PARAMETRIC INVESTIGATION ON THE AERODYNAMIC CHARACTERISTIC OF NACA2415 AIRFOIL AT LOW REYNOLDS NUMBER

MUHAMMAD HAFIZUDDIN BIN ISMAIL

BACHELOR OF MECHATRONIC ENGINEERING

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: **PARAMETRIC INVESTIGATION ON THE AERODYNAMIC CHARACTERISTIC OF NACA2415 AIRFOIL AT LOW REYNOLDS NUMBER**

SESI PENGAJIAN: 2009/2013

Saya **MUHAMMAD HAFIZUDDIN BIN ISMAIL (901105-11-5313)**

mengaku membenarkan tesis (Sarjana Muda/~~Sarjana~~ /~~Doktor Falsafah~~)* ini disimpan di perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

[ ]	**SULIT**	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
[ ]	**TERHAD**	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
√	**TIDAK TERHAD**	

Disahkan oleh:

(TANDATANGAN PENULIS)

(TANDATANGAN PENYELIA)

Alamat Tetap:

749, KG GONG TERAP
ALOR LINTAH,
22000,JERTEH,
TERENGGANU.

MR. SHAIFUL HAKIM BIN MOHAMED NOOR

Tarikh:

Tarikh:

CATATAN:
* Potong yang tidak berkenaan.
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai atau TERHAD.
♦ Tesis dimaksudkan sebagai tesis bagi Ijazah doktor Falsafah dan Sarjana secara penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

PARAMETRIC INVESTIGATION ON THE AERODYNAMIC CHARACTERISTIC OF NACA2415 AIRFOIL AT LOW REYNOLDS NUMBER

MUHAMMAD HAFIZUDDIN BIN ISMAIL

Report submitted in partial fulfillment of the requirement for the award of the degree of Bachelor of Mechatronic Engineering

Faculty of Manufacturing Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2013

EXAMINER'S APPROVAL DOCUMENT

I certify the thesis entitled "Parametric Investigation on the Aerodynamic Characteristic of NACA2415 Airfoil at Low Reynolds Number" by Muhammad Hafizuddin bin Ismail. I have examined the final copy of this thesis and in my opinion it is fully adequate in terms of scope and quality for the award of degree of Bachelor of Mechatronic Engineering. I herewith recommend that it be accepted in fulfillment of the requirement for the degree of Bachelor of Mechatronic Engineering.

Signature :

Name : ASSOC. PROF. DR. AZIZ BIN JAAFAR

Institution : UNIVERSITI MALAYSIA PAHANG

Date : 19 JUNE 2013.

SUPERVISOR'S DECLARATION

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

SUPERVISOR

Signature :

Name : MR. SHAIFUL HAKIM BIN MOHAMED NOOR

Position : LECTURER.

Date : 19 JUNE 2013.

STUDENT'S DECLARATION

I hereby declare that the work in this thesis entitled "Parametric Investigation On The Aerodynamic Characteristic of NACA2415 Airfoil At Low Reynolds Number" is my own except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature :

Name : MUHAMMAD HAFIZUDDIN BIN ISMAIL.

ID Number : FB09022.

Date : 19 JUNE 2013.

Dedicated to my parents

and Palestinian

ACKNOWLEDGEMENT

I am grateful and would like to express my sincere gratitude to my supervisor Mr. Shaiful Hakim bin Mohamed Noor for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. I am truly grateful for his tolerance of my naïve mistakes, and his commitment. I also would like to express very special thanks to my panel, Assoc. Prof. Dr. Aziz bin Jaafar for his suggestions and co-operation throughout the study. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

My sincere thanks go to all of the staff of the Manufacturing Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to all of my friends for their excellent co-operation, inspirations and supports during this study.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals

ABSTRACT

This study is to investigate the aerodynamic characteristic of an airfoil in low Reynolds number flow. NACA2415 airfoil is chosen as the test subject as it has been studied by many researches and documented. Coordinate point of NACA2415 is generated by using NACA 4 Digits Series Profile Generator, a web-based Java applet before being used by ANSYS Fluent to create 2D geometry of the airfoil. In the simulation, the boundary condition is set mimicking the streamline flowing at an angle ranging from 0 degree to 20 degree. As for the velocity, it is set for a flow condition under the Reynolds number of 200000 in incompressible viscous medium. From the simulation data, the C_d, C_l, C_p and the formation of Laminar Separation Bubble becoming the main concern for the simulation. Validation test is performed to approve the validity of the simulation result. Unfortunately, this investigation comes to a hold since the validation test is failed to capture the formation of LSB. The factor leading to the error is discussed.

ABSTRAK

Tujuan kajian ini ialah untuk menyiasat ciri aerodinamik keatas aerofoil bagi aliran dengan nombor *Reynolds* yang rendah. Aerofoil NACA2415 dipilih sebagai bahan kajian berikutan ianya telah dikaji ramai dan didokumentasikan dengan baik. Titik koordinat NACA2415 dijana menggunakan *NACA 4 Digits Series Profile Generator* sebuah aplikasi *Java* sebelum digunakan oleh ANSYS Fluent untuk membuat geometri 2 dimensi bagi aerofoil tersebut. Dalam simulasi, keadaan lapisan sempadan di tetapkan bagi membuat seolah-olah aliran datang dari sudut berjulat dari 0 darjah hingga 20 darjah. Untuk halaju aliran pula, ia di tetapkan untuk aliran seperti mana pada nombor Reynolds sebanyak 200000 dalam medium yang tidak boleh dimampat dan berkepekatan. Daripada data simulasi, nilai C_d, C_l, C_p dan formasi *Laminar Separation Bubble* menjadi tumpuan utama kajian. Ujian pengesahan dijalankan untuk mengesahkan kebergantungan terhadap keputusan simulasi. Malangnya, kajian ini menemui jalan buntu apabila ujian pengesahan gagal untuk merakam pembentukan *Laminar Separation Bubble*. Faktor yang menyebabkan kegagalan ini dibincangkan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

C_p	Coefficient of pressure
C_d	Coefficient of drag
C_l	Coefficient of lift
C_m	Coefficient of moment
ρ	Density of the fluid
V	Mean velocity
L	Characteristic linear dimension
μ	Kinematic viscosity
^{o}	Degree
%	Percent

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamic
Re	Reynolds number
RANS	Reynolds Averaged Navier-Stokes
Tu	Turbulence intensity
2D	2-Dimensional
3D	3-Dimensional
NACA	The National Advisory Committee for Aeronautics
LSB	Laminar Separation Bubble
MAVs	Micro-Air Vehicles
AOA	Angle of Attack
UMP	Universiti Malaysia Pahang

Chapter 1

INTRODUCTION

1.1 INTRODUCTION

This chapter discuss about the project background, the problem statement of the project, the objectives of the project and project scope.

1.2 PROJECT BACKGROUND

1.2.1 Aerodynamic characteristic of an airfoil.

The aerodynamic cross section of a body such as a wing that creates lift and drag on relative motion with the air is called an airfoil. Wing should be in proper shape for smooth lift. That is why airfoil size and shape playing an important role on airplane flight. Basic element of an airfoil is shown in Figure 1.1.

