AIRFOIL AERODYNAMICS ANALYSIS BY USING VISCOUS-INVISCID DIRECT ANALYSIS

MOHD FAIZAL BIN CHE MOHD HUSIN

A thesis submitted as partial fulfillment of the requirement for The Award of Masters of Mechanical Engineering.

Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia

MAY, 2011

ABSTRACT

The main objective of this research is to develop computer code for airfoil aerodynamics analysis. The flow assumed to be incompressible and pass through a streamline body at relatively high Reynolds number. As result the flow domain around the airfoil can be divided into two regions, namely the flow region closed to the airfoil surface and the flow region relatively far away from the surface which the viscosity effects can be ignored. The first flow region is known as Boundary layer region while the second flow region is called as inviscid flow domain. The inviscid flow problem can be solved by use a Panel Method. While the boundary layer region can be solved by two approaches depended on the governing equation had been used to represent the flow behavior inside the boundary layer. For boundary layer analysis, the present work used the combined Thwaites – Head's method and Keller Box Method. As result two computer codes had been developed. The first computer code called as The Present Computer Code 1 is the computer code which consists of Panel Method and the combined Thwaites – Head's Method, while the second code called as The Present Computer Code 2 represents the combination between Panel Method and Keller Box Method. Both computer codes use a Michele method for predicting the transition point laminar to turbulent. Both computer code applied for several test case of airfoil at different angle of attack and compared with result provided by XFOIL Code. Their comparison results show encouraging result and for the Keller Box method may need exploration further more in order improve their accuracy in turbulent part.

CHAPTER 1

INTRODUCTION

1.0 Introduction

An airfoil (in American English) or aerofoil (in British English) is the shape of a wing or blade (of a propeller, rotor or turbine) or sail as seen in cross-section (Wikipedia, 2010). Study on airfoil has begun around 100 years ago.

In former times, when no theoretical methods were available, relevant geometric parameters were varied systematically and the effect on the aerodynamic characteristics was measured in a series of wind-tunnel tests. The first extensive experimental investigations in this sense were performed at the AVA after completion of the G¨ottinger low-speed wind-tunnel in 1917/1918. After L. Prandtl developed the boundary-layer theory it became obvious that the outer-flow velocity distribution along the airfoil surface determines the airfoil performance. This gave reason to develop inverse methods which enable the calculation of the airfoil contour from a given velocity distribution. First methods for the approximate inverse solution of the Laplace equation were developed in the late twenties and the thirties (Lutz, 2000).

The National Advisory Committee for Aeronautics (NACA) recognized the importance of airfoils as a cornerstone of aeronautical research and development. The first series of airfoils, designated "M sections" for Max M. Munk, was tested in the Langley Variable-Density Tunnel. This series was significant because it represented a systematic approach to airfoil development as opposed to earlier, random, cut-and-try approaches. This empirical approach, which involved modifying the geometry of an existing airfoil, culminated in the development of the four- and five-digit-series airfoils in the mid 1930's (NACA, 2000).

F. X. Wortmann and Richard Eppler from Germany then engaged in laminar-flow airfoil design. Wortmann employed singularity and integral boundary layer methods to develop a catalog of airfoils intended primarily for sailplanes. Because the theoretical methods he used were relatively crude, however, final valuation of the airfoils was performed in a low-turbulence wind tunnel. Eppler pursued the development of more accurate theoretical methods .The successor to the NACA, the National Aeronautics and Space Administration (NASA), reentered the airfoil field in the 1960's with the design of the supercritical airfoils by Richard T. Whitcomb (NACA, 2000).

The development of transonic airfoils is transferred to the design of a series of turbulent-flow airfoils for low-speed aircraft. The basic objective of this series of airfoils was to achieve higher maximum lift coefficients than the earlier NACA airfoils. While these NASA, turbulent-flow airfoils did achieve higher maximum lift coefficients, the cruise drag coefficients were no lower than those of the NACA four- and five-digit-series airfoils. Emphasis was therefore shifted toward natural-laminar-flow (NLF) airfoils in an attempt to combine the low drag characteristics of the NACA 6-series airfoils with the high-lift characteristics of the NASA low-speed airfoils. In this context, the term 'natural-laminar-flow airfoil' refers to an airfoil that can achieve significant extents of laminar flow (30-percent chord) on both the upper and lower surfaces simultaneously, solely through favorable pressure gradients (no boundary-layer suction or cooling) (NACA, 2000).

Today, airfoils are being designed for an ever-widening range of applications. Examples include unmanned aerial vehicles, cooling-tower fans, sailplanes, wind turbines, rotorcraft, and general aviation, commuter, and transport aircraft (NACA, 2000).

1.2 Geometry and Aerodynamic Characteristic of Airfoil

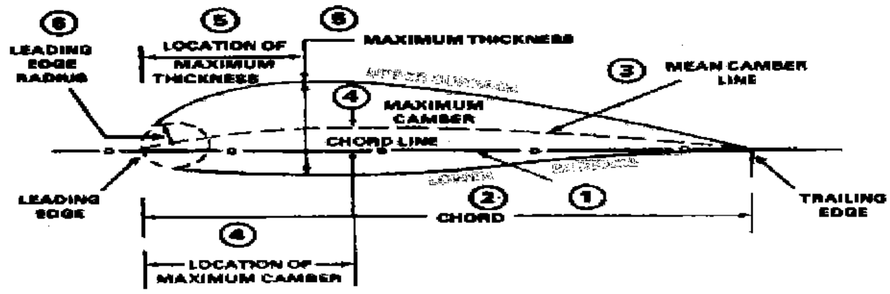

FIGURE 2-10. AIRFOIL TERMINOLOGY.

