

THE DEVELOPMENT OF REYNOLDS AVERAGED NAVIER STOKES SOLVER  
FOR A TWO DIMENSIONAL COMPRESSIBLE FLOW PROBLEM

HASAN TAHER M.ELKAMEL

A thesis submitted in  
fulfillment of the requirement for the award of the  
Doctor of Philosophy (Mechanical Engineering)

Faculty of Mechanical and Manufacturing Engineering  
University Tun Hussein Onn Malaysia

May 2017

*To my family, Brothers, Sisters, Uncles, wife  
and sweet daughter*

*I dedicate this work to the soul of my parents*



PTTA UTHM  
PERPUSTAKAAN TUNKU TUN AMINAH

## ACKNOWLEDGEMENT

For ever we offer our deep great thanks to Allah for this wide blessing.

I would like to express my sincere gratitude to my project's supervisors, **Dr. Ir. Bambang Basuno** and **Dr. Norzelawati** for the continuous support of my project, for their patience, motivation, enthusiasm, and immense knowledge. Their guidance helped me in all the time of research and writing of this thesis.

Last but not the least, I would like to thank my family: my uncles , brothers ,sisters, my wife, my daughter and the soul of my parents at the first place for supporting me spiritually throughout my life.



PT TAAUTHM  
PERPUSTAKAAN TUNKU TUN AMINAH

## ABSTRACT

The computational fluid dynamics represented by fluid dynamic science focuses on the way how to solve the flow problems numerically. The governing equation of fluid motion passing through an object flow can be presented in various forms depending on the assumption imposed to the flow problem in hand. Initially, in solving the flow problem passing through an object such as the flow passing through an aircraft, the flow is incompressible, irrotational, and inviscid flow. Resulting from the initial form of governing equation called the Navier-Stokes equations; the flow can be simplified as the Laplace equation. When the incompressible condition cannot be maintained, the compressibility effects have to be taken into account due to the increasing incoming velocity, while the inviscid and irrotational conditions are still maintained. The Navier-Stokes can be reduced to become a full potential equation. The Navier-Stokes equation becomes the Euler equations by ignoring the viscous effects. If the viscous effects are included, the presence of turbulent flow phenomena creates a small fluctuation to the flow variables resulting in the Navier-Stokes equation to reduce and become a Reynolds-averaged Navier-Stokes (RANS) equation. For instance, these various models of the governing equations had been formulated before the era of computer started.

The manner on how to solve the flow problem according to the level of governing equations is based on the achievement of computer technology. In 1960, the aerodynamic problems were solved when the computer capability was limited, which led to the change of the Laplace equation by the method known as the Panel Method. As the computer power became more available, the aerodynamic problems were solved through the full potential equation. Further improvement in computing power made the aircraft designers since 1980 to use Euler equation as the governing equation of motion for the flow problem in hand.

Continuous support gained from computer technology development has helped aircraft designers since 1990 by using the RANS equations in solving their flow problems. The success in the use of RANS equations depends on the manner in combining the numerical grid generation and scheme for discretizing the governing equation and turbulence model, which need to be provided in making the RANS equation solvable. In developing the RANS solver, the present research uses the unstructured grid for meshing the flow domain, combined with the Roe's finite volume scheme for discretizing the RANS equation and Spalart-Allmaras for fulfilling the required turbulent modeling.

For the purpose of validation, the result of the developed computer code was compared with the experimental result available in the literature and result through running the Fluent software. The validation was carried out by using airfoil NACA 0012 and RAE 2822. Both two airfoils have the experimental result in terms of distribution pressure coefficient along the airfoil surfaces at different angles of attacks and Mach numbers. The comparison result over these two airfoil models had found that the developed RANS solver was able to produce the results closed to the experimental result, as well as the Fluent software.

The developed computer code was applied to further evaluate the aerodynamic airfoil characteristics NACA 4415 and Supercritical Airfoil 26a at various angles of attacks and Mach numbers. For the airfoil NACA 4415, the aerodynamic analysis were carried by treating the flow problem as inviscid flow problems while the other as viscous flow problems. In other words, the flow problems in hand were solved by the Euler and RANS solvers. As for the results of the pressure coefficient distribution along the airfoil surface, there was a significant difference between the result provided by the Euler and RANS solvers. While for the supercritical airfoil, the result of the developed computer code as RANS solver found the position of the shock wave strongly influenced by the angle of attacks as well as the Mach number.

Combining Roe's finite volume scheme, the Spalart-Allmaras turbulent model, and unstructured grid made RANS solver developed successfully. In addition, developing the code for RANS solver simultaneously develops the Euler solver. When viscous term was set up to zero, the RANS solver became Euler solver. Hence, the present work developed both the RANS and Euler solver.

## CONTENTS

<b>TITLE</b>	<b>I</b>
<b>DECLARATION</b>	<b>II</b>
<b>DEDICATION</b>	<b>III</b>
<b>ACKNOWLEDGEMENT</b>	<b>IV</b>
<b>ABSTRACT</b>	<b>V</b>
<b>CONTENTS</b>	<b>VI</b>
<b>LIST OF FIGURES</b>	<b>X</b>
<b>LIST OF TABLES</b>	<b>XV</b>
<b>SYMBOLS AND ABBREVIATIONS</b>	<b>XVI</b>
<b>CHAPTER 1 INTRODUCTION</b>	<b>1</b>
1.1 Background Study	1
1.2 Problem Statement	4
1.3 Research Objectives	5
1.4 Research Goals	5
1.5 Scope of Research Study	5
1.6 Contribution to Knowledge	6
1.7 Thesis Outline	7
<b>CHAPTER 2 LITERATURE REVIEW</b>	<b>8</b>
2.1 Introduction	8
2.2 Literature Reviews of Approaches of Solving Aerodynamics Problem	9
2.3 Computational Fluid Dynamics CFD	10
2.4 Computational Fluid Dynamics CFD Codes	11
2.5 Equation of Fluid Dynamic	13

2.6	Discretization Methods	16
2.6.1	Finite Difference Method FDM	16
2.6.2	Finite Element Method FEM	17
2.6.3	Finite Volume Method FVM	18
2.7	Turbulence Modeling	20
2.7.1	The Boussinesq's eddy viscosity model	21
2.7.2	SST k- $\omega$ two-equation model	22
2.7.3	k- $\epsilon$ turbulence model	23
2.7.4	Spalart-Allmaras turbulence model	24
2.8	The Validation of The Present Code	26
2.9	Summary	28

