THE DEVELOPMENT OF EULER SOLVER BASED CODE FOR EXTERNAL AERODYNAMICS FLOW

FATIMAH BINTI MOHAMED YUSOP

UNIVERSITI TUN HUSSEIN ONN MALAYSIA

THE DEVELOPMENT OF EULER SOLVER BASED CODE FOR EXTERNAL AERODYNAMICS FLOWS

FATIMAH BINTI MOHAMED YUSOP

A thesis submitted in
fulfilment of the requirement for the award of the
Doctor of Philosophy

Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia

AUGUST 2016

This thesis is dedicated to

My parents, Mohamed Yusop Mat Dehari and Badariah Ismail,

My husband, Mohd Yurzie Nazierul Yahaya

and to

My lovely son, Muhammad Firas Hadif

ACKNOWLEDGEMENT

Firstly, I would like to express my sincere gratitude to my supervisor, Associate Prof. Dr. Zamri Bin Omar and my co-supervisor, Dr. Ir Bambang Basuno for the continuous support on my research, for their patience, motivation, and immense knowledge. Their guidance helped me throughout the research and writing of this thesis. I could not have imagined having a better advisors and mentors for my Ph.D study.

I am also thankful to all the staff members of the Centre Graduates Studies and Faculty of Mechanical and Manufacturing Engineering for their help and assistance.

I am also thankful to my husband, Mohd Yurzie Nazierul Yahaya for being understanding and for his encouragement. To my son, Muhammad Firas Hadif for bringing me cheerfulness and happiness.

Last but not the least, I would like to thank my parents Mohamed Yusop Mat Dehari and Badariah Ismail for bringing up my spirit and motivation to complete this thesis. Finally, to all my brothers and sister for supporting me spiritually throughout writing this thesis and my life in general.

ABSTRACT

Tinoco (2009) estimated that direct solution to the Navier-Stokes equation in line with the progress in the development of computing power and the algorithm in solving the non-linear partial differential equation can be made in the next 65 years from now [1]. Realizing the highly demanding computer power, the way to solve the flow problems is by adopting some sort of simplification of Navier-Stokes equation according to physical flow phenomena consideration. Such approach gives, for instance, to the case of flow past through a streamline body at high and at low Reynolds number to moderate angle of attack, the Navier-Stokes equation can be simplified by ignoring viscous effects resulting a new governing equation of fluid motion named Euler equation. However, this equation is still belongs to the class of non-linear partial differential equation, while result in numerical approach is required. The main features of compressible flow are the presence of discontinuity flow phenomena due to shock wave and a contact surface. Such flow phenomenon is always found if the flow in transonic or supersonic flow regimes. As a result, all the aircrafts designed to fly at transonic or supersonic speed will face such flow phenomenon. It is therefore, all numerical schemes designed to be a tool for aerodynamics analysis or aerodynamics design tool must be able to predict the presence of such discontinuity flow phenomena accurately. The present work focused on the development of computational fluid dynamics (CFD) aerodynamics analysis tool by solving the compressible Euler equation through a finite difference method (FDM) as well as a finite volume method (FVM). The FDM used the method developed based on Davis-Yee TVD scheme. While that, the FVM was developed by using Roe scheme. In view of TVD scheme, there were various methods in involving a limiter functions. The use of various limiter functions as part of the development of CFD software is also carried out in the present work. To validate the developed software, these two types of CFD code are applied for solving the high Mach number flow past through airfoil. The airfoils are (1) the conventional airfoil NACA0012 and

(2) the supercritical airfoil NASA SC (2)-0714. Then, the comparison is made by comparing their result with the ANYSYS-FLUENT software as well as with the experimental result as stated in the literature. Through results comparison for these two different airfoils operated at various angles of attack, it had been found that the developed CFD code based on Davis-Yee TVD scheme are able to produce the result close to the experimental result compared with the result produced by ANSYS-FLUENT software or the code developed based on Roe finite volume scheme.

ABSTRAK

Tinoco (2009) menganggarkan penyelesaian terus kepada persamaan Navier-Stokes adalah selaras dengan kemajuan komputer dan algoritma dalam menyelesaikan persamaan pembezaan separa yang boleh dicapai 65 tahun dari sekarang [1]. Menyedari keperluan yang sangat tinggi terhadap kuasa komputer, cara terbaik untuk menyelesaikan masalah aliran dengan meringkasakan persamaan Navier-Stokes dengan mengambilkira fizikal aliran. Dengan pendekatan ini, sebagai contoh untuk kes aliran melalui satu badan yang nipis pada nombor Reynold yang tinggi atau rendah dan juga sudut serangan yang sederhana, persamaan Navier-Stokes boleh dipermudahkan dengan mengabaikan kesan kelikatan yang mana persamaan baru digunakan ialah persamaan Euler. Walau bagaimanapun, persamaan ini masih tergolong dalam kelas persamaan pembezaan separa yang bukan linear, manakala pendekatan berangka diperlukan. Ciri utama aliran termampat adalah kehadiran fenomena aliran ketakselanjaran disebabkan gelombang kejutan dan permukaan sentuhan. Fenomena aliran seperti ini sering berlaku dalam aliran jika aliran dalam keadaan transonik dan supersonik. Oleh itu, semua pesawat yang direka untuk terbang pada kelajuan supersonik atau transonik akan menghadapi fenomena aliran tersebut. Maka, semua skim berangka yang direka sebagai alat untuk menganalisis aerodinamik atau alat merekabentuk aerodinamik perlu meramal fenomena kehadiran ketakselanjaran pada aliran secara tepat. Kajian ini memberi tumpuan kepada pembangunan pengkomputeran dinamaik bendalir (CFD) bagi alat untuk analisis aerodinamik dengan menyelesaikan persamaan Euler boleh mampat melalui kaedah perbezaan terhingga (FDM) dan juga sebagai kaedah isipadu terhingga (FVM). FDM menggunakan kaedah yang dibangunkan berdasarkan skim Davis-Yee TVD. Manakala FVM menggunakan skim Roe. Dalam konteks TVD, terdapat pelbagai kaedah yang melibatkan fungsi penghad. Dalam kajian ini turut menggunaan pelbagai jenis fungsi penghad sebagai sebahagian daripada pembangunan perisian CFD. Untuk mengesahkan perisian yang dibangunkan, kedua-

dua jenis kod CFD digunakan untuk menyelesaikan aliran pada nombor Mach yang tinggi melalui aerofoil. Aerofoil yang digunakan adalah (1) konvensional aerofoil NACA 0012 dan (2) superkritikal aerofoil NASA SC (2)-0714. Seterusnya, perbandingan dibuat dengan membandingkan hasil yang deperolehi dengan hasil daripada perisian ANSYS-FLUENT dan juga dengan hasil eksperimen yang sediada. Melalui keputusan perbandingan dalam pelbagai sudut serangan oleh kedua-dua aerofoil, didapati bahawa kod CFD yang dibangunkan berdasarkan skim Davis-Yee TVD mampu menghasilkan keputusan yang hampir dengan hasil eksperimen berbanding dengan keputusan yang dihasilkan oleh perisian ANSYS-FLUENT atau kod yang dibangunkan berdasarkan kaedah isipadu terhingga mengunakan skim Roe.

CONTENT

LIST OF FIGURE

LIST OF TABLE

LIST OF SYMBOL AND ABBREVIATIONS

a	-	Speed of sound
c	-	Chord length
C_p	-	pressure coefficient
e	-	Internal energy
e_t	-	Total energy
E	-	Flux vector in x-direction
$\overline{E}$	-	Flux vector of E in curvilinear coordinates
f	-	Vector of body force per unit mass
f	-	Body force per unit mass
F	-	Flux vector in y-direction
$\overline{F}$	-	Flux vector of F in curvilinear coordinates
F_c	-	Sum of convective flux
t	-	Time
M	-	Mach number
V	-	Vector velocity
u	-	Velocity in x-direction
v	-	Velocity in y-direction
w	-	Velocity in z-direction
p	-	Static pressure
Q	-	Vector of conservative variable
$\overline{Q}$	-	Vector of conservative variable in curvilinear coordinates
X	-	Eigenvector
α	-	Angle of attack
α	-	Eigenvalue
ρ	-	Density
Φ	-	Flux limiter

Ω	-	Cell volume
τ	-	Shear stress
∇	-	Vector operator
ξ , η, ζ	-	curvilinear coordinate system
CASA	-	Civil Aviation Safety Authority
J	-	Jacobian
FDM	-	Finite difference method
FVM	-	Finite volume method
FEM	-	Finite element method
PDE	-	Partial differential equation
ODE	-	ordinary differential equation
TVD	-	Total Variation Diminishing
TFI	-	Transfinite interpolation
NASA	-	National Aeronautics and Space Administration
NACA	-	National Advisory Committee for Aeronautics

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Background of study

The development of modern computational fluid dynamics (CFD) began with the introduction of the digital computer in the early 1950s. Since then, the aircraft manufactures such as Boeing (US), Airbus (Europe), Aerospatiale (France), British Aero Space, CASA and many others have started using CFD as a tool for design and analysis in development of new aircraft products. Basically, CFD has been used for more than three decades in the aircraft design activities. Boeing experiences indicated that since 1980, the number of hours in the use of wind tunnel involved in the design phase of new aircraft had been reduced up to 50 % [2].

