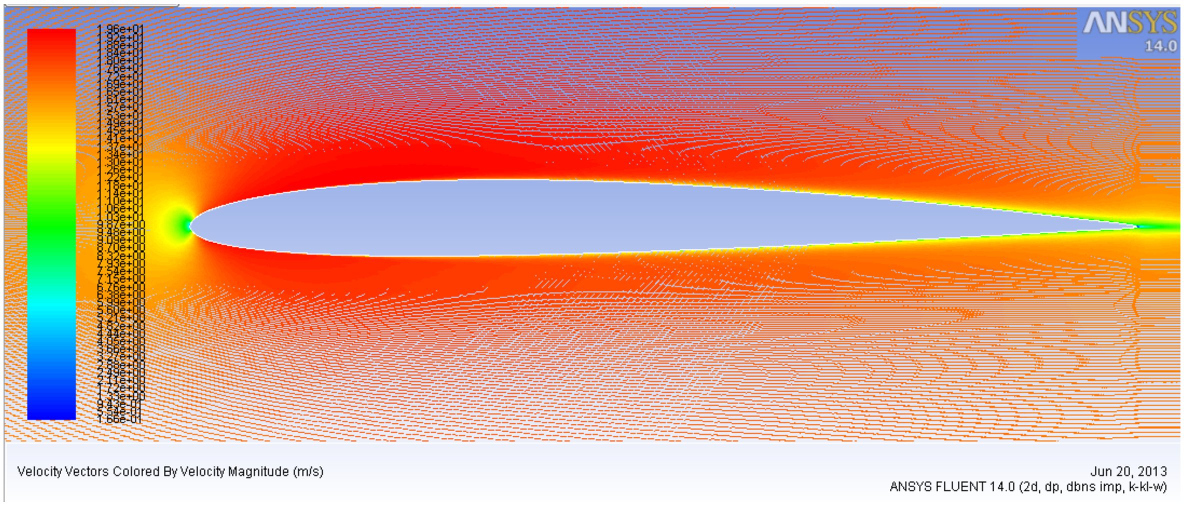

Figure 4.10: Velocity vector of NACA 1408

Furthermore, forces report on NACA 1408 was shown in Figure 4.11. Drag forces on NACA 1408 is 1.6587 N while drag coefficients is 0.0094.

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	0.29124214	1.3674786	1.6587207	0.0016453196	0.007725322	0.0093706415
Net	0.29124214	1.3674786	1.6587207	0.0016453196	0.007725322	0.0093706415

Figure 4.11: Forces report on NACA 1408 airfoil.

4.3.5 NACA 1410

Simulation result for NACA 1410 was shown in Figure 4.12 which shows the velocity vector that occurred at airfoil surfaces. Velocities were highest at upper and lower a surface which is 20 m/s while inlet velocity was 17 m/s.

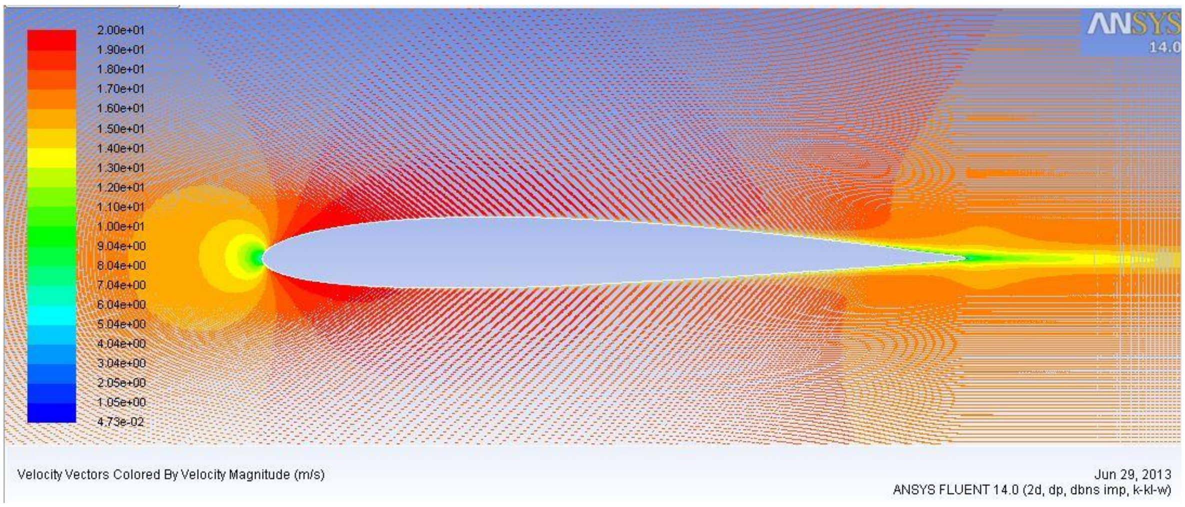

Figure 4.12: Velocity Vectors of NACA 1410

Furthermore, forces report on NACA 1410 was shown in Figure 4.13. Drag forces on NACA 1410 is 1.7882 N while drag coefficients is 0.0101.

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	0.40798928	1.3802126	1.7882019	0.0023048614	0.0077972605	0.010102122
Net	0.40798928	1.3802126	1.7882019	0.0023048614	0.0077972605	0.010102122

Figure 4.13: Forces report on NACA 1410 Airfoil

4.3.6 NACA 2408

Simulation result for NACA 1408 was shown in Figure 4.14 which shows the velocity vector that occurred at airfoil surfaces. Velocities were highest at upper and lower a surface which is 20.1 m/s while inlet velocity was 17 m/s.

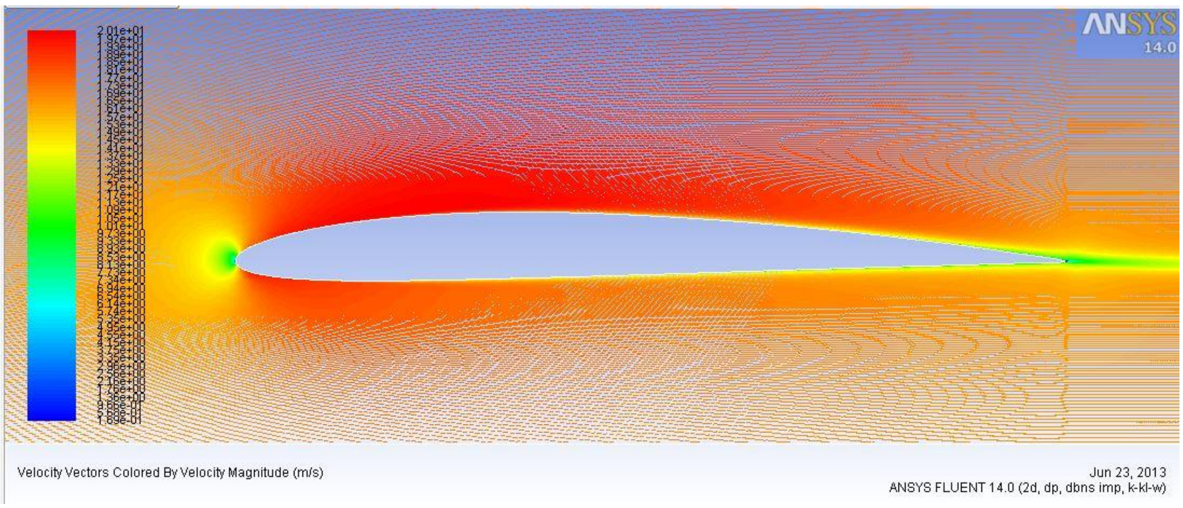

Figure 4.14: Velocity Vectors of NACA 2408

Furthermore, forces report on NACA 2408 was shown in Figure 4.15. Drag forces on NACA 2408 is 1.7016 N while drag coefficients is 0.0096.