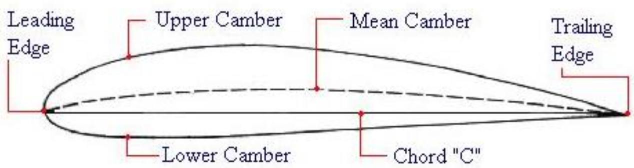
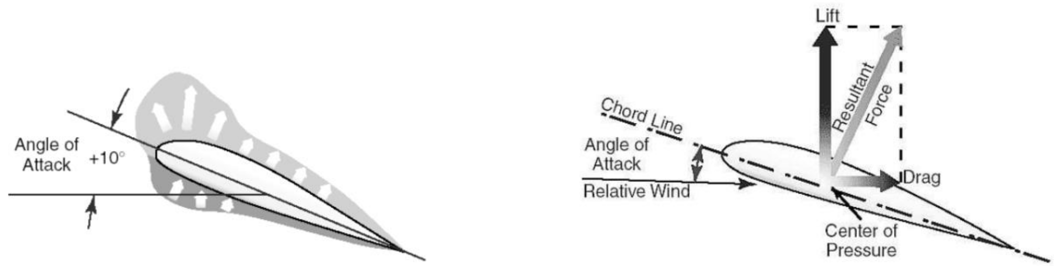


Figure 1.1: Basic elements of an airfoil

Source: http://www.aerospaceweb.org/question/airfoils/q0100.shtml

Lift is defined as force perpendicular to motion of the airfoil. The force parallel to the motion of the airfoil is called drag. As the air flow over an airfoil, the pressure over and under the airfoil changes due to the wind speed and circulation. To produce lift, a large part of the region over the wing has lower pressure than on the lower surface. Typical pressure distribution and resultant forces on an airfoil are shown in Figure 1.2 (a) and 1.2(b).



(a) Pressure distribution over an airfoil (b) Resultant forces on an airfoil

Figure 1.2: Airfoil characteristic.

Source: Jasvipul. S.C (2009)

For the purpose of analysis of air flow around an airfoil, the flow is divided into two regions: an outer region of inviscid flow, and a small flow region near the airfoil where viscous effects dominate. The region near the airfoil contains slow moving air and is known as boundary layer. The majority of drag experienced by a body in a fluid is created inside the boundary layer. The outer inviscid flow is faster moving air and determines the pressure distribution around the airfoil. The outer flow thus determines the lift force on the airfoil.

1.2.2 Flow in Low Reynolds number

The performance of airfoils operating at low relative wind speeds (low free stream velocities) has been of interest in modern subsonic aerodynamics. Typical applications where such airfoils can be used are wind turbines, remotely piloted vehicles, sail-planes, human powered vehicles, high altitude devices and many more. To characterize flows, the dimensionless Reynolds number (Re) is used. Reynolds number is defined as in Eq. (1.1) and gives a measure of the ratio of inertial forces to viscous forces and consequently quantifies the relative importance of these two types of forces for given flow conditions.

$$\mathrm{Re} = \frac{\rho V L}{\mu} \tag{1.1}$$

Where:

ρ = density of the fluid (kg/m³).

V = mean velocity of the object relative to the fluid (m/s).

L = characteristic linear dimension (m).

μ = kinematic viscosity (m²/s).

As Reynolds number is proportional to free stream velocity, the low wind speed flows (low free stream velocity) correspond to low Reynolds numbers. At low Reynolds numbers, the airfoils generate lesser lift, and encounter higher drags, bringing down the performance of the airfoil. This study gives a basic overview of low Reynolds number aerodynamic.

1.2.3 Computational Fluid Dynamics(CFD)

The effect of the Laminar Separation Bubble(LSB) and flow control method on low Reynolds number flow has been investigated by means of various experimental method, such as force measurement, velocity measurement, by using hot-wire anemometry and particle image velocimetry, pressure measurement with pressure transducer, flow visualization, with smoke wire, oil, Infrared thermography, etc. These systems are useful and accurate but also expensive and everyone cannot find the opportunity to use these methods. Therefore investigating all kind of aerodynamic phenomena via Computational Fluid Dynamic (CFD) is now popular and easy to use. By using CFD, the flow characteristic of a wing profile or any object can be easily analyzed. The biggest concern about software simulation is the validity of its results. There is no guaranty about the accuracy of the simulation result and the only way to validate it is by comparing the simulation result with the experimental result, both under the same flow condition. Specific parameters, condition, constant, and assumption need to be known under which the software yields the most accurate result compared to the experimental data.

1.2.4 Established work on NACA2415 and CFD simulation.

Many experiments had been done on NACA2415 in investigating the aerodynamic characteristic of low Re number flow. Serdar Genc, M et al. (2012) study experimentally the aerodynamics of NACA2415 at low Reynolds number. They conclude that, as the angle of attack increase, the separation and transition point moved towards the leading edge at all Reynolds number. They also found that as the Reynolds number increase, stall characteristic changed and the mild stall occurred at higher Reynolds number whereas the absurd stall occurred at lower Reynolds number. In addition, their result shows that stall angle decrease as the Reynolds number decrease.

Ghods, M. (2001) conducting an experiment to introduced the basic theory of wing and provide an introduction on wind tunnel testing involving NACA2415. Lift increases as the angle of attack increases between -5 and +17 degrees and at +17 degrees, maximum lift is generated. If the angle of attack is increased any further, drag becomes the dominant factor and the wing enters the stall mode.

Catalano, P (2009) analyzed on the aerodynamic of low Reynolds number flows focusing on the laminar separation bubbles. They conclude that laminar separation bubbles can be found by using the Spalart-Allmaras and the κ-ω SST turbulence models. Detail investigation on κ-ω SST did not predict correctly the viscous and logarithmic regions of the boundary layer at the lowest Reynolds number which is 6 x 104. The modified model, the κ-ω SST-LR, has provided a correct simulation of the viscous sub-layer and logarithmic region in the tests performed at high and low Reynolds numbers. The κ-ω SST-LR turbulence model can be used in a wide range of Reynolds numbers to simulate different flow aspects from the laminar separation bubbles to the shock-boundary layer interaction.

Kaynak, U in his study on aerodynamic characteristic of NACA64A006 airfoil using the k-ω SST turbulence model, k-ω SST transition model and k-k_L-ω transition model using FLUENT, conclude that, all numerical approaches give reasonably good result in the linear region, although the results begin to differ as the angle of attack gets larger. The k-k_L-ω transition model yields the best result whereas the k-ω SST turbulence model and k-ω SST transition model greatly under predict the lift coefficient. For the pressure coefficient, it is observed that the k-k_L-ω transition model also fares better than the other models.

1.3 PROBLEM STATEMENT

Predicting low-Reynolds number airfoil performance is a difficult task that requires correctly modeling several flow phenomena such as inviscid flow field with the presence of shock waves, laminar separation regions with presence of separation bubbles, transition to turbulence in the free shear layer and turbulent boundary layer. Especially the presence of the separation bubble may affect the results significantly. Constant pressure assumption across the boundary layer may not be valid across the bubble. Thus, correct modeling of the flow around the airfoils operating at low Reynolds numbers becomes a challenging research problem. Today, state of the art Reynolds Averaged Navier-Stokes (RANS) solvers are widely available for numerically predicting fully turbulent part of flow fields, but none of these models are adequate to handle flows with significant transition effects because of lack of practical transition modeling.