Currently, there are various CFD codes that have been developed. However, it is necessary to be noted that the earliest CFD code was the code developed based on the use of potential flow theory. This code was firstly developed by Hess and Smith in 1962 through the solving of Laplace’s equation according to a linear potential flow approach [3]. This method was known as a Panel method. In 1970, Murman and Cole, introduced a CFD code which was developed by solving the transonic small disturbance equation [4]. This code represents the enhancement of the capability in solving the aerodynamics problem. If the panel method is limited to case subsonic flow, the Murman and Cole approach will be able to solve the flow problem at transonic speed. However, there’s a limitation to the Murman and Cole approach where it is only suitable to the case of flow past through a streamline body with a small thickness. In other words, the presence of body immersed in the flow field only created a small disturbance to the flow field. To increase the capability in solving the flow problem, in 1973, Jameson had successfully developed and established the CFD code called FLO22 and FLO27, which solve the flow problem based on a transonic potential full equation. This code had been used by Boeing with the given code name as Boeing code A488 [5]. Another successful code based on potential flow is PANAIR and TRANAIR code. These codes are well established and have been used until today.

Another high level CFD code is the code that was based on Euler equation. The code was developed in early 1980s. In 1981, Jameson, Schmidt and Turkel developed FLO57 based on three-dimensional (3D) Euler equation [6, 7] This code has been considered as a great code and it has been used extensively by Boeing, with a code name A588. Aerodynamics software which was developed by solving Euler equation but dedicated for two-dimensional (2D) airfoil analysis and design was also known as an ISES code. This code was developed by Drela and Giles, it has been used extensively for the design and analysis of advanced airfoil. Besides ISES, there is another 2D Euler code called MSES code [8]. The success of solving flow problem based on Euler equation had opened up solutions to the flow problem based on the Time Averaged Navier-stokes. From 1993 until today, there have been various attempts to solve the flow problem based on this approach.

Considering the possibility of solving flow problems, various research agencies such NASA, ONERA, NLR, etc, have shown an interests in solving particular engineering problems. Besides that, there are a lot of universities involved in the development of a new numerical scheme for use in CFD as well as in the form of CFD code. Currently, the ability of CFD codes has been increasing rapidly. They can solve the flow problem with more complexities and varieties in their flow phenomena. CFD codes have reached their mature capability, which allows use CFD for flow analysis without needing a supporting experimental result for comparison. A commercial CFD code such as ANSYS-FLUENT and CFDRF may represent an example of a well-established CFD code. Unfortunately, the most well-established aircraft companies such as Boeing, Airbus, and MacDonald Douglas prefer to develop in-house CFD code by their own methods instead of using readymade commercial software. They consider these commercial CFD code are not yet suitable for a design tool in the Aircraft industry activities [9, 10].

Basically the computational approach used solved a mathematical equation by describing the governing equation of fluid motion. The governing equation of fluid motion is normally in the form of partial differential equation (PDE) which consist of terms involved with partial derivatives with respect to space and time.

There are three approaches that can be used for solving such equation. They are known as (i) finite difference method (FDM), (ii) finite element method (FEM) and (iii) the extension of FDM and FEM which is known as the finite volume method (FVM). FDM approach is the oldest method used and uses a Taylor expansion in converting the PDE into a discrete form. Meanwhile, FVM uses integral form of PDE which requires the volume or cell to solve the equation. FEM also uses same approach with FVM but usually used for solving a structure problem.

When all significant aspects have been taken into account in the fluid behaviour, fluid flow can be governed by the Navier-Stokes equation which applies mass, momentum and energy conservation. This equation has the highest degree in the hierarchy of fluid flow problem. Generally, it is very difficult to solve it. As a result, some assumptions were made to simplify the equation by considering the behaviour effects due to a viscous effect, a compressibility effect, vorticity and others.

Compressible flow is a group of fluid dynamics that deals with flows having significant changes in density properties. In general, gases are highly compressible and liquids have very low compressibility. Physical observation had found that gas flows under incompressible flow condition if the gas is flowing at the Mach number less than 0.3. Otherwise, the gas must be considered as a compressible flow. Basically compressibility of a fluid is measured by the change in density that will be produced in the fluid in the presence of a specific change of pressure and temperature. The change of density will be less than 5% if the fluid moves with speed below Mach number, $M = 0.3$.

Another branch of fluid dynamics is called an inviscid flow. This flow indicates that the viscous effects can be neglected. The assumption of inviscid flow is generally valid where viscous forces are small compared to the inertial forces. As a result the high Reynolds numbers flow indicates the type of flow in which the inertial forces are more significant than the viscous forces. The combination of flow under condition inviscid and compressible can be represented by the governing equation of fluid motion in the form of compressible Euler equation. This equation can be classified as a hyperbolic PDE if the flow is having a supersonic speed and it behaves as an elliptic-hyperbolic PDE if the flow is under transonic speed. However in view of CFD approach, the hyperbolic have been is extensively used to solve problems in CFD.

There are many numerical schemes that can be adopted for solving the PDE from the splitting method until high-order scheme. The high-order scheme may represent the most common scheme and currently practiced to solve the hyperbolic partial differential equation. Before more elaboration is made on discretisation schemes, it is important to know the basic idea of discretisation. There are two types of discretisation that is considered in a governing equation. They are temporal discretisation and spatial discretisation.

Temporal discretisation method can be divided into two approaches; they are explicit and implicit approach. Explicit method allows a direct computation of the dependent variable that can be made in terms of known quantities. In other words, the explicit method uses known data from the current time step in order to find the unknown data to the following time step. In 1981, Jameson et al introduced a very famous and convenient time step by using multi-stage time stepping [6, 7]. This scheme was called Runge-Kutta scheme. In contrast, the implicit approach is very hard and complex to implement, but it leads to a faster convergence. The present work focuses on explicit method with multi-stage time step.

Another part of discretisation is spatial discretisation. This discretisation approach is used to approximate the convective and viscous term. There are varieties of spatial discretisation scheme in literatures and researchers are continuously trying to find a more accurate, robust and cheap scheme. In general, there are two types of schemes, a central scheme and upwind scheme. The central scheme is less accurate in handling the discontinuities but easier to code. An example of the central scheme is the Lax scheme that was introduced in 1954 and being represented as the first-order explicit scheme. The Lax-Wendroff scheme was introduced in 1960 representing a second-order explicit scheme. In 1969, MacCormak introduced two-stages of central scheme. While that, the upwind scheme developed by considering the physical properties of Euler equation can be categorised into four groups, they are a Flux-Vector (FVS) scheme, a Finite Difference Splitting (FDS) scheme, a Total Variation Diminishing (TVD) scheme and a Fluctuation-Splitting scheme. In this respect, the present work will emphasize to two schemes, the FDS scheme and the TVD scheme.

FDS scheme can be stated as a second level of upwind scheme after FVS scheme. This scheme which is often called as approximate Riemann solver, in solving the flow problem is not only considering the direction of wave propagation but also the wave itself (expansion wave or compression wave or contact surface discontinuity). The idea on how to solve the problem is by considering the wave behaviour which was firstly introduced by Godunov in 1959 [11]. Through such approach, the computational effort required for solving the Riemann problem can be reduced. The approximate Riemann solver based Osher et al and Roe may represent the most popular scheme in CFD community [12]. However, the approximate Riemann solver provided by Roe offers a high accuracy to their solution compared to other approximate Riemann solver.

The TVD scheme was firstly introduced by Harten in 1983 [13, 14]. This scheme is based on the concept of avoiding a creation of new extrema points in the solution. This scheme with property of monotonic preserving and combinining a flux limiter (limiter) made the TVD condition fulfilled and the scheme has a second order accurate. Among the common TVD limiters are Davis-Yee, Harten-Yee, Roe-Sweaby, etc.

The present work deals with the flow problem related to streamline body of a relatively low angle of attack in which the viscous effect can be neglected. In solving this type of flow, the two computer codes are developed and designed to be able to solve a compressible flow past through airfoil in transonic speed. The first computer code is the code developed based on FDM according to Davis-Yee TVD scheme, while the second computer code was developed by the use of FVM according to Roe scheme. Two types of airfoil were tested for validation purpose. These two airfoils are (1) the conventional airfoil NACA0012 and (2) the supercritical airfoil NASA SC (2)-0714. The code validation was carried out by comparing their results with the available experimental result obtained from literature and the result obtained from running commercial CFD code ANSYS-FLUENT software.