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	0.33191684	1.3696493	1.7015661	0.0018751039	0.0077375848	0.0096126888
Net	0.33191684	1.3696493	1.7015661	0.0018751039	0.0077375848	0.0096126888

Figure 4.15: Forces report on NACA 2408

4.3.7 NACA 2418

Simulation result for NACA 2418 was shown in Figure 4.16 which shows the velocity vector that occurred at airfoil surfaces. Velocities were highest at upper and lower a surface which is 22.6 m/s while inlet velocity was 17 m/s.

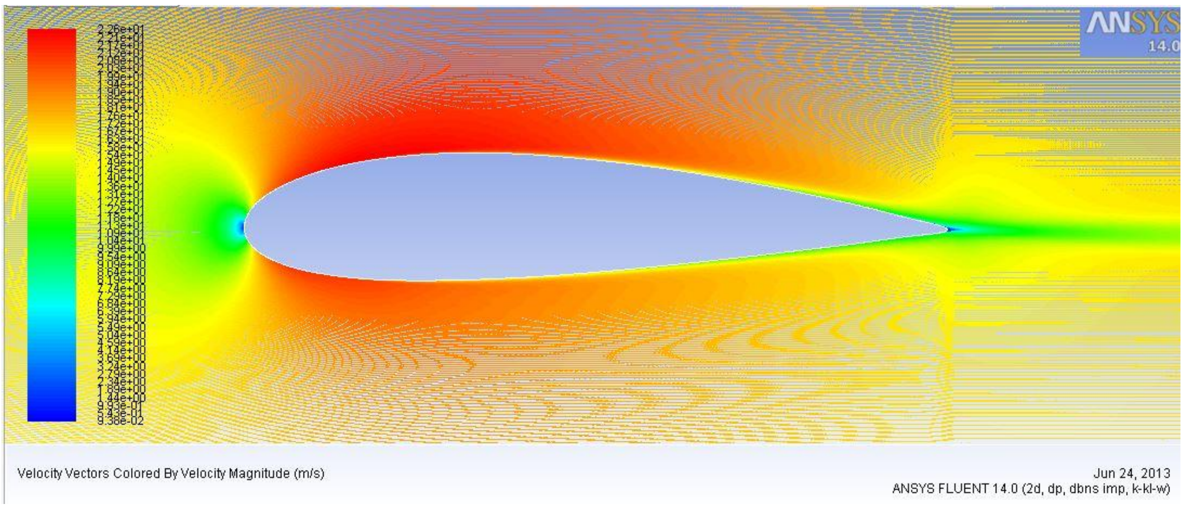

Figure 4.16: Velocity vectors of NACA 2418

Furthermore, forces report on NACA 2418 was shown in Figure 4.17. Drag forces on NACA 2418 is 2.2390 N while drag coefficients is 0.0126.

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	0.83253858	1.4064575	2.2389961	0.0047032756	0.0079455263	0.012648802
Net	0.83253858	1.4064575	2.2389961	0.0047032756	0.0079455263	0.012648802

Figure 4.17: Forces report on NACA2418 Airfoil

4.3.8 NACA 4424

Simulation result for NACA 4424 was shown in Figure 4.18 which shows the velocity vector that occurred at airfoil surfaces. Velocities were highest at upper and lower a surface which is 24.8 m/s while inlet velocity was 17 m/s.

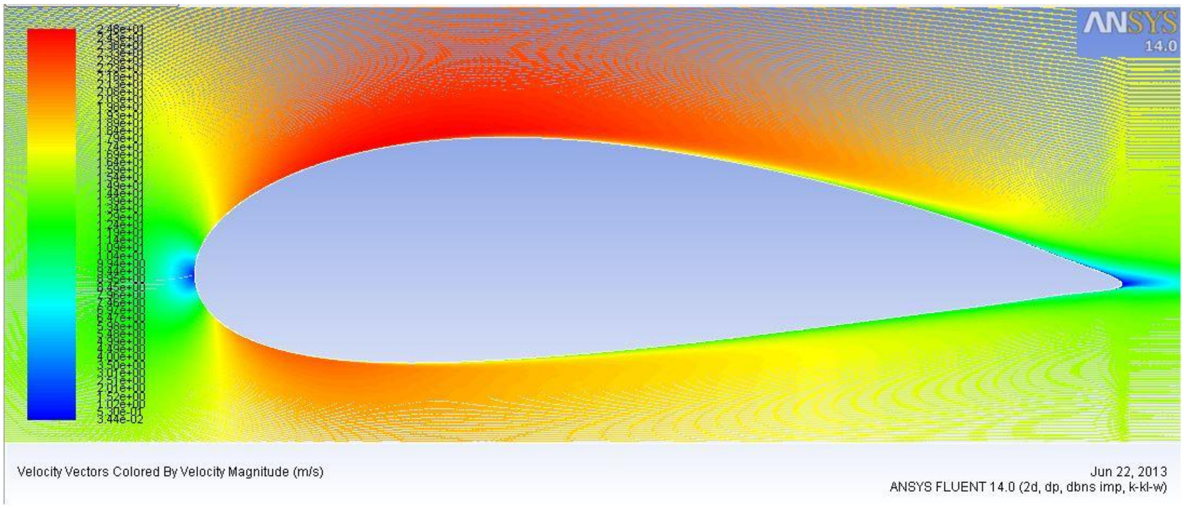

Figure 4.18: Velocity Vectors of NACA 4424

Furthermore, forces report on NACA 2418 was shown in Figure 4.19. Drag forces on NACA 2418 is 2.8135 N while drag coefficients is 0.0159.

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	1.3519172	1.4615404	2.8134576	0.0076374108	0.0082567071	0.015894118
Net	1.3519172	1.4615404	2.8134576	0.0076374108	0.0082567071	0.015894118

Figure 4.19: Force Report on NACA 4424 Airfoil

4.3.9 NACA 6409

Simulation result for NACA 6409 was shown in Figure 4.20 which shows the velocity vector that occurred at airfoil surfaces. Velocities were highest at upper and lower a surface which is 23.0 m/s while inlet velocity was 17 m/s.