Correlation with the development of a wind tunnel in the Faculty of Manufacturing Engineering of Universiti Malaysia Pahang (UMP), investigation by means of simulation needs to be done to get an in-depth view of what happen during the flight in low Reynolds number. Since the wind tunnel is still under development, this simulation study will enable the lectures and students to identify the instruments, devices, and methods in recording the specific parameter such as lift, drag, and pressure distribution from the wind tunnel testing.

1.4 PROJECT OBJECTIVES

The objective of this project is to investigate the aerodynamic characteristic of NACA2415 at low Reynolds number. It is also to find the critical angle of attack which may induce stall condition for the airfoil. As the angle of attack increasing, the shifting of separation, transition, and reattachment region is also part of this project objective.

1.5 PROJECT SCOPE

The scope of this project is limited to the investigation of NACA2415 airfoil profile only. It includes generating the coordinate points in creating the geometry of the airfoil in Ansys FLUENT 2D environment. By using FLUENT, suitable meshing technique and meshing parameters is investigated. To suit the simulation condition to the real life, simulation parameters, constants, and assumption is studied. This study also includes the identification of the best mathematical model to be used in order to get the most accurate result. For that purpose, simulation result is validated by comparing the result with the experimental result at the Reynolds number of 200,000. After the validation, the simulation is continued with the tested angle of attack within the range of 0^{o} to 20^{o}. As for the results, critical angle of attack, stall characteristics, and LSB is investigated.

Chapter 2

Literature Review

2.1 INTRODUCTION

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

2.2 NACA 4-DIGIT FAMILY

In the early 1930's, National Advisory Committee for Aeronautics (NACA) had published a report entitled "The Characteristics of 78 Related Airfoil Sections from Tests in the Variable Density Wind Tunnel". In this landmark report, the authors noted that there were many similarities between the airfoils that were most successful, and the two primary variables that affect those shapes are the slope of the airfoil means camber line and the thickness distribution above and below this line. They then presented a series of equations incorporating these two variables that could be used to generate an entire family of related airfoil shapes. As airfoil design became more sophisticated, this basic approach was modified to include additional variables, but these two basic geometrical values remained at the heart of all NACA airfoil series, as illustrated in Figure 2.1.

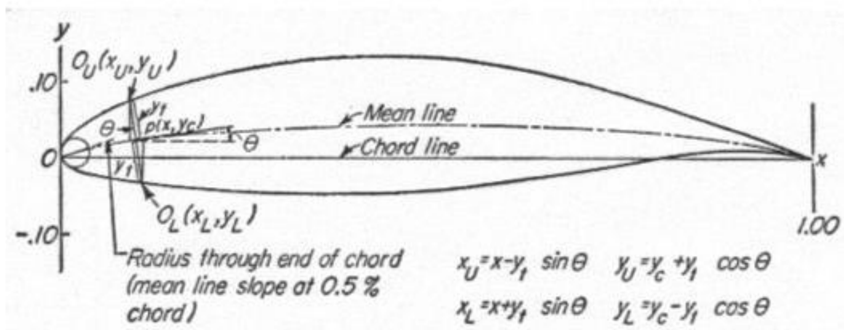


Figure 2.1: NACA airfoil geometrical construction.

Source: http://www.aerospaceweb.org/question/airfoils/q0041.shtml

2.2.1 NACA2415

NACA2415 has become the research subject for many researches in understanding the aerodynamic properties in low Reynolds number flight. It is in the family of NACA Four-Digit Series where the first digit specifies the maximum camber in percentage of the chord (airfoil length), the second indicates the position of the maximum camber in tenths of chord, and the last two numbers provide the maximum thickness of the airfoil in percentage of chord. So, NACA2415 has a maximum thickness of 15 % with a camber of 2 % located 40 % back from the airfoil leading edge.

2.3 ANGLE OF ATTACK (AOA)

The Angle of Attack (AOA) is the angle at which relative wind meets an airfoil. It is the angle formed by the chord of the airfoil and the direction of the relative wind or the vector representing the relative motion between the aircraft and the atmosphere (SKYbrary, 2011) as illustrate in Figure 2.2. In other words, AOA can be simply described as the difference between where a wing is pointing and where it is going.

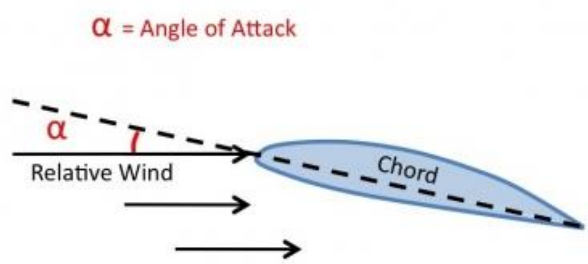


Figure 2.2: Definition of AOA.

Source: SKYbrary (2011)

Most commercial jet airplanes use the fuselage centerline or longitudinal axis as the reference line. AOA is sometimes confused with pitch angle or flight path angle. Referring to Figure 2.3, pitch angle (attitude) is the angle between the longitudinal axis (where the airplane is pointed) and the horizon. This angle is displayed on the attitude indicator or artificial horizon. AOA is the difference between pitch angle and flight path angle when the flight path angle is referenced to the atmosphere. Because of the relationship of pitch angle, AOA, and flight path angle, an airplane can reach a very high AOA even with the nose below the horizon, if the flight path angle is a steep descent.

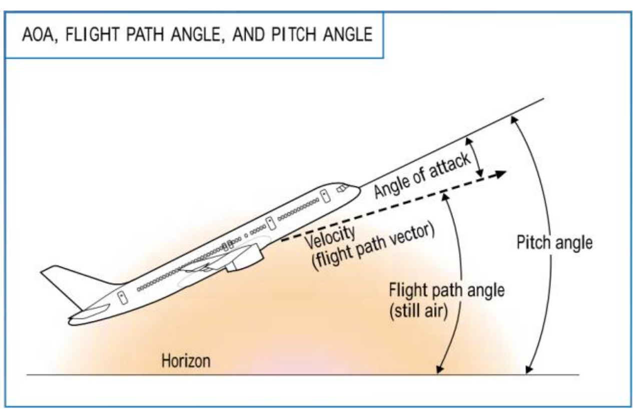


Figure 2.3: AOA, Flight Path Angle, and Pitch Angle.

Source: Aero

2.4 Laminar Separation Bubble.

At the low Reynolds number encounter in Micro-Air Vehicles (MAVs) and animal locomotion, the boundary layer over the body remain laminar over large distance and is thus prone to separation, a phenomena which is detrimental to aerodynamic performance (Uranga, A.,2011). Computational design strategies for low Reynolds number flying and swimming vehicles, as well as for model airplane and glider design, hence rely on the accurate prediction on separation. Furthermore, separation often induces transition to turbulence, which in turn can induce re-attachment. When a laminar boundary layer cannot overcome the viscous effects and adverse pressure gradients, it separates and transition may occur in the free-shear-layer-like flow near the surface and may reattach to the surface forming a LSB (Mayle, 1991). Flow in the region under the LSB, slowly circulates and reverse flow occurs in this region.