Figure 1.1:Nomenclature of Airfoil Geometry (Cantrell, 2004)

An airfoil-shaped body moved through a fluid produces an aerodynamic force. The main parameters which described the geometry of airfoil as shown in the Figure 1.1 are:

i. The chord line is a straight line connecting the leading and trailing edges of the airfoil.

ii. The chord is the length of the chord line from leading edge to trailing edge and is the characteristic longitudinal dimension of the airfoil.

iii. The mean camber line is a line drawn halfway between the upper and lower surface. The chord line connects the ends of the mean camber line.

iv. The shape of the mean camber is important in determining the aerodynamic characteristics of an airfoil section. Maximum camber (displacement of the mean camber line from the chord line) and the location of maximum camber help to define the shape of the mean camber line. These quantities are expressed as fractions or percentages of the basic chord dimension.

v. Thickness and thickness distribution of the profile are important properties of an airfoil section. The maximum thickness and its location help define the airfoil shape and are expressed as a percentage of the chord.

vi. The leading edge radius of the airfoil is the radius of curvature given the leading edge shape.

The component of this force perpendicular to the direction of motion is called lift. The component parallel to the direction of motion is called drag. The definition of the incoming velocity, lift and drag force for the flow past through airfoil are shown in the Figure 1.2 (Scott, 2004).

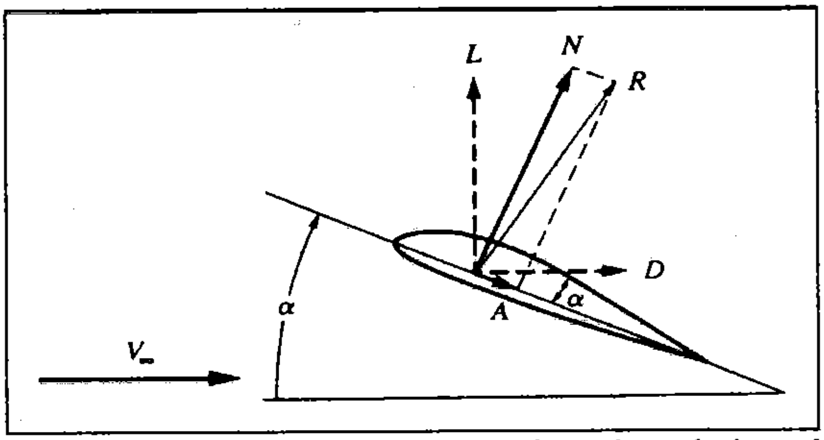

Figure 1.2: Definition of angle of attack, incoming velocity and force vector on airfoil (Scott, 2004)

The performance of an airfoil can be characterized by three quantities: the lift, moment and drag coefficients: C_l, C_m and C_d respectively. They represent the aerodynamic loads applied to the airfoil. The actual loads are proportional to the coefficients times the square of the flow velocity (Wauquiez, 2000).

C_l corresponds to the force acting on the airfoil in the direction orthogonal to the flow, which allows an aircraft to fly by compensating its weight.

- C_m corresponds to the moment of the aerodynamic force with respect to the quarter of the airfoil chord length. For the equilibrium of an aircraft, the pitching moment of the main wing has to be compensated by the moment of a negative-lift tail. C_m should therefore not be too large.

Finally, Cd corresponds to the component of the force in the flow direction, which hinders aerodynamic performances and causes fuel consumption.

Typical Aerodynamic characteristics of airfoil as function of angle of attack as shown in the Figure 1.3.

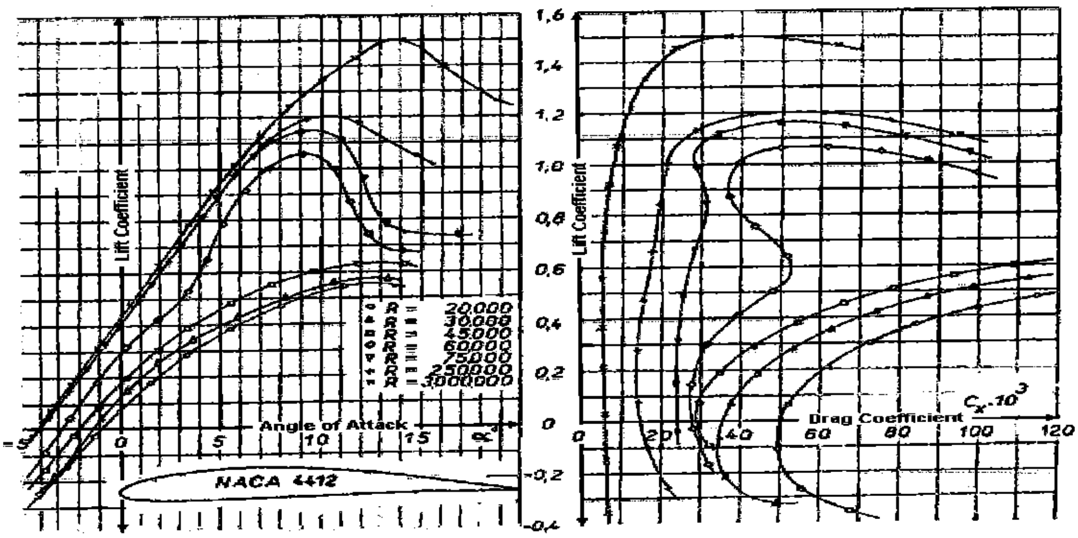

Figure 1.3: Lift and Drag as function of angle of attack on airfoil at different Reynolds Number for NACA 4412 (Model Aircraft, 2010)

The lift on an airfoil is primarily the result of its shape (in particular its camber) and its angle of attack. When either is positive, the resulting flow field about the airfoil has a higher average velocity on the upper surface than on the lower surface. This velocity difference is necessarily accompanied by a pressure difference, via Bernoulli's principle for incompressible inviscid flow, which in turn produces the lift force. The lift force can also be related directly to the average top/bottom velocity difference, without invoking the pressure, by using the concept of circulation and the Kutta-Joukowski theorem (Wikipedia, 2010).