1.2 Problem statement

Currently CFD had been considered as an important tool for solving engineering problems. The application of CFD had been used intensively in the aircraft industries in design a new aircraft or in the effort of improvement on the existing aircraft. In terms of CFD computer code, the CFD code differs with other may due to the difference in the numerical scheme have been used in developing that code. In the case of inviscid compressible flow, the CFD code may be developed by using MacCormack Scheme, Steger-Warming scheme or the method derived from TVD schemes. Basically, each of numerical schemes has its own capabilities in solving the flow problem. In the case of transonic flow past through an airfoil, the important features of this flow is the presence of shock wave in the flow field. This shock wave generates a change of flow properties sharply to form some kind discontinuity flow phenomena in the flow field. Any numerical scheme which capable to predict such discontinuity and the location where discontinuity occurred accurately will produce a CFD code which give capabilities to provide their results close to the aerodynamics experimental result. As the result, the corresponding CFD code would be truly useful tool for the aerodynamic analysis and design activities normally found in the aircraft industries. There are various numerical schemes had been developed for solving inviscid compressible flow problem, but to achieve successful CFD code dedicated to solving a 2D compressible flow accurately, it requires developing CFD code step by step. It needs to start with the development of CFD code applied to the case of 1D shock tube problem and 2D symmetric model as a stage for evaluating various numerical schemes before selecting one of them as the numerical scheme for solving 2D compressible flow. Therefore, through this study able to develop a robust CFD code that able to predict shock location close to experimental.

1.3 Objective

The aim of the research work is to develop a CFD code for aerodynamic analysis. In order to achieve this aim, the following specific objectives have to be accomplished:

i. To develop a CFD code which is applicable for solving 2D inviscid compressible flows.
ii. To compare developed CFD code and commercial CFD code in terms of value of flow variables such as pressure and Mach number distribution in the flow domain.
iii. To validate the aerodynamic properties through developed CFD code with the available experimental results.

1.4 Scope of study

In this research, there are a few scopes of this study addressed as below:

i. Develop CFD code by using FORTRAN programming language and ANSYS-FLUENT for commercial code.
ii. The study only focuses on 2D compressible flows.
iii. Limit to single-block topology.
iv. Numerical grid generation by using of algebraic grid generator or elliptic grid generator.
v. Solution on the governing equation of fluid motion based on FDM and FVM.
vi. Study effect of the flux limiter.
vii. The spatial discretisation in relationship with upwind scheme.
viii. The temporal discretisation in relationship with a multi-stage time stepping.

1.5 Importance of the research

Understanding aerodynamic characteristics are required in every design of a new aircraft activity. To obtain the aircraft aerodynamics characteristics, it can be done either through experimental aerodynamics by putting the aircraft model inside wind tunnel or by solving the governing equation of fluid motion. The first approach may be time consuming and costly since there are a lot of parameter geometry which give influence to the aerodynamics characteristics need to be assessed. Hence, determination of the aerodynamics characteristics through solving the governing equation of fluid motion may represent the most feasible manner in the early stage of aircraft design. In addition to this, the aircraft designers are always design their aircraft shapes belonging to the class of streamline body and operated at a relatively a low angle of attack. As a result, the corresponding flow problem needs to be treated and adequately represented as solving compressible inviscid flow problems.

The research work focused on the development of CFD code for providing detail on flow phenomena of compressible inviscid problem as in the case of quasi 1D compressible flow past through nozzle and in the case of 2D flow problems such as flow past through airfoil. The second case is useful for the aircraft designer in evaluating the aerodynamics characteristics of an airfoil or in selecting the types of airfoil which will be used as its wing cross section.

1.6 Organization of the thesis

This thesis is organized into five chapters. The first chapter describes the introductory of the study, the explanation of research objectives and the scope of study will be conducted.

Chapter 2 gives an explanation on the governing equation of fluid motions of the problem in hand including the existence of various level of governing equation of fluid motion. Overviews over available CFD code in the market as well as the commonly used codes in the aircraft manufacturers are presented. This chapter also provideds the literature on the numerical method that has been used for CFD as well as a literature study to the selected scheme used in the present research work.

Chapter 3 presents the numerical scheme used in the development of CFD code in view to the manner of the governing equation of fluid motion which are solved and the numerical grid generation required. This chapter included the pseuocode needed in developing the CFD code in the present work.

Chapter 4 presents the discussion results of the implementation CFD code in solving one- and two-dimensional inviscid compressible flow problems for various flow conditions. The comparison result between the developed CFD codes are carried out by comparing the experimental result available from literature and running the available CFD Code ANSYS-FLUENT.

Chapter 5 presents the conclusion and recommendations for future works on this research area.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

The governing equation of fluid motion is already well-established more than a century ago. This equation called the Navier-Stokes equation represents the governing equation of fluid motion which is capable of describing whatever flow phenomena that may exist in the flow field. Unfortunately, in the aeronautics applications there is no analytical solution for this kind of equation. Therefore, in order to solve the Navier-Stokes equation, it has to rely on the numerical approach. This approach opens up a possibility in solving the Navier-Stokes equation directly through the method known as Direct Numerical Simulation (DNS) method.

Unfortunately, solving the Navier-Stokes equation directly is very demanding on computer performance and highly time consuming. The flow domain of the flow problem needs to be discretised in such way that distance between two neighbouring control points must be less than the Kolmogorov scale, λ. This scale describes the distance of particle travel before it loses its identity. It has the order of magnitude 1×10^{-6} m. As a result to evaluate the flow behaviour which occupied the flow domain 1.0 m^3, it will require at least 1×10^{18} control points. The flow variables at each control point, which involves variables in terms of velocity components in three direction, pressure p, temperature T and the density ρ has to be updated through a time marching process to achieve a steady state solution. This approach make the process has to be carried out by time stepping method with time increment Δt which is very small to fullfil numerical stability requirement. Such requirements make the DNS through solving directly to the Navier-Stokes equation cannot be applied for solving the fluid dynamics problem of the interest of aeronautics applications.

DNS method can be applied in the aeronautics applications, especially in solving the aerodynamics problems become viable may after 2080 when more powerful computer resources and better algorithm in solving the partial differential equations than the present time are available [1]. It is true that the direct solution to the Navier-Stokes equation will give a very accurate and detailed solution to the physical flow phenomena that may appear in the flow field. However in the interest of engineering application, such detailed flow phenomena may not be required. The engineer may require overall solution such as the pressure distribution on the body surface in which the detail of particle movements in the flow field are not necessary to be known.

In this aspect, it may reduce the complexity of the Navier-Stokes equation to the level in which the present computer performance and numerical algorithm are able to solve the flow problems in reasonable CPU time and accuracy. There are various levels of governing equation of fluid motion that can be implemented and adopted in solving a particular flow problem. The following sub-chapter discusses the various forms of governing equation that can be introduced as part of reducing the complexity of the Navier-Stokes equation. Reducing the Navier-Stokes equation in the form of its lower level means to solve the problem in partial differential equation (PDE) in an ordinary differential equation (ODE) or others. It also has to deal with the problem of PDE as a non-linear equation but less demanding in spatial as well as temporal discretisation.

There are various numerical approaches that have been introduced on how to solve the governing equation of fluid motion in the form of derivative of the Navier-Stokes equation approach which is known as a Computational Fluid Dynamic (CFD). Hence the chronology of CFD code development since 1980s and also the application of CFD code especially in aircraft industries are presented also in the next sub-chapter. Finally, this chapter reviews experimental result for selected airfoil as a data comparison to validate the CFD code.

2.2 Hierarchy of CFD solver

Before further discussing CFD in detail, it is necessary to look at the map on how the CFD is developed. CFD is the way to solve the flow problem numerically. So, it starts from the governing equation of fluid motion. Figure 2.1 shows the road map in solving the flow problem by using CFD. The governing equation can be basically presented in either differential form or integral form. If the need chooses the governing equation in differential form, it may need to adopt a finite difference method (FDM) to solve that governing equation. Since the Euler equation represents the non-linear partial differential equation, a certain numerical technique needs to be introduced. With this, it will adopt particular manner of FDM for solving the Euler equation to be implemented and may also choose a MacCormack Scheme, Flux splitting Steger Warming or TVD scheme. The TVD scheme can be used according to Harten-Yee, Roe-Sweby or Davis-Yee TVD schemes. Meanwhile, if the need uses an integral form in representing the governing equation of fluid motion; it may use a finite volume method (FVM). In this respect, an idea of FVM based on cell vertex method or cell center method can be used. These two methods are basically concerned with the way the control point for each element is defined. While that, in terms of solving the non-linear partial differential equation, the method as introduced by Roe can be used. This method describes the flux of the flow variables. Hence the Roe method can also be used in the finite difference approach as well. The yellow bars and blue bars indicate the CFD code processes that have been developed in this study.