2.4.1 Effect of Laminar Separation Bubble.

Laminar separation bubble may cause adverse effects, such as decreasing of lift force, increasing of drag force, reducing stability of the aircraft, vibration, and noise (Nakano et al., 2007; Ricci et al., 2005; 2007; Zhang et al., 2008). Characteristics of LSB must be understood well to design control system to eliminate LSB or design new airfoils which do not affect from adverse effects of LSBs. As can be seen in Figure 2.4, a hump is seen on pressure distribution, this region illuminates the LSB, the region just after the maximum point of this hump indicates transition. If the flow is inviscid, LSB will not take place over the airfoil.

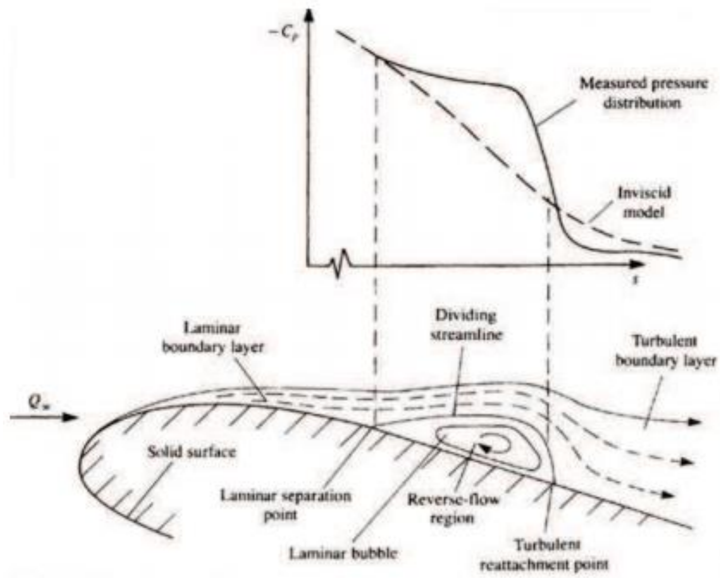


Figure 2.4: The effects of laminar separation bubble on pressure distribution.

Source: Serdar Genc, M. et al.

2.5 REYNOLDS AVERAGED NAVIER-STOKES (RANS) TURBULENCE MODELS

RANS models offer the most economic approach for computing complex turbulent industrial flows. Typical examples of such models are the k-ε or the k-ω models in their different forms. These models simplify the problem to the solution of two additional transport equations and introduce an Eddy-Viscosity (turbulent viscosity) to compute the Reynolds Stresses. More complex RANS models are available which solve an individual equation for each of the six independent Reynolds Stresses directly (Reynolds Stress Models – RSM) plus a scale equation (ε-equation or ω-equation). RANS models are suitable for many engineering applications and typically provide the level of accuracy required. Since none of the models is universal, users has to decide which model is the most suitable for a given applications.

Two models for transition prediction are available in ANSYS FLUENT, namely the SST-transition model and the k-kl Transition model. For many test cases, both models produce similar results. Due to its combination with the SST model, the SST-Transition model is favored. It is important to point out that only laminar-turbulent transition of wall boundary layers can be simulated with any of these two models.

Proper mesh refinement and specification of inlet turbulence levels is crucial for accurate transition prediction. In general, there is some additional effort required during the mesh generation phase because a low-Re mesh with sufficient streamwise resolution is needed to accurately resolve the transition region. Furthermore, in regions where laminar separation occurs, additional mesh refinement is necessary in order to properly capture the rapid transition due to the separation bubble. Finally, the decay of turbulence from the inlet to the leading edge of the device should always be estimated before running a solution as this can have a large effect on the predicted transition location. Physically correct values for the turbulence intensity (Tu) should be achieved near the location of transition.

Chapter 3

Methodology

3.1 INTRODUCTION

The methodology is one of important part of this project which required a lot of step to achieve the research objectives. This chapter will explain on the tools used for the CFD simulation. Beside, all the simulation detail and parameters setting is described here.

3.2 GENERATING NACA2415 AIRFOIL PROFILE

The NACA 2415 is chosen since it is characterized by the formation of LSB along its upper surface which is present across a range of Reynolds number and angle of attack, and has been the subject of several studies. Generating the coordinate points of an airfoil required both theoretical and mathematical work. Coordinates for many of these airfoils already exist in print or on the web. In addition, many programs and web sites now exist that can automatically compute the coordinates once the user enters the desired airfoil name or characteristics.

3.2.1 NACA 4 Digits Series Profile Generator.

In order to precisely plot the profile of NACA 2415, NACA 4 Digits Series Profile Generator is used. This Java web-based applet as shown in Figure 3.1 below, allows the user to generate any 2D airfoil profile within the NACA 4-digit family. The top three sliders are adjusted to create a non-symmetric 2415 airfoil. *# Point* and *Point Size* is change to 60 and 4 respectively. If too many points are used to define the airfoil, ANSYS Fluent Design Modeler would not be able to create the profile of the airfoil because the distance between adjacent points is too small. Other parameters are set as default. After clicking the *Show Point* button, coordinate points of NACA2415 are generated and displayed. But, the data points are not yet ready to be used in Fluent. Microsoft Excel is then used to format the airfoil data points before it is saves as “Text (Tab delimited)” format.

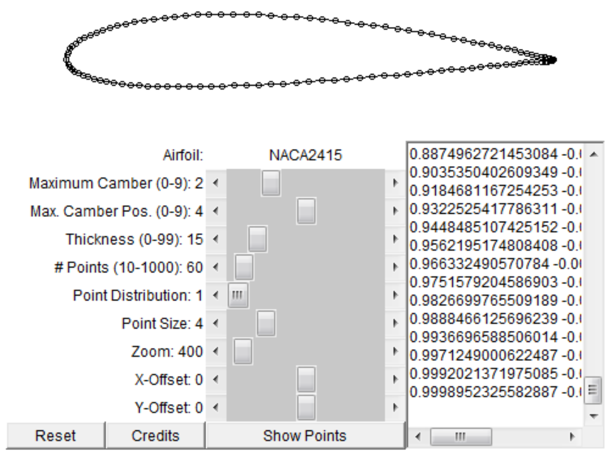


Figure 3.1: NACA 4 Digits Series Profile Generator

Source: http://www.ppart.de/aerodynamics/profiles/NACA4.html

3.3 ANSYS Fluent SIMULATION SETUP.

ANSYS, Inc. develops and globally markets engineering simulation software and technologies widely used by engineers and designers across a broad spectrum of industries. The ANSYS fluid dynamics solution is a comprehensive suite of products that allows user to predict the impact of fluid flows on the product throughout design and manufacturing as well as during end use. ANSYS fluid dynamics solutions gives a valuable insight into the product's performance regardless of fluid flow phenomena being studied such as single phase, multi-phase, isothermal or reacting, compressible and incompressible flow.

ANSYS Fluent software which is one of the ANSYS product contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants. Special models that give the software the ability to model in-cylinder combustion, aeroacoustics, turbomachinery, and multiphase systems have served to broaden its reach. So, ANSYS Fluent is an ideal tool to be used in the simulation. The version used is ANSYS 14.0.

Starting with the simulation, the first thing to do is selecting the Fluid Flow (FLUENT) Analysis System in ANSYS Workbench. This creates a new ANSYS FLUENT-based fluid flow analysis system in the Project Schematic which is composed of various cells (Geometry, Mesh, Setup, Solution and Result.) that represent the work flow for performing the analysis.