### CHAPTER 3 METHODOLOGY

3.1	Introduction	30
3.2	Numerical Grid Generation	31
3.2.1	Algebraic grid generation	41
3.2.2	Elliptic grid generation	43
3.3	Governing equations of fluid motion	45
3.3.1	Reynolds averaged Navier-Stokes equations	47
3.4	Turbulence model	49
3.4.1	Governing Equations of One-Equation Turbulence Models	50
3.4.2	Spalart-Allmaras One-Equation Turbulence Model:	54
3.4.3	Nondimensional form:	53
3.4.4	Spalart-Allmaras turbulence model in computational space:	54
3.4.5	Spalart-Allmaras turbulence model in integral form	57
3.5	The Finite Volume method for solving 2-D Flow Problems	59

3.3.1	Cell –Centered scheme	62
3.5.2	Time Stepping	65
3.5.3	Boundary Conditions	65
3.6	Fluent Software for Airfoils aerodynamics Analysis.	71
<b>CHAPTER 4</b>	<b>RESULTS AND DISCUSSIONS</b>	<b>74</b>
4.1	Introduction	74
4.2	Comparison Result of the Case of Inviscid Compressible Flow Problem.	75
4.2.1	Case Study of Flow Pass through Airfoil NACA0012 for Inviscid Flow	75
4.3	Aerodynamic Analysis of Viscous Flow	86
4.3.1	RANS solver to the study case of flow passing through Airfoil NACA0012	87
4.3.2	RANS Solver to the study case of flow passing through Airfoil RAE 2822	95
4.4	The Application of the developed computer code for case of flow analysis over an airfoil NACA 4415.	98
4.5	The Application of the developed computer code for case of flow analysys Over Supercritical Airfoil 26a.	103
4.6	Summary	106
<b>CHAPTER 5</b>	<b>CONCLUSION AND RECOMMENDATIONS</b>	<b>108</b>
5.1	Conclusion	108
5.2	Recommendations.	109
<b>REFERENCES</b>		<b>110</b>
<b>APPENDIX</b>		<b>117</b>
<b>PUBLICATIONS</b>		<b>138</b>



## LIST OF FIGURES

2.1	Approaches of solving aerodynamics problems (Blazek, 2005, Anderson, 1995)	10
2.2	The basic ingredient of Computational Fluid Dynamics CFD codes.	12
2.3	Discretizations on triangular meshes. Shaded area is the control volume for the solution at location $O$ . (a) Cell centered scheme, (b) vertex scheme	19
2.4	Levels for turbulence modeling	22
2.5	Model validation with experimental Data	26
2.6	Airfoil pressure coefficient $C_p$ distribution along an airfoil NACA 0012	27
2.7	Variation of pressure coefficient $C_p$ for $M_\infty=0.15$ and $\alpha=0^\circ, 10^\circ$ , and $15^\circ$ of Gregory & O'Reilly, NASA R&M 3726, Jan1970	27
2.8	Plot of the pressure coefficients on the RAE2822 airfoil $M_\infty=0.729$ and $\alpha=2.31^\circ$ by Cook et al., 1979	28
3.1	The transform of physical domain to computational domain	31
3.2	Two-dimensional structured and unstructured grids	32
3.3	Flow pass through an airfoil	33
3.4	The C-topology for the flow past through airfoil and the boundary condition.	34
3.5	The mesh flow domain in structured grid representation.	35
3.6	The mesh flow domain in unstructured grid representation	35
3.7	The flow chart of the grid generation algorithm in the case of two-dimensional Structured grid.	36
3.8	The flow chart of the grid generation algorithm in the case of two-dimensional Unstructured grid	37

3.9	Grid size for a case of viscous flow around airfoil, Total cells is 65536 cells for the viscous cases and C-Grid topology.	38
3.10	A structured and Unstructured grid over an airfoil.	39
3.11	Unstructure Grid of NACA0012 for viscous flow problem	39
3.12	Unstructure Grid of NACA 4415 for viscous flow problem	40
3.13	Effects of the control functions at $\eta = \text{const}$ , boundary. P controls the skewness and Q the spacing of the grid.	44
3.14	Turbulent velocity fluctuations $V'$ and statistical mean value $\bar{V}$ .	47
3.15	A general unstructured 2-D grid generation	61
3.16	Numbering of nodes and face vector of: a triangular element, C denotes the center of the element.	61
3.17	Control volume of a cell-centred scheme	62
3.18	Flow chart of cell centered scheme FVM computer code in case of 2-D unstructured grid generation.	68
3.19	Flow chart of the solver's iterations	69
3.20	The pseudo code for FVM Cell Centered Roe's solver with Spalart-Allmaras turbulent model	70
3.21	Create Farfield Boundary around an airfoil, c is the chord length of the airfoil	71
3.22	Mesh generation around an airfoil by use of GAMBIT software	72
3.23	Fluent software permits the selection of appropriate model for the problem under study	72
3.24	Setting up solver properties and plot for Fluent Software	73
4.1.a	The mesh flow domain over airfoil NACA 0012, mesh size is $128 \times 64 \times 2$	76
4.1.b	The mesh flow domain over airfoil NACA 0012	76
4.2.a	Comparison result of pressure coefficient distribution $C_p$ along the airfoil surfaces for the case Airfoil NACA 0012 at Angle of Attack $\alpha = 0^\circ$ .	77
4.2.b	Comparison result of $C_p$ along the airfoil surfaces for the case Airfoil NACA 0012 at Angle of Attack $\alpha = 10^\circ$	78

