

THE DEVELOPMENT OF DEDICATED CFD SOLVER FOR AERODYNAMIC
ANALYSIS ON INTERNAL AND EXTERNAL FLOW

ABOBAKER MOHAMMED ALAKASHI

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

Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia

JULY 2015

DEDICATION

I dedicate this work to my parents



ACKNOWLEDGEMENT

I would like to express my sincere thanks to my project supervisors, **Ir. Dr. Bambang Basuno** and **Dr. Safiah Binti Othman** for their efforts in helping me in the development of the project, through difficulties and in search for relevant literature are much appreciated.

Finally, I would like to thank my **Parents** in the first place for their constant support, encouragement and love all along my way. I wouldn't have been in this position if I hadn't had the support and best wishes from my parents. I owe everything to them.



PTTA UTHM
PERPUSTAKAAN TUNKU TUN AMINAH

Abstract

There are various engineering applications which required one has to solve an internal or an external flow problem. The design of flying vehicles such as aircraft, helicopter, missiles are examples which the ability to solve an external flow problem will determine the success in designing such flying vehicles. While for the case of flows pass through intake of the engine, compressor, turbine and nozzles, the ability to deal with internal flow problem is needed. Basically the external and internal flow problems are governed by the same equation, their difference may come from their difference in term of their boundary conditions. If both flow problems are under high Reynolds number condition and involves with the flow problems pass through a streamline body at relatively a low angle of attack, the viscous effects can be ignored. However if the flow is belong to class of a flow above a high subsonic flow, the suitable governing equation of fluid motion is a compressible Euler equations. Unfortunately, such a kind governing equation of fluid motion cannot be solved analytically; a numerical approach is required for solving it.

The present work focuses on the use of two types of Finite Volume methods. The first Finite volume method is a Cell-Centered Finite volume Scheme, while the second one is the Roe's Finite Volume Scheme. In parallel to the development of computer code based on finite volume schemes, the present works also carry out work on solving the governing equation of fluid motion by use of Finite Difference Approach. In this respect the present work focuses on the use of Steger Warming Scheme and MacCormack Scheme are applied to Nozzle flow problems.

To implement the Finite volume method as well as the Finite difference method in relating to the case of internal and external flow problems, the present work applies a combination of Algebraic grid generation and Elliptic Grid. In

addition to this, the developed computer codes are designed in such away to allow one solve the flow problem by use of structured grid or unstructured grid.

To validate the developed computer codes, their results compare with the result for the flow problem which the experimental results are available such as for the case airfoil NACA 0012. While for other test cases such the flow past rotor blades is compared with the result provided from solving the flow problem by use of Fluent software. Through comparison result with the Fluent software as well as the available experimental results indicate that the developed computer code are in a good agreement.



CONTENTS

CHAPTER	TITLE	PAGE
	TITLE	i
	DECLARATION	ii
	DEDICATION	iii
	ACKNOWLEDGEMENT	iv
	ABSTRACT	v
	CONTENTS	vii
	LIST OF FIGURES	xi
	LIST OF SYMBOLS AND ABBREVIATIONS	xvi
	LIST OF PUBLICATION	xviii
CHAPTER 1	INTRODUCTION	1
1.1	Background	1
1.2	Problem statement	4
1.3	Significance of the Study	4
1.4	Research objectives	5
1.5	Scope of Study	5
1.6	Contribution to knowledge	6
CHAPTER 2	LITERATURE REVIEW	9
2.1	Overview literature reviews	7
2.2	Summary	17

CHAPTER 3	RESEARCH METHODOLOGY	18
3.1	Introduction	18
3.2	General Background	18
3.2.1	Pre-processor	19
3.2.1.1	Computational Grid	19
3.2.2	Solver	21
3.2.2.1	solver Approaches	21
3.2.2.2	Governing Equations	23
3.2.2.3	Boundary Conditions	29
3.2.3	Postprocessor	30
3.3	Various Form of Numerical Approach For solving the Euler Equation	30
3.3.1	Complete System of the Euler Equations	31
3.4	The numerical Euler solver based on Finite Difference Approach	34
3.5	Implementation of MacCormack and Steger Warming Scheme	34
3.5.1	Conservative and non-conservative variables	34
3.5.2	MacCormack's Scheme	36
3.5.3	Steger-Warming flux vector Splitting	40
3.6	The numerical Euler solver based on Finite Volume Approach	42
3.7	Implementation of Cell-Centered and Roe's scheme	42
3.8	Governing Equation of fluid motion in The integral conservation laws	43
3.8.1	governing Equation of Flow pass Through Nozzle	43

3.8.2	Tow dimensional governing Equation of fluid motion in the form of the integral conservation laws based on Cell-centred scheme and Roe's scheme	47
3.9	Grid generation	54
3.10	Comparison between Finite Volume Method (FVM) Based On Inviscid and Viscous Flow with Experimental and Fluent Results	56
3.11	Comparison Between Structure And Unstructured Grid Based On Finite Volume Method Cell-Centered Approach	58
3.12	Comparison Between Cell-Centered Scheme And Roe's Scheme For Flow Pass Through A Turbine Blade Section	63
3.13	Implementation Of Cell -Centred Scheme For N Number Of Turbine Blade System	65
3.14	Summary	68
CHAPTER 4 RESULTS AND DISCUSSION		70
4.1	Solution of simple diverging nozzle problem by using MacCormak's and Steger-Warming	70
4.1.1	Comparison Result of Supersonic Divergent Nozzle	72
4.2	The Implementation of Cell-Centered Finite Volume Method over Five Nozzle Models in the case of Quasi One dimensional	75
4.3	Employment and verification the Finite Volume Method (FVM) Cell-centered scheme and Fluent software with experimental data in case two dimensional for NACA 0012 AIRFOILS	85
4.4	Comparison between Structure and unstructured grid based on Finite Volume	94



