Computational Investigation of Active Flow Control Parameters on the Aerodynamic Performance of NACA 2412 Airfoil

Bhanu Prakash Reddy Samala

SUPERVISED BY

Dr. Fernando Mellibovsky
Dr. Josep Maria Bergadà

Universitat Politècnica de Catalunya
Master in Aerospace Science & Technology
June 2015
Computational Investigation of Active Flow Control Parameters on the Aerodynamic Performance of NACA 2412 airfoil

BY
Bhanu Prakash Reddy Samala

DIPLOMA THESIS FOR DEGREE
Master in Aerospace Science and Technology

AT
Universitat Politècnica de Catalunya

SUPERVISED BY:
Dr. Fernando Mellibovsky
*Departament de Física Aplicada*

*Dr. Josep Maria Bergadà*
*Departament de Mecànica de Fluids*
This Page Intentionally Left Blank
ABSTRACT

The focus of this Master Thesis is to perform computational study, analyze and evaluate the implementation of Active Flow control (AFC) over NACA 2412 airfoil with an objective to delay the separation and improvise the aerodynamic parameters, lift and drag coefficient. The computations are performed at the Reynolds number of $3.1 \times 10^6$ (3.1 millions) with an inlet velocity ($U_\infty$) of 46 m/s within the angles of attack 12 to 18 degrees. Active flow control is implemented in such a way to optimize the parameters involved i.e., blowing/suction velocity magnitude, location of actuator and perturbation angle with an underlying motivation to modify the naturally occurring boundary layer phenomena. So, the present thesis is concentrated on two dimensional (2D) study of the viscid incompressible flow.

The computations are performed using Navier Stokes Equation solvers available in OpenFOAM, an open source Computational Fluid Dynamics software, in the cluster available at Department of Applied Physics, UPC-Barcelona Tech. The turbulence is modelled using Reynolds Averaged Navier Stokes (RANS) models mainly K Epsilon, $k-\omega$ SST and $k-kl-\omega$ for baseline cases. As $k-kl-\omega$ has predicted the turbulence better by providing the aerodynamic coefficients much closer to experimental values, it is used for active flow control cases. The numerical discretization methods, the turbulence models, turbulence parameters and linear solvers parameters are varied for the best accuracy with optimized computational resources. The mesh independency study with a focus on yPlus and validation with experimental results is done for baseline cases. Then Active flow control is applied using two techniques which include constant suction and constant blowing separately by varying the AFC parameters.

The results clearly illustrated that there is a dependency on all the parameters of actuator i.e., velocity amplitude, location, slot width and perturbation angle. The separation point is delayed by a maximum of 16.2%, 21% for angles of attack $12^\circ$ and $13^\circ$ through constant suction. There is always a positive impact on Lift Coefficient ($C_l$) where as both increase and decrease for drag coefficient ($C_d$). The increase in actuation velocity magnitude resulted in separation delay, increasing $C_l$ with a critical value for $C_d$ critical value. The suction worked better as the actuation is done away from separation point till a critical location whereas blowing is good for the slot location close to separation point. The suction at 90 degrees works better compared to angles less than $30^\circ$ for blowing. The increase in slot width has a positive impact on separation delay and $C_l$ with a decrease in $C_d$. The constant blowing has delayed the separation by 17.5%, increased $C_l$ by 1.55% and decreased $C_d$ by 3.74% for AOA 12 with a low increase in $C_l$ as compared to the suction cases.
ACKNOWLEDGEMENTS

First of all, I would like to deeply thank my supervisors Dr. Fernando Mellibovsky and Dr. Josep Maria Bergada for guiding me all through for the completion of the Master Thesis. Their technical support and personal motivation is an immense foundation for this thesis. Without high computational and numerical methods knowledge levels of Dr. Mellibovsky and top level scientific understanding possessed in Active Flow Control techniques by Dr. Bergada, it would have been a huge daunting task to reach the objectives of the thesis.

I would like to convey my best regards to the Department of Applied Physics at Campus Nord, UPC- Barcelona Tech for providing the access to the cluster to perform the simulations all through the duration of this Thesis. I take this opportunity to express my sincere thanks to colleague and friend, Nabil kakeh at the department who has been with me consistently all through supporting mentally and providing essential motivation. Also, I express my gratitude to Dr. Calvette, Jaime Alonso, Jordi, Angels , Rinsa and all others working in the department for providing all the necessary resources.

I am ever thankful to my parents, sister and wonderful cousins back home with whom I share an unbounded relation. I will be grateful to my friends Hema Vulchi and Anand Sanmukh for spending time and making interesting discussions that boosted my confidence during this time. Also I would like to thank Juan, Priya and Aditya for their suggestions and providing some inputs and resources. It is a pleasure to thank everyone who is directly or indirectly related to the completion of the present Thesis.

I would like to specially mention Dr. Ricard Gonzalez for coordinating the Aerospace Science and Technology Master Program and providing an opportunity to write my Master Thesis at an esteemed institution UPC-Barcelona Tech.
This Page Intentionally Left Blank
Table of Contents

CHAPTER 1 INTRODUCTION ........................................................................ 1
  1.1 Bibliographic Review ................................................................. 1
  1.2 Overview and Scope of Current Work ........................................ 3
  1.3 Objectives of Present Thesis ..................................................... 3
  1.4 Outline of Remaining Chapters .................................................. 5

CHAPTER 2 EQUATIONS AND MODELING ............................................... 6
  2.1 Basic Equations ............................................................... 6
  2.2 Finite Volume Discretization .................................................. 7
  2.3 Solution procedure for Steady and Transient State Problems ........ 9
  2.4 Calculation of turbulence parameters ..................................... 10
  2.5 Boundary Layer Theory and yPlus Calculation ....................... 11

CHAPTER 3 COMPLETE SIMULATION SETUP .......................................... 12
  3.1 Meshing ................................................................. 12
  3.2 OpenFOAM solver setup ...................................................... 14
  3.3 Implementation of active flow control .................................... 26
    3.3.1 Meshing .......................................................... 26
    3.3.2 Boundary condition implementation .............................. 27

CHAPTER 4 VALIDATION AND COMPUTATIONAL TESTING .................... 30
  4.1 Testing of computational Parameters ..................................... 30
    4.1.1 Variation of numerical discretization schemes ............... 30
    4.1.2 Variation of turbulence models .................................. 34
    4.1.3 Variation in turbulence parameters .............................. 36
    4.1.4 Mesh independent study with yPlus .............................. 39
  4.2 Validation with Experimental results ..................................... 41

CHAPTER 5 ACTIVE FLOW CONTROL ...................................................... 46
  5.1 Baseline cases for different locations ..................................... 46
  5.2 Suction ................................................................. 50
    5.2.1 Variation of location of actuation/ suction .................... 50
    5.2.2 Variation of velocity magnitude .................................. 56
    5.2.3 Variation of angle of suction ...................................... 62
    5.2.4 Variation of Slot width ............................................ 66
    5.2.5 Variation of Angle of Attack .................................... 70
  5.3 Blowing ................................................................. 76
    5.3.1 Variation with location ............................................... 76
    5.3.2 Variation of velocity ............................................... 77
    5.3.3 Variation of angle .................................................. 81

CHAPTER 6 CONCLUSION ....................................................................... 84
  6.1 Achievements ................................................................. 84
6.2 Future work ................................................................. 86

APPENDIX (Separately attached at the end)
List of Figures

Figure 1.1 Tomoscopy flow visualization of synthetic jet reattaching the separated boundary layer .......................................................... 2
Figure 1.2 Tomoscopy flow visualization of synthetic jet reattaching the separated boundary layer ...................................................... 2
Figure 1.3 Types of Active Flow control devices and their location of actuation and aerodynamic performance in steady and Unsteady cases ........................................ 2

Figure 3.1 Mesh Computational Domain .................................................. 13
Figure 3.2 Boundary labels of the mesh ....................................................... 13
Figure 3.3 Close up view of Mesh with near wall refinement ......................... 15
Figure 3.4 Case structure in OpenFOAM ....................................................... 16
Figure 3.5 Computational domain of the Active flow control implemented mesh.... 28
Figure 3.6 Close up view of active flow control mesh with the slot location at 70%C from the leading edge ................................................... 28
Figure 3.7 Exact Meshing around slot location in a close up view ................. 29

Figure 4.1 Drag Coefficient (Cd) vs order of discretization schemes ............ 31
Figure 4.2 Lift Coefficient (Cl) vs order of discretization schemes ................. 31
Figure 4.3 Computational time(sec) vs order of discretization schemes .......... 32
Figure 4.4 Residuals using first order schemes ............................................. 32
Figure 4.5 Residuals using first/second order schemes .................................. 33
Figure 4.6 Residuals using second order schemes ........................................ 33
Figure 4.7 Lift Coefficient vs AOA for different turbulence models .............. 35
Figure 4.8 Drag Coefficient vs AOA for different turbulence models .............. 35
Figure 4.9 Lift Coefficient vs turbulent Intensity [Data Source: Appendix table 3.3].... 37
Figure 4.10 Drag Coefficient vs turbulent Intensity [Data Source: Appendix table 3.3] 37
Figure 4.11 Leading and Trailing edge geometries of NACA 0015, TAU 0015 and TAU 015m airfoils .......................................................... 41
Figure 4.12 Lift Coefficient vs AOA for all three airfoils with validation .......... 42
Figure 4.13 Lift Coefficient vs AOA [Data Source: Appendix table 3.5] ............ 43
Figure 4.14 Drag Coefficient vs AOA [Data Source: Appendix table 3.6] ........... 44