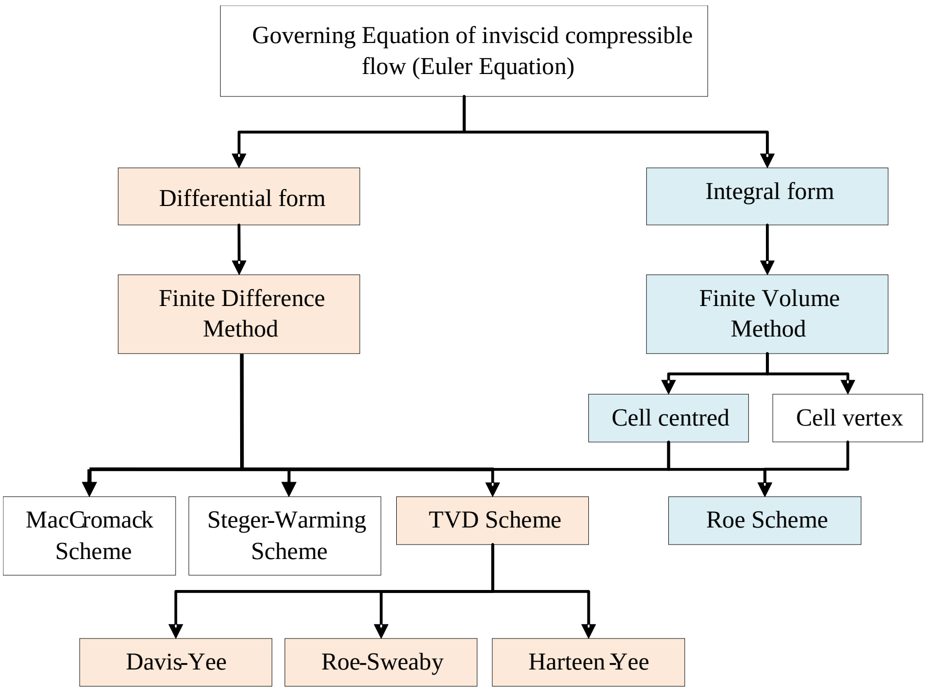

Figure 2.1: Hierarchy of CFD solver

2.3 Governing equation

Basically describing the flow behaviour can be done by writing it in the form of governing equation of fluid motion. This governing equation of fluid motion can be written in differential form or in the integral form. Both forms are formulated based on three physical conservation laws. They are known as (1) conservation of mass (continuity equation), (2) conservation of linear momentum (Newton’s second law) and (3) conservation of energy (first law of thermodynamics). The manner on how to derive the governing equation of fluid motion in differential form as well as in integral form can be obtained in various fluid mechanics text books [15-19] or CFD text books [20-25]

The governing equations of fluid motion expressed in differential form presented in vector notation and Cartesian are given in Eq. (2.1) to Eq. (2.5). The continuity equation can be written as below,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0 \tag{2.1}$$

In Eq. (2.1), the ρ, **V** and t represent the density of fluid, vector velocity and time respectively. The momentum equations in three components can be written as below,

x-component

$$\frac{\partial}{\partial t}(\rho u) + \nabla \cdot (\rho u \mathbf{V}) = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\tau_{xx} + \frac{\partial}{\partial y}\tau_{yx} + \frac{\partial}{\partial z}\tau_{zx} + \rho f_x \tag{2.2}$$

y-component

$$\frac{\partial}{\partial t}(\rho v) + \nabla \cdot (\rho v \mathbf{V}) = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\tau_{xy} + \frac{\partial}{\partial y}\tau_{yy} + \frac{\partial}{\partial z}\tau_{zy} + \rho f_y \tag{2.3}$$

z-component

$$\frac{\partial}{\partial t}(\rho w) + \nabla \cdot (\rho w \mathbf{V}) = -\frac{\partial p}{\partial z} + \frac{\partial}{\partial x}\tau_{xz} + \frac{\partial}{\partial y}\tau_{yz} + \frac{\partial}{\partial z}\tau_{zz} + \rho f_z \tag{2.4}$$

In Eq. (2.2) to Eq.(2.3) p is static pressure, τ is shear stress and f is flux vector. Meanwhile, u, v, w are the x, y, z components of the velocity. The energy equation can be written as below:

$$\begin{aligned}\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla\cdot\left[\rho\left(e+\frac{V^2}{2}\right)\mathbf{V}\right] &= \rho\dot{q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)\\ &+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}+\frac{\partial(u\tau_{xx})}{\partial x}\\ &+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}+\frac{\partial(v\tau_{xy})}{\partial x}+\frac{\partial(v\tau_{yy})}{\partial y}+\frac{\partial(v\tau_{zy})}{\partial z}\\ &+\frac{\partial(w\tau_{xz})}{\partial x}+\frac{\partial(w\tau_{yz})}{\partial y}+\frac{\partial(w\tau_{zz})}{\partial z}+\rho\mathbf{f}\cdot\mathbf{V}\end{aligned} \tag{2.5}$$

where ∇ is vector operator, **V** is vector velocity, **f** is vector of body force per unit mass. It can be defined as below

$$\nabla\equiv\mathbf{i}\frac{\partial}{\partial x}+\mathbf{j}\frac{\partial}{\partial y}+\mathbf{k}\frac{\partial}{\partial z} \tag{2.6}$$

$$\mathbf{V}=u\mathbf{i}+v\mathbf{j}+w\mathbf{k} \tag{2.7}$$

$$\mathbf{f}=f_x\mathbf{i}+f_y\mathbf{j}+f_z\mathbf{k} \tag{2.8}$$

Namely, Eq. (2.1) to Eq. (2.5) are representing the governing equation for viscous flow which considers the transport phenomena of friction and thermal conduction. These equations are known as Navier-Stokes equation.

2.4 Euler equation

If viscosity effect can be neglected in a fluid flow, the flow is considered to be non-viscous or inviscid. That means all elements of friction and thermal conduction will be neglected. As a result, the continuity in Eq. (2.1) can be written as below.

$$\frac{\partial\rho}{\partial t}+\nabla\cdot(\rho\mathbf{V})=0 \tag{2.9}$$

and momentum equation in Eq. (2.2), Eq. (2.3) and Eq. (2.4) can be written as, *x-component*

$$\frac{\partial}{\partial t}(\rho u) + \nabla \cdot (\rho u \mathbf{V}) = -\frac{\partial p}{\partial x} + \rho f_x \tag{2.10}$$

y-component

$$\frac{\partial}{\partial t}(\rho v) + \nabla \cdot (\rho v \mathbf{V}) = -\frac{\partial p}{\partial y} + \rho f_y \tag{2.11}$$

z-component

$$\frac{\partial}{\partial t}(\rho w) + \nabla \cdot (\rho w \mathbf{V}) = -\frac{\partial p}{\partial z} + \rho f_z \tag{2.12}$$

For energy equation in Eq. (2.5), it can be written as,

$$\begin{aligned} \frac{\partial}{\partial t}\left[\rho\left(e + \frac{V^2}{2}\right)\right] + \nabla \cdot \left[\rho\left(e + \frac{V^2}{2}\right)\mathbf{V}\right] = \rho q - \frac{\partial (up)}{\partial x} - \frac{\partial (vp)}{\partial y} - \frac{\partial (wp)}{\partial z} \\ + \rho \mathbf{f} \cdot \mathbf{V} \end{aligned} \tag{2.13}$$

In the above equations, Eq. (2.9) until Eq. (2.13), are known as Euler Equation. These equations can be used whether the flow problem belongs to the class of compressible flow or incompressible flow. Both may be in steady or unsteady flow conditions.

2.5 Available CFD code for air vehicle design and analysis

The application of CFD especially in aerodynamics field has been widely use since 30 years ago. CFD together with wind tunnels and flight test may represent the main tools in the aerodynamic analysis and design of flying vehicles. However, as the CFD starts in use as aerodynamics tool, the aircraft manufacturers are no longer depending too much on the use of wind tunnels. These are due to the fact that, for the same purposes, the use of wind tunnels are very costly and time consuming compared to the use of CFD. However, wind tunnels are still needed to confirm the results provided by CFD that are still in line with the result produced by wind tunnel test. In addition to this, it can be said that the wind tunnel and CFD can complemented each others. The wind tunnel result can be used to validate the CFD results, on other hand the CFD results can be used to calculate the wind tunnel wall correction factors. Other benefits offered by CFD would be that, this approach can narrow down the number of design constraints and parameters in which further detailed flow analysis can be conducted through wind tunnel tests. This means that CFD will identify some important flow regions where further study purposes will be carried out in wind tunnel tests by using some instrumented models [26].

Basically, the development of CFD throughout the history of Boeing Commercial Airplane in the process of producing their various types of commercial passenger aircrafts can be looked on. Figure 2.2 shows the Boeing aircraft generation and the corresponding CFD code associated with the development of these aircrafts. The Boeing 767 and Boeing 757 represent two examples of passenger type aircraft in which their aerodynamics analysis and design are obtained by using a CFD code named as PANAIR. While that, Boeing 737-300 series are using CFD code called FLO22 and TRANAIR. The aircraft Boeing 787 series as the most lastet aircraft are designed on the CFL3D OVERFLOW's CFD code [2].