	Method (FVM)	
4.5	Comparison between the develop computer codes and Fluent software in flow pass through two dimensional Blazek blades	97
4.6	Solution of Cell- centred and Roe's approach based computer codes	99
4.7	The implementation of Cell-centered scheme Code to case of For Transonic Flow Pass Through array of Turbine Stator Blades	108
4.8	Summary	121

CHAPTER 5 CONCLUSION AND RECOMMENDATION 122

5.1	Conclusion	122
5.2	Recommendation	123

REFERENCE 124

Appendix		133
-----------------	--	-----



PT TA UTHM
PERPUSTAKAAN TUNKU TUN AMINAH

LIST OF FIGURES

3.1	Rectangular grid segment	37
3.2	Schematic of the grid for time marching	37
3.3	Control volume for the 1-D Euler solver	44
3.4	Control volume of a cell-centred scheme (in one dimension)	45
3.5	Cells in the finite-volume approach	49
3.6	Effects of the control functions P controls the skewness and Q the spacing	56
3.7	Cross section of NACA 0012 AIRFOILS	58
3.8	Grid generation of NACA 0012 AIRFOILS cross section	58
3.9	Cross section for structured grid generation of Bump Channel	61
3.10	Cross section for Unstructured grid generation of Bump channel	61
3.11.a	Cross section for structured grid generation of Blazek turbine blades	62
3.11.b	Cross section for unstructured grid generation of Blazek turbine blades	62
3.12	Cross section for Unstructured grid generation of of Naca 0012	63
3.13.a	Cross section of Blazek turbine blades	64
3.13.b	Cross section of SE1050 turbine blades	64
3.14.a	Cross section of N number Blazek turbine blades	67
3.14.b	Cross section of N number of SE1050 turbine blades	67
3.14.c	Cross section of N number of VCE turbine blades	68
3.15	Flowchart of methodology	69
4.1	Domain of interest of the diverging nozzle	71

4.2	Computational domain	71
4.3	Comparison of Mach Number distribution	72
4.4	Comparison of velocity distribution	73
4.5	Comparison of pressure distribution	73
4.6	Comparison of density distribution	74
4.7	Comparison of temperature distribution	74
4.8.a	Anderson Nozzle -A	76
4.8.b	Anderson Nozzle -B	76
4.9.a	Pressure distribution over Anderson's Nozzle- A	77
4.9.b	Mach number distribution over Anderson's Nozzle- A	77
4.10.a	Pressure distribution over Anderson's Nozzle- B	78
4.10.b	Mach number distribution over Anderson's Nozzle-B	78
4.11.a	Hoffmann Nozzle	79
4.11.b	Pressure distribution over Hoffmann Nozzle	80
4.11.c	Mach number distribution over Hoffmann Nozzle	80
4.12	Geometry of NASA TM 88926 Nozzle	81
4.13.a	Pressure distribution over NASA TM 88926 Nozzle	82
4.13.b	Mach number distribution over NASA TM 88926 Nozzle	82
4.14.a	Geometry of Blazek Nozzle	83
4.14.b	Pressure distribution over Blazek Nozzle	83
4.13.c	Mach number distribution over Blazek Nozzle	84
4.15.a	Behaviour of flow at Mach Number 0.8 on NACA 0012 AIRFOIL at the inlet angle of attack 0^0	86
4.15.b	Behaviour of flow at Mach Number 0.8 on NACA 0012 AIRFOIL at the inlet angle of attack 1.25^0	86
4.15.c	Behaviour of flow at Mach Number 0.8 on NACA 0012 AIRFOIL at the inlet angle of attack 2.5^0	87
4.15.d	Distribution of Mach Number on NACA 0012 AIRFOILS	87
4.15.e	Distribution of Pressure on NACA 0012 AIRFOILS	88

4.15.f	Behaviour of flow for angle of attack 1.25^0 on NACA 0012 AIRFOIL the inlet at Mach Number 0.6	88
4.15.g	Behaviour of flow at angle of attack 1.25^0 on NACA 0012 AIRFOIL at the inlet Mach Number 0.8	89
4.15.h	Behaviour of flow at angle of attack 1.25^0 on NACA 0012 AIRFOIL at the inlet Mach Number 1	89
4.15.i	Distribution of Mach Number on NACA 0012 AIRFOILS	90
4.15.j	Distribution of Pressure on NACA 0012 AIRFOILS	90
4.16.a	C_p Distribution over the NACA 0012 airfoil at angle of attack 1.25^0	92
4.16.b	C_p Distribution over the NACA 0012 airfoil at angle of attack 2.00^0	92
4.16.c	C_p Distribution over the NACA 0012 airfoil at angle of attack 2.340^0	93
4.16.d	C_p Distribution over the NACA 0012 airfoil at angle of attack 2.410^0	93
4.17.a	Distribution of (M) on the Bump channel	95
4.17.b	Distribution of (P) on the Bump channel	95
4.17.c	Mach Number distribution along the airfoil surface NACA 0012	96
4.17.d	Distribution of (P) along the airoil surface Naca 0012	96
4.18.a	Distribution of M over the Blazek blade	98
4.18.b	Distribution of P over the Blazek blade	98
4.19.a	Distribution of Mach number for Blazek blade using Cell centred Scheme	100
4.19.b	Distribution of Mach number for Blazek blade using Roe's Scheme	100
4.19.c	Distribution of Mach number over the Blazek blade	101
4.20.a	Distribution of Pressure on Blazek blade using Cell centred Scheme	102
4.20.b	Distribution of Pressure on Blazek blade using Roe Scheme	102
4.20.c	Distribution of Pressure over the Blazek blade	103
4.21.a	Distribution of Mach number on SE1050 blade by using Cell centred Scheme	104
4.21.b	Distribution of Mach number for SE1050 blade by using Roe Scheme	104
4.21.c	Distribution of Mach number over SE1050 blade	105
4.22.a	Distribution of Pressure for SE1050 blade by using Cell centred	106