Airfoil design is a major facet of aerodynamics. Various airfoils serve different flight regimes. Asymmetric airfoils can generate lift at zero angle of attack, while a symmetric airfoil may better suit frequent inverted flight as in an aerobatic aeroplane. In the region of the ailerons and near a wingtip a symmetric airfoil can be used to increase the range of angles of attack to avoid spin-stall. Thus a large range of angles can be used without boundary layer separation. Subsonic airfoils have a characteristic shape with a rounded leading edge, followed by a sharp trailing edge and often with asymmetric camber. Foils of similar function designed with water as the working fluid are called hydrofoils. Such rounded leading edge is naturally insensitive to the change of angle of attack.Modern aircraft wings may have different airfoil sections along the wing span, each one optimized for the conditions in each section of the wing (Wikipedia, 2010).

Movable high-lift devices, flaps and sometimes slats, are fitted to airfoils on almost every aircraft. A trailing edge flap acts similar to an aileron, with the difference that it can be retracted partially into the wing if not used.A laminar airfoil has a maximum thickness in the middle camber line. Analyzing the Navier-Stokes equations in the linear regime shows that a negative pressure gradient along the flow has the same effect as reducing the speed. So with the maximum camber in the middle, maintaining a laminar flow over a larger percentage of the wing at a higher cruising speed is possible (Wikipedia, 2010).

Figure 1.4 shows various type of airfoil had been developed as the aircraft technology take in progress. The airfoil can be classified in respect to time as early airfoil development such as: Wright, RAF Airfoil, Gottingen, Clark Y etc. The second period of airfoil development known as Airfoil NACA, there are hundreds of airfoil had been created during period time of 1931 to 1946. During these period 3 classes airfoil had been introduced, they are namely: Airfoil NACA Series: four, five and six digits airfoil. In response to fulfill a better aircraft performance, the airfoil development still continue, as result there are new airfoils which can be grouped as a modern airfoils. Those airfoil for instance: Lissaman Airfoil, General Aviation Whitcomb (GAW), Liebeck airfoil and supercritical airfoil (Raymer, 1992).

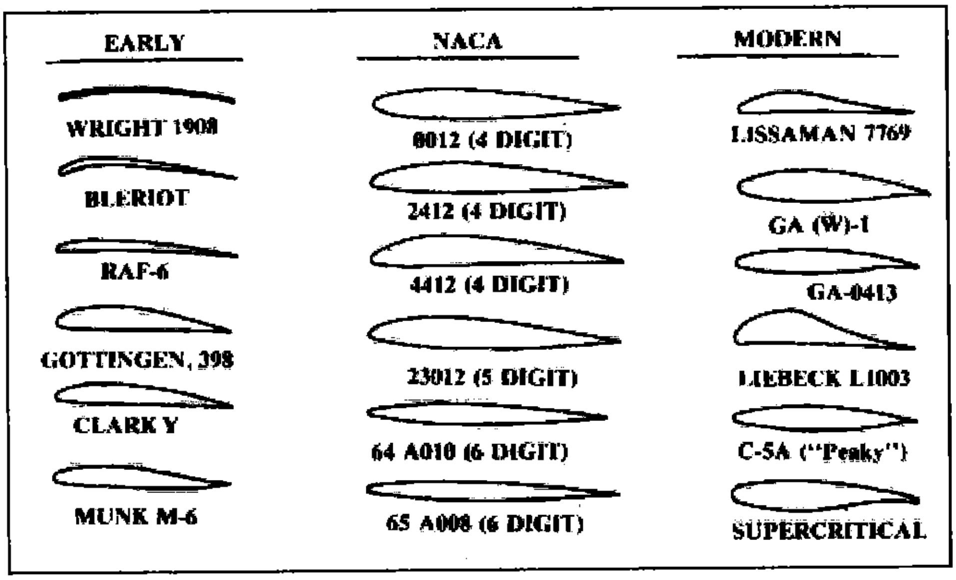

Figure 1.4: Various type of airfoils respect on period time (Raymer, 1992)

Airfoil plays important role in the aircraft design, the aerodynamics characteristic of the airplane cannot be separated with the aerodynamic characteristic of the airfoil. As result understanding of aerodynamics characteristics is required for success in aircraft design. Recognizing the importance of aerodynamics characteristic of the airfoil, various schemes which design to be able to predict the aerodynamics characteristics airfoil had been developed. There are various aerodynamics airfoil analysis software available in the market, such as: PROFOIL, XFOIL and AeroFoil (Wikipedia, 2010). Beside that the most of CFD software which designed to solve the general flow problem can also be used as tool for aerodynamics airfoil analysis.

1.3 Overview of the Viscous-Inviscid Model

The general Navier-Stokes equations are very powerful since they give a complete description of all possible flow situations. However it is very time-consuming to obtain a numerical solution using them. In particular case, at high Reynolds number, the viscous effects are important only in a small region near the profile. In this region, the Navier-Stokes equations can be approximated by the so-called boundary layer equations. Outside, viscous effects can be neglected, and one can use an inviscid flow model (Wauquiez, 2000).

It is true that certain flow problem, the solution of inviscid flow in term of pressure distribution over the airfoil surface is in a good agreement with the experimental result. Such good agreement is normally obtained if the airfoil is very streamline and at relative low angle of attack. As the angle of attack is increasing deviation result of inviscid solution to the experimental result can be avoided. This discrepancy may be eliminated if the viscous affect taken account in the inviscid solution. Such idea had been introduced a new approach in the way how to solve the aerodynamic airfoil analysis. This new approach is known as Viscous – Inviscid Interaction Scheme or sometimes called a viscous – inviscid coupling. Figure 1.5 shows the geometry of viscous and inviscid aerofoil (Wauquiez, 2000).