4.2.c	Comparison result of pressure coefficient distribution $C_p$ along the airfoil surfaces for the case Airfoil NACA 0012 at Angle of Attack $\alpha = 15^\circ$	78
4.3.a	Mach Contour, NACA 0012 airfoil at $\alpha=0^\circ$ by using the present Eluer inviscid code.	79
4.3.b	Mach Contour, NACA 0012 airfoil at $\alpha=0^\circ$ by using the Fluent software	80
4.3.c	Mach Contour, NACA 0012 airfoil at $\alpha=15^\circ$ by using the present Euler inviscid code	80
4.3.d	Mach Contour, NACA 0012 airfoil at $\alpha=15^\circ$ by using the Fluent software	81
4.4	Distribution of Mach Number on NACA0012 at $M_\infty=0.7$ and $\alpha = 1.49^\circ$ by using Euler code	82
4.5.a	Mach Contour, NACA 0012 airfoil at $M_\infty=0.7$ and $\alpha=1.49^\circ$ by using Euler code	82
4.5.b	Mach Contour, NACA 0012 airfoil at $M_\infty=0.7$ and $\alpha=1.49^\circ$ by using inviscid Fluent software.	83
4.6.a	The mesh flow domain over airfoil RAE 2822 with the mesh size is 16384 elements	84
4.6.b	Close up view of mesh flow domain over the RAE 2822 airfoil	84
4.6.c	Mach Contour, RAE2822 airfoil at $M=0.729$ and $\alpha = 2.31^\circ$ by using inviscid code	85
4.6.d	Mach Contour, RAE2822 airfoil at $M=0.729$ and $\alpha = 2.31^\circ$ by using Fluent software	85
4.7	$C_p$ Distribution on RAE2822 $M=0.729$ and $\alpha = 2.31^\circ$ by using inviscid code	86
4.8.a	The mesh flow domain over airfoil NACA 0012, mesh size is 65536 for the case of viscous flow problem	87
4.8.b	Close up view of mesh flow domain over the NACA 0012 airfoil for the case of viscous flow problem	88

4.9.a	Comparison pressure distribution $C_p$ on NACA 0012 at Mach Number $M=0.15$ for different angle of attack $\alpha = 0^\circ$ for the case of viscous flow problem	89
4.9.b	Comparison pressure distribution $C_p$ on NACA 0012 at Mach Number $M=0.15$ for different angle of attack $\alpha = 10^\circ$ for the case of viscous flow problem	89
4.9.c	Comparison pressure distribution $C_p$ on NACA 0012 at Mach Number $M=0.15$ for different angle of attack $\alpha = 15^\circ$ for the case of viscous flow problem	90
4.10.a	Mach number distribution over the flow domain for NACA0012 airfoil at $M=0.15$ and at angle of attack $\alpha = 0^\circ$ by using RANS code	90
4.10.b:	Mach number distribution over the flow domain for NACA0012 airfoil at $M=0.15$ and at angle of attack $\alpha = 0^\circ$ by using Fluent software	91
4.10.c:	Mach number distribution over the flow domain for NACA0012 airfoil at $M=0.15$ and at angle of attack $\alpha = 10^\circ$ by using RANS code	91
4.10.d:	Mach number distribution over the flow domain for NACA0012 airfoil at $M=0.15$ and at angle of attack $\alpha = 10^\circ$ by using Fluent software	92
4.10.e:	Mach number distribution over the flow domain for NACA0012 airfoil at $M=0.15$ and at angle of attack $\alpha = 15^\circ$ by using RANS code	92
4.10f:	Mach number distributions over the flow domain for NACA0012 airfoil at $M=0.15$ and at angle of attack $\alpha = 15^\circ$ by using Fluent software	93
4.11.	A Comparison pressure distribution $C_p$ on airfoil NACA0012 at $M_\infty=0.7$ and $\alpha = 1.49^\circ$ by using RANS code	94
4.12.a:	Mach number contour in the flow field of airfoil NACA0012 at $M_\infty=0.7$ and $\alpha = 1.49^\circ$ by using RANS code.	94
4.12.b:	Mach number contour in the flow field of airfoil NACA0012 at $M_\infty=0.7$ and $\alpha = 1.49^\circ$ by using RANS code.	95
4.13:	Close-up view of the mesh flow domain over airfoil RAE 2822 with 65536 elements of mesh size	96
4.14	$C_p$ Distribution on RAE 2822 $M=0.729$ and $\alpha = 2.31^\circ$ for viscous flow	97

4.15.a: Mach Contour, RAE2822 airfoil at $M=0.729$ and $\alpha = 2.31^\circ$ by using RANS code	97
4.15.b: Mach Contour, RAE2822 airfoil at $M=0.729$ and $\alpha = 2.31^\circ$ by using Fluent software code	98
4.16 Geometry airfoil NACA 4415	99
4.17 The Mesh Flow domain over NACA 4412 for inviscid flow solution	100
4.18 The Mesh Flow domain over NACA 4412 for viscous flow solution	100
4.19: Distribution of Mach Number on NACA4415 $M = 0.3$ and $\alpha = 5^\circ$ and $10^\circ$	101
4.20: Distribution of pressure coefficient on NACA4415 $M = 0.5$ and $\alpha = 3^\circ$ and $5^\circ$	102
4.21: Geometry Supercritical Airfoil 26a	103
4.22: The Mesh Flow domain over Supercritical Airfoil 26a	103
4.23: Distribution of pressure coefficient for supercritical airfoil 26a, at $M_\infty = 0.74$ and $\alpha = 2^\circ, 4^\circ, 5^\circ$ and $6^\circ$	104
4.24: Distribution of pressure coefficient for supercritical airfoil 26a, at $\alpha = 3.5^\circ$ and $M_\infty = 0.65, 0.7, 0.75$ and $0.8$	105



## LIST OF TABLES

2.1	some of the relative properties of the three methods FVM, FDM, and FEM	20
3.1	Date arrangement of the unstructured grid generation	41
4.1	Summary of Inviscid Flow Test Cases	75
4.2	Data Set Flow conditions in the aerodynamics analysis over Airfoil NACA 4415	99
4.3	position of shock wave on the supercritical airfoil 26a, at $M_{\infty} = 0.74$ and $\alpha = 2^{\circ}, 4^{\circ}, 5^{\circ}$ and $6^{\circ}$	104
4.4	position of shock wave on the supercritical airfoil 26a, at $\alpha = 3.5^{\circ}$ and $M_{\infty} = 0.65, 0.7, 0.75$ and $0.8$	106