Figure 5.1 Separation Point vs slot location for baseline cases *[Data Source: Appendix table 2.1] ....................................................... 47
Figure 5.2 Wall Shear Stress vs time ............................................................. 48
Figure 5.3 Wall Shear Stress vs time ............................................................. 49
Figure 5.4 Separation point vs Slot Location for different cases of velocity ratio *[Data Source: Appendix table 2.2 - 2.5] ........................................ 51
Figure 5.5 Lift Coefficient vs Slot Location for different cases of velocity ratio *[Data Source: Appendix table 2.2-2.5] ........................................... 52
Figure 5.6 Drag Coefficient vs Slot Location for different cases of velocity ratio *[Data Source: Appendix table 2.2-2.5] ........................................... 52
Figure 5.7 Separation point vs Velocity ratio for different cases of Slot Location *[Data Source: Appendix table 2.6 - 2.11] ........................................ 57
Figure 5.8 Lift Coefficient (CL) vs Velocity Ratio for different cases of Slot Location
[Data Source: Appendix table 2.6 - 2.11] ................................................................. 57
Figure 5.9 Drag Coefficient (Cd) vs Velocity Ratio for different cases of Slot Location
[Data Source: Appendix table 2.6 - 2.11] ................................................................. 58
Figure 5.10 Separation Point vs Angle of Suction [Data Appendix table 2.12] ........ 64
Figure 5.11 Lift Coefficient vs Angle of Suction [Data: Appendix table 2.12] ........ 65
Figure 5.12 Drag Coefficient vs Angle of Suction [Data: Appendix table 2.12] ..... 65
Figure 5.13 Separation Point (%C) vs Angle of Suction [Data Source: Appendix
table 2.13] ............................................................................................................ 66
Figure 5.14 Lift Coefficient (CL) vs Angle of Suction [Data Source: Appendix table
2.13] ...................................................................................................................... 67
Figure 5.15 Drag Coefficient (Cd) vs Angle of Suction [Data Source: Appendix table
2.13] ...................................................................................................................... 67
Figure 5.16 Separation Point (%C) vs slot width (b) [Data Source: Appendix table
2.14] ...................................................................................................................... 68
Figure 5.17 Lift Coefficient (CL) vs slot width (b) [Data Source: Appendix table 2.14] 69
Figure 5.18 Drag Coefficient (Cd) vs slot width (b) [Data Source: Appendix table
2.14] ...................................................................................................................... 69
Figure 5.19 Separation Point (%C) vs slot width (b) [Data Source: Appendix table
2.14] ...................................................................................................................... 71
Figure 5.20 Lift Coefficient (CL) vs slot width (b) [Data Source: Appendix table 2.14] 71
Figure 5.21 Drag Coefficient (Cd) vs slot width (b) [Data Source: Appendix table
2.14] ...................................................................................................................... 71
Figure 5.22 Velocity magnitude for AOA 12 without Active flow Control .......... 72
Figure 5.23 Velocity magnitude for AOA 12 with Active flow Control through suction 72
Figure 5.24 Wall shear stress indicating the Point of Separation (blue) (No AFC) .... 73
Figure 5.25 Wall shear stress indicating the Point of Separation (blue) (AFC) ........ 73
Figure 5.26 Velocity magnitude for AOA 13 without Active flow Control .......... 74
Figure 5.27 Velocity magnitude for AOA 13 with Active flow Control through suction 74
Figure 5.28 Wall shear stress indicating the Point of Separation (blue) (No AFC) .... 75
Figure 5.29 Wall shear stress indicating the Point of Separation (blue) (AFC) ........ 75
Figure 5.30 Separation Point (%C) vs Slot location [Data Source: Appendix table
3.1] ...................................................................................................................... 78
Figure 5.31 Lift Coefficient (CL) vs Slot location [Data Source: Appendix table 3.1] ..... 78
Figure 5.32 Drag Coefficient (Cd) vs Slot location [Data Source: Appendix table 3.1] . 79
Figure 5.33 Separation Point (%C) vs Velocity Ratio (U_b/U_∞) [Data Source: Ap-
 pendix table 3.2] ................................................................................................. 80
Figure 5.34 Lift Coefficient (CL) vs Velocity Ratio (U_b/U_∞) [Data Source: Appendix
table 3.2] .......................................................................................................... 80
Figure 5.35 Drag Coefficient (Cd) vs Velocity Ratio (U_b/U_∞) [Data Source: Appendix
table 3.2] .......................................................................................................... 81
Figure 5.36 Separation Point (%C) vs Blowing angle [Data Source: Appendix table
3.3] ...................................................................................................................... 82
Figure 5.37 Lift Coefficient (CL) vs Blowing angle [Data Source: Appendix table 3.3] .. 82
Figure 5.38 Drag Coefficient (Cd) vs Blowing angle [Data Source: Appendix table
3.3] ...................................................................................................................... 82
List of Tables

Table 4.1 Computational and Physical Parameters ................................................. 30
Table 4.2 Computational and Physical Parameters ................................................. 34
Table 4.3 Computational and Physical Parameters ................................................. 36
Table 4.4 Variation of viscosity ratio at higher turbulence intensity ................... 38
Table 4.5 Variation of $\mu$ with I ....................................................................... 39
Table 4.6 Variation of $y_{+}$ .................................................................................. 40
Table 5.1 Computational and Physical Parameters ................................................. 46
Table 5.2 Variation of Separation Point with time .................................................... 47
Table 5.3 Variation of Separation Point with time .................................................... 48
Table 5.4 Node distribution around slot ................................................................. 48
Table 5.5 Computational and Physical Parameters ................................................. 51
Table 5.6 Brief overview on parameters varied ....................................................... 51
Table 5.7 Best case with respect to separation delay .............................................. 53
Table 5.8 Brief overview on parameters varied ....................................................... 53
Table 5.9 Best case with respect to separation delay .............................................. 54
Table 5.10 Brief overview on parameters varied ..................................................... 54
Table 5.11 Best case with respect to separation delay ............................................. 55
Table 5.12 Brief overview on parameters varied ..................................................... 55
Table 5.13 Best case with respect to separation delay ............................................. 56
Table 5.14 Brief overview on parameters varied ..................................................... 57
Table 5.15 Best case with respect to separation delay ............................................. 59
Table 5.16 Brief overview on parameters varied ..................................................... 59
Table 5.17 Best case with respect to separation delay ............................................. 60
Table 5.18 Brief overview on parameters varied ..................................................... 60
Table 5.19 Best case with respect to separation delay ............................................. 61
Table 5.20 Brief overview on parameters varied ..................................................... 61
Table 5.21 Best case with respect to separation delay ............................................. 62
Table 5.22 Brief overview on parameters varied ..................................................... 62
Table 5.23 Best case with respect to separation delay ............................................. 63
Table 5.24 Brief overview on parameters varied ..................................................... 63
Table 5.25 Brief overview on parameters varied ..................................................... 63
Table 5.26 Best case with respect to separation delay ............................................. 64
Table 5.27 Brief overview on parameters varied ..................................................... 65
Table 5.28 Best case with respect to separation delay ............................................. 66
Table 5.29 Brief overview on parameters varied ..................................................... 66
Table 5.30 Best case with respect to separation delay ............................................. 68
Table 5.31 Computational and Physical Parameters ................................................. 76
Table 5.32 Brief overview on parameters varied ..................................................... 76
Table 5.33 Best case with respect to separation delay ............................................. 77
Table 5.34 Brief overview on parameters varied ..................................................... 79
Table 5.35 Best case with respect to separation delay ............................................. 79
Table 5.36 Brief overview on parameters varied ..................................................... 81
Table 5.37  Best case with respect to separation delay
Chapter 1

INTRODUCTION

1.1 Bibliographic Review

Designing an aircraft without conventional control surfaces is of interest to aerospace community. Active flow-control (AFC) is a fast growing multidisciplinary science and technology thrust aimed at altering a natural flow state or development path into a more desired state (or path). The ability to passively and actively manipulate a flow field has become one of the most researched topics in fluid mechanics since Prandtl’s first boundary layer control experiments in 1904 [1]. The change in aerodynamic performance may take the form of enhanced lift; reduced drag; controlled unsteadiness; and reduced noise or delayed transition. Alternatively, benefits for the same levels of performance may be accrued through reduced system complexity; less weight; less maintenance or reduced life cycle costs [2]. The experimental and computational investigations of active control of flow past airfoils at high angles of attack is an area of active research as extending the usable angles of attack has many important applications. A comprehensive review and analysis was provided by Lachman [3] and more recently by Gadel- Hak et al. [4], Gad-el-Hak [5].

Active Flow control (AFC) is the control of the local airflow surrounding the airfoil or blade to improve the aerodynamic performance in wind turbines, manned and unmanned airplanes, rotorcraft, gas turbines and other applications by delaying/advancing transition, suppressing/enhancing turbulence, or preventing/promoting separation. The impact of the active flow control can be depicted in the figures 1.1 and 1.2.