3.3.1 Creating the Airfoil Geometry in ANSYS DesignModeler

To begin with, the coordinate of NACA2415 is imported as 3D Curve to create the geometry used in the simulation. From the curve generated, a surface is created resemble the cross section of NACA2415 with 1 meter chord length. As a preparation to create the mesh for the simulation, a C-Mesh domain is creates by sketching the C-Mesh surface on the same plane as the airfoil. Then, the C-Mesh is split into 4 quadrants with the airfoil in the middle as show in Figure 3.2. The dimension of the arc radius is set at 12.5 meter, whereas the sides of other two squares are also set at 12.5 meter.

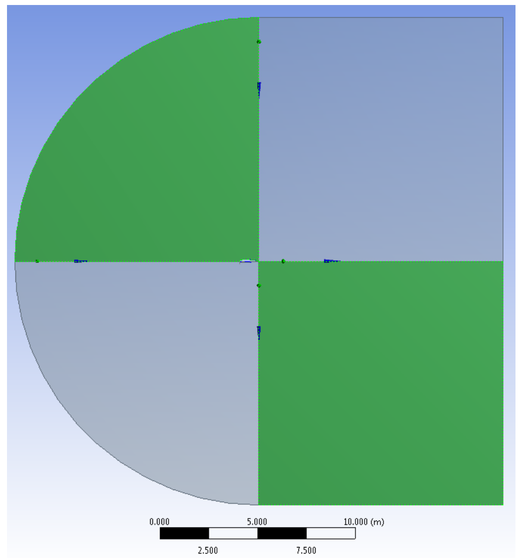


Figure 3.2: C-Mesh domain

3.3.2 Meshing the Geometry in the ANSYS Meshing Application

As for the mesh, ANSYS Fluent offer 3 methods of sizing the mesh which are element size, number of division, and sphere of influence. The mesh sizing used in the simulation is referred to the number of divisions where each side of the 4 quadrants is divided into 200 divisions with the bias factor set at 300 toward the airfoil. The biasing is made in such a way the meshing element is concentrated at the surface of the airfoil since it will improve the accuracy of the simulation result at the boundary layer of the airfoil. The final mesh is shown in Figure 3.3 and the up-close view of the mesh is shown in Figure 3.4 and Figure 3.5. Before creating the boundary condition for the mesh, name for some of the edges are assigned as inlet, outlet, and airfoil as shown in Figure 3.6. The edges highlighted red is named as outlet, and green edges are named as inlet. The edges making up the airfoil profile are named as airfoil.

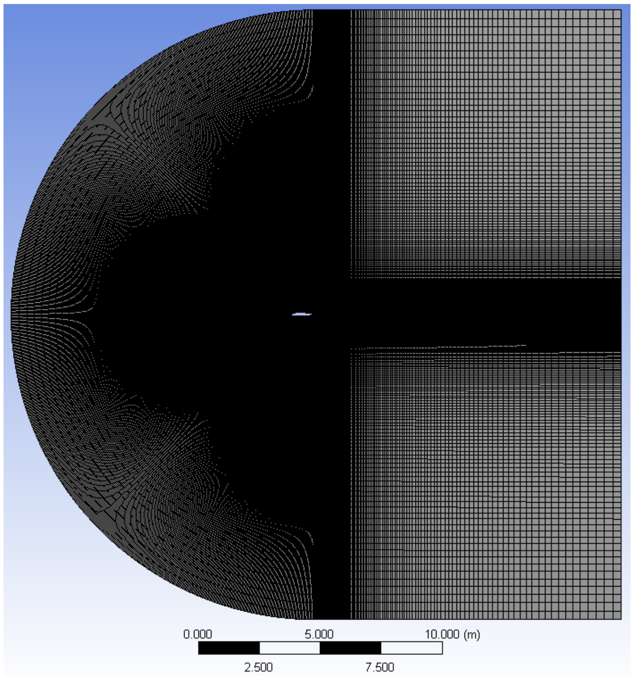


Figure 3.3: Mesh for the simulation.

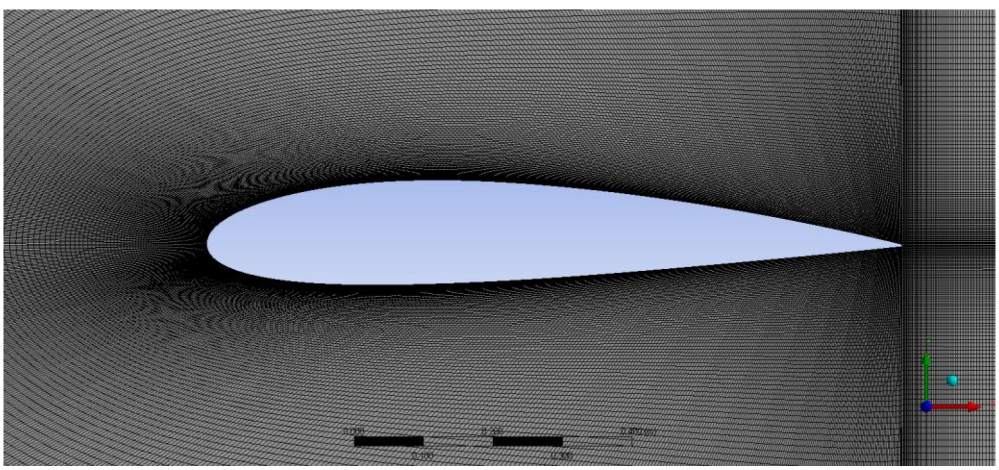


Figure 3.4: Up-close view of the mesh.

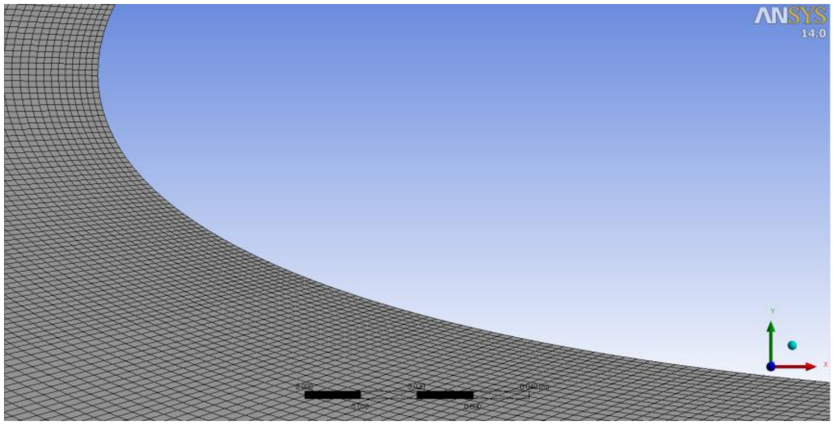


Figure 3.5: Mesh structure at the surface of the airfoil.