PTFAUTHM  
PERPUSTAKAAN TUNKU TUN AMINAH

## LIST OF SYMBOLS AND ABBREVIATIONS

CFD	Computational Fluid Dynamic
FVM	Finite Volume method
FEM	Finite Element method
FDM	Finite Difference method
2D	Two-Dimensional
PDEs	partial differential equations
DNS	Direct Numerical Simulation
IVP	Initial Value Problem
IBVP	Initial Boundary Value Problem
CDC	Control Data Cooperation
RANS	Reynolds Averaged Navier-Stokes
N	Number of elements
TVD	Total Variation Diminishing
NACA	The National Advisory Committee for Aeronautics
NASA	National Aeronautics and Space Administration

$U(x)$	Scalar function
$M$	Mach Number
$\partial$	Differentiation
$t$	Time
$\vec{W}$	The Vector Of Conserved Variable
$\Omega$	Control Volume
$\vec{F}_c$	Convective Flux
$\vec{F}_v$	The Vector Of Viscous Fluxes
$S$	Cross-Section Area
$\vec{Q}$	Source Term
$\rho$	Static Density
$u$	Velocity Component in x-Direction
$v$	Velocity Component in y-Direction
$w$	Velocity Component in z-Direction
$E$	Total Energy
$V$	Contravariant Velocity
$H$	Total Enthalpy
$e$	Internal Energy
$P$	Pressure
$P_r$	Prandtl Number



$k$	Turbulent Kinetic Energy
$Re$	Reynolds Number
$S_p$	The Generation Source Term
$S_D$	The Destruction Source Term
$D$	The Diffusion Term
$d$	Distance
$\tau_{tij}$	Reynolds Stress
$\mu_t$	The Eddy Viscosity
$S_{ij} \equiv (\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i})/2$	The Average Velocity Strain Rate Tensor
$\lambda$	Stokes' Hypothesis
$N_F$	The Number Of Faces
$d_t$	The Distance From Field Point
$w_t$	The Wall Vorticity At The Trip
$\Delta_q$	The Difference Between The Velocities
$\vec{R}_I$	The Residual.
$\tilde{\lambda}_i$	The Eigenvalue Of The Jacobian Matrix
$\vec{A}$	Jacobian Matrix
$\alpha_k$	The Stage Coefficients
$R$	Riemann Invariants

$C_{b1} \dots$ etc	Empirical Constant In The Turbulence Model
$c$	Chord Of An Airfoil
$C_p$	Pressure Coefficient
$C_l$	Lift Coefficient
$C_d$	Drag Coefficient
$\delta$	Thickness Of The Shear Layer
$\delta^*$	Displacement Thickness
$\nu$	Kinematic Molecular Viscosity
$f_{v2} \dots$ etc	Empirical Constant In The Turbulence Model
$g, r, \tilde{S}$	Intermediate variables
$H \equiv \delta^*/\theta$	Shape Factor
$S$	Measure of the deformation tensor
$u_\tau$	Friction Velocity
$u_i$	Fluctuating velocity components
$U$	Mean Velocity in x direction
$k$	Karman Constant Taken as 0.41
$\nu_t$	Kinematic Turbulent , or eddy viscosity
$\tilde{v}$	Working Variable of The Turbulent Model
$\sigma$	Turbulent Prandtl Number
$\theta$	Momentum Thickness
$\tau$	Shear Stress

## CHAPTER 1

### INTRODUCTION

#### 1.1 Background of Study

The publication of the equation of fluid dynamic flow with friction called for the “Navier-Stokes equation” in 1840s. This scenario, which can be considered as the governing equation of fluid motion to allow for describing all flow phenomena to appear in the real fluid flow, has opened up the door for scientists to go deeply in the field of fluid dynamics as well as in aerodynamic. The advent of high-speed computers in the last 30 years dramatically changed the nature of the application of the basic principles of theoretical fluid mechanics and heat transfer in solving engineering problems. Along with the development of conventional methods such as the analytical and experimental methods, the development of the third method called Computational Fluid Dynamics (CFD) has grown rapidly. This method has been used for solving various engineering designs ranging from the problems faced in the automotive design to the problems found in the aerospace flying vehicle design. The CFD capability has contributed significantly in reducing the design cost and shortening the required time for completing design process.

Although the CFD capability has been improved significantly, CFD designer must not leave the necessary experimental work in the design process. This is because the experiment will continuously play a very important role in the design process for design validation purposes.

As the governing equation of fluid motion in the form is a nonlinear partial differential equation, in which there is no analytical solution, the manner on how to solve the flow problem needs a numerical approach. There have been various numerical methods introduced for solving the governing equation of fluid motion. The growth in the popularity of numerical methods as a tool for solving the flow problem faced in the aircraft industries is due to having more available computing power. The speed and computer memory capacity have increased exponentially, especially due to the presence of super computer since 1964. In the earlier time, the first super computer named CDC 6600 produced by Control Data Cooperation has speed at  $3.0 \times 10^6$  FLOPS with CPU memory at  $128 \times 10^3$  bytes. After 50 years of computer technology development, the current speed of supercomputer capability is around  $93.0 \times 10^{15}$  FLOPS and the computer memory is around  $13.102 \times 10^{13}$  Bytes. The specifications are provided by a supercomputer named the Sunway TaihuLight located at the National Supercomputing Wuxi, China (Fu. H and *et al.*, 2016 ) (A. Petitet and *et al.*, 2016) The first supercomputer had contributed significantly in the aircraft design activities, when the Boeing aircraft manufacturer designed the well-known aircrafts; Boeing 737 and 747 (Marshall, and Jameson, 2010).

The availability CDC 6600 allows the aerodynamic engineer of the aircraft company to evaluate the aerodynamic characteristics at their full aircraft configuration by using a panel method. This method is conducted by assuming that the Navier-Stokes equation can be simplified by ignoring the viscous effects and flow behaving as an irrotational flow. As the progress of computer technology develops better, the manner on how to solve the flow problem is changed. The attempt of aerodynamic to solve the flow problem is done by using the equation closer to Navier-Stokes. The flow problem is treated with no viscous effect, only with the possibility that the flow may behave as a compressible and rotational flow. These flow conditions can be used to reduce the Navier-Stokes equation to a new governing equation of fluid motion called the Euler equations. The solution of this equation allows to capture the presence of shock wave and vortex flow phenomena, which can be found if an aerodynamic designer solves the flow problem passing through a delta wing model. Various methods have been developed through various studies for solving the Euler equation such as the Flux

Splitting Method (Klaus A.Hofmann and Steve T.Chiang., 2000), Maccormack Scheme (Pletcher, R.H., and Tannehill., 2012), Beam–Warming Scheme (Beam, R.M., Warming, 1982) , and TVD scheme(Yee, 1985). The Euler equation has been used as a model of the governing equation for solving the flow passing through aircraft configuration starting in the 1980s. Then, Boeing has started to apply RANS since 1990 in solving the problem faced in their aircraft design activities (Johnson, and Tinoco., 2005).