	Scheme	
4.22.b	Distribution of Pressure for SE1050 blade by using Roe Scheme	106
4.22.c	Distribution Pressure over SE1050 blade	107
4.23.a	Distribution of Mach Number on stator blades using computer code at $P_e = 62830$ Pa	109
4.23.b	Distribution of Mach Number on stator blades using computer code at $P_e = 67830$ Pa	109
4.23.c	Distribution of Mach Number on stator blades using computer code at $P_e = 72830$ Pa	110
4.23.d	Mach Number distribution on blades domain using computer code	110
4.24.a	Distribution of static pressure on stator blades using computer code at $P_e = 62830$ Pa	111
4.24.b	Distribution of static pressure on stator blades using computer code at $P_e = 67830$ Pa	111
4.24.c	Distribution of static pressure on stator blades using computer code at $P_e = 72830$ Pa	112
4.24.d	Static pressure distribution on blades domain using computer code	112
4.25.a	Behavior of flow for Mach Number for N number of Blazek blades at back pressure 62830 Pa	113
4.25.b	Behavior of flow for Mach Number for N number of Blazek blades at back pressure 67830 Pa	113
4.25.c	Behavior of flow for Mach Number for N number of Blazek blades at back pressure 72830 Pa	114
4.25.d	Distribution of Mach Number over Blazek blades turbine stator	114
4.26.a	Behavior of flow for back pressure for N number of Blazek blades at back pressure 62830 Pa	115
4.26.b	Behavior of flow for back pressure for N number of Blazek blades at back pressure 67830 Pa	115
4.26.c	Behavior of flow for back pressure for N number of Blazek blades at back pressure 72830 Pa	116

4.26.d	Distribution of Pressure over Blazek blades turbine	116
4.27.a	Behavior of flow for Mach Number for N number of VCE blades at back pressure 72830 Pa	117
4.27.b	Behavior of flow for Mach Number for N number of VCE blades at back pressure 77830 Pa	117
4.27.c	Behavior of flow for Mach Number for N number of VCE blades at back pressure 82830 Pa	118
4.27.d	Distribution of Mach Number over VCE blades turbine stator	118
4.28.a	Behavior of flow for static pressure for N number of VCE blades at back pressure 72830 Pa	119
4.28.b	Behavior of flow for static pressure for N number of VCE blades at back pressure 77830 Pa	119
4.28.c	Behavior of flow for static pressure for N number of VCE blades at back pressure 82830 Pa	120
4.28.d	Distribution of static pressure over VCE blades turbine stator	120



LIST OF SYMBOLS AND ABBREVIATIONS

ϕ	Derivative
f_i	Interpolation functions
∂	Differentiation
ρ	Static density
t	Time
p	Pressure
T	Temperature
F	Flux vector
Q	The conserved flow variables
E	Total energy
e	Internal energy
M	Mach number
V	Velocity
v	Speed of sound
γ	Specific heat ratio
η	Computational space in y direction
ξ	Computational space in x direction
Φ	Difference of characteristic variable

α	Numerical viscous derivative function
Ω	Control volume
\vec{F}_c	Convective flux
\vec{F}_d	Diffusive flux
\vec{I}	Identity tensor
$\vec{\tau}$	Viscous shear stress tensor
$\vec{\sigma}$	Stress tensor
k	Diffusivity
q_h	Heat transferred
$\vec{\nabla}$	Compounds in x,y and z
Q_v	Volume source terms
c_p	Pressure coefficient
$\vec{\omega}$	Angular velocity
\vec{V}_{rel}	Velocity relative
\vec{r}	Position vector relative
Δs	Face length
u_n	Normal velocity
u_t	Tangential velocity
S	Cross-sectional area
\vec{U}	Covariance velocity
C_v	Constant volume specific heat

Published Papers:

- 1. Title:** Comparison Results Between MacCormack Scheme and Steger Warming Scheme For the Case of Supersonic Flow Pass Through Divergent Nozzle
ID: 72.
Published in: Applied Mechanics and Materials 315 (2013): 268-272, Malaysia
- 2. Title:** The Implementation of Cell-Centred Finite Volume Method over Five Nozzle Models
ID: 168
Published in: Applied Mechanics and Materials 393 (2013): 305-310, Malaysia
- 3. Title:** Comparison Between Finite Volume Methods and Ansys CFD code to case of Transonic Flow Pass Through a Turbine Blade Section
ID: 183.
Published in: International Conference on Mechanical Engineering and Mechatronics (ICMEM'13), Canada
- 4. Title:** Comparison Between Roe's Scheme and Cell-centered scheme For Transonic Flow Pass through a Turbine Blades Section
ID: 4
Published in: IOP Conference Series: Earth and Environmental Science (2013). Vol. 19. No. 1. IOP, Indonesia

- 5. Title:** Comparison Between Cell-centered scheme's Computer Code and Fluent Software for a Transonic Flow Pass through an Array of Turbine Stator Blades
ID: 74
Published in: Applied Mechanics and Materials 437 (2013): 271-274., China
- 6. Title:** Numerical study on heat transfer and fluid flow characteristic of tube bank with integral wake splitter- effect of wake splitter length
ID: 183
Published in: Applied Mechanics and Materials 315 (2013): 650-654, Malaysia
- 7. Title:** Two Dimensional Compressible Flow Analysis Over a Generic Cruise Missile Model
Paper ID: 811
Published in: Applied Mechanics and Materials 465 (2014): 358-362, Malaysia
- 8. Title:** Comparison between Structured and Unstructured Grid Generation on Two Dimensional Flows Based on Finite Volume Method (FVM)
Published in: International Journal of Mining, Metallurgy & Mechanical Engineering (IJMMME) Volume 2, Issue 2 (2014) ISSN 2320-4060
- 9. Title:** Comparison between Finite Volume Method (FVM) Based On Inviscid and Viscous Flow with Experimental and Fluent Results
Published in: Applied and Computational Mathematics Special Issue: New orientations in Applied and Computational Mathematics. Vol. 4, No. 1-1, 2015, pp. 12-17. doi: 10.11648/j.acm.s.2015040101.13. USA
- 10. Title:** The Development of Dedicated CFD Solver for Aerodynamic Analysis on Internal and External Flow
Submitted: American Society of Mechanical Engineers (ASME)

CHAPTER 1

INTRODUCTION

In this chapter, the background of the research is outlined, followed by problem statements, the objectives and the scopes of the research, process of methodology and lastly contributions.