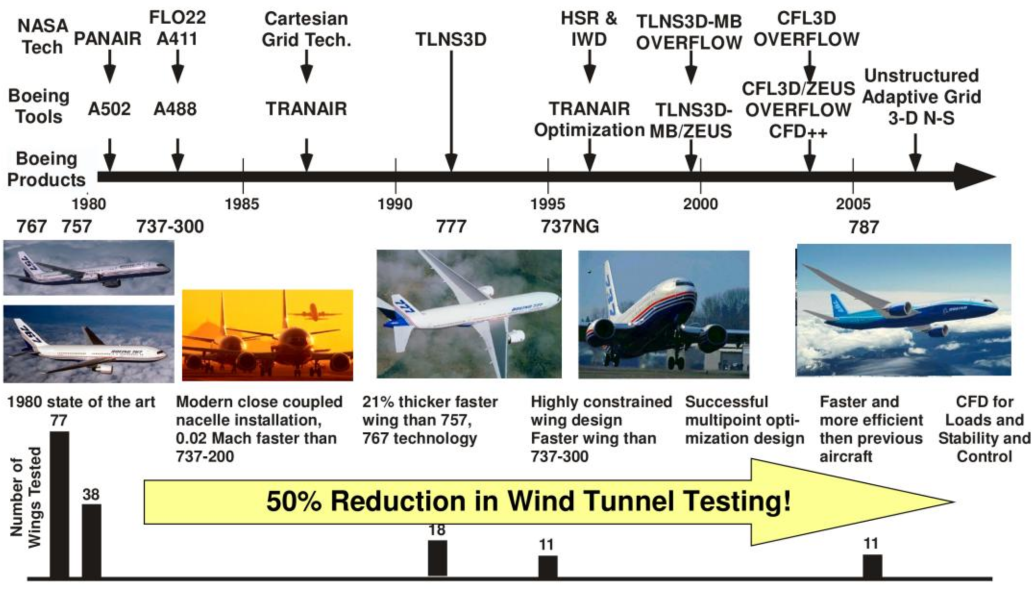

Figure 2.2: Chronology of Boeing aircraft production [27]

The presence of various softwares such that is being used in Boeing aircraft cannot separate the existence of various levels of governing equations of fluid motion. Basically in a manner of the flow problems that can be solved are not unique. They depend on the kind of solutions they are looking for. Strictly speaking, the governing equation of fluid motion can be divided into several levels of governing equation as shown in Figure 2.3.

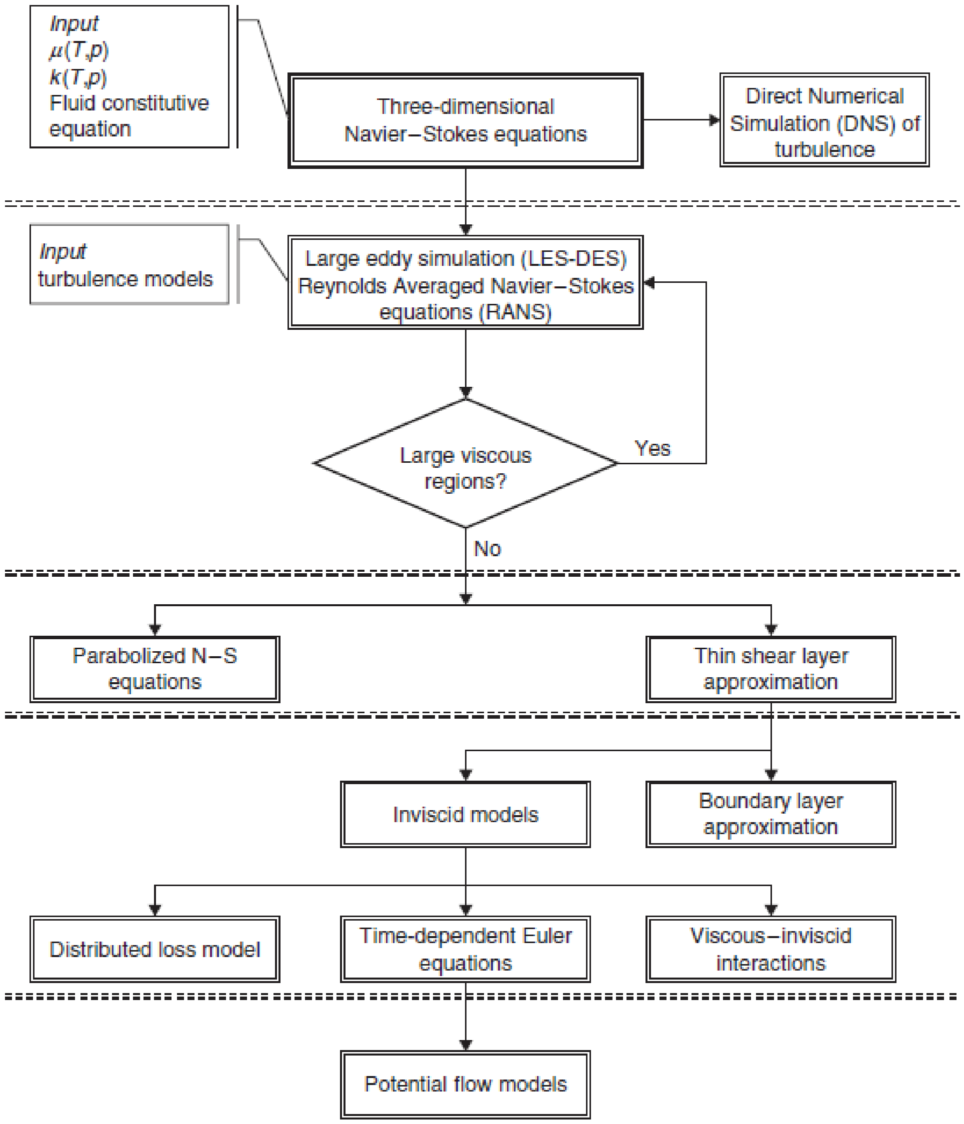

Figure 2.3: Hierarchy of governing equation [28]

Figure 2.3 describes that the highest level of governing equation of fluid motion is the three-dimensional (3D) Navier-Stokes equation. This equation is derived directly from three conservation laws, mass, momentum and energy. Basically, this equation can be solved directly; it needs to supplement additional information concerning flow properties of viscosity and thermal conductivity. For special case of laminar flow, any experiment in laminar flow regime can be accurately duplicated from the solution of this equation. Unfortunately, most engineering problem of interest enters the flow into a particular form of instability, called turbulence. This instability phenomenon occurs in almost all flow situations when the velocity, or more precisely, the Reynolds number, defined as the product of representative scales of velocity and length divided by the kinematic viscosity, exceeds a certain critical value. The particular form of instability generated in the turbulent flow regime is characterized by the presence of statistical fluctuations to the all flow quantities. These fluctuations can be considered as superimposed on mean or averaged values and can attain, in many situations, the order of 10% of the mean values, although certain flow regions, such as separated zones, can attain much higher levels of turbulent fluctuations. The numerical description of the turbulent fluctuations is a formidable task which puts very high demands on computer resources. As mentioned earlier, the numerical approach designed to solve the Navier-Stokes equation directly is DNS that is possible to be used for industrial applications which may become viable after 2080. The present computer capability is not yet sufficient enough to fulfil the DNS requirements. An attempt on reducing computer demand can be done through deep observation of the flow behaviour. It has been found that most flow problems have two types of turbulent fluctuations, a large and small scale fluctuations. In this respect, the large scale turbulent fluctuations are solved directly from the Navier-Stokes equation while a smaller scale is through its simplified form of the Navier-Stokes equation. Such approach is known as Large Eddy Simulations (LES). Unfortunately the implementation of this method to deal with industrial applications is still time consuming, it can be fully utilized after 2045 as shown in Figure 2.4 [1].

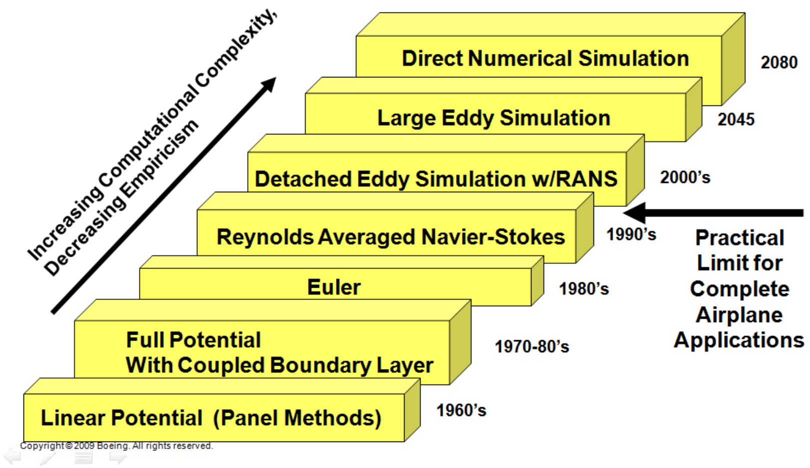

Figure 2.4: Usage of CFD on aircraft development [1]

The third level of the governing equation of fluid motion is the Reynolds Averaged Navier-Stokes equation (RANS). This equation can be obtained through representing turbulent flow phenomena consisting of two quantities which are the average value and the fluctuated value. If the ratio between fluctuated value and the average value is small and the average value of the fluctuated quantity goes to zero for sufficient time, then the Navier-Stokes equation can be reformulated to become a RANS. However in solving this equation, it has to introduce a turbulence modelling. It is true that the spatial and temporal discretisation requirements are far less than the DNS or LES approach, but their solution are depending on the turbulence modelling invoked and also the numerical scheme in use.

RANS can be further simplified if the flow problem belongs to the class of high Reynolds number flow with the flow separation covered up in small flow domain. In this situation, it can ignore the viscous and turbulent diffusion terms in the main stream direction. Here the RANS equation becomes an equation known as Thin Shear Layer Equation. While that, if the pressure gradient in the normal direction of the body surface is equal to zero, then the RANS equation becomes a Parabolized Navier-Stokes equation.