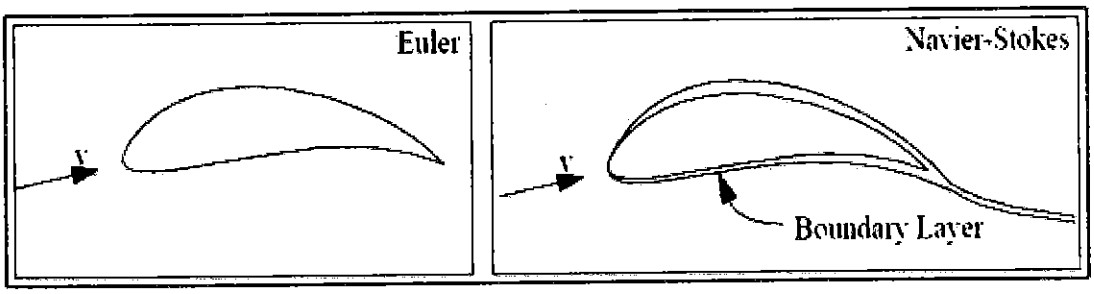

Figure 1.5: Left: airfoil occupied in inviscid flow, and Right: Airfoil occupied in viscous flow (Wauquiez, 2000)

The coupling between the two models is as follows:

iii. The effect of the boundary layer is that it modifies the shape of the airfoil as seen by the external flow. This gives the inviscid flow a zero normal velocity condition on a boundary obtained by adding the boundary layer displacement thickness to the airfoil's surface.

iv. The boundary layer equations depend on the external tangential velocity distribution.

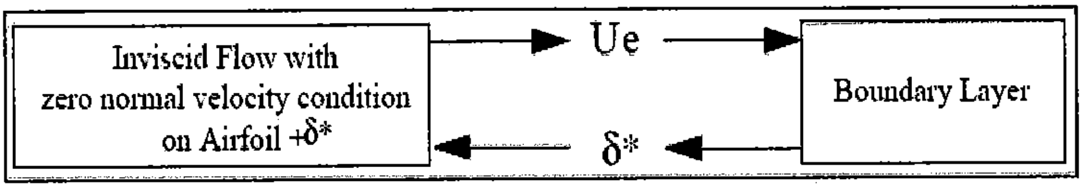

Figure 1.6: Two different approaches are possible for the coupling of inviscid - boundary layer flows (Wauquiez, 2000)

1.3.1 Coupled Computation

In the two-way coupled computation case, the modification of the boundary where the zero normal velocity condition has to be met due to the boundary layer thickness is taken into account. The solution begins with the inviscid flow problem, which produces the velocity field. This data is then fed into the boundary layer model which results the local wall friction coefficient and the displacement thickness. Then a second iteration is performed, now with modified surface geometry (Wauquiez, 2000).

This modification can be obtained by displacing the body panels according to the local displacement thickness, and the procedure is reiterated until a converged solution is obtained. Another way to account for the displacement effects is to modify the boundary condition instead of the geometry. In this case the normal flow is made non-zero to account for the effect of δ^*. This formulation, known as the transpiration velocity concept, states that:

$$V.n = \frac{d}{dx}(U_e\delta^*) \quad \text{on the airfoil surface}$$

Using a two-ways coupled model, the total drag is found by adding the friction drag, obtained by integrating the skin friction coefficient Cf, to the pressure drag, obtained by integrating the inviscid pressure distribution. A recent approach is the one used in ISES and Xfoil. The transpiration velocity concept is used, and the entire nonlinear equation set is solved simultaneously as a fully coupled system by a Newton-Raphson method. This method provides more robust results than the iterative approach (Wauquiez, 2000).

In the case One-Way Coupled Computation, the effect of the boundary layer thickness is neglected. One, single iteration is performed: the external tangential velocity is computed by the inviscid model with the condition V.n=0 on the airfoil's surface, and then fed into the boundary layer model. The drag coefficient is obtained using the Squire-Young formula, which in effect computes the momentum deficit. It is a function of some of the boundary layer results (momentum thickness and shape factor) at the trailing edge.

Compared to coupled computations, the accuracy of the lift is very good, but not as good for the drag. However it is the sensitivity of the results with respect to airfoil shape or flow parameters is well reproduced (Wauquiez, 2000).

1.4 Problem Statement

Flow over the airfoil surface taking important place in aerodynamic study. Since Prandtl introduce boundary layer concept over a decade ago, this study shows drastically improvement as we can see in aircraft industry. In study airfoil problems, the coupling between viscous and inviscid has been emphasized since those terms: viscous-inviscid, is the key role which from here, aerodynamic characteristics such as lift coefficient C_l, drag coefficient C_d, moment coefficient C_m, and that sort of properties can be determined.

It is true that various solutions had been represented in aerodynamic history in response to physical problem that they deal with. The solutions whether it was analytical or numerical solution shows us how knowledge was grew up keep along with time. However, how sophisticated was it, definitely it might be began from the basic understanding. In aerodynamic study, there are several methods to solve viscous-inviscid problem relied on fluid motion case and airfoil geometry given. Potential flow concept is good enough for basic inviscid flow problem while boundary layer concept is appropriate to applied in viscous flow analysis.

Numerous models have been introduced in order to make the analysis become easier. Thus, the analyzer which occupied on fluid flow had to involve with computational scheme so that all complex data can be taken as well as and analysis also can be undergone.

1.5 Objective

Based on statement of problem, two objectives have been highlighted as the guideline for this study as follows:

1. To develop computer code airfoil aerodynamics analysis by using a viscous inviscid direct analysis.

2. To understand the aerodynamics characteristics of the airfoil

1.6 Scope

1. At first some assumption might be state here such as flow is in an incompressible fluid, subsonic flow case, 2-Dimension, and irrotational.
2. Study on the potential flow concept for inviscid flow phenomena.
3. Study on the boundary layer concept for viscous flow phenomena.
4. Develop computer code for inviscid flow analysis
5. Develop computer code for viscous flow analysis
6. Combined two developed computer code to become a viscous inviscid direct analysis.
7. Use XFOIL software for comparison result with the developed computer code.

1.7 Expected Result

There are so many papers about such this study have been done by previous researches and they emphasized on correlation pressure distribution respects to airfoil geometry. The results will be looked like as in Figure 1.7 which taken from (Katz and Plotkin). This Figure shows pressure distribution on a NACA 0012 airfoil at zero angle of attack in a laminar flow where pressure coefficient C_p is lower relative to turbulent flow.