Basically, there are various problems in solving the flow problem numerically, whether the problem has to be solved through Euler equation or RANS equation. The first problem is in relation with the discretization of the flow or mesh flow domains. In the flow passing through a simple geometry, the mesh flow domain may be easily defined by a single block mesh. The associated numerical solution can be easily transformed into the computer code. However , when the flow problem related with a flow passing through a complex geometry such as flow past through multi component airfoils or multi surface such as flow passing through a complete full aircraft configuration or missile, the meshing of the flow domain becomes difficult and one must use a multi block mesh approach. As a result, the associated computer code in implementing the numerical approach whether using TVD scheme or MacCormack or others becomes more complicated. The complexity in the way to solve numerically is increased if the governing equation of fluid motion that must be solved is RANS. The complexity appears due to a finer grid requirement. For the same flow problem, using the finite volume method for solving the Euler equation through the Flow domains needs to be divided into  $N$  number elements, so that when a designer solves through the RANS equation may needs at least  $16 \times N$  number of elements. Besides that, and has to provide a turbulence modeling in order to make the RANS Equation solvable.

The present work focused on the development of computer code for solving the flow problems based on the RANS equation. This equation was solved by using Roe's finite volume Scheme (J.Blazek, 2008) with Turbulence modeling according to the Spalart-Ammaras model (Spalart, P. R. and Allmaras, S. R., 1992). The meshing flow is defined according to the unstructured grid model which can be obtained by combining algebraic grid and elliptic grid generator. The developed computer code was applied to

the case of flow passing through airfoil NACA 0012 and RAE 2822 for various flow conditions, ranging from the low to high subsonic Mach numbers (Agard, 1992). These two airfoils were chosen since their aerodynamic characteristics in terms of pressure distribution resulted from the experiments was available. Therefore, through comparison results, the code validation was carried out. In addition to this, the comparisons were also conducted by comparing the result obtained through running the ANSYS-FLUENT software (Stolarski, 2011). The comparison between results provided by FLUENT as well the experiment result indicated that the present codes are in good agreement whether the flow problem under investigation is at the low or high subsonic flow condition. The application of the developed computer code over the flow passing through airfoil NACA 4415 confirmed that there was a significant difference between the viscous and inviscid solution as the Mach number and angle of attack of the flow under investigated were increasing.

## **1.2 Problem Statement.**

Numerical methods for solving problems of aerodynamic are actively developed and widely used in various industries. The growth in the popularity of the numerical methods is largely due to modern supercomputers. It is true that the most accurate result as a complete result in providing all flow phenomena may appear in the flow field and solve the Navier-Stokes equation directly. This method is known as the Direct Numerical Simulation (DNS) (Jasak, H., 2009). Unfortunately, the availability of computing power and computer memory in the current computer technology is still insufficient to fulfill DNS requirement, especially in the case of the flow problem related to practical engineering applications. As a result, most efforts in solving the flow problems are still based on solving the governing equations of fluid motion such as RANS. However, no analytic solution for this type of equation is available and therefore a numerical approach is required. Unlike the flow problems which are solved through the Euler equation as its governing equation of fluid motion, this flow model made the corresponding solver (Euler Solver) in providing an accurate solution, which depends on the manner mesh of flow domain is defined and the numerical scheme in use. On the

other hand, the RANS solver depends on the mesh and numerical schemes, which also depend on the types of turbulent model in use. Hence, combining these three ingredients (mesh, numerical scheme, and turbulent model) may correctly lead to producing an accurate RANS solver.

### **1.3 Research Objectives.**

The aim of the research work is to develop a CFD code for two dimensional compressible flow, in order to achieve this aim, the following objectives have to be accomplished:

1. To develop an unstructured C-Grid generation code for meshing flow domain over an airfoil.
2. To develop computer code for 2D Euler solver based on Roe's Cell Centered Finite Volume method. .
3. To develop the extension of above 2D Euler solver as 2D Reynold Averaged Navier Stokes equations with Spalart–Allmaras turbulent model.
4. To validate the aerodynamic properties through developed CFD code with the available experimental results and results produced by Fluent software.

### **1.4 Research Goals**

End of this research will produce an integrated computer code between numerical code designed for creating mesh systems and CFD solver dedicated for solving two dimensional aerodynamics problems as viscous or inviscid flow problem for any given flow condition from a low to high subsonic Mach number for different angle of attacks.

### **1.5 Scope of Research Study.**

To achieve such objective as mentioned above, sequential research work need to be developing step by step started from:

1. Study on the implementation of the Finite Volume Method for a simple flow model ( Quasi One Dimensional Compressible Flow ). The result of this study applied to the case of flow past through Nozzle presented in the appendix-A.
2. Study on the manner how meshing flow domain past surrounding airfoil based on the C-topology developed. .
3. Understanding the 2D Euler solver based on Roe's cell center finite volume method applied to the case of flow past through an airfoil.
4. Understanding the way how to solve a 2D Reynold Averaged Navier Stokes Solver with Spalart – Allmaras turbulence modeling.
5. Finding the experimental result which the available data can be used for a validation purposes beside the use of Fluent software.

### **1.6 Contribution to knowledge**

The present work provides a new CFD code which allows the aerodynamic designers to carry out the aerodynamic analysis of the two-dimensional flow through airfoil with viscous effect as part of their flow solution. The code developed by using the second level of the governing fluid equations is named as RANS. Currently, most of the aircraft manufacturer industries use this type of equation to solve their flow problem in their aircraft design activities. For instance, another approach newly introduced called the DNS scheme gives a more complete and accurate solution. However, this approach can only be used in the aircraft design process when computer power is highly demanded. The present work combines Roe's finite volume scheme as a numerical scheme for solving the governing equation, Spalart-Allmaras as its turbulent model, and unstructured grid scheme for meshing flow domain to become an integrated solver for solving a turbulent flow past through any airfoil types. The developed solver can be used easier than the CFD designers using the Fluent software, since users are only required to input the airfoil geometry and the free stream flow condition (Angle of Attack, Mach number, and Reynolds number) in a simple manner. The developed code will produce the result of pressure, density and Mach number distribution over the flow field domain, similar with the result provided by the Fluent software.