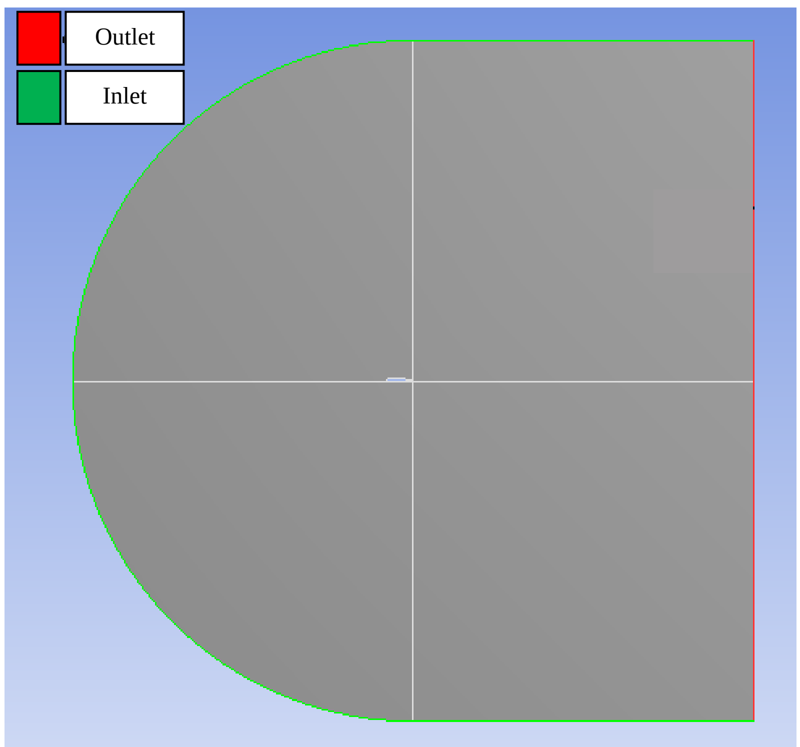


Figure 3.6: Edges name.

3.3.3 Setting Up the CFD Simulation in ANSYS Fluent

After finishing the mesh, parameters setting for the simulation need to be defined. Under FLUENT simulation setup, density-based solver type is used. 4 models were tested for validation which were Spalart-Allmaras, k-ε, k-ω standard, Transition k-k_L-ω, and Transition SST based on their ability to simulate the Laminar separation bubble. Air is chosen for the material type with the density and viscosity were kept as default at 1.225 kg/m^3 and 1.7894×10^{-5} kg/m-s respectively. At the inlet, the magnitude and direction of flow is set as shown in Figure 3.7. As the flow is to enter at angle, θ, X-component of flow direction is cos θ, and the Y-component flow velocity is sin θ. Gauge pressure at the inlet and the outlet is set at 0 pascal. Whereas, the boundary condition for wall is define as wall type. The velocity magnitude is calculated from the definition of Reynolds number as in Eq. (1.1).

With ρ and μ taking the default value and L equals to 1 meter which represent the chord length of the airfoil and Reynolds number equals to 200 000, the value of V can be obtain. From the calculation the value of V used in the simulation is 2.9215 m/s.

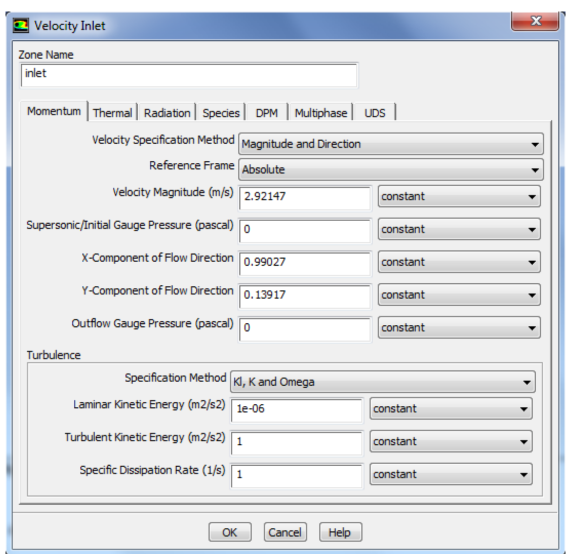


Figure 3.7: Parameter setting at the inlet.

The last thing to do before running the simulation is to acknowledge the reference values which are compute from the inlet. The values that need to be specified are shown in Figure 3.8.

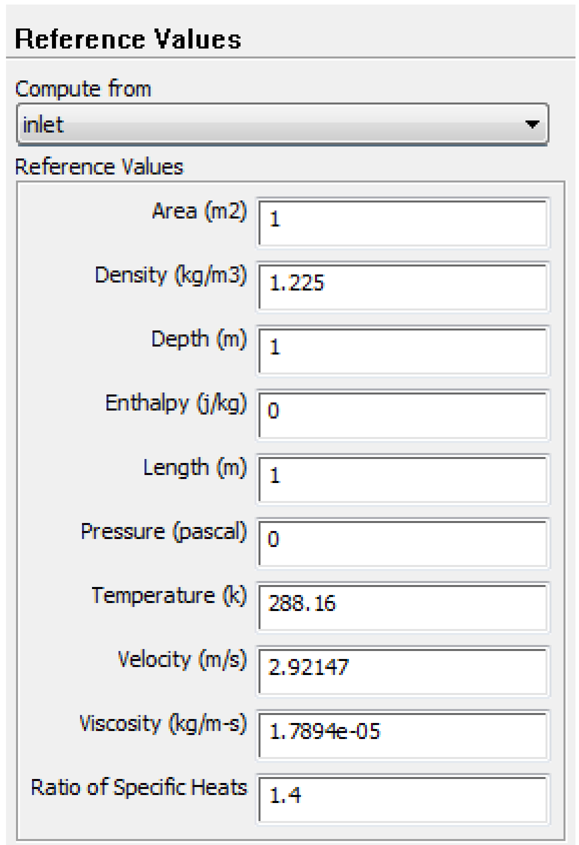


Figure 3.8 : Reference values of the simulation.

3.3.4 Simulation Solution.

In the solution menu, there are several things that can be monitored while the simulation is running. As the code iterates, residuals are calculated for each flow equation. Residual represent a kind of average error in the solution. The smaller the residual, the more converge the solution. Since there are four differential equations to be solved, there are four residuals to be monitored for convergence which are continuity, x- velocity, y-velocity, and k_L value. The default convergence criteria are 0.001 for all four of these and this value is generally not low enough for proper convergence. So, the default value is changed from 0.001 to 0.00001. Lastly, after initializing the solution, run the simulation and wait for the calculation to converge. Upon convergence, a message is displayed in the main FLUENT window as shown in Figure 3.9.

```
  iter continuity x-velocity y-velocity          k     epsilon        Cl-1        Cd-1      time/iter
!38641 solution is converged
 38641 9.9810e-07 5.2969e-07 7.2242e-07 4.7232e-06 9.3068e-06 7.9551e-01 1.7002e-02  0:00:00 200000
 38642 7.5718e-05 8.4882e-07 7.3431e-07 4.7229e-06 9.3093e-06 7.9556e-01 1.5413e-02 55:33:19 199999
 38643 3.6500e-05 5.8110e-07 7.2606e-07 4.7226e-06 9.3078e-06 7.9556e-01 1.5413e-02 55:33:18 199998
 38644 2.4223e-05 5.4585e-07 7.2501e-07 4.7223e-06 9.3073e-06 7.9556e-01 1.5412e-02 44:26:38 199997
 38645 1.5589e-05 5.3755e-07 7.2608e-07 4.7220e-06 9.3070e-06 7.9556e-01 1.5411e-02 46:39:57 199996
 38646 1.0096e-05 5.3493e-07 7.2684e-07 4.7217e-06 9.3066e-06 7.9556e-01 1.5411e-02 48:26:36 199995
 38647 7.6629e-06 5.3383e-07 7.2672e-07 4.7214e-06 9.3061e-06 7.9557e-01 1.5410e-02 49:51:55 199994
```

Figure 3.9: Convergence message in the main Fluent menu.