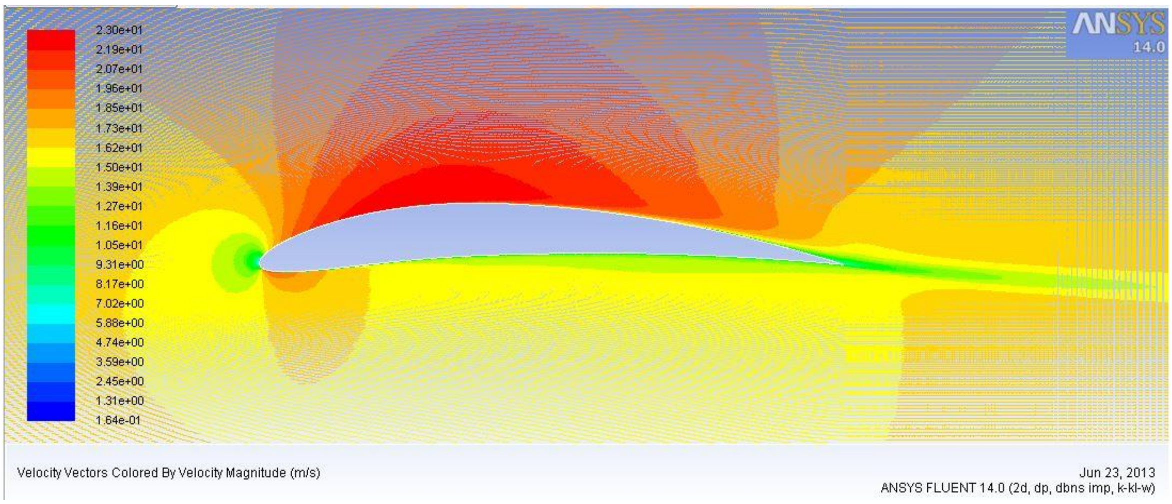

Figure 4.20: Velocity Vectors of NACA 6409

Furthermore, forces report on NACA 6409 was shown in Figure 4.21. Drag forces on NACA 6409 is 2.1539 N while drag coefficients is 0.0122.

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	0.80672363	1.3472169	2.1539406	0.0045574387	0.0076108575	0.012168296
Net	0.80672363	1.3472169	2.1539406	0.0045574387	0.0076108575	0.012168296

Figure 4.21: Force Report on NACA 6409 Airfoil

4.3.10 NACA 6412

Simulation result for NACA 6409 was shown in Figure 4.22 which shows the velocity vector that occurred at airfoil surfaces. Velocities were highest at upper and lower a surface which is 23.6 m/s while inlet velocity was 17 m/s.

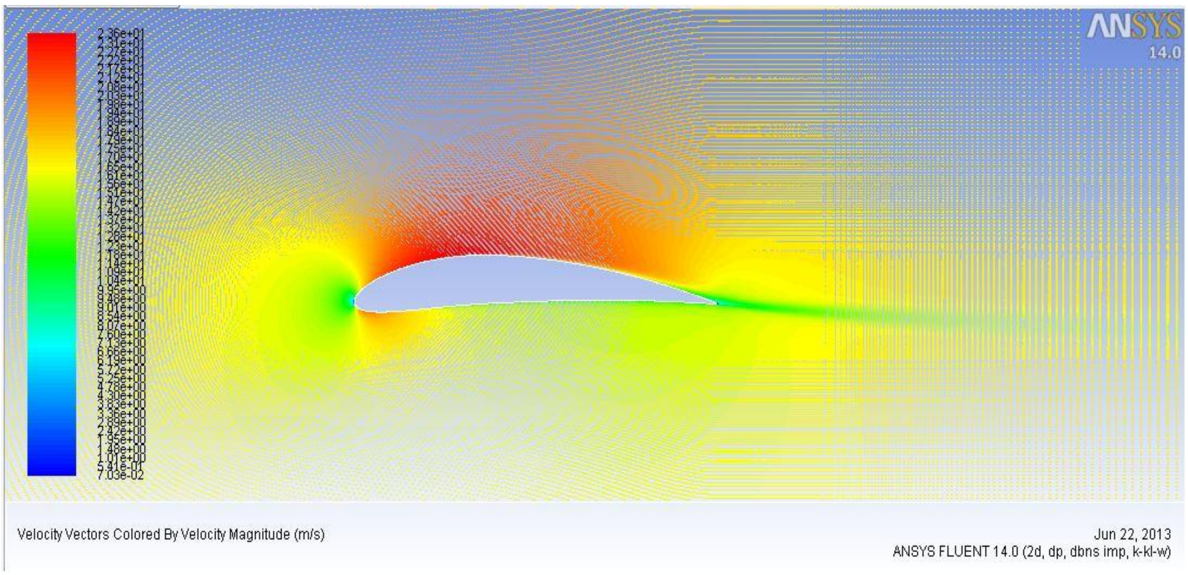

Figure 4.22: Velocity Vectors of NACA 6412

Furthermore, forces report on NACA 6412 was shown in Figure 4.23. Drag forces on NACA 6412 is 2.3540 N while drag coefficients is 0.0133.

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	0.97719353	1.3767806	2.3539741	0.0055204774	0.0077778718	0.013298349
Net	0.97719353	1.3767806	2.3539741	0.0055204774	0.0077778718	0.013298349

Figure 4.23: Force Report on NACA 6412 Airfoil

4.4 Summary of Simulation Result

Graph in Figure 4.24 shows the type of NACA 4 digit airfoil against drag coefficient that obtains from ANSYS 14.0 software result. In order to obtain result as shown in Figure 4.24, analyst used the parameter as follow in Table 4.3.