1.1 Research Background

Computational Fluid Dynamics, known today as CFD, is defined as the set of methodologies that enable the computer to provide us with a numerical simulation of fluid flows. Used the word ‘simulation’ to indicate that we use the computer to solve numerically the laws that govern the movement of fluids, in or around a material system, where its geometry is also modelled on the computer. Hence, the whole system is transformed into a ‘virtual’ environment or virtual product [1]. This can be opposed to an experimental investigation, characterized by a material model or prototype of the system. The use of Computational Fluid Dynamics (CFD) to predict external and internal flows, such as an aircraft, car model in a wind tunnel, and missiles for the external flows. In other hand, when predicting the flow properties in a prototype of an engine, compressor, turbine or nozzle, it will related in manner of one solve the internal flow problem. [2]

Generally in fluid mechanics, external flow is such a flow that boundary layers develop freely, without constraints imposed by adjacent surfaces. Accordingly, there will always exist a region of the flow outside the boundary layer in which velocity, temperature, and/or concentration gradients are negligible. Such flow condition can be defined as the flow of a fluid around a body that is completely

submerged in it. While, internal flow is a flow for which the fluid is confined by a surface [3]. Hence the boundary layer is unable to develop without eventually being constrained. The internal flow configuration represents a convenient geometry for heating and cooling fluids used in chemical processing, environmental control, and energy conversion technologies [4]. Basically the external and internal flow problems are governed by the same equation. If the compressibility effect can be ignored and the flow under high Reynolds number condition which allowing one to ignore the viscous effect, these two types of flow can be well represented by the compressible Euler equations. In this respect, the difference between the internal and external flow came from their boundary conditions. The same numerical approach can be used for solving the governing equation for these two types of flow. [5, 6]

In solving the governing equation of fluid motion as an internal or as an external flow problems numerically, basically there are three families of methods can be used. They are namely Finite Difference Method (FDM), Finite Volume Method (FVM) and Finite Element Method (FEM). Each of these methods carries various approximation techniques in solving the governing equation of fluid motion. [7, 8]

The finite difference method is based on the properties of Taylor expansions and on the straight forward application of the definition of derivatives. It is perhaps the simplest method to apply, particularly on uniform meshes, but it requires a high degree of regularity of the mesh. In particular, the mesh must be set up in a structured way, whereby the mesh points, in an n -dimensional space, are located at the intersections of n family of rectilinear or curved lines. These curves appear as a form of numerical coordinate lines and each point must lie on one, and only one, line of each family. Finite Volume Method FVM is developed as a special case of FDM. [9]

However, it gained much popularity due to the fact that FVM is more physically based. That is to say, each term in the calculation represent a physical phenomenon. Most of the commercial CFD package programs use FVM. In any FVM code, first, the governing equations are integrated over the control volumes (cells). Secondly, the discretisation is applied by representing all the physical terms of the flow (like convection, diffusion and sources) as finite difference approximations. Lastly, the unknown properties in each cell are solved simultaneously and iteratively. The basic idea behind the FVM is that all the governing equations represent the conservation of some property. Finite Element

Method FEM is originally developed for structural stress analysis, but it can also be used for fluid mechanics. This method is more mathematically based and uses the principle that the field variables can be approximated by linear combinations of simple piecewise functions locally. Under these approaches there are various methods. [10, 11]

The present work used the combine technique of FDM and FVM in particular, the MacCormack's scheme (Steger - Warming) will be applied for the FDM, while both Cell-Centred scheme and Roe's scheme used for FVM. Based on these methods, the developed Computer codes used to investigate aerodynamic behaviours of internal flow problems as well as external flow problems. In order to solve various types of flow problem, the present work involves the development of a number of numerical grid generation codes. In order to ensure appropriate boundary conditions,, A series of assessments were carried out to investigate their influence on both types of flow problems.

It is necessary to be noted that the present work focused more on the development of CFD code based on the use of Finite Volume approach rather than, based on the Finite Difference Approach. This is due to the fact, the Finite Volume Method give more flexibility in defining the mesh flow domain. In addition to this, it had been found, the spatial discretization of the flow domain will determine in manner how the computer code based on the Finite Volume approach to be implemented. The spatial discretization can be carried by use of a structured grid or an unstructured grid [12, 13]. It is therefore, two computer codes need to be developed if one deals with structured grid and in other cases one to use unstructured grid. In order to achieve a well-developed computer codes, the present work carried out step by step code developments. It starts with applying the finite volume and finite difference methods for solving a shock tube problem, followed with the test case of flow pass through nozzle, flow pass through a bump, airfoil, and single cascade configuration up to multi cascade configurations. For each study case as mentioned previously, data are compared to the available previous numerical work, experiment or present generated FLUENT software.