In particular, the control of flow separation from internal and external aerodynamic surfaces has gained significant interest as summarized by the proceedings of the biennial AIAA Flow Control Conferences which started in 2002. A multitude of passive and active means to control flow separation have been proposed and investigated. Some of the most common approaches involve vortex generators [8] [9], miniature trailing edge devices [10], continuous and oscillatory blowing and suction [11] [12] [13] [14] [15], microjets [16] [17] [18] [19], and zero-net-mass-flux actuation (e.g., plasma actuators [20] and synthetic jets [21] [22] [23] [24] [25] [26] [27] [28] [29]). These methods and their feasibility for full-scale applications have been reviewed and evaluated by numerous authors [30] [31] [32] [33] [34] [35] [36].

The classification of 15 AFC devices [37] on the type of actuation, location, and aerodynamic performance impact and time dependency can be observed in figure 1.3.

Although there have been a significant experiments being conducted in the implementation of active flow control, many challenges still remain to be resolved for considering this technology for practical purpose mainly commercial aviation [32]. Notable
Figure 1.1 Tomoscopy flow visualization of synthetic jet reattaching the separated boundary layer

[6]

Figure 1.2 Tomoscopy flow visualization of synthetic jet reattaching the separated boundary layer

[7]

Figure 1.3 Types of Active Flow control devices and their location of actuation and aerodynamic performance in steady and Unsteady cases
achievements were made in the download alleviation of the V-22 tilt rotor aircraft by Wygnanski, I et.al. Apart from that, AFC was implemented in Renalut altica concept car using the company’s patented synthetic jet technology [38]. Recently NASA and Boeing implemented AFC experimentally in a wind tunnel using sweeping jet actuators in Boeing 757 series especially to improve the rudder effectiveness there by reducing the size of vertical tails [39]. Hence there is a lot of scope for research in AFC which can be implemented practically.

1.2 Overview and Scope of Current Work

As aforementioned, there is a lot of scope and research prospects in the Active Flow Control for Aerospace and automobile applications although this technology can be applied for other sectors like wind energy, combustion and turbo machinery. However there are limited numerical investigations utilizing high end turbulence models like Large Eddy Simulation (LES), Detached Eddy Simulation (DES) and Direct Numerical Simulation (DNS). But the trend is to perform the experimental testing in the wind tunnel and supporting the obtained results using computational results.

So, the numerical methods and computational parameters are less analyzed in terms of their effectiveness in implementing AFC. Although the interests are more towards performing LES or DES, it is not logical to proceed directly without proper idea on the influence of the AFC parameters on aerodynamic performance as utilizing LES at high Reynolds number considered in the present case is a computationally intense project.

So, the present master thesis is more focused on setting the computational base to move further by providing a perfect understanding on the influence of the AFC parameters in the turbulence reduction. It is absolutely logical to perform two dimensional steady state simulations for incompressible flow before considering any complicated flow situations. It provides the flexibility to vary the AFC parameters and study their impact due to the fewer requirements of computational resources for these flow conditions. So, within the thesis, a detailed understanding on parameters of AFC for constant blowing and constant suction is provided.

Also by testing the computational parameters along with the validation simulations in the present thesis, a clear idea on the computational setup required for AFC simulations is obtained. For the present thesis only k-ω SST and k-kl-ω turbulence model simulations are considered.

1.3 Objectives of Present Thesis

The major objective of the present thesis is to evaluate the influence of different parameters of active flow control (AFC) through steady suction and steady blowing. The parameters considered include actuation velocity ratio \( U_s / U_\infty \) for suction or \( U_b / U_\infty \) for blowing, slot width \( b \), angle of suction or blowing \( \beta \) and location of actuation \( L_s \) for suction or \( L_b \) for blowing. As the active flow control implementation is done in a
constant magnitude with time, the frequency of actuation is not considered. The flow considered is two dimensional, incompressible and of high Reynolds number.

By varying the parameters, it is proposed to study the impact of each of these AFC parameters on the Separation delay, Lift coefficient (Cl) and Drag Coefficient (Cd). All these aerodynamic performance evaluation parameters depend on the instabilities and turbulence level involved on the suction side of the airfoil. The effect of the AFC parameters on the aerodynamic performance is related to its effectiveness in the reduction of the instabilities and turbulence around the airfoil. Hence the objective of the computational study is to analyze the impact of AFC implementation on these flow phenomena.

The secondary objective includes the validation of computational results obtained by OpenFOAM with the experimental aerodynamic data available for the different angles of attack (AOA) at high Reynolds number. As the computational tool being utilized to study the AFC implementation is OpenFOAM, an open source software, it is necessary to validate the computational results in order to check the accuracy and stability of the tool for the present thesis. Also, by utilizing the RANS turbulence models and steady state computations, it is a daunting task to obtain the exact aerodynamic data as in experiments performed. Hence, the knowledge of this expected computational tolerance with experiments will be definitely necessary when the AFC simulations are to be compared for their effectiveness.

Before performing a validation study, the different aspects involved in the Computational Fluid Dynamics (CFD) simulations will be studied in detail to fulfil the secondary objective of obtaining the results close to the experimental data. This include discretization schemes, turbulence models, turbulence parameters, mesh independent study with respect to yPlus and the lateral node distribution on the airfoil. The turbulence persisted in the wind tunnel experiments is not provided and hence a detailed study is conducted to match the practical turbulence conditions existed in the tunnel. Along with fulfilling the objective of the thesis, this computational parametric study is intended to benefit the open source community that is involved in developing the computational tool and it would be giving back for utilizing the software free of cost.

The third objective is to provide a knowledge resource for performing computational simulations of AFC implementation. As AFC is one of the viable technologies that can enhance the aerodynamic performance, performing computational study to understand and making it effective using open source software will be beneficial for the future researchers that might participate in this project. Hence the computational setup utilized in OpenFOAM is described in detail in the present thesis.

The fourth and minor objective is the setting up of a proper research base in CFD for Active flow control and also obtaining all the preliminary results required for continuing the thesis with transient, 2D and 3D cases using URANS and LES modelling. The results obtained from the primary objective with a detailed examination of AFC parameters can create the resources required for this. Also, this thesis will act as a
potential application for applying supercomputing time at Barcelona Supercomputing Center and also for the industry doctorate application for conducting research in Active Flow control.

1.4 Outline of Remaining Chapters

This master thesis is documented mainly in six chapters. The first chapter being Introduction to the present Thesis is majorly sub divided into four topics with Outline and scope of current work, Bibliographic review and previous work, Objectives of the present thesis and the outline of the chapters.

The Chapter 2 consists of Equations and Modelling which include the continuity and Navier Stokes momentum equations being solved by the computational tool for resolving the flow phenomena. The both turbulence models considered in this thesis $k$-$kl$-$\omega$ and $k$-$\omega$ SST are briefly elaborated. Along with, finite volume discretization is briefly mentioned. In addition, the solution procedure used in the OpenFOAM solvers for this thesis mainly Simple Foam which utilizes Simple algorithm and Pimple Foam which utilizes Pimple algorithm is elaborated.

The Chapter 3 includes the complete computational setup with a focus on the subtopics like Meshing, Computational Parameters available in OpenFOAM and also the ones used for the present thesis. Along with, the implementation of AFC in OpenFOAM mainly meshing and boundary conditions are discussed. Also, the transient simulation setup is discussed in brief as it was utilized for few simulations.

The Chapter 4 consists of Validation and Testing of Parameters in OpenFOAM. This is the first part of the Results and Discussion obtained from this thesis. All the parameters mentioned in the secondary objective are varied and the results along with the discussion are provided in this chapter. This is followed by the results of validation of the computational results with the experimental data.

The Chapter 5 includes the AFC results obtained from the constant suction and constant blowing. The parametric study is conducted in detail for both techniques and results are illustrated using the graphical and post processed images with the discussion followed by each case of computations for each parameter. All the parameters mentioned in the primary objective that plays a significant role in AFC implementation are considered.

The Chapter 6 is focused on the conclusions of the results obtained from Chapter 4 and 5 along with the future work that will be performed for drafting this thesis into research publications and also suggestions in general for research community.