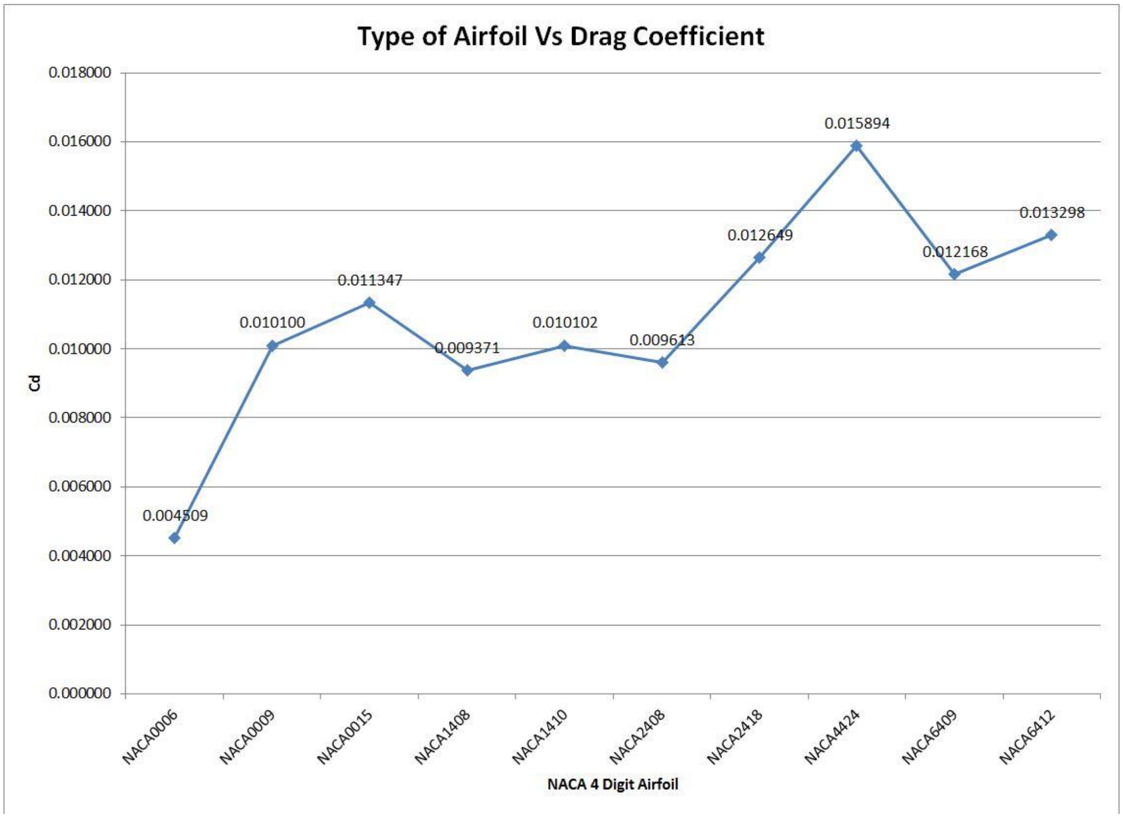

Figure 4.24: Type of NACA 4 digit Airfoil against Drag Coefficient

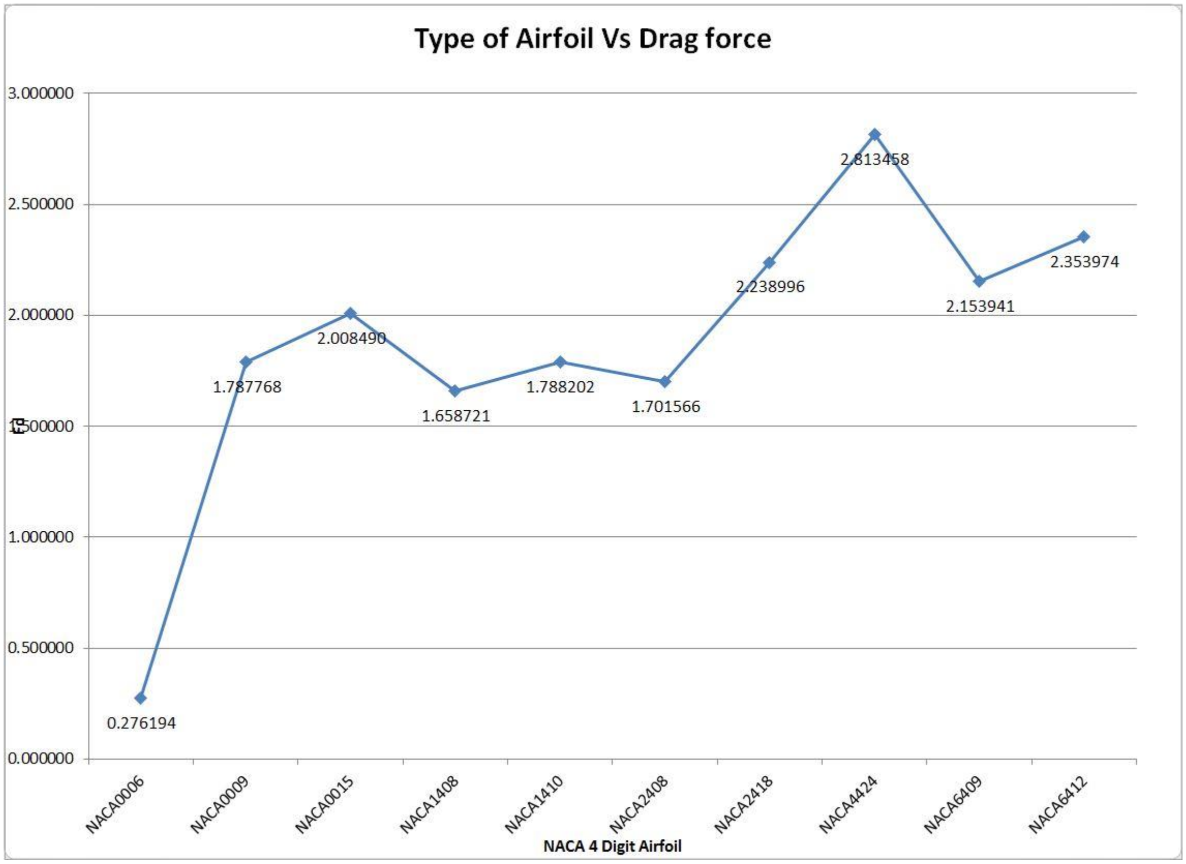

Figure 4.25: Type of NACA 4 digit Airfoil against Drag Force

Graph in Figure 4.24 shows the NACA 4424 have the highest drag coefficient which have the highest thickness than others NACA airfoil. From the plot and figures above, the airfoils aerodynamic properties can be analyzed. All this airfoil was tested on the same parametric condition which same inlet velocity and same mathematical model to find the different between them.

From these findings, there are several factors that drag can occur. Firstly, the geometry of the airfoil was obviously different from each other. From the graph shown, NACA 0006 lower than NACA 0009 by the different of the thickness in geometry aspect. Furthermore, NACA 1408 and NACA 2408 share the same thickness but different by maximum chord located in airfoil geometry and the result shows small differences between NACA 1408 and NACA 2408.

In Addition, NACA 2418 which have higher thickness than NACA 6409 but the result show there are small different of its drag coefficients. But there are differences of the symmetrical of airfoil. NACA 2418 was a symmetrically between upper and lower surfaces while, NACA 6409 was not a symmetrically between upper and lower surfaces. In this case, we look for the other factor which is separations of laminar flow to turbulence on the airfoils surfaces. The separations can be seen in figures above that shown in velocity vectors on airfoils surfaces. The early separations can contribute to high of drag coefficients. In addition, drag coefficients will influence the drag force, if the drag coefficients was high then drag force will be high as shown in Figure 4.25. Besides that, the contours of pressure coefficients along the NACA airfoil can refer on Appendices.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 Conclusion

Drag Coefficients for turbulence model was completely simulate using ANSYS 14.0 software. The drag coefficients obtained from simulation result was 0.0012 different to experimental result. However, the drag was generated was too small to make any significant change to the airfoil performance. In addition, drag force was affected by NACA airfoil geometry which have different maximum chord, camber, thickness and others. As shown in the result, ten different types of airfoils can be accurately described using the tools provided. CFD is a field that is gaining popularity to enables engineers analyze components accurately and efficiently.