1.2 Problem statement

The analysis of internal or external flow field which are generated by the external compression surfaces such as occurred in the external flow pass through aircraft, airfoils or missiles and the internal flow through compressor, turbine[1], and nozzles [14] are complicated due to the formation of multiple shock waves [15, 16]. In the case of external flow pass a streamline body at relatively low to moderate angle of attack and the case of flow through duct with gradual varying cross section [17, 18], both flow problems can be treated as inviscid flow. Basically whether the flow is internal or external flow problem, they are governed by the same governing equation of fluid motion called as The Navier Stokes Equation [16, 19]. Unfortunately for solving the Navier Stokes Equation for an arbitrary flow problems are very difficult and if it is possible will require computer resources may beyond the computer capability at present time. However for a special flow problems as mentioned above [20, 2], the viscous term can be eliminated from the Navier Stokes equation resulted the new governing equation of fluid motion called the Euler Equation. This equation can be applied for solving the external as well as the internal flow problem [21, 22]. Considering this equation just ignoring the viscous effect, as result, the Euler equation will represent the governing equation which allows the discontinuity flow phenomena due to shock wave or vortex and the density variation in the flow field are appeared in their solution[23]. Unfortunately, the Euler equations are still representing a non-linear differential equation which there is no analytical solution. As a result, a numerical approach are required in solving the Euler equations. [24, 25]

1.3 Significance of study

The outcome of the current research work is aimed to provide a useful solver for aerodynamic designers to carry out their aerodynamics analysis in solving the internal and external flow problems they are facing. This flow solver will provide the solution of the flow field that involves flow variables in term of velocity in both x and y direction, density, temperature, pressure and Mach number distribution. Such information are essential, since the aerodynamics designers can deduce the presence

of the shock wave, type of shock wave as weak or strong shock wave, the pressure gradient along the body surface to indicate the possibility of flow separation, the overall aerodynamics characteristics in term of lift coefficient, moment pitching coefficient to case of external flow problem or the excessive pressure drop may occurred if one deals with flow pass through turbine blade. Such information will decrease the cost and saving the time of design process. [26, 27, 28]

1.4 Research objectives

1. To develop a CFD software for aerodynamic analysis of two dimensional inviscid compressible flow as well as laminar viscous flow applicable for internal as well as external Flow problem.
2. To develop codes of mesh flow domain under structured grid and unstructured grid model.

1.5 Scope of study

1. Develop computer code for compressible inviscid flow as well as laminar viscous flow starting from the case of one dimensional flow problem, quasi one dimensional flow problem and two dimensional flow problems with various numerical schemes.
2. Develop numerical structured and unstructured grid generation tool based on algebraic and elliptic approach.
3. Develop Euler Solver for the test case as above by using Finite Volume Methods (FVM) Cell-Centred scheme and Roe's scheme
4. Grid generation of the flow domain with Gambit software and flow calculation by Fluent software.
5. Further extension on the works for the case of N number of blade cross sections.

1.6 Contribution to knowledge

This work will provide a new computer algorithm in manner how to solve one and two dimensional compressible flow pass through Internal and External Flows. The basic numerical schemes for the flow solver are Cell-Centered and Roe's Finite Volume scheme and the numerical grid generation developed based on multi block scheme. In addition, this work will provide a useful solver for aerodynamic designers to carry out their aerodynamics analysis in solving the internal and external flow problems that they are facing. Such information will decrease the cost and saving the time of design process aerodynamic designers. [24].



REFERENCE

1. Versteeg, Henk Kaarle, and Weeratunge Malalasekera. 2nd .edition. An introduction to computational fluid dynamics: the finite volume method. Pearson Education, 2007.
2. Hirsch, Charles. 2nd .Edition .Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics: The Fundamentals of Computational Fluid Dynamics. Vol. 1. Butterworth-Heinemann, 2007.
3. Dadone, A., and B. Grossman. Surface boundary conditions for the numerical solution of the Euler equations. *AIAA journal* 32.2. 1994. 285-293.
4. Thompson, Kevin W. Time-dependent boundary conditions for hyperbolic systems, II. *Journal of Computational Physics* 89.2. 1990. 439-461.
5. C, H. Numerical computation of internal and external flows. Computational methods for inviscid and viscous flows. Butterworth-Heinemann. 1990. 536-556.
6. Liou, Meng-Sing. A generalized procedure for constructing an upwind based TVD scheme. *AIAA journal*. 1987. 87-0355.
7. LeVeque, Randall J. and Randall J. Le Veque. 2nd .edition .Numerical methods for conservation laws. Vol. 132. Basel: Birkhäuser, 1992.
8. Ferziger, Joel H., and Milovan Perić. 3rd. edition. Computational methods for fluid dynamics. Vol. 3. Berlin: Springer, 2002.
9. Anderson, John David. 1st. edition. Computational fluid dynamics. Vol. 206. New York: McGraw-Hill. 1995.
10. Blazek, Jiri. 1st. edition. Computational Fluid Dynamics: Principles and Applications: Principles and Applications. Elsevier. 2001.
11. Botte, Gerardine G., James A. Ritter, and Ralph E. White. Comparison of finite difference and control volume methods for solving differential equations. *Computers & Chemical Engineering* 24.12. 2000. 2633-2654.