This will be followed by References section and Appendix.
Chapter 2

EQUATIONS AND MODELING

2.1 Basic Equations

The equations governing Newtonian fluid flow can be found in numerous fluid mechanics textbook[e.g. White (1974), Schlichting (1979)]

In addition to the assumptions used to derive these equations, the following assumptions are made for this Thesis:

1. Steady state conditions ( \( \frac{\partial}{\partial t} = 0 \) )
2. Incompressible flow (Mach number < 0.3)
3. Two dimensional flow

With these assumptions, the continuity equation can be written in tensor notation as

\[
\frac{\partial u_i}{\partial x_i} = 0
\]  

(2.1)

As it is assumed that density is constant and hence there is no need to differentiate the constant. Writing the continuity equation in this form ensures that mass is conserved for these types of flows. The Navier Stokes equation can now be written in tensor form as:

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = f_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_i \partial x_j}
\]  

(2.2)

where \( f_i \) is a vector representing external forces

The governing equations in the Cartesian coordinate systems are summarized below with the usage of del and laplacian operators.

\[
\nabla . U = 0
\]  

(2.3)

\[
\frac{\partial U}{\partial t} + \nabla . (UU) = g - \nabla p + \nabla . (\nu \nabla U)
\]  

(2.4)
Equations 2-1 to 2-3 apply directly to laminar flow. However, for turbulent flow, the time averaged equations will be solved. This is accomplished by decomposing the dependent variables into a mean and a fluctuating component. These decomposed variables are then substituted back into the Navier Stokes and continuity equations and these equations are averaged. This yields the following set of equations for the Cartesian coordinate system

Among the below turbulence models, RANS based eddy viscosity models generally model the turbulent viscosity by assuming the Boussinesq hypothesis whereas the Reynolds stress transport models do not depend on eddy viscosity but the algebraic and transport equations are directly solved for the calculation of Reynolds stress term. In Large Eddy simulation (LES), the velocity field is separated into resolved (large eddies) and sub-grid part (small eddies) of which sub grid scale turbulence models generally employ Boussinesq hypothesis whereas the equations of motion for the resolved part that represents large eddies are directly solved. Detached Eddy Models and other hybrid models attempts to treat near-wall regions in a RANS-like manner, and treat the rest of the flow in an LES-like manner. A direct numerical simulation (DNS) is a simulation in computational fluid dynamics in which the Navier-Stokes equations are numerically solved without any turbulence model

Although there are many available turbulence models in OpenFOAM as described in the section 3.2, it should be clear that the computing resources availability and the simulation time play an important role choosing the high accuracy turbulence models. Due to the limited computational power available and the huge number of parameters to be analyzed in active flow control implementation, this Thesis is restricted to the usage of RANS turbulence models mainly Menter $k-\omega$ SST turbulence model and $k-\omega$ turbulence model. These are best available RANS models for the near wall treatment. The major physical difference between the two models is the inclusion of laminar to turbulence transition phenomena in $k-\omega$ model by incorporating a third equation within the existing $k-\omega$ turbulence models

The $k-\omega$ model with the equations for $k$ and $\omega$ with other parameters for calculation of turbulent viscosity are present in the reference of Menter’s $k-\omega$ SST [40].

The $k-\omega$ model with the equations for $k$, $\omega$ with other parameters for calculation of turbulent viscosity are present in the reference of Walters and Cokljat [41].

### 2.2 Finite Volume Discretization

The solution of the partial differential equations from the models in the continuum requires the process of discretization in order to be solved by a finite capacity machine, in terms of CPU power and memory. This objective can be achieved by several methods mainly the Finite Difference Method (FDM), the Finite Element Method (FEM) and the Finite Volume Method (FVM).
Finite Volume Method is predominantly used in commercial and open source CFD softwares available followed by Finite Element Method in relatively low number compared to FVM. The other methods like FDM, Spectral Methods, Boundary Element method, Vorticity based method and Lattice Boltzmann methods are relatively even low with few of them being researched intensively to provide efficient alternatives for FVM. However, the main stream implementation is yet to be achieved which leaves FVM with some monopoly currently.

The purpose of any discretization practice is to transform one or more partial differential equations into a corresponding system of algebraic equations. The solution of this system produces a set of values which correspond to the solution of the original equations at pre-determined locations in space and times, provided certain conditions are satisfied. The discretization process can be divided into two steps: the discretization of the solution domain and equation discretization.

The discretization of the solution domain produces a numerical description of the computational domain which includes the positions of points and the description of the boundary. The space is divided into a finite number of discrete regions, called control volumes or cells. For transient simulations, the time interval is also split into a finite number of time-steps. Equation discretization gives an appropriate transformation of terms of governing equations into algebraic expressions.

In general, a CFD solver based on FVM is progressed in space and time with the following steps for the conservation equation of a variable. However the detailed equations for this present thesis are mentioned in section 2.1

- Integration of conservation equations in each cell.
- Calculation of face values in terms of cell-centered values.
- Collection of variables or physical parameters at each time step for transient simulation or overall iteration for steady state simulation.

In time discretization, the solution is obtained by marching in time from the prescribed initial condition. It is therefore sufficient to prescribe the size of the time-step that will be used during the calculation. The discretization procedure of Navier Stokes can be found in many references with its OpenFOAM implementation in this reference [42]

The form of the equations shows linear dependence of velocity on pressure and vice-versa. These inter-equation coupling requires a special treatment. PISO [43], SIMPLE [44] and their derivatives are the most popular methods of dealing with inter-equation coupling in the pressure-velocity system.
2.3 Solution procedure for Steady and Transient State Problems

The solution procedure for steady-state incompressible turbulent flow using SIMPLE for inter-equation coupling is as follows:

1. Set all field values to some initial guess.
2. Assemble and solve the under-relaxed momentum predictor equation.
3. Solve the pressure equation and calculate the conservative fluxes. Update the pressure field with an appropriate under-relaxation. Perform the explicit velocity correction.
4. Solve the other equations in the system using the available fluxes, pressure and velocity fields. In order to improve convergence, under-relax the other equations in an implicit manner as below

   If the velocities are needed before the next momentum solution, the explicit velocity correction is performed.
5. Check the convergence criterion for all equations. If the system is not converged, start a new iteration on step 2.

The solution procedure for transient incompressible turbulent flow with PISO algorithm for pressure velocity coupling is as below:

1. Set up the initial conditions for all field values.
2. Start the calculation of the new time-step values.
3. Assemble and solve the momentum predictor equation with the available face fluxes.
4. Go through the PISO loop until the tolerance for pressure-velocity system is reached. At this stage, pressure and velocity fields for the current time-step are obtained, as well as the new set of conservative fluxes.
5. Using the conservative fluxes, solve all other equations in the system. If the flow is turbulent, calculate the effective viscosity from the turbulence variables.
6. If the final time is not reached, return to step 2.

However in the present thesis, the very few transient cases are analyzed due to the limited computational resources available. But for those transient simulations, PIMPLE algorithm is used which is an extension of PISO and is more advantageous by being flexible with the courant number. For PISO algorithm, the courant number has to be strictly less than 1 all over the domain but it can be greater than 1 until few hundreds depending on the problem being simulated in the case of PIMPLE algorithm. The solution procedure of PIMPLE is very similar to PISO combined with the flexibility of SIMPLE algorithm implementation at each time step.
2.4 Calculation of turbulence parameters

Depending on the turbulence model being utilized for modelling the fluid flow, the turbulence parameters are to be initialized along with their conditions on the boundaries. The boundary conditions for different turbulence parameters required for k-kl-ω turbulence model and k-ω SST turbulence model are described in the section 3.2.1. However the calculation of the initial values for these parameters is explained in this section.

When the flow enters the domain at an inlet, outlet, or far-field boundary, OpenFOAM requires specification of transported turbulence quantities. This section provides guidelines for the most appropriate way of determining the inflow boundary values. The parameters required are of two types which include the physical parameters causing the turbulence and turbulence parameters calculated from these physical parameters. The two major physical parameters that are characterized due to various effects within the wind tunnel experiments include turbulent intensity (I) and viscosity ratio (µt/µ). The detailed study involving computational simulations to determine these two parameters is provided in the section 4.2.3.

The turbulence intensity (I) is defined as the ratio of the root-mean-square of the velocity fluctuations (u), to the mean flow velocity (u_avg). A turbulence intensity of 1 % or less is generally considered low and turbulence intensities greater than 10 % are considered high. Ideally, you will have a good estimate of the turbulence intensity at the inlet boundary from external, measured data. For example, if you are simulating a wind-tunnel experiment, the turbulence intensity in the free stream is usually available from the tunnel characteristics. In modern low-turbulence wind tunnels, the free-stream turbulence intensity may be as low as 0.05%. But in the present experimental data, I is not specified with in the experimental results.

After obtaining these parameters from either experimental conditions of turbulence persisted in the wind tunnel or from the validation results that give close values of aerodynamic coefficients close to experimental values which in itself is a trial and error method, the calculation of turbulence parameters is done. For k-kl-ω model, the three parameters kt, kl and omega are to be determined and for k-ω SST model k and omega are to be calculated from I and µt/µ. The kl in k-kl-ω model is initialized to zero at all boundaries due to its non-presence within the incoming flow.

The detailed explanation of the consideration of the below mathematical expressions for the calculation of k and omega can be found in the reference [45]. In short the below expressions will be utilized for the calculation of turbulence parameters.

The relationship between the turbulent kinetic energy (k), and turbulence intensity (I), is

\[ k = \frac{3}{2} (u_{avg}I)^2 \]  

(2.5)
2.5 Boundary Layer Theory and yPlus Calculation

The boundary layer theory is explained in many papers available including the references [46]. Also the one of major parameter for turbulence modeling calculation is done as in the reference [47].
Chapter 3

COMPLETE SIMULATION SETUP

3.1 Meshing

Firstly, ANSYS ICEM CFD 15.0 is chosen as meshing software due to its advantages in diverse blocking techniques to create structured meshes. As there is no complex geometry involved in the airfoil simulations, alternate meshing tools that are available commercially and also as open source can be utilized. However due to the wide range of resources available in the internet for meshing the airfoil using ICEM CFD, high quality meshes can be produced with minimal effort. It provides a variety of flexible tools that can take the model from any geometry to any solver in one modern and fully scriptable environment.

The computational domain as shown in the figure 3.1 is comprised of parabolic domain instead of conventional rectangular domain or semicircle upstream and rectangular downstream. This provides two major advantages which includes

- Flexibility in using the same mesh for different angles of attack by only shifting the tail part of mesh at airfoil trailing edge in accordance with the specific angle of attack. It is made sure that the edge of parabolic domain has enough vertical length to contain the incoming flow for angles of attack less than $25^\circ$.