## REFERENCES

- A. Petitet, R. C. Whaley, J. Dongarra, A. Cleary (2016). "*HPL – A Portable Implementation of the High-Performance Linpack Benchmark for Distributed-Memory Computers*". ICL – UTK Computer Science Department.
- AGARD (1992), *Engineering Methods in Aerodynamic Analysis and Design of Aircraft*. Advisory Group for Aerospace Research & Development.
- Ainslie D. French (2004) *Solution of the Euler equations on Cartesian grids*. Elsevier Science.
- Asbhy D.L and Dudley M.(1988) *Development of an Advanced Low Order Panel Method* . NASA TM - 101024.
- A.Jameson , D. Mavriplis. (1985) *Finite Volume Solution of the Two-Dimensional Euler Equations on a Regular Triangular Mesh* AIAA 23rd Aerospace Sciences Meeting
- Andersson, B., Andersson, R., Håkansson, L., Mortensen, M., Sudiyo, R., & Van Wachem, B. (2011). "*Computational fluid dynamics for engineers*". Cambridge University Press.
- Anderson Jr, J. D. (2010). "*Fundamentals of aerodynamics*." Tata McGraw-Hill Education.
- Bayliss, A., & Turkel, E. (1982). Far field boundary conditions for compressible flows. *Journal of Computational Physics*, 48(2), 182-199.
- Beam, R.M., Warming, R.F. and Yee, H.C., (1982). "*Stability analysis of numerical boundary conditions and implicit difference approximations for hyperbolic equations*". *Journal of Computational Physics*, 48(2), pp.200-222.
- Boris Diskin, James L. Thomas, Eric J. Nielsen, Hiroaki Nishikawa and Jeffery A. White (2010) *Comparison of Node-Centered and Cell-Centered Unstructured Finite-Volume Discretizations: Viscous Fluxes*. AIAA 10.2514/1.44940.
- Boris Andreianov, Mostafa Bendahmane, Ricardo Ruiz-Baier and Ricardo Ruiz-Baier, (2011) *Analysis Of A Finite Volume Method For A Cross-Diffusion Model In Population Dynamics*. World Scientific Vol. 21, No. 2. doi: 10.1142/S0218202511005064.

- Boris Diskin, James L. Thomas (2011). *Effects of mesh irregularities on accuracy of finite-volume discretization schemes*. AIAA
- Bandopadhyay, s., & kim, d. H. (2014). *Numerical Analysis of Unsteady Transonic Flow over Rocket Nose*. WSEAS Transactions on Fluid Mechanics,9.
- BANDOPADHYAY, S., & KIM, D. H. (2014). *Numerical Analysis of Unsteady Transonic Flow over Rocket Nose*. WSEAS Transactions on Fluid Mechanics,9.
- Blazek, J., (2015). “*Computational fluid dynamics: principles and applications*”. Butterworth-Heinemann.
- Başkut, E., & Akgül, A. (2011). *Development of a coupling procedure for static aeroelastic analyses*. Scientific Technical Review, 61(3-4), 39-48.
- Carlson, J. R. (2011). *Inflow/outflow boundary conditions with application to FUN3D*. NASA-TM-2011-217181
- Cebeci T. et.,al. (2005) *Computational Fluid Dynamics For Engineers* , Horizon Publishing, California, USA.
- Charles Hirsch (1994) , *Numerical Computation of internal and external flow*. John Wiley & Sons.
- Chung, T. J. (2010). *Computational Fluid Dynamics*. 2<sup>nd</sup> ED. Cambridge University Press.
- Chi- Wang Shu. (2001), *High Order Finite Difference and Finite Volume WENO Schemes and Discontinuous Galerkin Methods for CFD*. NASA/CR-2001-210865 ICASE Report No. 2001-11.
- Corrigan, A., Camelli, F. F., Löhner, R., & Wallin, J. (2011). *Running unstructured grid-based CFD solvers on modern graphics hardware*. International Journal for Numerical Methods in Fluids, 66(2), 221-229.
- Cook, P.H., M.A. McDonald, M.C.P. Firmin, "Aerofoil RAE 2822 - Pressure Distributions, and Boundary Layer and Wake Measurements," Experimental Data Base for Computer Program Assessment, AGARD Report AR 138, 1979.
- Corrigan, A., Camelli, F. F., Löhner, R., & Wallin, J. (2011). *Running unstructured grid-based CFD solvers on modern graphics hardware*. International Journal for Numerical Methods in Fluids, 66(2), 221-229.
- D. M. Causon, C. G. Mingham and Dr. L. Qian, (2011), *Introductory Finite Volume Methods for PDEs*.