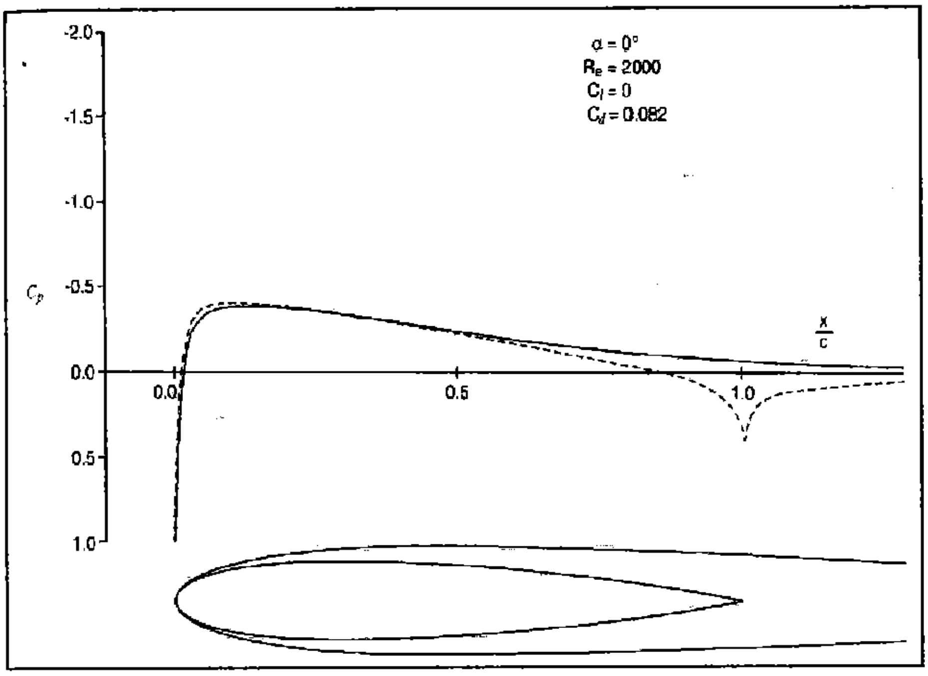

Figure 1.7: Calculated pressure distribution on a NACA 0012 airfoil at zero angle of attack. Dashed line stands for the potential solution and the solid line for the flow with viscous interaction. The airfoil and displacement thickness are depicted at the lower part of the figure

For turbulent flow, the results will be demonstrated as in Figure 1.8 (Katz and Plotkin) where pressure coefficient C_p initially increases sharply and begins to drop after certain point.

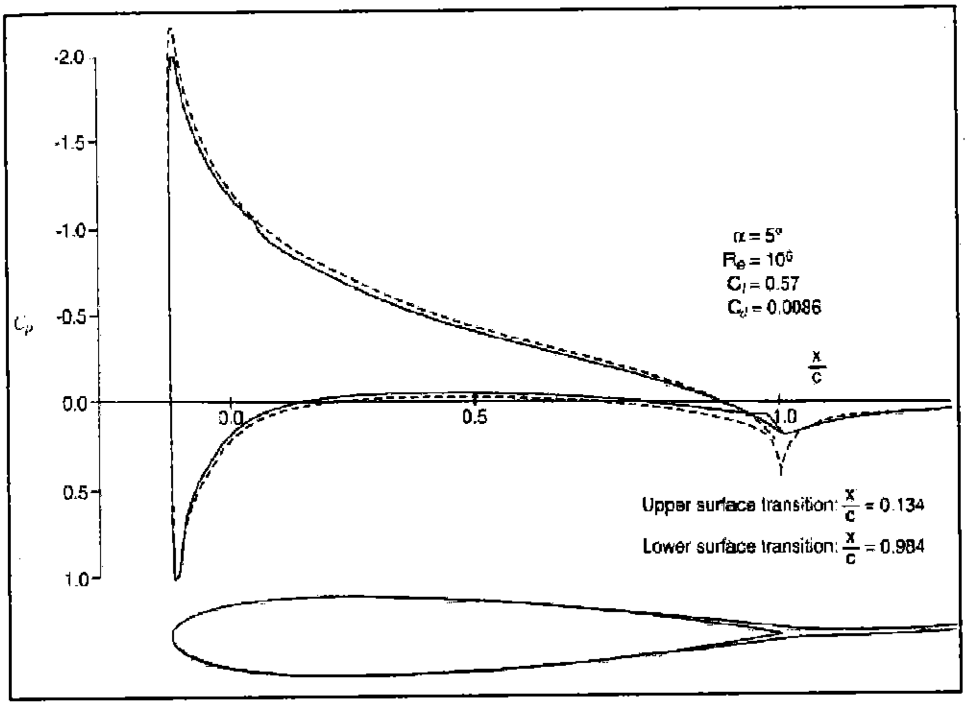

Figure 1.8: Calculated pressure distribution on a NACA 0012 airfoil at 5° angle of attack. Dashed line stands for the potential solution and the solid line for the flow with viscous interaction. The airfoil and displacement thickness shapes are depicted at the lower part of the figure

The angle of attack also resulted as function of coefficient of lift C_l and coefficient of drag C_d. As an example, a result of RAF 32 is taken in order to expect what will obtain in this study. See Figure 1.9 this figure shown relationship between coefficient of lift C_l and coefficient of drag C_d respect to the angle of attack α. The correlation between is α and C_l, and α with C_d is shown in Figure 1.10.