- Reduction of number of cells in the mesh in the farfield (top and bottom). The parabolic part of the mesh cuts through both the conventional meshing domains aforementioned in the internal which signifies the reduction of the mesh count qualitatively. However, the quantitative reduction is not provided here due to the clear illustration in the qualitative approach.

The figure 3.1 shows the computational domain used for the present thesis. The conventional C-grid meshing is done in ICEM for the airfoil in 2D case for all the simulations. Due to the parabolic domain, the mesh has limited boundaries with INLET, OUTLET, WALL_UP, WALL_DOWN, SLOT (only in AFC implementation). WALL_UP and WALL_DOWN are the suction and pressure sides of the airfoil. The slot is incorporated in the suction side of the airfoil for AFC simulation cases. The boundaries are as marked in the figure 3.2.

The general mesh node count is $350 \times 360$ nodes where the former corresponds to the upstream and downstream of airfoil whereas the latter is the mesh node count in the total vertical direction. In addition the airfoil is split into 500 nodes laterally and the node count on airfoil is varied in different angles of attack to have more number of nodes in the high AOA’s to model the turbulence phenomena properly.
Figure 3.1 Mesh Computational Domain

Figure 3.2 Boundary labels of the mesh
The upstream of computational domain is 13 chords before the airfoil leading edge and 25 chords to the downstream of airfoil. The vertical edge of the parabola is 25 chords in each direction. The ratio of growth in the next cell layer is 1.05 near the leading edge, pressure and suction sides of airfoil. It is 1.1 downstream of trailing edge. However, to reduce the orthogonality and maintain good angles in the mesh cells, the far field vertical and horizontal cell layer growth ratio is 1.01. This is the tradeoff for the structured mesh in general but in this case due to the focus of the study close to the airfoil edges, this would be beneficial for the simulation.

With reference to the blocking techniques in ICEM, the 2D planar blocking is used for the parabolic domain. The front, top, bottom of the 2D block are associated with the inlet and the back part is associated with the outlet using the Associate Edge to Curve tool. Then a C grid is generated within the domain by using a combination of O grid with an edge using O grid block tool. Then the blocks are split accordingly to generate a mesh as shown in the figure 3.1. Then the mesh lines are divided in nodes and ratios with first cell layer distances are prescribed using Pre Mesh Parameters tool. The mesh quality tool is utilized to check the Aspect Ratio, Angle, 2x2 Determinant, Orthogonal quality, Ericsson Skewness with maximum and minimum values.

The close up view of the meshing around the airfoil is depicted in the figure 3.3. The boundary layer is very finely meshed to provide more than the required number of cells within the outer layer of the boundary for the implementation of $k-\omega$ and $k-\omega_t$ turbulence models. These models require $y+$ strictly less than 1 to model the boundary layer physics accurately. But as the present thesis requires clear resolution of the boundary layer phenomena, $y+$ is considered around 0.3. Hence the fine grid as shown in the figure 3.3 is obtained with such a value of $y+$. The grid dependency study is conducted with respect to both $y+$, lateral nodes on the airfoil and the results are provided in the section 4.1.4 and section 4.1.5.

### 3.2 OpenFOAM solver setup

This chapter presents a brief overview on implementation of OpenFOAM-2.2.1 with and without active flow control. Also the meshing strategy used in ANSYS ICEM CFD 15.0 is mentioned in section 3.1. This would be helpful for anyone to implement or continue the work done in present thesis for future.

OpenFOAM is a free, open source CFD software package developed by OpenCFD Ltd at ESI Group and distributed by the OpenFOAM Foundation. It has an extensive range of features to solve anything from complex fluid flows involving chemical reactions, turbulence and heat transfer, to solid dynamics and electromagnetics. Although there are wide range of commercial and open source CFD softwares that are as good as OpenFOAM for implementing active flow control which is the present case, it is specifically chosen to the below reasons.

- Free software. Other software packages are rather costly.
• Open source software. It is possible to adapt the code and build new functionalities.

• Is community driven. Various communities work on various fields of applications. This enables a fast evolution. Increasing popularity in industries.

• Can easily work in parallel with inbuilt OpenMPI environment.

• Already used for active flow control implementation.

• Wide range of options available for turbulence models, discretization schemes, linear solvers, boundary conditions and iteration monitoring

• Flexible with different preprocessed meshing file formats and also provides output for multiple post processing softwares.

• Although interface is not GUI (Graphical User Interface), it is easy to define parameters due to the perfectly organized case structure as shown in the figure 3.4

The figure 3.4 represents the exact case which is simulated with OpenFOAM in parallel. In brief, the 0 folder represents the boundary conditions of the parameters at all the boundaries in the mesh along with the initial values of the parameters mentioned under the ”0” in the figure 3.4 at time 0. The constant folder contains a full description of the case mesh in a subdirectory polyMesh and files specifying physical properties for the application concerned in the RAS, turbulent, transport properties files. The system folder is for setting parameters associated with the solution procedure itself. It contains at least 3 files controlDict, fvSchemes and fvSolution in which iterations and time stepping, discretization schemes and linear solvers are declared.
Boundary conditions and initial values

The 0 folder as shown in the above flow chart (figure 3.4) contains the files to define boundary conditions and initial values for pressure (p) and velocity (U) which are common for any case that needs to be simulated. The other parameters vary depending on the turbulence model being implemented which in this case is $k-\omega$ model. The turbulent kinetic energy ($k_t$), laminar kinetic energy ($k_l$), specific dissipation rate ($\omega$) and turbulent viscosity ($\nu_t$) are defined accordingly. For $k-\omega$ SST turbulence model, it would be only turbulent kinetic energy ($k$), specific dissipation rate ($\omega$) and turbulent viscosity ($\nu_t$). There is a wide range of boundary conditions as specified in detail in OpenFOAM [48].

The pressure is defined as shown in Appendix 1.1. As it is velocity inlet with specifically defined velocity magnitude, pressure is declared as zeroGradient at inlet. The outlet is 25 chord lengths downstream of airfoil and hence the fluid is allowed to stabilize and totally resolved before leaving the computational domain. Also as there would be no back flow in this case, the outlet is defined as zeroGradient for pressure. The wall (airfoil) is divided into WALL_UP and WALL_DOWN in this case and pressure condition is zeroGradient. In the active flow control implementation, the slot is made on the suction side (WALL_UP) with wall and zeroGradient boundary conditions. In the interior domain, the pressure is initialized to uniform value of 0 which refers to atmospheric pressure in the OpenFOAM solvers.

The Velocity is defined as shown in Appendix 1.1. The inlet velocity and the internal field of computational domain are of fixed value with a magnitude of 46 m/s which corresponds to a Reynolds number of $3.1 \times 10^6$. The outlet velocity is considered as zero gradient due to the availability of computational domain to the downstream of airfoil to stabilize and fully resolve the flow. The suction and pressure sides of airfoil
are considered as no slip walls with a fixed value velocity of zero. For implementing active flow control, the velocity is initialized with fixed value and a magnitude according to the velocity ratio \((U/U_\infty)\) being considered. The details of implementation of active flow control can be found in section 3.3

The boundary and initial conditions of k-kl-\(\omega\) turbulence model and its parameters are prescribed separately using \(k_t\), \(k_l\), \(\omega\) and \(\nu_t\). The laminar kinetic energy \((k_l)\) is defined with fixed value and an initial value of zero all across inlet, outlet, airfoil and internal field as specified in the Appendix 1.1. It is the amount of kinetic energy possessed by the flow until the transition phase occurs i.e. within the laminar flow section.

The turbulent kinetic energy \((k_t)\) is defined on boundaries involved and initialized as shown in Appendix 1.1. The boundary conditions of \(k_t\) at inlet and internal fields are declared as fixed value with the initial value calculated from the numerical formulas defined in section 2.4. This is similar to the turbulence introduced in the flow for wind tunnel experiments due to the non-ideal physical and atmospheric conditions which cannot create an atmospheric boundary layer. Hence the introduced undesired turbulence in the wind tunnel provides an error in the calculation of Pressure and viscous forces, which is to be corrected. So, the initialization of the same turbulence in the flow before reaching the test specimen has to done in the computational simulations also. Similarly, suction side and pressure side are defined with fixed Value condition with numerical value of zero. At outlet, the flow is stabilized due to the size of computational domain and the boundary condition is taken as zero gradient.

The specific dissipation rate \((\omega)\) is defined at all boundaries using the same physical phenomena mentioned in the turbulent kinetic energy \((k_t)\) boundary condition. Hence, it is fixed value at inlet and in internal field with the numerical value calculated from the mathematical expressions mentioned in the section 2.3. Similarly, it is zero gradient at outlet and a fixed value of zero at suction side and pressure side. The turbulent viscosity \((\nu_t)\) is not iterated in the computational simulation as it can be obtained from turbulent kinetic energy \((k_t)\) and specific dissipation rate \((\omega)\) as mentioned in the mathematical expressions in the section 2.3. Hence, it is defined as fixed value of zero in the internal field and the OpenFOAM primitive type boundary condition "calculated" in all other boundaries of computational domain. This boundary condition is not designed to be evaluated; it is assumed that the value is assigned via field assignment. Thus, the OpenFOAM calculates the numerical value for this parameter using the aforementioned parameters.