5.2 Recommendations

This report shows the performance of an airfoil, there are various conditions that need to be known. Sometimes, these assumptions do not work well and must be changed. This 10% change is necessary to get values that are close to their actual ones. The mathematical model used was adequate for this type of analysis. There were various mathematical models that were not used since the aim of this report was to only touch the surface of CFD. ANSYS (Fluent) can be used for complex airfoil analysis in 2D and 3D.

In the future, CFD will solve problems that have not even been recognized today which before CFD appear engineers need a week to solve but now it can solve in few hours. This software should be one of subject to teach at university in order to produce competent engineers in Fluid Dynamics specialized.

REFERENCES

Bhargava, Ajay V. (1987). *Design and calibration of strain gage balance*, 17-28.

Suhariyono, Arief., Jong, Hyun Kim., Nam, Seo Goo., Hoon, Cheol Park. & Kwang, Joon Yoon. (2005). Aerospace Science and Technology. *Design of precision balance and aerodynamic characteristic measurement system for micro aerial vehicles*, 2-8.

Wells, Adam Joseph. (2005). *Experimental Investigation of an Airfoil with Co-Flow Jet Flow Control*. Degree Master of Science Thesis. University of Florida. Retrieved September 30, 2012, from University of Florida Theses.

Miguel Gonzalez, Jose Miguel Ezquerro, Victoria Lapuerta, Ana Laveron and Jacobo Rodriguez (2011). *Components of a Wind Tunnel Balance: Design and Calibration*, Wind Tunnels and Experimental Fluid Dynamics Research, Prof. Jorge Colman Lerner (Ed.), ISBN: 978-953-307-623-2, InTech, Available from: http://www.intechopen.com/books/wind-tunnels-and-experimental-fluid-dynamics-research/components-of-awind- tunnel-balance-design-and-calibration.

American Institute of Aeronautics and Astronautics. (2003). C*alibration and Use of Internal Strain- Gage Balances with Application to Wind Tunnel Testing journal*. 16-22.

Ghods, Mehrdad. (2001). Technical Communication for Engineers. *Theory of Wings and Wind Tunnel Testing of a NACA 2415 Airfoil*. 7-10.

Miller, S, D. (2008). *Lift, Drag and Moment of NACA 0015 Airfoil*. Department of Aerospace Engineering. The Ohio State University. 8-13.

Scarbrough, W, T. (1992). *NACA Four-Digit Airfoil Section Generation Using Cubic Parametric Curve Segments and the Golden Section*. Degree Master of Science of Mechanical Engineering Thesis. Rochester Institute of Technology. Retrieved May 25, 2013, from Rochester Institute of Technology Theses.

Chandok, T. (2010). *Analysis of an airfoil using Computational Fluid Dynamics.* Independent research thesis. Georgia Institute of Technology,