12. Vladimir, D. Liseikin. 2nd .edition. Grid generation methods. Springer. 2010.
13. Hoffmann, Klaus A., and Steve T. Chiang. 4th. edition. Computational fluid dynamics, Vol. 3. Wichita, KS: Engineering Education System. 2000.
14. Hoffmann, Klaus A., and Steve T. Chiang. 4th. edition. Computational fluid dynamics, Vol. 1. Wichita, KS: Engineering Education System. 2000.
15. Hoffmann, Klaus A., and Steve T. Chiang. 4th. edition. Computational fluid dynamics, Vol. 2. Wichita, KS: Engineering Education System. 2000.
16. Blazek, Jiri. Computational Fluid Dynamics: Principles and Applications: Principles and Applications. 2nd .edition. Elsevier, 2005.
17. Reese L. Sorenson. A Computer Program to Generate Two-Dimensional Grids About Airfoils and Other Shapes by the Use of Poisson's Equation. *National Aeronautics And Space Reproduced By Us. Department Of Commerce National Technical Information Service Springfield*. 1980. Va 221 61. 505-31-11.
18. Lutton, Mark J. Comparison of C-And O-Grid Generation Methods Using A NACA 0012 Airfoil. No. Afit/Gae/Eny/89D-21. *Air Force Inst Of Tech Wright-Patterson Afb Oh School Of Engineering*. 1989.
19. Drela, Mark, and Michael B. Giles. Viscous-inviscid analysis of transonic and low Reynolds number airfoils. *AIAA journal* 25.10. 1987. 1347-1355.
20. Shih, T. P., Bailey, R. T., Nguyen, H. L., & Roelke, R. J. Grid.2D/3D: A computer program for generating grid systems in complex-shaped two-and three-dimensional spatial domains. Part 1: Theory and method. *NASA Lewis Research Center; Cleveland, OH, United States*. 1990. 1.15:102453.
21. Pulliam, Thomas H. Artificial dissipation models for the Euler equations. *AIAA journal* 24.12. 1986. 1931-1940.
22. Whitfield, D. L., A. Jameson, and W. Schmidt. Computation of transonic viscous-inviscid interacting flow. Springer Berlin Heidelberg. 1984. 978-3-662-09014-5.
23. Chyu, W. J., Takaichi Kawamura, and D. P. Bencze. Calculation of external-internal flow fields for mixed-compression inlets. *Computer Methods in Applied Mechanics and Engineering* 64.1. 1987. 21-37.
24. Hoffmann, Klaus A., and Steve T. Chiang. 4th. edition. Computational fluid dynamics, Vol. 4. Wichita, KS. *Engineering Education System*. 2000.

25. Rivera, Jose A., Bryan E. Dansberry, Robert M. Bennett, Michael H. Durham, and Walter A. Silva. NACA0012 benchmark model experimental flutter results with unsteady pressure distributions. *AIAA journal*. 1992. 23665-5225.
26. Wu, Gangfeng, Zhiguo He, and Guohua Liu. Development of a Cell-Centered Godunov-Type Finite Volume Model for Shallow Water Flow Based on Unstructured Mesh. *Mathematical Problems in Engineering*. 2014. pp. 1–13.
27. NASA Conference Publication 2166. Numerical Grid Generation Techniques. Proceedings of a workshop sponsored by the *National Aeronautics and Space Administration and the Institute for Computer Applications in Science and Engineering and held at NASA Langley Research Center Hampton, Virginia*. 1980. N81-14690.
28. Thompson, Joe F., Bharat K. Soni, and Nigel P. Weatherill, eds. 1st. edition. Handbook of grid generation. CRC press. 2010.
29. Shyy, Wei, H. S. Udaykumar, Madhukar M. Rao, and Richard W. Smith. Computational fluid dynamics with moving boundaries. Courier Corporation, 2012. 978-0486458908.
30. Magoulès, Frédéric, ed. 1st. edition. Computational fluid dynamics. CRC Press. 2011.
31. Murrone, Angelo, and Hervé Guillard. Behavior of upwind scheme in the low Mach number limit: III. Preconditioned dissipation for a five equation two phase model. *Computers & fluids* 37.10. 2008. 1209-1224.
32. T. H. Pulliam. Solution Methods in Computational Fluid Dynamics. *NASA Ames Research Center*, 1986.
33. Yee, Helen C., and Paul Kutler. 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*. 1983. NAS 1.15:85845.
34. Shi, Jian, and E. F. Toro. Fully discrete high-order TVD schemes for a scalar hyperbolic conservation law. Cranfield College of Aeronautics, Department of Aerospace Science. 1993. 978-1871564648.
35. Liang, Dongfang, Roger A. Falconer, and Binliang Lin. Comparison between TVD-MacCormack and ADI-type solvers of the shallow water equations. *Advances in water resources* 29.12. 2006. 1833-1845.

36. Göncü, L.O. Computation of External Flow Around Rotating Bodies. *Mechanical Engineering Department, METU*. 2005.
37. Renwei Liu, Dongjie Wang, Xinyu Zhang, Wang Li, and Bo Yu. Comparison Study on the Performances of Finite Volume Method and Finite Difference Method. *Journal of Applied Mathematics*. 2013. 596218.
38. M. Goodarzi, M. R. Safaei, A. Karimipour, K. Hooman, M. Dahari, S. N. Kazi, and E. Sadeghinezhad. Comparison of the finite volume and lattice Boltzmann methods for solving natural convection heat transfer problems inside cavities and enclosures. *Abstract and Applied Analysis*. Vol. 2014. 762184.
39. Vosoughifar, Hamid Reza, Azam Dolatshah, and Seyed Kazem Sadat Shokouhi. Discretization of Multidimensional Mathematical Equations of Dam Break Phenomena Using a Novel Approach of Finite Volume Method. *Journal of Applied Mathematics*. 2013. 642485.
40. Sheng Bi, Jianzhong Zhou, Yi Liu, and Lixiang Song. A Finite Volume Method for Modeling Shallow Flows with Wet-Dry Fronts on Adaptive Cartesian Grids. *Mathematical Problems in Engineering*. 2014. 209562.
41. Stefano Ferraris, Ivan Bevilacqua, Davide Canone, Davide Pognant, and Maurizio Prevati. The Finite Volume Formulation for 2D Second-Order Elliptic Problems with Discontinuous Diffusion/Dispersion Coefficients. *Mathematical Problems in Engineering* 2012. 187634.
42. Li, Zheng, Mo Yang, and Yuwen Zhang. Hybrid Lattice Boltzmann and Finite Volume Method for Natural Convection. *Journal of Thermophysics and Heat Transfer* 28.1 (2014): 68-77.
43. Breviglieri, Carlos, Joao Luiz F. Azevedo, and Edson Basso. An implicit spectral finite volume method for unstructured grid aerodynamic simulations. *European Conference on Computational Fluid Dynamics* J. C. F. Pereira and A. Sequeira (Eds) Lisbon, Portugal. 2010.
44. Lee, Kyung Rock, Jung Ho Park, and Kyu Hong Kim. High Order Interpolation Method for Overset Grid using FVM. *19th AIAA Computational Fluid Dynamics*. 2009. 3660.
45. Seth C. Spiegel, Douglas L. Stefanski, Hong Luo, and Jack R. Edwards. A cell-centered finite volume method for chemically reacting flows on hybrid grids. *AIAA Paper*. 2010. 1083.