**Discretization schemes**

The fvSchemes dictionary in the system directory sets the numerical schemes for derivative and interpolation terms along with the specification of terms that require the generation of flux. The terms that must typically be assigned a numerical scheme in fvSchemes range from derivatives which include gradient, divergence, laplacian, surface normal gradient and time derivative terms along with interpolation terms for the values that are to be interpolated from one set of points to another. The exact
terms of the aforementioned types that are discretized are mentioned in the section 2.1.

OpenFOAM offers an unrestricted choice to the user with respect to each of these terms as mentioned in the reference [49] for gradient, divergence, laplacian, surface normal gradients, time derivatives and interpolation schemes respectively. The syntax to be used for different schemes is available in OpenFOAM user guide along with the reference [49] along with the OpenFOAM user guide.

Interpolation schemes

As it is described in the section 2.2, finite volume method generally performs the calculations of variables by resolving equations at each face for a single cell considered. However, the initial data for every iteration is available only at cell centers and hence using this data the interpolation of the values has to be done to the face of each cell in order to utilize these values while resolving the equations. This is where interpolations schemes come into picture and OpenFOAM provides wide range of options as mentioned in the reference [49] along with the OpenFOAM user guide.

The present thesis focuses only using Linear Interpolation scheme for General interpolation with Gaussian Integration.

Gradient schemes

The gradient schemes available within OpenFOAM are provided in the reference [49] along with the OpenFOAM user guide. Of these, the present thesis uses cell limited version of the Gauss integration with linear interpolation scheme as mentioned in the Appendix 1.1 under fvSchemes sub section. The least squares and fourth order schemes diverged and provided more in-stable results in this case. There are two types of limiting: cell and face. Cell limiting determines the limited gradient along a line connecting adjacent cell centers. Face limiting determines the limited gradient on the face itself. There is a MD version for each of these which stand for Multi-Dimensional limiter whereby the gradient is clipped in the direction normal the cell faces. The limited gradient schemes in OpenFOAM listed from least to most dissipation are cellMDLimited, cellLimited, faceMDLimited and faceLimited. Reducing dissipation in this case also has the potential benefit of better accuracy but increases the risk of instability. Hence the constant value 0.5 used here indicates both the stability and accuracy are given the importance as 1 turns the limiter completely and 0 does not use the limiter.

Divergence Schemes

The computational study with the variation of divergence schemes with respect to the order of the numerical behavior is done in the section 4.1.1. The divergence schemes available within OpenFOAM are provided in the reference [49] along with the OpenFOAM user guide. The TVD schemes which include limitedLinear, van-Leer, MUSCL, limitedCubic which as the names say add limiters to the second order schemes and try to provide the first order schemes stability. The normalized variable
differencing schemes like SFCD (Self-filtered central differencing) and Gamma Differ-
encing schemes are generally diffusive in nature for complicated problems. Also due
the high turbulence involved in this case, bounded schemes will try to prevent the di-
vergence of iterations mainly in turbulence parameters. The simulations were done by
using linear, linear upwind, upwind, limitedLinear, SFCD and Gamma schemes f or the
present thesis. However, the linear upwind and limited linear schemes provided stable
results but the latter is more accurate due to its nature of addition of limiter to a second
order scheme. So, limited linear scheme is mostly used throughout the computations
done in this thesis.

Laplacian schemes

The typical Laplacian term in fluid dynamics, $\nabla(\nabla U)$ is given the word identifier lapla-
cian(nu,U) in the fvSchemes as provided in the fvSchemes file of Appendix 1.1. The
Gauss scheme is the only choice of discretization and requires a selection of both
an interpolation scheme for the diffusion coefficient and a surface normal gradient
scheme, i.e. $\nabla U$. To summarize, the entries required are: Gauss interpolation-
Scheme, snGradScheme.

The interpolation scheme can be chosen from the reference [49] along with the
OpenFOAM user guide. However in the present thesis along with the general cases,
it is common to use linear interpolation scheme. With respect to the discretization
terms for the surface normal gradient terms, the scheme can be chosen from the table
3.4 based on the order of accuracy required and the boundedness of the scheme. In
the present thesis, corrected scheme is mostly used for the surface normal gradient
term with in the laplacian term. In some cases where the instability phenomenon is
predominant, limited scheme with a value of 0.5 for the corrector $\Psi$ is tested.

Surface Normal Gradient Schemes:

A surface normal gradient is evaluated at a cell face; it is the component, normal to
the face, of the gradient of values at the centers of the 2 cells that the face connects.
It is required to evaluate a Laplacian term using Gaussian integration as mentioned in
section (d) above. The schemes with their description are provided in the reference
[49] along with the OpenFOAM user guide. In most of the present cases, the corrected
scheme is utilized which can account for non-orthogonal correction. But however if the
mesh is good, limited scheme with the limiting parameter $\Psi$ with a value of 0.5 is used.

Time Schemes

The first time derivative ($\frac{\partial}{\partial t}$) terms are specified in the ddtSchemes sub-dictionary
of fvSchemes file as specified in the Appendix 1.4. The schemes available for time
discretization are provided in the reference [49] along with the OpenFOAM user guide.
In the transient cases analyzed in the present thesis, Euler schemes are utilized to
optimize the computational requirement with the results required.
**Linear Solvers**

The equation solvers, tolerances and algorithms are controlled from the `fvSolution` dictionary in the system directory. The `fvSolution` contains a set of sub dictionaries that are specific to the solver being run. However, there is a small set of standard sub dictionaries that cover most of those used by the standard solvers. These sub dictionaries include solvers, relaxation Factors, PISO and SIMPLE which are described in the remainder of this section.

Depending on the solver chosen in OpenFOAM, the `fvSolution` file has to be modified with the parameters being solved by that particular solver. The list of solvers available in OpenFOAM is huge [50] and depending on the problem being simulated, it has to be chosen. In the present case, Simple Foam is used for solving steady state cases and Pimple Foam is used to solve transient simulation cases. The `fvSolution` files for both of them are provided in Appendix 1.1 and 1.4. It should be noted that certain parameters are varied with the iterations as mentioned in the section 3.2.6 but the main solvers remain the same in most of the computations performed.

The first sub dictionary in the `fvSolution` file requires the solver to be defined for various parameters being analyzed. This generally includes pressure, velocity, turbulence parameters like kl, kt, and omega in the present thesis. However, depending on the case the parameters that are to be linearly solved are decided. The solvers available in OpenFOAM are as shown in the reference [51] along with the OpenFOAM user guide. Of these it is generally common to use either GAMG or PCG/PBiCG. However due to the many advantages provided by GAMG solver for multigrid problems, it is used as a default solver after no significant impact from the PBiCG solver. GAMG is faster than standard methods when the increase in speed by solving first on coarser meshes outweighs the additional costs of mesh refinement and mapping of field data.

Within the GAMG solver, the approximate mesh size at the most coarse level in terms of the number of cells is defined using the parameter `nCoarsestCells`. This parameter has to be played around in order to find the one that speeds up the considered simulation. For this case, it is generally 100 and is varied depending on the size of the mesh. The agglomeration of cells is performed by the algorithm specified by the `agglomerator` keyword which by default is `faceAreaPair` method. The `mergeLevels` keyword controls the speed at which coarsening or a refinement level is performed. It is considered to be 1 by default except when the meshes are very simple. The number of iterations for the parameter can be defined within the solver sub section.

Smoothing is specified by the smoother within the solver using the options provided in the reference [51] along with the OpenFOAM user guide. Generally GaussSeidel is the most reliable option, but for bad matrices DIC can offer better convergence. In some cases, additional post-smoothing using GaussSeidel is further beneficial, i.e. the method denoted as DICGaussSeidel. In this thesis, Gauss Siedel is utilized in most of the cases. The number of sweeps used by the smoother at different levels of mesh density is specified by the `nPreSweeps`, `nPostSweeps` and `nFinestSweeps` keywords. The `nPreSweeps` entry is used as the algorithm is coarsening the mesh, `nPostSweeps`
is used as the algorithm is refining, and nFinestSweeps is used when the solution is at its finest level.

There are a range of options for preconditioning of matrices in the conjugate gradient solvers, represented by the preconditioner keyword in the solver dictionary. The different preconditioners available in OpenFOAM are available in the reference [51] along with the OpenFOAM user guide. In this GAMG preconditioner is utilized.

Before solving an equation for a particular field, the initial residual is evaluated based on the current values of the field. After the each iteration by solver, the residual is re-evaluated. The solver stops if either of the following conditions is reached:

- the residual falls below the solver tolerance, tolerance
- the ratio of current to initial residuals falls below the solver relative tolerance, relTol
- the number of iterations exceeds a maximum number of iterations, maxIter

The tolerances, tolerance and relTol must be specified in the dictionaries for all solvers; maxIter is optional. In this thesis these parameters are varied according to the case being simulated and also depending on the parameter. For the pressure, reltol is generally 0.01 or 0.001 and for other parameters, it is 0.1. The tolerance is generally higher for pressure as compared to other parameters with a value of $10^{-6}$ to $10^{-8}$ and other parameters from $10^{-8}$ to $10^{-10}$.