The fourth level of governing equation of fluid motion is split into two equations models. The first equation describes the governing equation of inviscid flow motion while the second one describes the influence of viscous effects. Such flow phenomenon may exist if the flow problem deals with the flow at high Reynolds number past through a streamline body at relatively low angle of attack. Here, the flow domain around the body can be divided into two regions, the flow domain that are relatively away from the body surface where the flow will behave as inviscid flow while the flow domain close to the body surface gets a strong influence of the viscosity. As a result, the flow domain relatively far away from the body surface is governed by Euler equations. This equation is simply obtained from the Navier-Stokes equation through eliminating viscous term. While that, the governing equation of fluid motion for the flow close to the body surface which can be obtained through the implementation order of magnitude analysis upon the Navier-Stokes equation resulting in the governing equation known as the Boundary Layer Equation. The Euler equation still represents a non-linear partial differential equation and it is not easy to be solved. In the absence of viscous effects, in the inviscid flow domain may impose the flow is irrotational flow. Imposing such flow condition, the Euler equation can be reduced to become the the continuity equation that needs to be solved. The momentum equation was initially in the form of differential equation; they can be converted to become an algebraic equation in relating the flow variables of pressure and flow velocity. The reduced form of the Euler Equation is known as a Full Potential Equation. Through a Full potential Equation further simplification can be done by considering the presence of the body immersed in the flow field creating a small perturbation into the flow field. Such flow condition makes the Full Potential Equation can be simplified to become a Prandtl-Glaured equation. This equation is applicable to the case of compressible flow at high subsonic flow or supersonic flow. In the case of transonic flow, the Full Potential Equation becomes a Transonic Small Perturbation Equation (TSP-equation). If the inviscid flow problem applied to the incompressible flow and in addition with the flow considered as irrotational flow, the Euler equation can be reformulated to become a Laplace equation. This represents the lowest level of the governing equation of fluid motion. The level of complexity of the governing equation of fluid motion is shown in Figure 2.5.

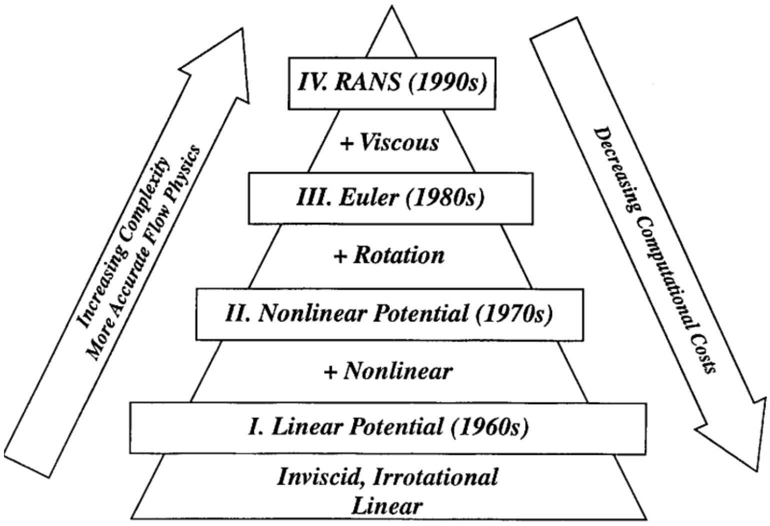

Figure 2.5: The level of complexity of the governing equation of fluid motion

2.5.1 CFD Linear potential flow equation

Most aeronautics applications are involved with the flow problem past through a streamline body. Aircraft as a flying vehicle was designed for having a streamline body and operated at a low angle of attack and high Reynolds number. Such flow conditions make their flow problems can be treated as inviscid flow problems and irrotational flow condition can be invoked. As a result, the governing equation of fluid motion can be formulated in the form of Laplace equation if the flow belongs to the class of incompressible flow or in the form of Prandtl-Glaured equation if the compressible effect needs to be taken into account. Both equations correspond with each other. The method was used for solving the Laplace equation for solving the Prandtl-Glaured equation. Initially an attempt to solve the Laplace equation is carried out by using an analytical approach. Using a complex variable, the two-dimensional (2D) flow past through shape like airfoil can be solved, so the pressure distribution along the airfoil surface can be predicted.

The advancement of computer technology had opened up ways on how the Laplace equation can be solved in order to obtain the aerodynamics characteristics over an arbitrary body shape. The first attempt for such flow problem solving is carried out by Hess and Smith in 1962. The manner on how the Laplace equation is solved is called as the Panel method [3]. This method was developed based on the Laplace equation invention which represents the governing equation in the form of partial differential equation that can be converted into an integral form. The flow problem which represents the field problem becomes the body surface problem, since the unknown of the quantity of the flow is now determined on the body surface. The success of Hess and Smith in solving the Laplace equation by using Panel method had encouraged researchers around the world to develop other versions of Panel method. Table 2.1 shows some list of well known Panel method.

Table 2.1 List of some major Panel Method [29]

Originator and Method Name	*Year*	*Panel Geometry*	*Source Type*	*Doublet Type*	*Boundary Conditions*	*Restrictions*	*Comments*
MCAIR [1] (McDonnell)	1980	flat	constant	quadratic			design option
PAN AIR [2] (Boeing)	1980	continuous piecewise flat	continuous linear	continuous quadratic	arbitrary in ϕ, $\nabla\phi$		subsonic and supersonic
Hess II [3] (Douglas)	1981	parabolic	linear	quadratic	normal flow		
VSAERO[4] (AMI)	1981	flat	constant	constant	exterior and interior normal flow		subsonic
QUADPAN [5] (Lockheed)	1981	flat	constant	constant			
PMARC [6] (NASA Ames)	1988	flat	constant	constant			unsteady, wake rollup

Among those Panel method as mentioned in above table, PANAIR, VSAERO and PMARC will be described further in the following sub-chapter.

2.5.1.1 PANAIR code [30]

This code was developed by Boeing and funded by a variety of government agencies such NASA. PANAIR (an abbreviation for "panel aerodynamics") is a state-of-the art computer program developed to predict inviscid subsonic and supersonic flows. It is about an arbitrary configuration by means of a higher-order Panel method. A Panel method can solve a linear partial differential equation numerically by approximating the configuration surface by a set of panels on which unknown "singularity strengths" are defined, imposing the boundary conditions at a discrete set of points, thereby generating a system of linear equations relating the unknown singularity strengths. These equations solved problems for singularity strengths which provide information on the properties of the flow. PANAIR method differs from earlier Panel method by employing a "higher-order" panel method wherethe singularity strengths are not constant on each panel. This is necessitated by the more stringent requirements on the supersonic problem. The potential for numerical error is greatly reduced in the PANAIR program by requiring the singularity strength to be continuous. It is also this "higher order" attribute which allows PANAIR to be used to analyse flow on arbitrary configurations. PANAIR can handle the simple configurations considered in the preliminary design phase and later serves as the "analytical wind tunnel" which can analyse the flow on the final detailed, complex configurations. In general, the aircraft surface is partitioned into several networks of surface grid points, such as a fore body network, a wing network, and so forth. The coordinates of the input grid points must be computed and entered by the user. The theoretical background on how to develop this code according to the Panel method is described in [30-33]. While that, [34-36] gives some examples on the application of the PAN AIR code for solving various flow problems

2.5.1.2 VSAERO code [37]

This code was developed by Ames Research Centre by AMI (Analytical Mechanics Inc). The code implements the surface singularity panel method using quadrilateral panel on which doublet and source singularities distributed piecewise constant form.
The panel source values are directly determined by the external Neuman boundary condition controlling the normal local resultant flow. The doublet values are solved after imposing the internal Direchlet boundary condition of zero perturbation potential at the centres of the panels simultaneously. Surface perturbation velocities are obtained from the gradient of the doublet solution while field velocities are obtained by direct summation of all singularity panel contributions. In order to accommodate the ability to predict the non-linear aerodynamic characteristics, the vortex separation and vortex/surface interaction are treated in an iterative wake-shape calculation procedure, while the effects of viscosity are treated in an iterative loop coupling potential flow and integral boundary layer calculations. The user manual on how to use the VSAERO Code including its theoretical back ground can be referred to [32, 38, 39].

2.5.1.3 PMARC [40]

PMARC is a computer code for aerodynamics analysis around a complex 3D geometries by using a Low order Panel Method. PMARC stand for “A Panel Method of Ames Research Centre”. Basically PMARC represents the extension of VSAERO code. This computer code has the same capabilities as the VSAERO codes; it has several advanced features such as the ability in dealing with the internal flow model, a simple jet wake model, and a time-stepping wake model. In addition to that, the data management within the code has been optimized by the use of an adjustable size arrays for rapidly changing the size capability of the code, reorganization of the output file and adopting a new plot file format. The Panel method which had been used in developing PMARC code may be referred to [41, 42].