46. Thompson, Joe F., Zahir UA Warsi, and C. Wayne Mastin. Numerical grid generation: foundations and applications. Vol. 45. *Amsterdam: North-holland*, 1985.
47. Udaykumar, H. S., R. Mittal, P. Rampunggoon, and A. Khanna. A sharp interface Cartesian grid method for simulating flows with complex moving boundaries. *Journal of Computational Physics* 174.1. 2001. 345-380.
48. Farrashkhalvat, M., and J. P. Miles. 1s. edition. Basic Structured Grid Generation: With an introduction to unstructured grid generation". *Butterworth-Heinemann*, 2003.
49. Zhang, Yaixin, Yafei Jia, and S. Y. Wang. A Conservative Multi-block Algorithm for Two-dimensional Numerical Model. *J. Math. Sci* 1.2. 2007. 13.
50. Luo, Sheng, Chaoqun Liu, and Hau Shan. A Multigrid Method for Elliptic Grid Generation Using Finite Volume Method. *texas univ at arlington dept of mathematics*, 2001.
51. Thomas, James L., Boris Diskin, Achi Brandt, and Jerry C. South Jr. General framework for achieving textbook multigrid efficiency: one-dimensional Euler example. *Frontiers of computational fluid dynamics*. 2002. 61-80.
52. Thompson, Joe F., Bharat K. Soni, and Nigel P. Weatherill, eds. Handbook of grid generation. *CRC press*. 2010.
53. Barth, Timothy J., and Paul O. Frederickson. Higher order solution of the Euler equations on unstructured grids using quadratic reconstruction. *AIAA paper* 90. 1990. 0013.
54. Sorenson, Reese L., and Joseph L. Steger. Numerical generation of two-dimensional grids by the use of Poisson equations with grid control at boundaries. *NASA. Langley Research Center Numerical Grid Generation Tech*. 1980. 81-14690 05-64.
55. Chung, T. J. Computational fluid dynamics. *Cambridge university press*. 2010.
56. Zienkiewicz, Olgierd Cecil, and Robert Leroy Taylor. 7th. edition. The finite element method: Solid mechanics. Vol. 2. *Butterworth-heinemann*. 2000.
57. Cebeci, Tuncer. 1st. edition. Computational fluid dynamics for engineers: from panel to navier-stokes methods with computer programs. *Springer*. 2005.

58. Thomée, Vidar. From finite differences to finite elements: A short history of numerical analysis of partial differential equations. *Journal of Computational and Applied Mathematics* 128.1. 2001. 1-54.
59. Hirsch, Charles. Numerical Computation of Internal and External Flows, Fundamentals of Numerical Discretization. *University of California. Wiley.* 1991.
60. Oosthuizen, Patrick H., and William E. Carscallen. Compressible fluid flow. *New York: McGraw-Hill.* 1997.
61. Frank M. White. Fluid Mechanics. *Fourth Edition.* 1998.
62. Hannemann, V., A. Siegmund, K. Oßwald, P. Birken, K. Weinman, and A. Meister. Dissipation of upwind schemes at high wave numbers. *Seventh International Conference on Computational Fluid Dynamics (ICCFD7), 1. Big Island, Hawai., 9-13.* 2012.
63. Blevins, Robert D. Applied fluid dynamics handbook. *New York, Van Nostrand Reinhold Co., 1984, 568 p.* 1. 1984.
64. Faber, Tom E. Fluid dynamics for physicists. *Cambridge University Press.* 1995.
65. Katate Masatsuka. 2nd. edition. I do like CFD, VOL.1, Governing Equations and Exact Solutions. 2013.
66. Thomas H. Pulliam. The Euler equation. *Nasa Ames Research Center.* 1994.
67. Gilding, B. H. A numerical grid generation technique. *Computers & fluids* 16.1. 1988. 47-58.
68. Morton, Keith W., and David Francis Mayers. Numerical solution of partial differential equations: an introduction. *Cambridge university press.* 2005.
69. Munson, Bruce Roy, Donald F. Young, and Theodore Hisao Okiishi. Fundamentals of fluid mechanics. *New York.* 1990.
70. Andersson, B., Andersson, R., Håkansson, L., Mortensen, M., Sudiyo, R., & Van Wachem, B. Computational fluid dynamics for engineers. *Cambridge University Press.* 2011.
71. Van Leer, Bram. Towards the ultimate conservative difference scheme. IV. A new approach to numerical convection. *Journal of computational physics* 23.3. 1977. 276-299.
72. Nordström, J., Gong, J., Van der Weide, E., & Svärd, M. A stable and conservative high order multi-block method for the compressible Navier–