The second sub-dictionary of fvSolution that is often used in OpenFOAM is relaxationFactors which controls under-relaxation, a technique used for improving stability of a computation, particularly in solving steady-state problems. As mentioned in section 2.3, the relaxation factor $\alpha$ can be tuned as per the case being simulated. An under-relaxation factor $\alpha$, $0 \leq \alpha \leq 1$ specifies the amount of under-relaxation. An optimum choice of $\alpha$ is one that is small enough to ensure stable computation but large enough to move the iterative process forward quickly.

- No specified $\alpha \implies$ no under-relaxation.
- $\alpha = 1 \implies$ guaranteed matrix diagonal equality/dominance.
- $\alpha$ decreases $\implies$ under-relaxation increases.
- $\alpha = 0 \implies$ solution does not change with successive iterations

In the present thesis, generally a value 0.3 for pressure and 0.7 for other parameters or 0.2 for pressure and 0.5 is used for other parameters depending on the stability of the simulation. Also in most of the cases, the simulation is monitored during the run time and the relaxation parameters are changed accordingly.
The third sub dictionary is used to define the pressure-velocity coupling algorithm. Most fluid dynamics solvers in OpenFOAM use the pressure-implicit split-operator (PISO) or Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithms. PIMPLE algorithm is a combination of both PISO and SIMPLE algorithms. Generally SIMPLE algorithm is used for steady state solvers with PISO and PIMPLE being used for transient solvers.

All the algorithms are based on evaluating some initial solutions and then correcting them. SIMPLE only makes 1 correction whereas PISO and PIMPLE requires more than 1, but typically not more than 4. The user must therefore specify the number of correctors in the PISO and PIMPLE dictionary by the nCorrectors. The number of non-orthogonal correctors is specified by the nNonOrthogonalCorrectors keyword. It can be 0 for an orthogonal mesh and increasing with the degree of non-orthogonality up to, say, 20 for the most non-orthogonal meshes. In general either 1 or 2 is used and it is changed during the run time for some cases.

Also, in a closed incompressible system, pressure is relative: it is the pressure range that matters not the absolute values. In these cases, the solver sets a reference level of pRefValue in cell pRefCell where p is the name of the pressure solution variable. Where the pressure is $p_{\text{rgh}}$, the names are $p_{\text{rghRefValue}}$ and $p_{\text{rghRefCell}}$ respectively. It is generally declared as 0 in this computational setup due to the interest in the relative magnitudes of the parameters and not the absolute values.

The more detailed information on the fvSolution file and its sub dictionaries can be obtained from the OpenFOAM users guide or from the reference [51].

Turbulence Models

OpenFOAM offers a large range of methods and models to simulate turbulence. The detailed list of turbulence models available within each of these methods [?]. The methods include:

- Reynolds-average simulation (RAS)
- Large eddy simulation (LES)
- Detached eddy simulation (DES)
- Direct numerical simulation (DNS)

The choice of turbulence model is dependent on the various reasons as mentioned in the section 2.1; the present thesis is focused on using RAS turbulence models or Reynolds Averaged Navier Stokes Turbulence models. Again, there is a wide range of options available depending on the nature of the flow [53]. Though some initial test cases were done using Spalart Allmaras and k Epsilon, due to the focus on studying the boundary layer, the choice of turbulence model with RAS models is highly dependent on the near wall treatment being considered with in the model.
The best model available within the list of options in RAS models for modelling near wall flow physics better is k Omega SST. But it does not consider the laminar flow that persists before the transition and the flow is considered fully turbulent. This has resulted in issues regarding the prediction of drag forces and there by the drag coefficient which highly depends on the resolving of near wall fluid flow. Hence, k-kl-omega turbulence model is considered as default in our case with validation of computational results being done also with k Omega SST model.

The implementation of turbulence model is to be done in two files namely turbulent properties in which the simulation type has to be defined which in this case is RAS Model and depending on the method used; this has to be changed accordingly. Then in the RAS Properties file of OpenFOAM, the specific RAS Model has to be chosen, whether the turbulence is on or the flow is laminar and the option to print coefficients has to be given. The two files, RAS Properties and turbulence Properties can be found in the Appendix 1.1. These are the same for with and without active flow control cases.

ControlDict

The OpenFOAM solvers set up a database with I/O controls and, since output of data is usually requested at intervals of time during the run, time is an inextricable part of the database. The controlDict dictionary sets input parameters essential for the creation of the database. The keyword entries can be found in the reference [54]. The below are the standard settings used in this thesis. Depending on the solver, if the case is transient, pimpleFoam might be used too. All the other parameters are as the same except the endTime which is the number of the global iterations or end time in transient simulations and can be changed accordingly as described in the reference [54].

The other major factor is the runTimeModifiable which is true in this case but if the simulation is properly setup with no other changes required during the iterations are progressed, it can be specified as false which can reduce the computational time by reducing the time needed to check for all the parameters before every iteration if they are changed or kept constant. The other parameters are described in detail with in the OpenFOAM user guide or the reference provided.

```
application simpleFoam;
startFrom latestTime;
startTime 0;
stopAt endTime;
endTime 35000;
deltaT 1;
writeControl timeStep;
writeInterval 5000;
purgeWrite 0;
writeFormat ascii;
writePrecision 6;
writeCompression off;
timeFormat general;
```
Complete Simulation Setup

```c
timePrecision 6;
runTimeModifiable yes;
```

Also for the calculation of lift and drag coefficients along with the forces, the lib-\forces.so has to be used within the control Dictionary and all the required parameters are to be mentioned. The code required for this implementation is obtained from the online forums available in reference [55]. It is also attached with in the controlDict file provided in the appendix 1.1. The major parameters changed within this code are the lift and drag axis depending on the angle of attack being simulated and accordingly the axes has to be shifted with reference to the OpenFOAM Cartesian coordinates which is the conventional coordinate system. The cosine and sine of the AOA being used has to be computed and utilized according to the axis.

**Parameters varied during the computation**

The Computational and physical parameters involved in the simulation are varied within the wide range of Options available in OpenFOAM and some of the results are documented with in the Testing of Parameters of OpenFOAM in section 4.1. The parameters that were played around include discretization schemes, boundary and initial conditions, linear solvers and its parameters, relaxation factors and tolerance values, mesh independent study with respect to yPlus and the lateral node distribution on the airfoil, Reynolds number, turbulence models and turbulence parameters. However this was mostly done before the validation of the computations with experiments is completed.

After the validation phase of the present thesis, the computational setup is mostly the same all through the baseline and active flow control implementation cases as provided in the appendix 1.1 and appendix 1.2 for steady state with and without active flow control cases. Also as provided in the appendix 1.4 for transient cases.

However, as the simulation is progressed, certain parameters are changed for better convergence depending on the progress. This includes reltol in fvSolution solvers subdictionary which is described in the section 2.2.3. By increasing the value of it from 0.01 to 0.001 for the pressure solver, the residuals within each iteration or time step are allowed to have a difference of order of two or three magnitudes which will result in the pressure value for the next iteration or time step accordingly. This will be 0.01 till the simulation is stabilized and once this happens, it is reduced to 0.001

The nCoarsestCells in the pressure solver GAMG which increases or decreases the number of the cells within the coarsest level being resolved by the solver which generally is changed from 50 to 150 if the iterations slow down as they are progressed with residuals n control.

The nPreSweeps, nPostSweeps and nFinestSweeps are changed from 0 to 2 to refine or stabilize the solution. The relaxation factors which are described in the section 2.3 and also in the section 5.2.3 are varied majorly as the simulation progresses with
0.1 to 0.3 for the pressure relaxation factor and 0.3 to 0.7 for the relaxation factors of U, \( k_t \), \( k_l \) and omega to stabilize or speed up the simulation. The \text{nOrthogonalcorrectors} which increases the number of overall iterations of pressure in order is changed to compensate for the pressure errors caused by the orthogonality of the mesh.

The \text{maxIter} parameter which describes the number of iterations that will be performed by each solver and is specified with in the solver is also changed. In general, the default value is 1000 but if there is not better convergence or the maximum iterations are being reached, the value is increased to 1500 or 2000 which also increases the computational time.

The end time in the \text{control Dict} is changed as the simulation is progressed after monitoring the residuals. Generally the number of iteration are either 25000 or 30000 for this case to see better convergence irrespective of the relaxation factors but however it is changed to 35000 to 45000 if the simulation has to be stabilized and a better convergence in \( c_d \) and \( c_l \) is required. Also the \text{write Interval} is changed from 5000 to 3000 if the iterations are to be monitored closely.

**Parallelization for implementing in the cluster**

The parallel implementation in OpenFOAM is straight forward and it is required to use the inbuilt openmpi modules for MPI parallelization of the computational domain for solving the case in the cluster. Parallelisation is robust and integrated at a low level, so in general, new applications require no 'parallel-specific' coding; they will run in parallel by default.

The utilities used for the parallel implementation of the present case within the cluster available at Department of Applied Physics, UPC- Barcelona Tech are mentioned within the job script used for launching simulations into the cluster. The major utilities include \text{decomposePar}, \text{renumberMesh}, \text{reconstructPar} and \text{mpirun} command. OpenFOAM employs domain decomposition, with its \text{decomposePar} utility, to split the mesh and fields into a number of sub-domains and allocate them to separate processors. The \text{renumberMesh} utility renumbers the cell list in order to reduce the bandwidth, reading and renumbering all fields from all the time directories. The \text{reconstructPar} utility reconstructs a decomposed case run in parallel. The syntax of this utilities is quite simple and in general they can just be used by mentioning the name of the utility but many number of options are available within each utility and can be used depending on the case. Along with the \text{mpirun} command is utilized to start and progress the iterations. The syntax of these utilities and commands in the present thesis are mentioned in appendix 1.3.