APPENDIX A

METHODOLOGY FLOWCHART

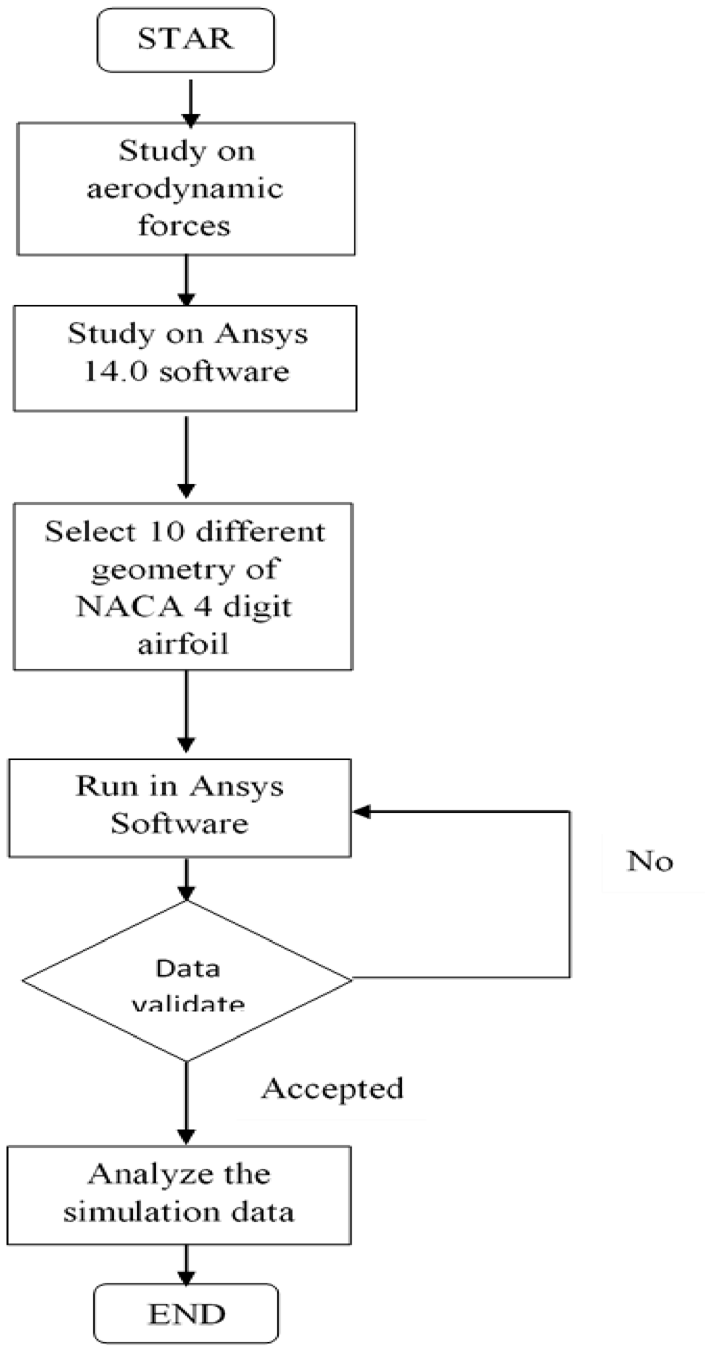

Figure 3.1: Flow Chart of the Methodology

APPENDIX B

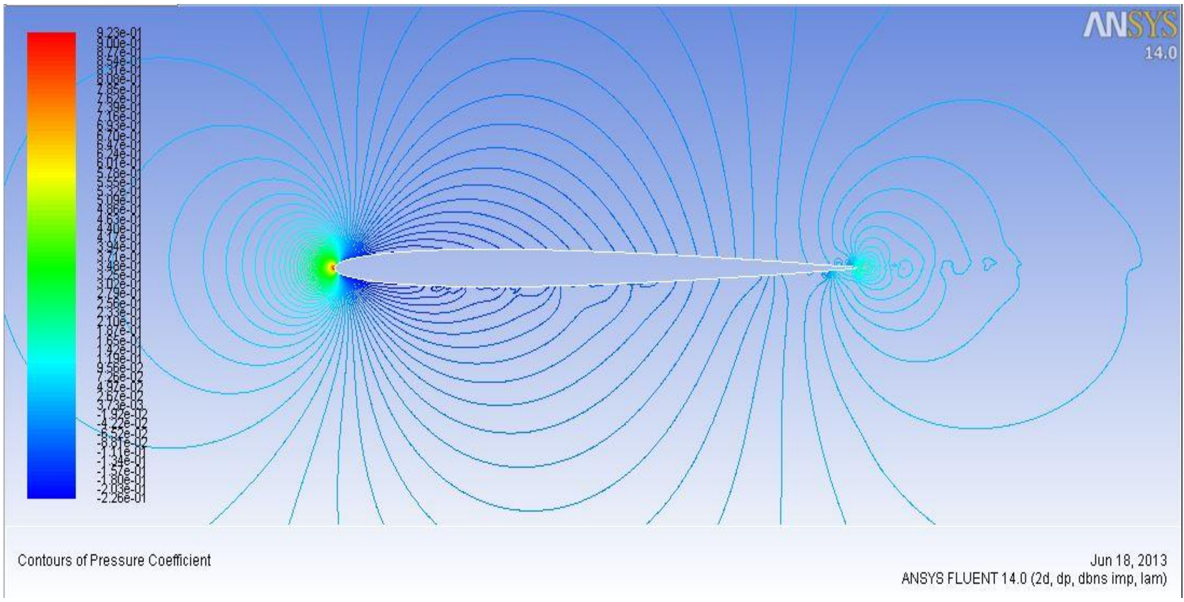

Figure 4.26: Contours of Pressure Coefficient on NACA 0006

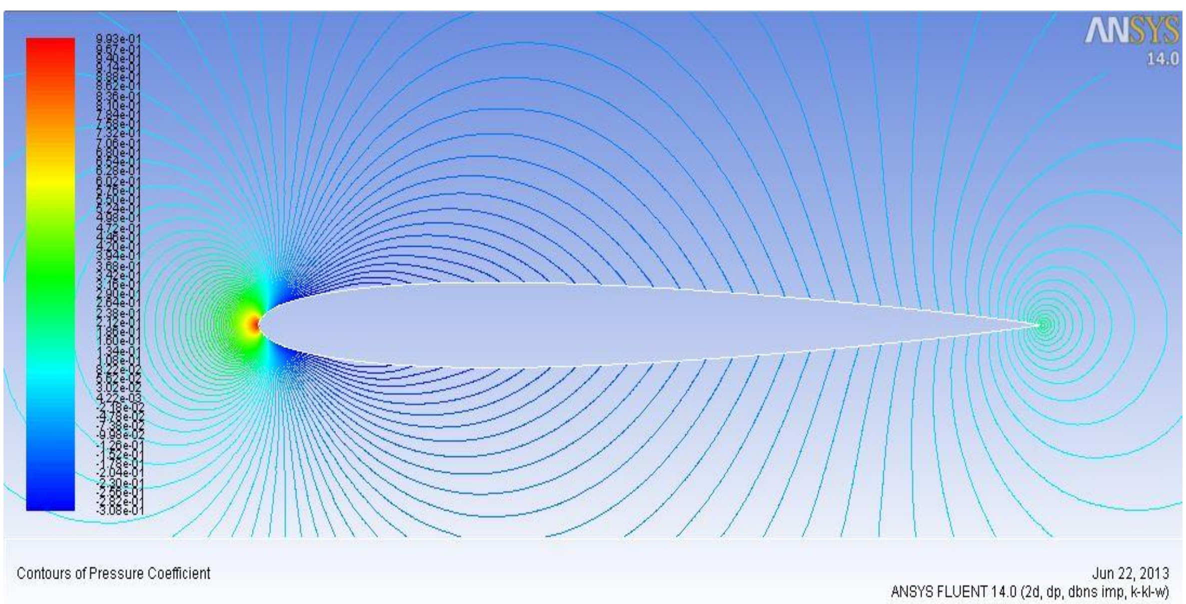

Figure 4.27: Contours of Pressure Coefficient on NACA 0009

APPENDIX B (Continued)