2.5.2 Potential flow equation

Basically, there are three conservative formulations used for inviscid transonic flow. They are followed by transonic small-disturbance equation, full potential equation and Euler equation for the exact inviscid formulation. Transonic small-disturbance is suitable to solve transonic flow with simple geometry. Transonic small-disturbance has been solved for lifting, sweat-wings and simple wing-fuselage combination. Meanwhile, the full potential for complex body includes bodies of revolution, asymmetric and planar inlet nacelles and yawed [43]. The solver for transonic-small disturbance has been developed in the early 1970s by Murman and Cole by using concept within subsonic regions and backward differences within supersonic regions. Then the modification of Murman and Cole scheme has been used to solve full potential flow equation [4].

2.5.2.1 FLO22 & FLO27 [5]

These codes were developed by Jameson for Boeing Company by using rotated finite volume scheme. The FLO22 code is the first transonic potential flow solution for 3D swept wing. This code has been used for the wing design of the Canadair Challenger, later marketed as XFLO22 by the Dutch NLR. The code is still in use today for preliminary design at Boeing and other aircraft companies. It is useful in this role, as it is capable of computing 3D flow fields on grids containing about 150,000 cells in less than 15 seconds on a current laptop computer. The next is FLO27 and it has been incorporated in Boeing A488 software.

2.5.2.2 TRANAIR [44]

This code was also developed by Boeing Company and NASA. This code has been used heavily on commercial transport designs since the 777 – 200. TRANAIR code development began under contract with NASA as a feasibility study in 1984. The technology to analyze transonic flow with a uniform orthogonal field grid was developed under this initial contract. Further development under a second NASA contract led to the development of grid refinement techniques. Today TRANAIR code is a fully functional analysis and design tool with continuous development in design, adaptive grid refinement, coupled boundary layer and design capability

2.5.3 Euler equation

The highest level of the inviscid flow is Euler equation. For many practical aerodynamic applications, this equation is relatively accurate for representing the flow field which includes both rotational and discontinuous (shock) phenomena in the flow and providing an excellent approximation for lift induced drag and wave drag. Furthermore, a robust Euler solver is an essential part of any Navier-Stokes solver. In addition to this, Euler equations promised to provide more accurate solutions of transonic flows.

The Euler equation was used by Jameson in developing computer code for solving a 3D flow problem named FLO57 code in 1981. It was used to develop other codes called a MGAERO code. MGAERO code is unique in being a structured Cartesian mesh code. Besides that, Jameson also developed the AIRPLANE code which made use of unstructured tetrahedral grids. In the 2D, Drela and Giles developed the ISES code for airfoil design and analysis. This code first became available in 1986 and has been further developed to design, analyze and optimize single or multi-element airfoils, also known as MSES code.

2.5.3.1 FLO57 [6]

This code was developed by Jameson and had been used extensively in Airbus Company. FLO57 code uses 3D Euler code to analyse the inviscid transonic flow

REFERENCES

1. Tinoco, E. N. The Impact of High Performance Computing and Computational Fluid Dynamics on Aircraft Development. *20th Anniversary Dinner & Symposium CASC.* Washinton, DC: Coalation for Academic Scientific Computation. 2009.
2. Johnson, F.T., Tinoco, E. N., & Yu, N. J. Thirty years of development and application of CFD at Boeing Commercial Airplanes, Seattle. *Computers & Fluids.* 2005. 34: 1115-1151.
3. Hess, J. L. & Smith, A. M. Calculation of non-lifting potential flow about arbitrary three-dimensional bodies: DTIC Document. 1962.
4. Cole, J. D. & Murman, E. M. Calculation of plane steady transonic flows. *AIAA Journal.* 1971. 9: 114-121.
5. Jameson, A. & Caughey, D. A. Caughey. A finite volume method for transonic potential flow calculations. *AIAA Paper.* 1977. 635.
6. Jameson, A. & Schmidt, W. and Turkel, E. Numerical solutions of the Euler equations by finite volume methods using Runge-Kutta time-stepping schemes. *AIAA Paper.* 1981. 1259.
7. Jameson, A. & Schmidt, W. and Turkel, E. Numerical solution of the Euler equations by finite volume methods using Runge Kutta time stepping schemes. *14th Fluid and Plasma Dynamics Conference*. American Institute of Aeronautics and Astronautics. 1981.
8. Drela, A. Newton solution of coupled viscous/inviscid multielement airfoil flows. *21st Fluid Dynamics, Plasma Dynamics and Lasers Conference*. American Institute of Aeronautics and Astronautics. 1990.
9. Jameson, A. & Fatica, A. Using computational fluid dynamics for aerodynamics. Stanford University. 2006.

10. Jameson, A., Martinelli, L. & Vassberg, J. Using computational fluid dynamics for aerodynamics - a critical assessment. *Proceedings of ICAS*. 2002: 1-10.

11. Godunov, S. A. difference method for numerical calculation of discontinuous solutions of the equations of hydrodynamics. *Matematicheskii Sbornik.* 1959. 89: 271-306.

12. Roe, P. L. Approximate Riemann solvers, parameter vectors, and difference schemes. *Journal of Computational Physics.* 1981. 43: 357-372,.

13. Harten, A. High resolution schemes for hyperbolic conservation laws. *Journal of Computational Physics.* 1983. 49: 357-393.

14. Harten, A. On the symmetric form of systems of conservation laws with entropy. *Journal of Computational Physics.* 1983. 49:151-164.

15. Subrahmaniyam, S. *Fluid mechanics*. New Delhi. Capital Publishing. 2007.

16. Munson, B. R. *Fundamentals of fluid mechanics* (6th ed.). Hoboken NJ. Wiley. 2010.

17. Munson, B. R. *Fluid mechanics* (10th ed.). Hoboken, NJ. Wiley. 2013.

18. White, F. M. *Fluid Mechanics*. Boston. McGraw-Hill,.2008.

19. Janna, W. S. *Introduction to fluid mechanics.* Boca Raton. CRC Press. 2010.

20. Blazek, J. *Computational Fluid Dynamics: Principles and Applications*. Elsevier. 2005.

21. Hoffman, K. A. & Chiang, S. T. *Computational Fluid Dynamics vol. II.* (4th ed). USA: www.EESbooks.com. 2000.

22. Hoffman, K. A. &Chiang, S. T. *Computational Fluid Dynamics vol. I.* (4th ed.). USA: www.EESbooks.com. 2000.

23. Chung, T. J. *Computational fluid dynamics*. Cambridge university press, 2010.

24. Tannehill, J. C. & Anderson, D. A. & Pletcher, R. H. *Computational Fluid Mechanics and Heat Transfer* (2nd ed). United State of America. Taylor & Francis. 1997.

25. Anderson, J. D. *Computational Fluid Dynamics : The Basic with Application*. New York. McCraw-Hill. 1995.

26. Cosner, R. R. & Roetman, E. L. Application of computational fluid dynamics to air vehicle design and analysis. *Aerospace Conference Proceedings*. IEEE. 2000. pp. 129-142.

27. Jameson, A. Computational Past, Present and Future. *AMS Seminar Series*. Moffett Field, CA: 2012.

28. Hirsh, C. *Numerical Computation of Internal and External Flows* (2nd ed). Great Britian. Elsevier. 2007.

29. Henne, P. A. *Applied computational aerodynamics*. Washington, DC. American Institute of Aeronautics and Astronautics. 1990.

30. Magnus, A. E. & Epton, M. A. PAN AIR-A Computer Program for Predicting Subsonic or Supersonic Linear Potential Flows about Arbitrary Configurations Using a Higher Order Panel Method. Volume I. Theory Document (Version 1.0). NASA. CR-3251. 1980.

31. Sidwell, K. W., Baruah, P. K.,Bussoletti, J. E., Medan,T. R. & Conner, S. PAN AIR - A Computer Program for Predicting Subsonic or Supersonic Linear Potential Flows About Arbitrary Configurations Using A Higher Order Panel Method, Vol. II. User's Manual (Version 1.0). NASA. 1980.

32. Larry, L. E. Panel Method - An Introduction. Ames Research Centre, Moffett Field, California. Nasa Technical Paper 2995.1990.

33. Derbyshire, T. & Sidwell, K. W. PAN AIR Summary Document, (Version 1.0). NASA. 1982.

34. Alexander, J. I. D., Johnson, F. T. & Freeman, L. M. Application of a higher order panel method to realistic supersonic configurations. *Journal of Aircraft.*1980. 17: 38-441980.

35. Lee, K. D. Numerical Simulation of the Wind Tunnel Environment by a Panel Method. *AIAA Journal*, 1981. 19: 470-475.

36. Chen, A. W. & Tinoco, E. N. PAN AIR applications to aero-propulsion integration. *Journal of Aircraft.* 1984. 21:161-167.

37. Maskew, B. Prediction of Subsonic Aerodynamic Characteristics: A Case for Low-Order Panel Methods. *Journal of Aircraft.* 1982. 19: 157-163.

38. Maskew, B. *Program VSAERO theory Document: a computer program for calculating nonlinear aerodynamic characteristics of arbitrary configurations* . 4023. NASA. 1987.

39. Quick, H. A. Estimation of aerodynamic load distributions on the F/A-18 aircraft using a CFD panel code. DTIC Document. 1995.