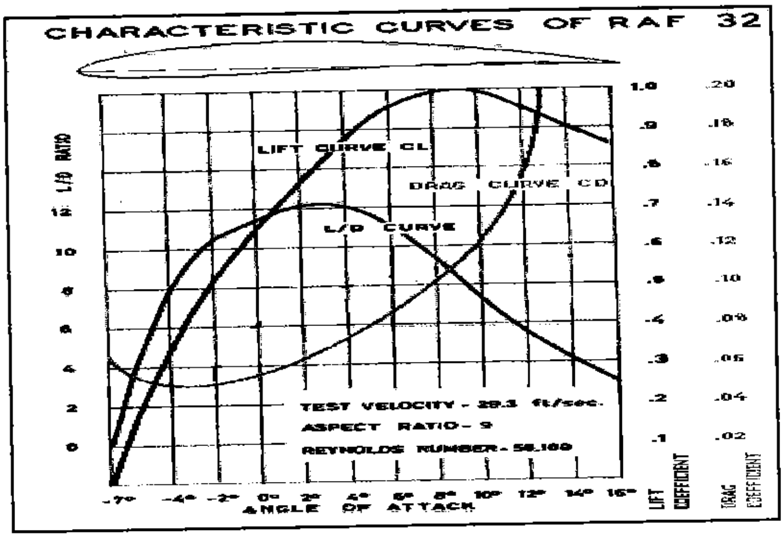

Figure 1.9: The graph on left shows Lift and Drag Coefficients along with Lift/Drag ratio of a whole wing with aspect ratio of 9 and airfoil RAF 32 at Re 56,100 (Model Aircraft, 2010)

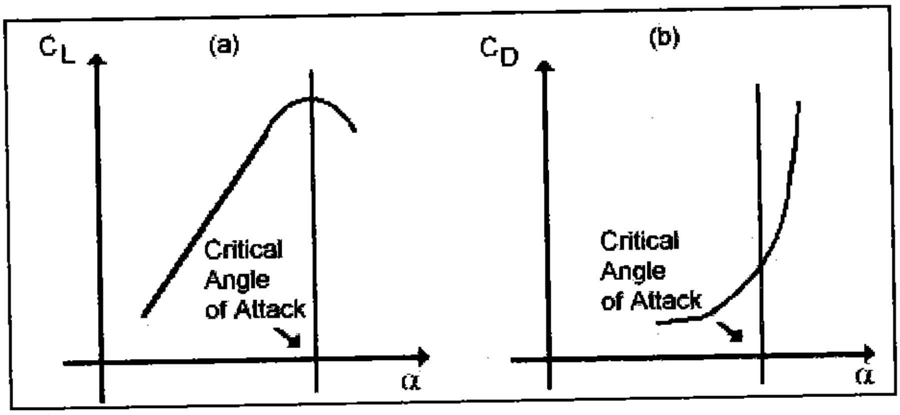

Figure 1.10(a): The relationship between the Angle of Attack and Lift Coefficient, (b) The relationship between the Angle of Attack and Drag Coefficient (Pilot's Web, 2005).

The result of this study is expected exhibit similar trend with previous study which if we look at the above graph we can see that the increment of C_l and C_d as the angle of attack α is increases. At a certain point, the lift begins to drop while the drag increases sharply. This point is defined as the Critical Angle of Attack. If the angle of attack increases passed the Critical Angle of Attack, at one point all lift will be lost while the drag continues to increase.

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

This chapter will explain the fundamental study of viscous-inviscid interaction, potential flow concept and boundary layer concept. It also covered historical review of viscous – inviscid interaction and some example airfoil in their practical application. The governing equation had been used in this study will be presented in this chapter and it became as basic of derivation in this study. The derivation here is useful in explanation the methodology which will be given in the next following chapter.

2.1 Viscous-Inviscid Interaction

The viscous-inviscid interaction flows, such as boundary layer separation, interaction between boundary layer and shock wave, etc., have been investigated for many engineering problems during the last forty years (Veldman, 2000). This history started in 1904 when Ludwig Prandtl presented the 'boundary layer' at the Third International Mathematical Congress in Heidelberg. From that moment on the flow field around a body was derided into parts: a thin shear layer where viscosity plays a role and the remaining outer part where flow can be considered inviscid (Veldman & Coenen, 2000).

Several methods have been developed from there. The oldest technique is known as the direct method, which solves the boundary layer with a prescribed velocity (pressure) distribution Ue(x). In turn, the boundary layer expresses its presence in terms of a virtually thickened profile, called displacement thickness δ^* (Veldman & Lai, 1999).One of the success stories in theoretical fluid dynamics in the second half of the twentieth century was the development of the theory of interaction between the boundary layer and external inviscid flow, usually referred to as 'viscous–inviscid interaction'. It was first published by Stewartson & Williams (1969) and Neiland (1969) in relation to boundary-layer separation in supersonic flow, and by Stewartson (1969) and Messiter (1970), who studied incompressible flow near the trailing edge of a flat plate. Then, the theory of viscous–inviscid interaction, also known as the 'triple-deck theory', was applied to a variety of fluid flows, and by the mid-1970s, it had become clear that the viscous–inviscid interaction played a key role in many fluid dynamic phenomena (Kravtsova, Zametaev & Ruban, 2005).

In addition to different forms of separation, it was shown to govern upstream influence in the supersonic and hypersonic boundary layers, development of different modes of instabilities, bifurcation of the solution and possible hysteresis in separated flows. The incipience of separation in the boundary layer near a corner point of a rigid body contour was one of the first problems to which the theory was applied. The set of asymptotic equations governing the flow behavior near the corner was formulated by Stewartson (1970), who also was able to construct analytically the solution of the linearized version of the problem. It describes a preseparated flow regime when the angle of rotation of the tangent to the body contour at the corner point θ is sufficiently small to cause only weak perturbations in the boundary layer (Kravtsova, Zametaev & Ruban, 2005).

A numerical solution to the nonlinear problem was first produced by Ruban (1976), who observed the formation of a local separation region in the boundary layer on convex and concave corners. Similar calculations were also performed by Smith and Merkin (1982). The problem was later re-examined by Korolev (1992), who discovered that the solution is non-unique for the flow over convex corners. It appeared that with the same body geometry, two different solutions are possible; one with a 'short' separation region forming behind the corner point, and the other with a 'long' separation region (Kravtsova, Zametaev & Ruban, 2005).