3.4 DATA VALIDATION

As for the Ansys simulation data validation, the simulation result is compared to the previous experimental result presented by Serdal Genc, M. He is experimenting on the aerodynamic properties of NACA2415 airfoil at low Reynolds number in low speed, suction-type wind tunnel with a square working section of 500mm x 500mm which are similar to the wind tunnel at the Faculty of Manufacturing in UMP. For validation purpose, air flow around NACA2415 at 8^o angle of attack with Reynolds number of 200000 is simulated and the pressure coefficient, C_p distribution over the airfoil from the simulation is compare with the experimental. Another parameter for validation is the value of C_d and $C_{l.}$ When the simulation result agreed with the experimental result, it mark that the simulation model is in close representative of the real world flow and the same simulation setting will be used to simulate the airflow for the other angles of attack.

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

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 the entire 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 VALIDATION RESULT

4.2.1 Experimental result of C_p distribution over NACA2415 airfoil.

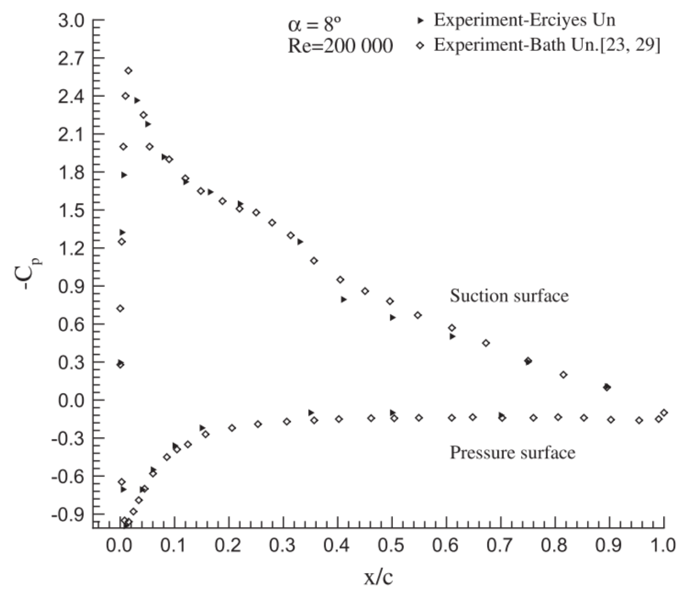


Figure 4.1: Experimental result of C_p distribution over the NACA2415 airfoil

Source: Serdar Genc, M. (2012)

Figure 4.1 shows the experimental result of C_p distribution over NACA2415 airfoil at AOA of 8^o and Re = 200,000 at the University of Erciyes, Turkey. Notice that the C_p distribution on the suction surface shows a hump starting from 0.1 to 0.4 on x/c scale. This indicates the present of transition or LSB at that area. The minimum and maximum C_p value on the suction surface is -2.4 and 0.15 respectively. Whereas, on the pressure surface, the maximum C_p value of 1.02 is recorded.

4.2.2 Simulation result of C_p distribution over NACA2415 airfoil.

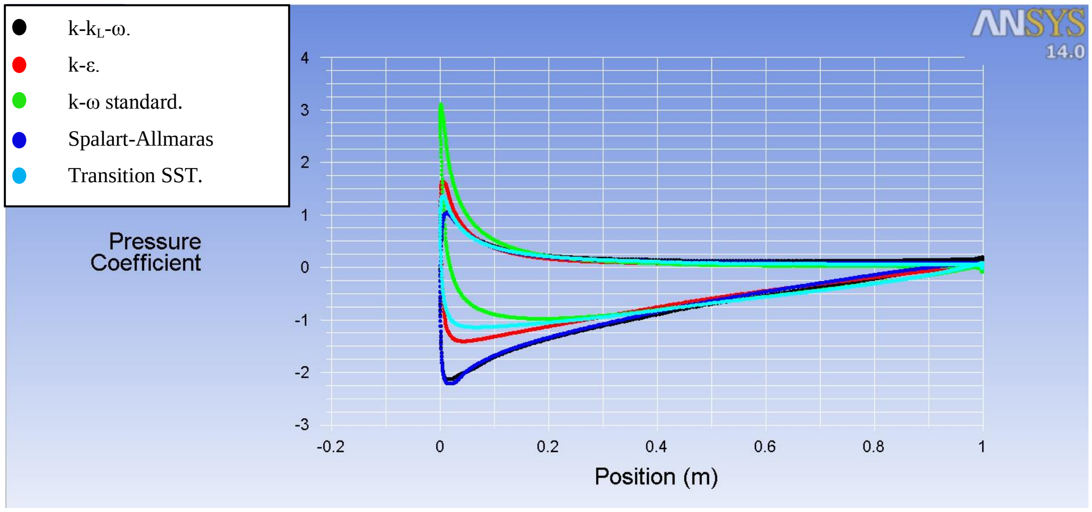


Figure 4.2: Simulation result of C_p distribution over the NACA2415 aerofoil.

Figure 4.2 shows the comparison of C_p distribution over NACA2415 airfoil of different simulation model at AOA of 8^o and Re = 200,000, which is in the same flow condition as the experiment at the University of Erciyes, Turkey. The presentation of this simulation result is actually inverted about the horizontal axis from the graph of experimental result. Comparing the simulation result from Figure 4.1 and the experimental result in Figure 4.2, the general C_p distribution line of all simulation models are the same as the experimental one. From 0.5 to 1 chord length, the C_p distribution is about the same value. But, a big variation in C_p value is recorded from 0 to 0.5 chord length. Interestingly, both k-k_L-ω and Spalart-Allmaras model shows only a slight difference for their C_p distribution line. Spalart-Allmaras model minimum and maximum C_p value on the suction surface is -2.25 and 0.12 respectively. Whereas, on the pressure surface, the maximum C_p value of 1.03 is recorded. These values are very close to the experimental result. As for the k-ε, k-ω standard, and Transition SST model, all of them under-predict the magnitude of Cp on the suction surface and over-predict the magnitude of C_p on the pressure surface.

Unfortunately, all of them fail to capture the formation of LSB since none of them show the present of a hump as in the experimental result.

4.2.3 Experimental result of C_d and C_l over NACA2415 airfoil.

Figure 4.3 shows the relationship between C_l, C_d, and C_m with angle of attack at the Reynolds number of 200 000. The value of C_l, and C_d at AOA of 8^o is tabulated in Table 4.1.