40. Ashby, D. L., Dudley, M. R. & Iguchi, S. K. *Development and validation of an advanced low-order panel method*: NASA, Ames Research Center. 1988.
41. Hess, J. L. Calculation of potential flow about arbitrary three-dimensional lifting bodies. DTIC Document. 1972.
42. Ashby, D. L., Dudley, M. R., Iguchi,S. K., Browne, L. & Katz, J. Potential Flow Theory and Operation Guide for the Panel Code PMARC 12. NASA, Ames Research Center. 1992.
43. Caughey, D. A. & Jameson, A. Numerical Calculation of Transonic Potential Flow about Wing-Body Combinations. *AIAA Journal.* 1979. 17: 175-181.
44. Samant, S., Bussoletti, J., Johnson, F., Burkhart, Everson, R., Melvin, B. R., Young, D., Erickson, L. & Madson, M.TRANAIR - A computer code for transonic analyses of arbitrary configurations. *25th AIAA Aerospace Sciences Meeting*: AIAA. 1987.
45. Giles, M., Drela, M. & Thompkins, W. J. R. Newton solution of direct and inverse transonic Euler equations. *7th Computational Physics Conferene:* AIAA. 1985.
46. Jameson,A., Baker, T. & Weatherill, N. Calculation of Inviscid Transonic Flow over a Complete Aircraft. *24th Aerospace Sciences Meeting*: AIAA. 1986.
47. Tidd, D., Strash, D. Epstein, B., Luntz, A., Nachshon, A. & Rubin, T. Application of an efficient 3-D multigrid Euler method (MGAERO) to complete aircraft configurations. *9th Applied Aerodynamics Conference*: AIAA. 1991.
48. Mohamad, M. A. H., Basri, S. & Basuno, B. One-dimensional high-order compact method for solving Euler equations. 2013.
49. Zulkafli, B., Fadhli, M. Omar, A. A. E. & Asrar W. Numerical analysis of FDV method for one-dimensional Euler equations. 2012: 1-6.
50. Wahi, N. & Ismail, F. Numerical shock instability on 1D Euler equations. *AIP Conference Proceedings.* 2013. 1522: 376-383.
51. Roslan, N. K. H. & Ismail, F. Evaluation of the entropy consistent euler flux on 1D and 2D test problems. *The 4th International Meeting Of Advances In Thermofluids.* 2012: 671-678.
52. Ghuanghui, H. *Numerical simulations of the steady Euler equation on unstructured grid*. Thesis Ph.D. Hong Hong Baptist University; 2009.

53. Botte,G. G., Ritter, J. A. & White, R. E. Comparison of finite difference and control volume methods for solving differential equations. *Computers & Chemical Engineering. 2*000. 24: 2633-2654.

54. Terzi, D. V., Linnick, M., Seidel, J. & Fasel, H. Immersed boundary techniques for high-order finite-difference methods. *15th AIAA Computational Fluid Dynamics Conference*: AIAA. 2001.

55. Thomas, P. High Order Accurate Finite-Difference Methods: as seen in OVERFLOW. *20th AIAA Computational Fluid Dynamics Conference*: AIAA. 2011.

56. Ray, H., Nallasamy, M. & Scott, S. A Method for the Implementation of Boundary Conditions in High-Accuracy Finite-Difference Schemes. *43rd AIAA Aerospace Sciences Meeting and Exhibit*: AIAA. 2005.

57. Mead H. R. & Melnik. R. E. GRUMFOIL: A Computer Code for the Viscoues Transonic Flow Over Airfoils. NASA CR-3806. 1985.

58. Yee, H. C.Upwind and symmetric shock-capturing schemes. NASA Ames Research Center; Moffett Field, CA, United States NASA-TM-89464. 1987.

59. Harten, A. On a Class of High Resolution Total-Variation-Stable Finite-Difference Schemes. *SIAM Journal on Numerical Analysis.* 1984. 21:1-23.

60. Yee, H. C. & Kutler, P. Application of second-order-accurate total variation diminishing (TVD) schemes to the Euler equations in general geometries. NASA Ames Research Center; Moffett Field, CA, United States. NASA-TM-85845. 1983.

61. Yee, H. C., Warming, R. F. & Harten, A. Implicit total variation diminishing (TVD) schemes for steady-state calculations. *6th Computational Fluid Dynamics Conference Danvers*. 1983.

62. Yee, H. C., Warming, R. F. & Harten, A. Implicit total variation diminishing (TVD) schemes for steady-state calculations. *Journal of Computational Physics.* 1985. 57: 327-360.

63. Yee, H. C. & Harten, A. Implicit TVD schemes for hyperbolic conservation laws in curvilinearcoordinates. *AIAA JournaL.* 1985. 25:

64. Yee, H. C. & Harten, A. Implicit TVD schemes for hyperbolic conservation laws in curvilinearcoordinates. *7th Computational Physics Conference*. 1985.

65. Van Leer, B. Towards the ultimate conservative difference scheme. V. A second-order sequel to Godunov's method. *Journal of Computational Physics.* 1979. 32: 101-136.

66. Sohn, S.-I. A new TVD-MUSCL scheme for hyperbolic conservation laws. *Computers & Mathematics with Applications.* 2005. 50: 231-248.

67. Davis, S. F. A Simplified TVD Finite Difference Scheme via Artificial Viscosity. *SIAM Journal on Scientific and Statistical Computing.* 1987. 8: 1-18.

68. Kroll, N., Gaitonde, D. & Aftosmis, M. A systematic comparative study of several high resolution schemes for complex problems in high speed flows. in *29th Aerospace Sciences Meeting*: AIAA. 1991.

69. Chen, M.-H., Hsu, C.-C. & Shyy, W. Assessment of TVD schemes for inviscid and turbulent flow computation. *International Journal for Numerical Methods in Fluids.* 1991. 12: 161-177.

70. Qureshi K. R. & Lee C. H. Behavior Of Tvd Limiters On The Solution Of Non-Linear Hyperbolic Equation. *Modern Physics Letter.* 2005. 19: 1507-1510.

71. Chang,Y. L. *Development of a CFD code using TVD schmes and advanced turbulence models for incompressible flow simulations*. Thesis Ph.D. Michigan Technological University; 1997.

72. Glaister, P. Flux difference splitting for the Euler equations in one spatial co-ordinate with area variation. *International Journal for Numerical Methods in Fluids.* 1988. 8: 97-119.

73. Glaister, P. A weak formulation of Roe's scheme for two-dimensional, unsteady, compressible flows and steady, supersonic flows. *Computers & Mathematics with Applications,* 1995. 30: 85-93.

74. Abgrall, R. M. An extension of Roe's upwind scheme to algebraic equilibrium real gas models. *Computers & Fluids,* 1991. 19: 171-182.

75. Mottura, L., Vigevano, L. & Zaccanti, M. An Evaluation of Roe's Scheme Generalizations for Equilibrium Real Gas Flows. *Journal of Computational Physics*. 1977. 138: 354-399.

76. Chakravarthy, S. & Osher, S. A new class of high accuracy TVD schemes for hyperbolic conservationlaws. *23rd Aerospace Sciences Meeting*: AIAA. 1985.

77. Anderson, W. K., Thomas, J. L. & Van Leer, B. Comparison of finite volume flux vector splittings for the Euler equations. *AIAA Journal*. 1986. 24: 1453-1460.

78. Jameson, A. Requirements and trends of computational fluid dynamics as a tool for aircraft design. *Proceedings of the 12th NAL symposium on aircraft computational aerodynamic.* Tokyo, Japan. 1994.

79. Maciel, E. S. G. TVD Algorithms Applied to the Solution of the Euler and Navier-Stokes Equations in Three-Dimensions. *WSEAS Transactions on Mathematics*. 2012. 11: 546-572.

80. Maciel, E. S. G. Explicit and Implicit TVD and ENO High Resolution Algorithms Applied to the Euler and Navier-Stokes Equations in Three-Dimensional Turbulent Results. 2013.

81. Shigeki, H., Justin, A., Klaus, H. &Ramesh, A. Development of a modified Runge-Kutta scheme with TVD limiters for the ideal 1-D MHD equations. *13th Computational Fluid Dynamics Conference*. 1997.

82. Shigeki, H., Klaus, H. & Justin, A. Development of a modified Runge-Kutta scheme with TVD limiters for the ideal two-dimensional MHD equations. *36th AIAA Aerospace Sciences Meeting and Exhibit*: American Institute of Aeronautics and Astronautics. 1998.

83. Anderson, J. D. *Introduction to flight*. New York: McGraw-Hill. 2015.

84. Harris, C. D. Two-dimensional aerodynamic characteristics of the NACA 0012 airfoil in the Langley 8 foot transonic pressure tunnel. 1981.

85. Gregory, N. & O'Reilly, C. L. *Low-Speed aerodynamic characteristics of NACA 0012 aerofoil section, including the effects of upper-surface roughness simulating hoar frost*: HM Stationery Office. 1973.

86. Hess, R. W. Unsteady pressure measurements on a supercritical airfoil at high Reynolds numbers. 1989.

87. Thompson, J., Soni, B. & Weatherill, N. *Handbook of Grid Generation*. Washington, D. C.: CRS Press. 1999.