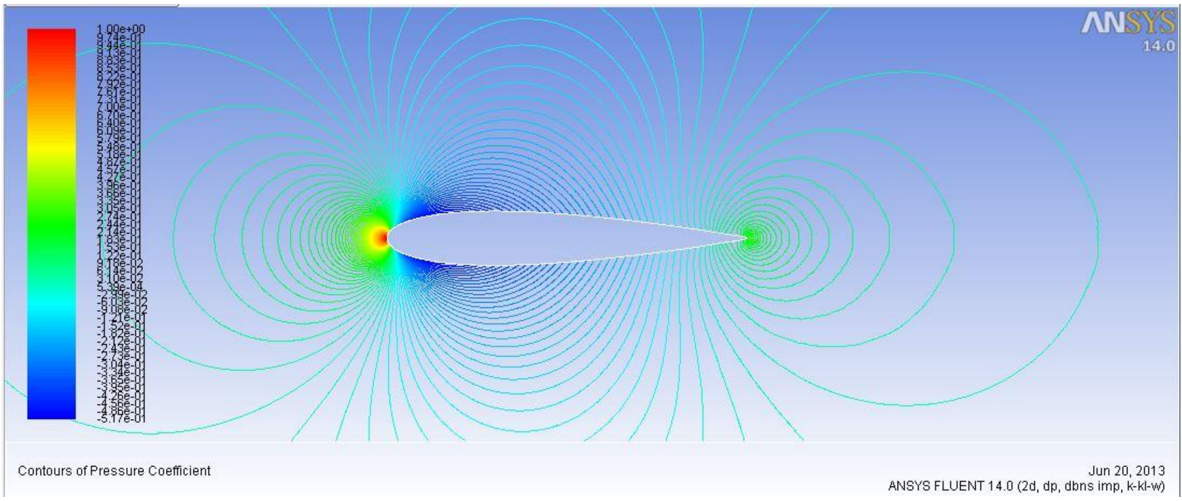

Figure 4.28: Contours of Pressure Coefficient on NACA 0015

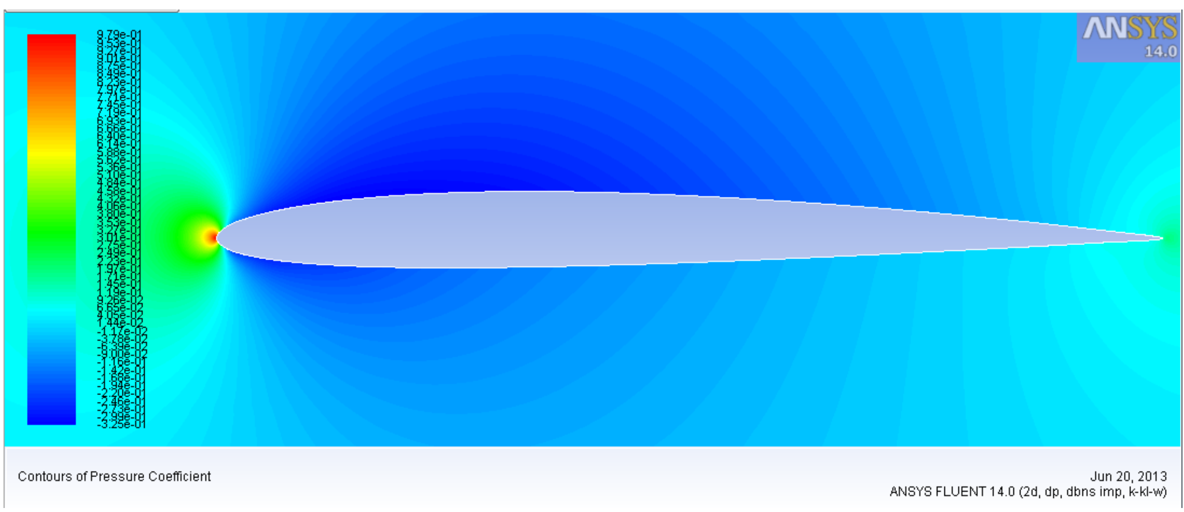

Figure 4.29: Contours of Pressure Coefficient on NACA 1408

APPENDIX B (Continued)

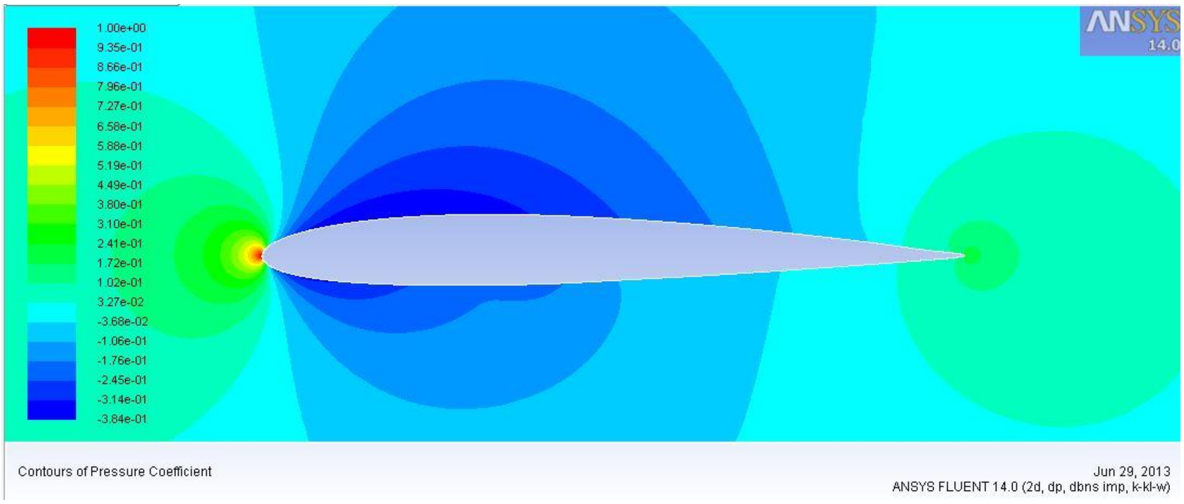

Figure 4.30: Contours of Pressure Coefficient on NACA 1410

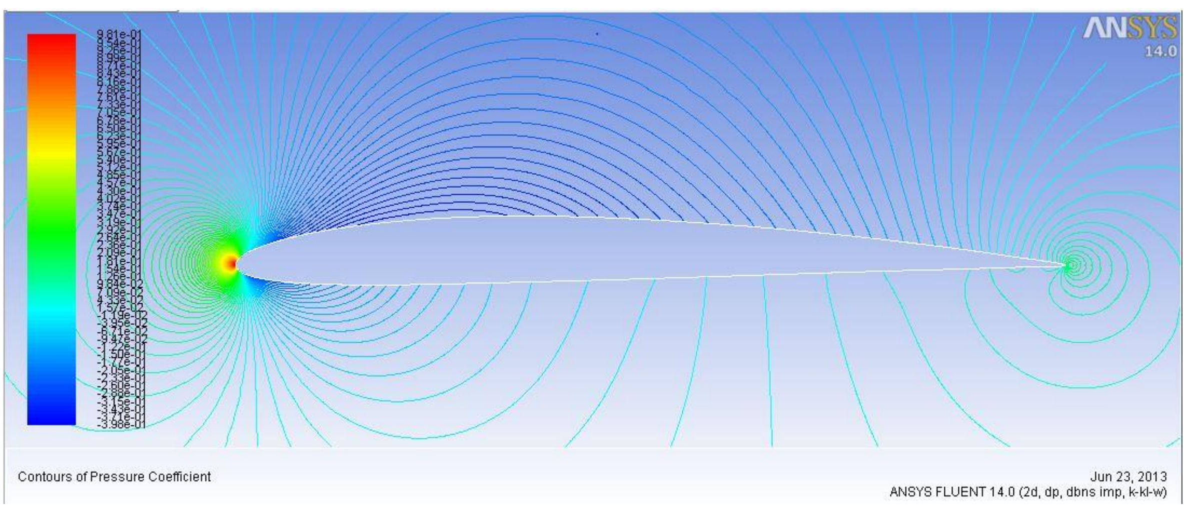

Figure 4.31: Contours of Pressure Coefficient on NACA 2408

APPENDIX B (Continued)