In summarize, there are mainly three theoretical approaches. One is the triple-deck theory (TDT), governing the small separation flow problem. TDT is an asymptotic theory of high Reynolds number flows and can give a fine structure of the flow in the neighborhood of a separation point. It has been applied to many flows of practical interest for example attached trailing-edge flows, leading edge separation flows, etc. Another approach is the interacting-boundary-layer theory (IBLT) which couples inviscid and the boundary layer equation and takes the viscous displacement effect into account. IBLT exploits the classical the boundary layer equation in handling many viscous flow problems involving separation. The third approach is the simplified-Navier-Stokes (SNS) equation theory. SNS equations contain an exact representation of the inviscid flow-field as well as the viscous effect at least in the range of IBLT and TDT approximations (Gao Zhi, 1990).

In this study, two approaches will be used to capture fluid phenomena of viscid-inviscid interaction. One, Potential Flow solution will be solved inviscid flow while the viscous part of the flow will be solved by using Boundary layer theory. In order to understand aerodynamics characteristics of the aerofoil, some assumption had been made so that the analysis can be avoided from complication of mathematical problem. The assumptions mentioned are the fluid is incompressible, Newtonian, 2-Dimension analysis, and high Reynolds number high Reynolds number flow.

2.2 Fundamental Of Potential Flow Theory For Incompressible: Inviscid And Irrotational Fluid Flow

One, this study deal with inviscid flows, even all real fluids are viscous. However, in high-Reynolds number flows, viscous effects are confined to within thin boundary layers near solid surfaces, to wakes and to mixing regions. When there is an adverse pressure gradient, separation may occur making the wake wide. Outside boundary layers, wakes and mixing regions, the flow can reasonably be considered to be inviscid. The equations of motion have a particularly simple form in an incompressible, inviscid flow. They can be solved by straightforward analytically or use of computational techniques (Nickels, 2009).

For inviscid fluid, $\mu = 0$, incompressible fluid flow, 2-Dimension and steady, the governing equation are (Darus, 1994):

i. Euler equation

$$\frac{\partial \boldsymbol{V}}{\partial t} + V.\nabla V = \mathbf{f} - \frac{\nabla P}{\rho}$$

Or in steady flow, the equation becomes:

$$V.\nabla V = \mathbf{f} - \frac{\nabla P}{\rho} \qquad (2.1)$$

ii. Continuity equation, is:

$$\frac{D\rho}{Dt} + \rho\nabla.\boldsymbol{V} = 0$$

Assumed flow was steady, and 2-Dimension, the equation becomes

$$\nabla . \boldsymbol{V} = 0$$

Or in Cartesian coordinate, equation can be written as,

$$\frac{\partial \mathrm{u}}{\partial \mathrm{x}} + \frac{\partial v}{\partial y} = 0 \qquad (2.2)$$

iii. Irrotational Flow

In Cartesian coordinates the vorticity components are:

$$\xi_x = 2\omega_x = \left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)$$

$$\xi_y = 2\omega_y = \left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)$$

$$\xi_x = 2\omega_x = \left(\frac{\partial w}{\partial x} - \frac{\partial v}{\partial y}\right)$$

Irrotational means that there is no vorticity. Here a two-dimensional flow in the x-y plane assumed and therefore the velocity points in the z-direction. The left-hand side in this equation is the rate at which vorticity is accumulated, which is equal to the rate it being generated (near the solid boundaries of solid surfaces). It is clear that for high Reynolds number flows, vorticity generation is small and can be neglected outside the boundary layer (Katz & Plotkin, 2001). For better understanding, Kelvin's theorem can be referred to Katz & Plotkin's book(Low-

Speed Aerodynamics, 2001), which provides good information for rate of change of circulation.

Thus for an Irrotational fluid reduces to

$$\frac{\partial u}{\partial y} - \frac{\partial v}{\partial x} = 0 \qquad (2.3)$$

In Cartesian coordinates the velocity components are given by:

$$u = \frac{\partial \Phi}{\partial x}, \qquad v = \frac{\partial \Phi}{\partial y}, \qquad w = \frac{\partial \Phi}{\partial z} \qquad (2.4)$$

The substitution of Eq. (2.4) into the continuity Eq. (2.2) leads to the following differential equation for the velocity potential

$$\frac{\partial^2 \Phi}{\partial x^2} + \frac{\partial^2 \Phi}{\partial y^2} = 0 \qquad (2.5)$$

Or can be written as:

$$\nabla^2 . \Phi = 0$$

Which is Laplace's equation (named after the French mathematician Pierre S. De Laplace (1749-1827)). It is statement of the incompressible continuity equation for an irrotational fluid. Note that Laplace's equation is a linear differential equation. For an irrotational, inviscid, and incompressible flow is not appears that the velocity field can be obtained from a solution of Laplace's equation for the velocity potential. Once the velocity field is obtained it is necessary to also obtain the pressure distribution on the body surface to allow for a calculation of the aerodynamic forces moments (Katz & Plotkin, 2001).

To obtain pressure distribution along the aerofoil surface, Bernoulli equation has been applied. Bernoulli equation that derived from Euler equation Eq. (2.1) will be more useful by comparing the quantities at two points in the fluid; the first is an arbitrary point and the second is a reference point at infinity. The equation becomes

$$\left[\frac{V^2}{2}+\frac{P}{\rho}+gz\right]_2=\left[\frac{V^2}{2}+\frac{P}{\rho}+gz\right]_1=\text{constant}$$

Or can be written

$$P+\frac{\rho V^2}{2}=P_e+\frac{\rho {V^2}_e}{2} \qquad (2.6)$$

$$C_p=\frac{P_e-P}{\frac{1}{2}\rho U^2}=1-\left(\frac{U_e}{U}\right)^2 \qquad (2.6(a))$$

Eq. 2.6 called Bernoulli equation. Most important to remind in case that constant B is not only valid on certain streamline in fluid field, though B is valid for all streamline in that field (Katz & Plotkin, 2001). From Eq. (2.6), most important parameter in this study, pressure coefficient C_p Eq. (2.6(a)) has been obtained (Schlichting, 1995).