- Dacles-Mariani, J., Zilliac, G. G., Chow, J. S. and Bradshaw, P. (1995.)  
*"Numerical/Experimental Study of a Wingtip Vortex in the Near Field"*, AIAA Journal,  
 33(9), pp. 1561-1568,
- Eleuterio F. Toro. (2009), *Riemann Solvers and Numerical Methods for Fluid Dynamics*.  
 Springer-Verlag Berlin Heidelberg.
- Enrico Bertolazzi (2007) *on vertex reconstructions for cell-centered finite volume  
 approximations of 2d anisotropic diffusion problems*. World Scientific.
- Edisson Sávio de Góes Maciel. (2007) *Comparison Among Structured First Order Algorithms in  
 the Solution of the Euler Equations in Two- Dimensions*. ABCM October- Vol. XXIX,  
 No. 4 / 421.
- Elkamel, H.T.M., Alakashi, A.M. and Basuno, B., (2013). "*Comparison Results between  
 MacCormack Scheme and Steger Warming Scheme for the Case of Supersonic Flow Pass  
 Through Divergent Nozzle*". In Applied Mechanics and Materials (Vol. 315, pp. 268-  
 272). Trans Tech Publications. Vancouver
- Elkamel, M., Taher, H., Basuno, B. and Asmuin, N., (2013). "*Investigation the behaviour of cell  
 centered finite volume scheme to the convergent divergent nozzle flow problems*".
- Enrico Bertolazzi And Gianmarco Manzini, (2004) *A Cell-Centered Second-Order Accurate  
 Finite Volume Method For Convection Diffusion Problems On Unstructured Meshes*.  
 World Scientific , Vol. 14, No. 8 (2004) 1235-1260.
- Eppler , (2000) *Airfoil Design And Analysis Code*. Richard Eppler, c.2000.
- Timothy Barth and Mario Ohlberger. (2004) *Finite volume methods: foundation and analysis*.  
 John Wiley & Sons, Ltd.
- Edisson Sávio De Góes Maciel.(2012) *MGD Application to a Blunt Body in Two-Dimensions*.  
 Wseas transactions on fluid mechanics, Issue 1, Volume 7,
- Fu, H., Liao, J., Yang, J., Wang, L., Song, Z., Huang, X., Yang, C., Xue, W., Liu, F., Qiao, F.  
 and Zhao, W, (2016). "*The Sunway TaihuLight supercomputer: system and applications*".  
 Science China Information Sciences, 59(7), p.072001.
- Frank Dzaak (1995) *Solving 2D Euler equations on a multi-processor network*. Elsevier Science.
- Frink, Neal T., Magnus Tormalm, and Stefan Schmidt. (2011). *Unstructured CFD Aerodynamic  
 Analysis of a Generic UCAV Configuration*. NATO RTO-MP-AVT-170 PAPER NBR –  
 25

- Gang Wang \*, Axel Schwöppe †, and Ralf Heinrich. (2010) *Comparison And Evaluation Of Cell-Centered And Cell-Vertex Discretization In The Unstructured Tau-Code For Turbulent Viscous Flows*. ECCOMAS CFD 2010 Lisbon, Portugal, 14–17.
- Georg May and Antony Jameson. (2006), *A Spectral Difference Method for the Euler and Navier-Stokes Equations on Unstructured Meshes* . AIAA 2006-304.
- Gordon, W.N.; Hall, C.A. (1973): “*Construction of Curvilinear Coordinate Systems and Application to Mesh Generation*”. Int. J. Num. Methods in Engineering, pp. 461-477.
- H K Versteeg and W Malalasekera (2007), *An Introduction to Computational Fluid Dynamics the finite volume method*, Pearson Education Limited.
- H. Versteeg (Author), W. Malalasekera. (2007). *An Introduction to Computational Fluid Dynamics: The Finite Volume Method (2nd Edition)* Paperback
- Mahawadiwar, H.V., Dhopte, V.D., Thakare, P.S. and Askhedkar, D.R., (2012). “*Cfd Analysis Of Wind Turbine blade*”. International Journal of Engineering Research and Applications, 2(3), pp.3188-3194.
- Hirsch, C. (2007). “*Numerical computation of internal and external flows: The fundamentals of computational fluid dynamics*”. Butterworth-Heinemann.
- Jameson and D. Mavriplis, (1985) ‘*Finite volume solution of the two-dimensional Euler equations on a regular triangular mesh*’, AIAA Paper 85-0435.
- Jameson, A. (1983). *Solutions of the Euler Equations for Two-Dimensional Transonic Flow by a Multigrid Method*. MAE report No.1613,
- Jameson, W. Schmidt, and E. Turkel. (1981). “*Numerical simulation of the euler equations by finite volume methods using runge-kutta time stepping schemes*”. AIAA Paper 81-125.
- Jasak, H., (2009). “*Turbulence Modeling for CFD*”. NUMAP-FOAM Summer School, Zagreb, pp.2-15.
- J. Blazek , (2008) *Computational Fluid Dynamics: Principles and Applications*. Elsevier science.
- Jonathan A. Parker (2007) *Euler equations*. Princeton, NJ 08544-1013.
- Jiequan Li (2002) *Global Solution of an Initial-Value Problem for Two-Dimensional Compressible Euler Equations*. Elsevier Science.
- John D. Anderson (1995) *Computational Fluid Dynamics, The basics with applications*

- Jones, W. P., and Launder, B. E. (1972), *"The Prediction of Laminarization with a Two-Equation Model of Turbulence"*, International Journal of Heat and Mass Transfer, vol. 15, 1972, pp. 301-314.
- Johnson, F.T., Tinoco, E.N. and Yu, N.J., (2005). *"Thirty years of development and application of CFD at Boeing Commercial Airplanes"*, Seattle. Computers & Fluids, 34(10), pp.1115-1151.
- Klaus A.Hofmann and Steve T.Chiang.(2000). *Computational fluid dynamic*.
- Karman, S.L. Jr.(1995), *Splitflow: A 3D Unstructured Cartesian/Prismatic Grid CFD Code for Complex Geometries*, AIAA Paper 95-0343.
- Liseikin, V. D (2006). *A Computational Differential Geometry Approach to Grid Generation*. Springer
- Miranda L.R (1984). *Application of Computational Aerodynamics to Airplane Design*," AIAA *Journal of Aircraft*, Vol. 21, No. 6, pp 355-370.
- Marshall, D.D. and Jameson, K.K., (2010),. *"Overview of Recent Circulation Control Modeling Activities at Cal Poly"*. In 48th AIAA Aerospace Sciences Meeting (pp. 2010-0348).
- Marc Bernot , Alessio Figalli and Filippo Santambrogio .(2009) *Generalized solutions for the Euler equations in one and two dimensions*. Elsevier Science.
- Maskew B. (1987). *Program VSAERO Theory Document – A Computer Program for Calculating Non Linear Aerodynamics Characteristics of Arbitrary Configuration* ‘ NASA CR – 4023.
- Magnus Svard, Jing Gong and Jan Nordstrom. (2006) *An Accuracy Evaluation of Unstructured Node-Centered Finite Volume Methods*. NASA/CR-2006-214293 NIA Report No. 2005-04.
- Maciel, E. S. G. (2012) *MGD Application to a Blunt Body in Two-Dimensions*. WSEAS TRANSACTIONS ON FLUID MECHANICS 7: 23-47.
- Menter, F. R. (1993). *"Zonal Two Equation  $k-\omega$  Turbulence Models for Aerodynamic Flows"*, AIAA Paper 93-2906.
- Menter, F. R. (1994), *"Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications"*, AIAA Journal, vol. 32, no 8. pp. 1598-1605.
- Moukalled, F., Mangani, L. and Darwish, M., (2016). *"The finite volume method in computational fluid dynamics"*. Springer International Publishing Switzerland, 113, DOI 10.1007/978-3-319-16874-6-1