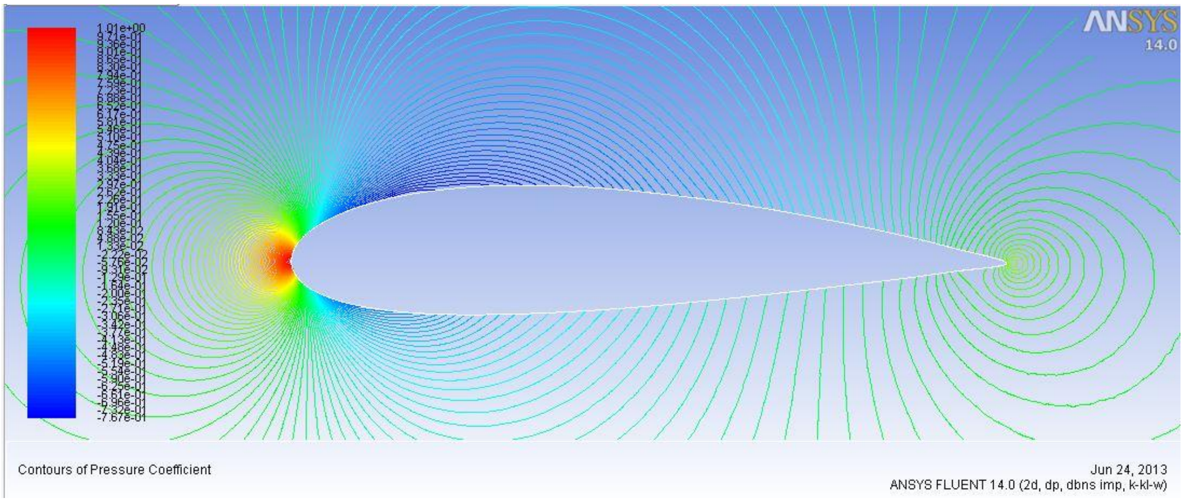

Figure 4.32: Contours of Pressure Coefficient on NACA 2418

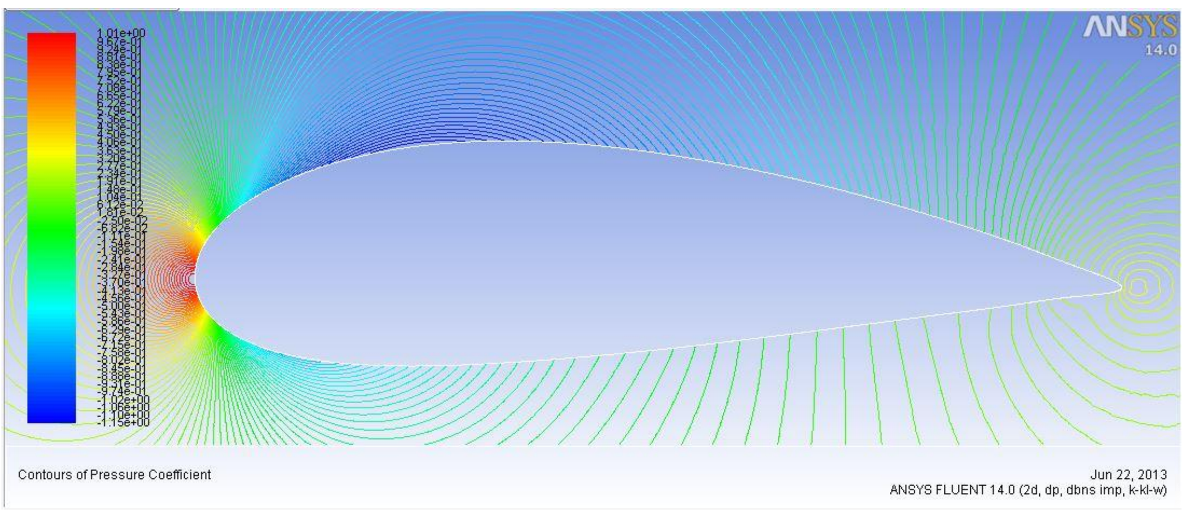

Figure 4.33: Contours of Pressure Coefficient on NACA 4424

APPENDIX B (Continued)

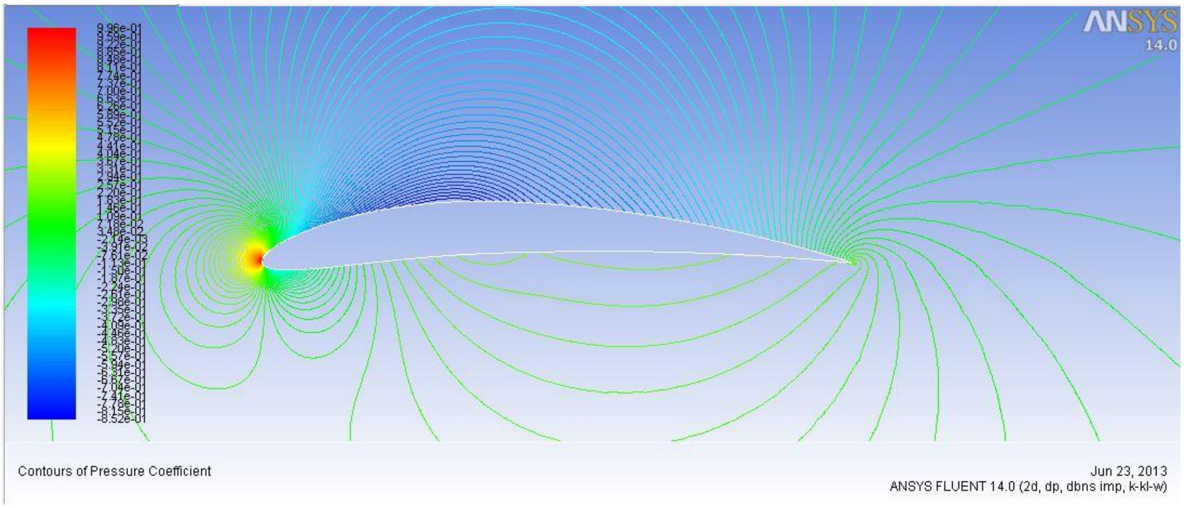

Figure 4.34: Contours of Pressure Coefficient on NACA 6409

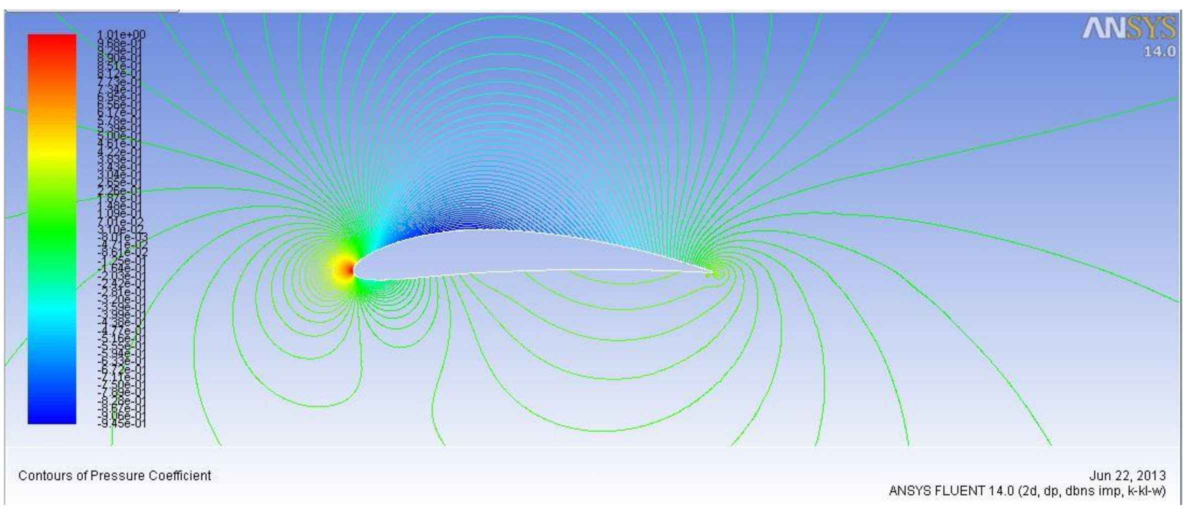

Figure 4.35: Contours of Pressure Coefficient on NACA 6410