2.2.1 The General Solution, based on Green's Identity

Referred to Katz & Plotkin (2001), the velocity component normal to the body's surface or submerged body in the fluid, must be zero, and in a body-fixed coordinate system:

$$\nabla\Phi.n=0 \qquad (2.7)$$

Here $\mathbf{n}$ is a vector normal to the body's surface, and $\nabla\Phi$ is measured in a frame of reference attached to the body. Also, the disturbance created by the motion should decay far $r \to \infty$ (Katz & Plotkin, 2001):

$$\lim_{r\to\infty}(\nabla\Phi - v) = 0 \qquad (2.8)$$

Where r=(x,y,z) and v is the relative velocity between the undisturbed fluid in V and the body (or the velocity at infinity seen by an observer moving with the body).

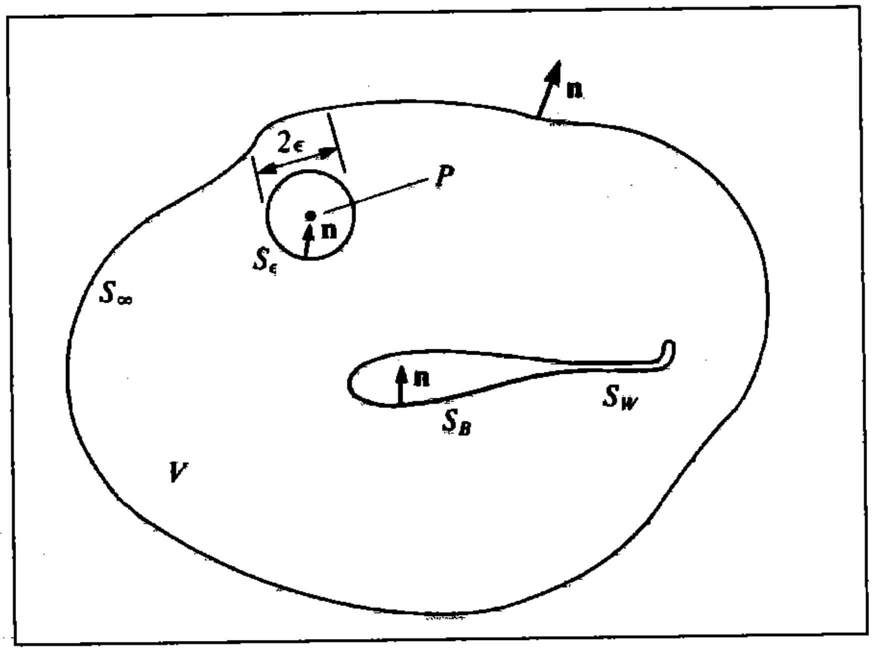

Figure 2.1: Nomenclature used to define the potential flow problem (Katz & Plotkin, 2001)

Laplace's equation for the velocity potential must be solved for an arbitrary body with boundary S_B enclosed in a volume V, with the outer boundary S_∞, as shown in Figure 2.1. The boundary conditions in Eqs. (2.7) and (2.8) apply to S_B and S_∞ respectively. The normal $\mathbf{n}$ is defined such that it always points outside the region of interest V. The vector which is derived from divergence theorem replaced by vector $\Phi_1 \nabla \Phi_2 - \Phi_2 \nabla \Phi_1$, where Φ_1 and Φ_2 are two scalar function of position (Katz & Plotkin, 2001). The result is;

$$\int_S (\Phi_1 \nabla \Phi_2 - \Phi_2 \nabla \Phi_1) \cdot \boldsymbol{n} ds = \int_V (\Phi_1 \nabla^2 \phi_2 - \Phi_2 \nabla^2 \Phi_1) dV \quad (2.9)$$

This equation is one of Green identities. Here the surface integral is taken over all boundaries S, including a wake model S_w which might model a surface across which a discontinuity in the potential or the velocity may occur (Kreyszig, 1999).

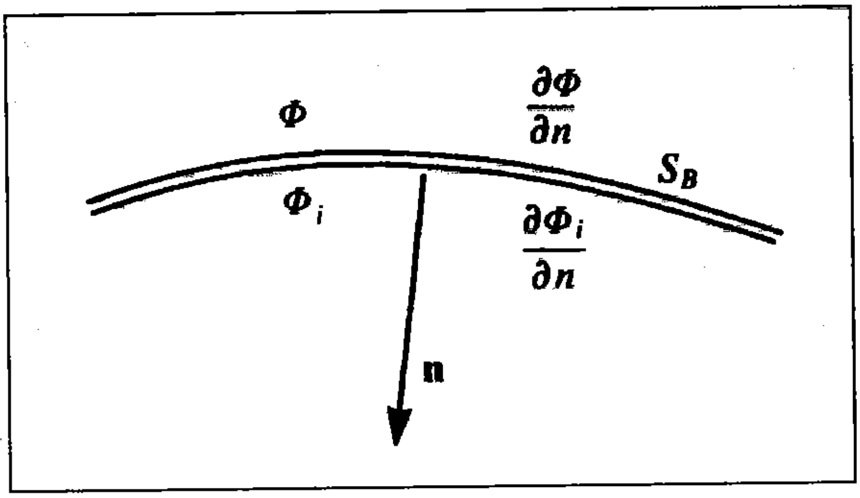

Figure 2.2: The velocity potential near a solid boundary S_B (Katz & Plotkin, 2001)

Therefore, the problem is reduced to determining the value of these quantities on the boundaries. For, example, consider a segment of the boundary S_B as shown in Fig. (2.2); then the difference between the external and internal potentials can be defined as (Katz & Plotkin, 2001):

$$-\mu = \Phi - \Phi_i \quad (2.10)$$