- Nasser Ashgriz & Javad Mostaghimi (2002) *An Introduction to Computational Fluid Dynamics*.  
 Nasser Ashgriz & Javad Mostaghimi. (1998). *An Introduction to Computational Fluid Dynamics*.  
 University of Toronto Toronto, Ontario
- Oktay Baysal and Victor R. Lessard, (1990), *An Overlapped Grid Method For Multigrid, Finite Volume/Difference Flow Solvers – MAGGIE*,. NASA Contractor Report 182008.
- Paul Schallhorn and Alok Majumdar (2012) *Implementation of Finite Volume based Navier Stokes Algorithm within General Purpose Flow Network Code*. 50th AIAA January, 2012 in Nashville, Tennessee.
- Pletcher, R.H., Tannehill, J.C. and Anderson, D., (2012). “*Computational fluid mechanics and heat transfer*”. CRC Press.
- Qiang ZHAO, Yina SHI, Guangwei YUAN, and Zhiwei DONG. (2010) *A Cell-centered Diffusion Finite Volume Scheme and it's Application to Magnetic Flux Compression Generators*. World Academy of Science, Engineering and Technology 46 2010.
- Richard L. Fearn (2008) *Airfoil Aerodynamics Using Panel Methods*. Wolfram Media, Inc.
- Randall J. Leveque , ( 2004 ), *Finite Volume Methods for Hyperbolic Problems*. Cambridge University Press.
- Richard T. Medan. (1981) , PAN AIR - *A Computer Program for Predicting Subsonic or Supersonic Linear Potential Flows About Arbitrary Configurations Using A Higher Order Panel Method*. NASA CR-3253.
- R.V. Chima, D.J. Arend, R.S. Castner, and J.W. Slater. (2010) *CFD Models of a Serpentine Inlet, Fan, and Nozzle*. NASA/TM—2010-216349.
- Robert E. Bartels. (2012) *Development, Verification and Use of Gust Modeling in the NASA Computational Fluid Dynamics Code FUN3D*. NASA/TM–2012-217771
- Stolarski, T., Nakasone, Y. and Yoshimoto, S., (2011). “*Engineering analysis with ANSYS software*”. Butterworth-Heinemann.
- Smith, A.M.O. and Cebeci, T. "Numerical solution of the turbulent boundary layer equations", Douglas aircraft division report DAC 33735. 1967.
- Stephen Nicholson. (1985). Development of a Finite Volume Time Marching Method. NASA/Turbomachinery Research Group Report No. Jn/85-3.
- Spalart, P. R. and Allmaras, S. R. (1992). "A One-Equation Turbulence Model for Aerodynamic Flows", AIAA 92-0439.
- Spalart, P. R. and Allmaras, S. R. (1994), "A One-Equation Turbulence Model for Aerodynamic Flows", La Recherche Aeronautique n 1, 5-21.

- Sorenson, R.L.( 1980): “A Computer Program to Generate Two-Dimensional Grids About Airfoils and Other Shapes by the Use of Poisson's Equation”. NASA TM-81198,
- Thomas H. Pulliam (1986), *Artificial Dissipation Models for the Euler Equations*. AIAA VOL. 24, NO. 12
- Thompson, J.F. (1987): “A General Three-Dimensional Elliptic Grid Generation System Based on a Composite Block Structure” *Comp. Meth. Appl. Mech. and Eng.*, 64 pp. 377-411.
- Tinoco E. N . (1990). *CFD Application on Complex Configuration: A Survey “*, in *Applied computational aerodynamics* , Editor Hanne P.A , Progress in Astronautics and Aeronautics Vol. 125.
- Taniguchi, T. (2006). *Automatic Mesh Generation for 3D FEM, Robust Delauny Triangulation*. Morikita Publishing.
- Thompson, J. F, Soni, B, Weatherill, N. P. (1999). *Handbook of Grid Generation*. CFC Press.
- Thompson, J. F, Warsi, Z.U.A, Mastin, C. W. (1985). *Numerical Grid Generation, Foundations and Applications*. North Holland
- Veluri, S. P., Roy, C. J., Choudhary, A., & Luke, E. A. (2009). *Finite volume diffusion operators for compressible CFD on unstructured grids*. In 19th AIAA computational fluid dynamics conference, AIAA Paper (Vol. 4141)
- XU, L. and LUO, H.X., (2008). “The Technology of Numerical Simulation Based on ANSYS ICEM CFD and CFX Software [JJ]”. *Mechanical Engineer*, 12, p.049.
- Yee, H.C., Warming, R.F. and Harten, A., (1985). “Implicit total variation diminishing (TVD) schemes for steady-state calculations”. *Journal of Computational Physics*, 57(3), pp.327-360.
- Yusop, Fatimah, Bambang Basuno, and Zamri Omar.(2013) "*Application of Modified 4th Order Runge Kutta-TVD Scheme for the Flow Past through Symmetrical Model*." *Applied Mechanics and Materials*. Vol. 315. Trans Tech Publications,.
- Yen Liu, , Marcel Vinokur, , Z.J. Wang,( 2006). “Spectral (finite) volume method for conservation laws on unstructured grids V: Extension to two-dimensional systems”. *Journal of Computational Physics*, Volume 212, Issue 2.
- Yu-Xin Ren and Yutao Sun (2006) *A multi-dimensional upwind scheme for solving Euler and Navier–Stokes equations*. Elsevier Inc.
- [Ref. <http://m-selig.ae.illinois.edu/ads/aircraft.html>]
- [<https://web.stanford.edu/class/me469b/handouts/turbulence.pdf>]
- [[www.southampton.ac.uk/~nwb/lectures/.../Turbulence\\_Notes\\_Fluent-v6.3.06.pdf](http://www.southampton.ac.uk/~nwb/lectures/.../Turbulence_Notes_Fluent-v6.3.06.pdf)]