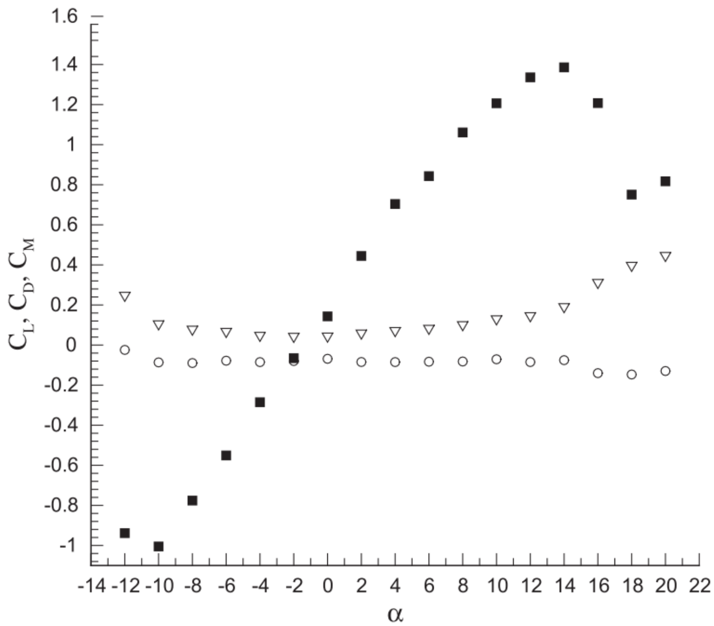


Figure 4.3: Experimental result of C_l, C_d, and C_m

Source: Serdar Genc, M. (2012)

Table 4.1 : Cd and Cl values of experimental result.

Angle of Attack	C_d	C_l
8^o	0.1000	1.0800

4.2.4 Simulation result of C_d and C_l over NACA2415 airfoil.

Table 4.2: Cd and Cl values of simulation result.

Model	C_d	C_l
Transition k-k_L-ω.	0.0945	1.0051
k-ε	0.0154	0.7956
k-ω standard.	0.1555	0.7703
Spalart-Allmaras	0.0991	0.9319
Transition SST.	0.0253	0.8256

Table 4.2 shows the C_d and C_l values of all the model used in the validation test. Comparing the simulation result with the experimental result in Table 4.1, the most accurate prediction of C_d and C_l comes from Spalart-Allmaras and Transition k-k_L-ω. k-ε model and Transition SST model under predict both the value of C_d and C_l with a large differences. Whereas, for k-ω standard model, it over predict the value of Cd and under predict the value of Cl.

Based on the validation data, it is clearly shown that the only acceptable models to be used in this project are Spalart-Allmaras and Transition k-k_L-ω. Both models show a high accuracy when it comes to predict the value of C_d and C_l. But still, both models fail to capture the formation of LSB which will greatly influence the aerodynamic characteristic of an airfoil in low Re. Surprisingly, Transition k-k_L-ω model was once shows the formation of LSB. Unfortunately, the result of it was not documented right away. When the result was displayed for the second time, the C_p distribution was not showing the same result as before. The evident of the Formation of LSB captured by that model is gone. Even after numerous trial using the Transition k-k_L-ω model, the result are still frustrating without the formation of LSB. The reason for that change in the result is still unknown. As a conclusion, the model tested still need to be study for them to be able to predict the formation of LSB. The validation test is a failure. So, further work is at hold until the simulation model pass the validation test.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 CONCLUSION

In this project, the aerodynamic characteristic of NACA2415 airfoil is investigated. Low Re number flow condition is simulated using ANSYS Fluent. It can be conclude that, there are several parameters related to the performance of an airfoil which are C_p,C_d,C_l, and C_m. The factor influencing the value of that parameters, are angle of attack, and the Re number of the flow. At low Re number, the flow are prone to the formation of LSB on the suction surface of the airfoil which will greatly reduce the performance of an airfoil. By increasing the angle of attack, the amount of lift generated increase linearly until the airfoil is at its critical angle of attack. At this point, airfoil experiences its maximum lift force. Further increase in AOA will result in a rapid drop in the lift force and stall occurs. As for airfoil drag, the differences in AOA will give a relatively small effect. But, during stall condition, the drag force will become significant as it will increase rapidly. Unfortunately, all of these cannot be simulated as the validation result is did not good enough. Spalart-Allmaras and Transition k-k_L-ω model yield a good agreement with the experimental data in term of C_d and C_l while the other model gives a relatively large error in their simulation result. Validation process is still pending due to the absent of LSB in the simulation result. Further work need to be done to correct the mistake and find the factor leading to the failure.

5.2 RECOMMENDATIONS

Since the validation test is not a success, but still, there are a lot to learn and to discover from this failure. In searching for the corrective action, there are several factors which are believed to be the source of error. The first recommendation is by refining the geometry of the trailing edge of the airfoil. In the simulation, close-up view show that the trailing edge is rounded instead of pointed. This may cause undesired disturbance in the air flow and affecting the result. Another improvement for the simulation is to study more on the meshing parameter since the mesh is a big factor influencing the accuracy of the result. A correct Y-plus value for the mesh need to be investigated and implemented.

REFERENCES

Mayle, R.E. (1991). The Role of Laminar-Turbulent Transition in Gas Turbine Engines. Journal of Turbomachinery, Vol. 113, 509-537.

Nakano, T., Fujisawa, N., Oguma, Y., Takagi, Y., Lee, S. (2007). Experimental studyon flow and noise characteristics of NACA0018 airfoil. Journal of Wind Engineering and Industrial Aerodynamics, Vol. 95, pp. 511-531.

Ricci, R. & Montelpare, S.A. (2005). Quantative IR Thermographic Method to Study the Laminar Separation Bubble Phenomenon. International Journal of Thermal Sciences, Vol. 44, pp. 709-719.

Ricci, R., Montelpare, S.A., Silvi, E. (2007). Study of acoustic disturbances effect on laminar separation bubble by IR thermography. Experimental Thermal and Fluid Science, Vol. 31, pp. 349-359.

Zhang W., Hain R., Kahler C.J. (2008). Scanning PIV Investigation of the Laminar Separation Bubble on a SD7003 Airfoil. Experiment in Fluids, Vol. 45, pp. 725-743.

Katz, J. & Plotkin, A. (1991). Low-Speed Aerodynamics from Wing Theory to Panel Methods, McGraw-Hill, Inc.

Cohen P. Kundu, Fluid Mechanics, chapter Dynamic Similarity, pp. 279–294, Academic Press, 2008.

Catalano, P. & Tognaccini, R., (2010). Turbulence ModelingforLow-Reynolds-NumberFlows. AIAAJournal, Vol. 48, pp. 1673-1685.

Genç, M.S., Kaynak, Ü., Yapici, H. (2011). Performance of Transition Model for Predicting LowReAerofoilFlowswithout/with SingleandSimultaneous Blowingand Suction, EuropeanJournalof Mechanics B/Fluids, Vol. 30, pp. 218-235.

Genç, M.S., Karasu, ˙I., Açıkel, H.H. (2012). An experimental study on aerodynamics of NACA2415 aerofoil at low Re numbers, Experimental Thermal and Fluid Science, DOI:10.1016/j.expthermflusci.2012.01.029, in press.

Karasu, ˙I. (2011). Experimental and numerical investigations of transition to turbulence and laminar separation bubble over aerofoil at low Reynolds number flows (In Turkish), MSc. Thesis, Graduate School of Natural and Applied Sciences, Erciyes University, Kayseri, TURKEY.