- Stokes equations. *Journal of Computational Physics* 228.24. 2009. 9020-9035.
73. Leer, B. Lee W., and P. Roe. Characteristic time-stepping or local preconditioning of the Euler equations. *Ann Arbor* 1001. 1991. 48109-2140.
 74. Nordsletten, D. A., P. J. Hunter, and N. P. Smith. Conservative and non-conservative arbitrary Lagrangian–Eulerian forms for ventricular flows. *International journal for numerical methods in fluids* 56.8. 2008. 1457-1463.
 75. Davis, Stephen F. A simplified TVD finite difference scheme via artificial viscosity. *SIAM journal on scientific and statistical computing* 8.1. 1987. 1-18.
 76. Steger, Joseph L., and R. F. Warming. Flux vector splitting of the inviscid gasdynamic equations with application to finite-difference methods. *Journal of computational physics* 40.2 1981. 263-293.
 77. Garcia, Reinaldo, and Rene A. Kahawita. Numerical solution of the St. Venant equations with the MacCormack finite-difference scheme. *International Journal for Numerical Methods in Fluids* 6.5. 1986. 259-274.
 78. Shuen, Jian-Shun, Meng-Sing Liou, and Bram Van Leer. Inviscid flux-splitting algorithms for real gases with non-equilibrium chemistry. *Journal of Computational Physics* 90.2. 1990. 371-395.
 79. Whitfield, David L., and L. K. Taylor. Discretized Newton-relaxation solution of high resolution flux-difference split schemes. *American Institute of Aeronautics and Astronautics*, 1991. A91-40714.
 80. LeVeque, Randall J. 1st. edition. Finite volume methods for hyperbolic problems. *Cambridge university press*. 2002.
 81. Courant, Richard, and Kurt Otto Friedrichs. Supersonic flow and shock waves. Vol. 21. Springer Science & Business Media, 1999.
 82. Rao, G. V. R. Exhaust nozzle contour for optimum thrust. *Journal of Jet Propulsion* 28.6. 1958. 377-382.
 83. Diskin, Boris, James L. Thomas, Eric J. Nielsen, Hiroaki Nishikawa, and Jeffery A. White. Comparison of node-centered and cell-centered unstructured finite-volume discretizations: Viscous fluxes. *AIAA journal* 48, no. 7. 2010. 1326-1338.
 84. Eymard, Robert, Thierry Gallouët, and Raphaële Herbin. Finite volume methods. *Handbook of numerical analysis* 7. 2000. 713-1018.

85. Eymard, R., T. Gallouët, and R. Herbin. A cell-centred finite-volume approximation for anisotropic diffusion operators on unstructured meshes in any space dimension. *IMA Journal of Numerical Analysis* 26.2. 2006. 326-353.
86. Guillard, Hervé, and Angelo Murrone. On the behavior of upwind schemes in the low Mach number limit: II. Godunov type schemes. *Computers & fluids* 33.4. 2004. 655-675.
87. Harten, Amiram, Peter D. Lax, and Bram van Leer. On upstream differencing and Godunov-type schemes for hyperbolic conservation laws. *SIAM review* 25.1. 1983. 35-61.
88. DeZeeuw, Darren, and Kenneth G. Powell. An adaptively refined Cartesian mesh solver for the Euler equations. *Journal of Computational Physics* 104.1. 1993. 56-68.
89. Roe, Philip L. Approximate Riemann solvers, parameter vectors, and difference schemes. *Journal of computational physics*. 43.2. 1981. 357-372.
90. Powell, Kenneth G., Philip L. Roe, Timur J. Linde, Tamas I. Gombosi, and Darren L. De Zeeuw. A solution-adaptive upwind scheme for ideal magnetohydrodynamics. *Journal of Computational Physics* 154, no. 2. 1999. 284-309.
91. Deconinck, H., Philip L. Roe, and R. Struijs. A multidimensional generalization of Roe's flux difference splitter for the Euler equations. *Computers & fluids* 22.2. 1993. 215-222.
92. Steinberg, Stanley. Fundamentals of Grid Generation. *CRC Press*. 1993.
93. Eiseman, Peter R., and Robert E. Smith. Mesh generation using algebraic techniques. *Numerical Grid Generation Tech.* 1980. 81-14690 05-64.
94. Sorenson, Reese L. The 3DGRAPE Book: Theory, Users Manual, Examples. *National Aeronautics and Space Administration, Ames Research Center*. 1989.
95. Häuser, Jochem, and Yang Xia. Modern Introduction to Grid Generation. *Department of Parallel Computing, Center of Logistics and Expert Systems, Salzgitter, Deutschland*. 1996.
96. Date, Anil Waman. Introduction to computational fluid dynamics. *Cambridge: Cambridge University Press*. 2005.

97. Pulliam, Thomas H. Solution methods in computational fluid dynamics. *Notes for the von Kármán Institute For Fluid Dynamics Lecture Series*. 1986.
98. Lehtim-aring, Reijo, and ki. Elliptic grid generation. *AIAA journal* 37.6. 1999. 768-770.
99. Knupp, Patrick M. acobian-weighted elliptic grid generation. *SIAM Journal on Scientific Computing* 17.6. 1996. 1475-1490.
100. Spekrijse, Stephanus Petrus. Elliptic grid generation based on Laplace equations and algebraic transformations. *Journal of Computational Physics* 118.1. 1995. 38-61.
101. Thompson, Joe F., and Nigel P. Weatherill. Structured and unstructured grid generation. *Critical reviews in biomedical engineering* 20.1-2. 1991. 73-120.
102. Dervieux, A. Steady Euler simulations using unstructured meshes. *In Von Karman Inst. for Fluid Dynamics Computational Fluid Dynamics*. 1985. N86-18652 09-34.
103. Landon, R.H. Compendium of unsteady aerodynamic measurements. *AGARD, AGARD-R-702*. 1982.
104. Ramesh, V., and S. M. Deshpande. Least squares kinetic upwind method on moving grids for unsteady Euler computations. *Computers & fluids* 30.5. 2001. 621-641.
105. http://qnet-ercoftac.cfms.org.uk/w/index.php/Silver:AC_6-12_Description
106. Yin, Zhao, Xiang-jun Fang, Si-yong Liu, Ping Wang, and Zhi-gang Liu. Research of Design Method of Variable Area Nozzle Turbine for Variable Cycle Engine. *In ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition. American Society of Mechanical Engineers*. 2011. 493-501
107. Pletcher, Richard H., John C. Tannehill, and Dale Anderson. Computational fluid mechanics and heat transfer. CRC Press. 2012.
108. Richard E. Sonntag, Claus Borgnakke. 2nd. edition. Introduction to Engineering Thermodynamics. Wiley. 2001.