In addition, the numbers of subdomains are to be specified in the \text{decomposeParDict} file and the division of domains along the \( x \), \( y \) and \( z \) axes of the computational domain is also to be mentioned. Also the method \text{simple} is used to decompose the domain and there are other methods available too. The number of subdomains is in general 8, 12 or 16 depending on the size of the mesh and availability of the nodes. However
the n in simpleCoeffs sub dictionary which specified the number of subdomains along the different axes of computational domain is in general with 2 domains on the y axes and the rest along x axes. This division has provided results within less computational time as any change in the y subdomains has increased the computational time. The file used for the present thesis is provided in the appendix 1.3.

**Utilities in OpenFOAM**

There is wide range of utilities available within OpenFOAM and they are described within the OpenFOAM user guide along with the online reference [56]. However in the present case the utilities used include:

- **mapFields** to map volume fields from one mesh to another, reading and interpolating all fields present in the time directory of both cases. Parallel and non-parallel cases are handled without the need to reconstruct them first.

- **fluentMeshToFoam** to convert a Fluent mesh to OpenFOAM format including multiple region and region boundary handling.

- **checkMesh** to check validity of a mesh and also outputs the quality of mesh along with the mesh details.

- **wallShearStress** to calculate and write the wall shear stress, for the specified times when using RAS turbulence models.

- **yPlus** to calculate and report yPlus for all wall patches, for the specified times when using RAS turbulence models. This utility is the improved version of the yPlusRAS available in OpenFOAM which works only with wall functions. But the present thesis does not include any wall functions and hence the improved version of the yPlusRAS available in the cfd-online forum is used.

- **decomposePar** to automatically decompose a mesh and fields of a case for parallel execution of OpenFOAM.

- **redistributePar** to redistribute existing decomposed mesh and fields according to the current settings in the decomposeParDict file

- **reconstructParMesh** to reconstruct a mesh using geometric information only.

### 3.3 Implementation of active flow control

#### 3.3.1 Meshing

The meshing for active flow control is done the same way as mentioned in the section 3.1 but a slot is incorporated within the suction side of the airfoil by slicing the airfoil geometry without any changes to the curve. Depending on the slot location, the mesh is regenerated using the same techniques. However, the node number is varied for each slot location in order to maintain the required number of later nodes before and after the slot on the airfoil. The overview of the computational domain and the close up view of mesh near airfoil are attached in the figure 3.5 and 3.6 respectively. Due
to the more number of nodes used on the airfoil for reducing the aspect ratio, the AFC implementation is not clearly observed in the computational domain (figure 3.5). The aspect ratio is generally high due to the close location of the first mesh layer from the airfoil that is around $6 \times 10^{-6}$ m for most of the cases.

It should be observed in the figure 3.6 that the meshing behind the slot is much dense as compared to the mesh density in the pre slot region. This is attributed to study the control of instabilities of the turbulence around airfoil by the implementation of the Active Flow control. Also from the figure 3.7, it can be observed that the transition from the pre slot to slot to post slot location regions over the suction side of the airfoil are made to have a smooth transition with respect to the mesh cells.

The vertical mesh edges at the slot are divided in the same cell growth ratio along with the same distance of first layer of the mesh from the airfoil as the vertical mesh edges near leading and trailing edges of airfoil. The blocking is done in such a way that the final blocked mesh without any active flow control is divided with a separate block of mesh for slot. The upper edge of this block is associated with the slot location in the airfoil. And the mesh transition at all edges near slot is made smooth by having the same lateral spacing of the cells. The boundary file obtained from the .msh file generated from the ICEM can be found in the Appendix 1.2

### 3.3.2 Boundary condition implementation

The main implementation of the AFC for computational setup lies in the boundary condition and also the initial condition declaration. This provides a major advantage in the CFD without really have to manufacture the slot for actuators as in the experimental setups. The accuracy of the CFD simulation depends on a lot of parameters and sometimes it is even difficult to predict the source of the error.

The meshing is done in such way that the .msh file which is outputted from the ICEM has the airfoil as WALL UP for the suction side of the airfoil without slot, wall down for the pressure side of the airfoil and slot for the actuator which also is located in the suction side of the airfoil. The boundary type of the slot is also defined as wall but with slip due to the presence of the non-zero velocity near the wall.

The major parameter to be introduced in the boundary conditions is the velocity at the slot location. The slot has to be introduced in the velocity file located in 0 folder and it has to be defined as the fixed value type with a fixed velocity equal to the magnitude of the suction or blowing being evaluated computationally. Although we evaluate with respect to the velocity ratio with the inlet, $U_s/U_\infty \text{ or } U_b/U_\infty$, the velocity has to be provided with its magnitude in the x-direction and y-direction with respect to the regular Cartesian coordinate system used by the OpenFOAM for the reference. However, the angle of velocity being sucked or injected is evaluated with respect to the tangent at the slot. Hence this velocity vector has to be converted into x and y direction magnitudes with respect to OpenFOAM coordinate system just the same way as the inlet velocity vector is changed for different angles of attack by varying its x and y direction magnitudes.
Figure 3.5 Computational domain of the Active flow control implemented mesh

Figure 3.6 Close up view of active flow control mesh with the slot location at 70% C from the leading edge.
The pressure is defined using the zeroGradient type boundary condition and not being the same calculated boundary conditions. However, it is assumed that turbulence is introduced in the same manner as at inlet and internal field and hence a fixedValue with the numerical value calculated as mentioned in the section 2.4. It is pertinent to all the turbulence parameters involved like laminar kinetic energy ($k_l$), turbulent kinetic energy ($k_t$) and omega ($\omega$) for $k$-$k_l$-$\omega$ turbulence model. Similarly, turbulent kinetic energy ($k$) and omega ($\omega$) for $k$-$\omega$ SST turbulence model. The turbulent intensity ($I$) is considered the same as inlet along with the viscosity ratio which are 0.06% and 0.01 respectively.

It should be noted that these parameters are considered by making a cross check using the same setup with the diameter of pipe as the slot width and the flow is assumed to be fully developed within the pipe. Using this flow condition with the same velocity as at slot, the turbulence parameters are calculated with the internal flow within the pipe as reference and a fixed length pipe. However the values of the parameters $k_l$, $k_t$ and $\omega$ are the same as compared to the values obtained by utilizing the turbulent intensity ($I$) and viscosity ratio as considered at the inlet and internal field. Hence the initial values of turbulence parameters are considered with good assumptions.

Apart from the computational parameters, another parameter changed for active flow control implementation is the inclusion of slot in the forces and force coefficients calculation with in the respective functions described in the controlDict dictionary. Also relaxation parameters are defined with low magnitude values in the beginning of the computation till the flow reaches a stable state due to the modeling of the flow required near the slot.
Chapter 4

VALIDATION AND COMPUTATIONAL TESTING

4.1 Testing of computational Parameters

4.1.1 Variation of numerical discretization schemes

The numerical schemes used to discretize partial differential equations involved are varied with respect to the order of the schemes mainly strict first order, first/second order and strict second order schemes. These are varied typically for the divergence, gradient and Laplacian terms in the PDE's. The schemes that are available in OpenFOAM are described clearly in the section 4.4 of the OpenFOAM user guide provided by CFD Direct, the Architects of Open FOAM [http://cfd.direct/openfoam/user-guide/]. The numerical schemes for these three different cases are mentioned in the Appendix 1.4.

Table 4.1 Computational and Physical Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Values or Schemes</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_\infty$</td>
<td>46 m/s</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>$3.1 \times 10^6$</td>
</tr>
<tr>
<td>Discretization Schemes</td>
<td>Appendix 1.4</td>
</tr>
<tr>
<td>Linear Solver and parameters</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Time and data Input</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Boundaries</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>k-kl-$\omega$ model</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>$12^\circ$</td>
</tr>
</tbody>
</table>

The lift and drag coefficients for these three cases are as illustrated in the figures 4.1 and 4.2 respectively. As it can be clearly observed from the figures 4.4-4.6, the residuals with first order upwind scheme are stable. But the results are not on par with experimental results as seen in figures 4.1- 4.2. The second order linear scheme results are very close to experimental results but the stability has to be compromised and in this case it can be depicted in the huge oscillations of omega residual. However the First/second order schemes are a blend of both orders i.e., a limiter is added to the second order Linear scheme and utilized as limited Linear scheme. This enhances the stability of the second order scheme and calculates aerodynamic coefficients that are close to experimental results but the accuracy is compensated by the properly converged results.

However, as the present thesis is focused on near stall angles of attack, like 12, 13 and 14, it was obvious to consider the stability on par with accuracy due to the high Reynolds number and turbulence modeling involved. Also, the computational
Figure 4.1 Drag Coefficient (Cd) vs order of discretization schemes.

Figure 4.2 Lift Coefficient (Cl) vs order of discretization schemes.
Figure 4.3 Computational time (sec) vs order of discretization schemes.

Figure 4.4 Residuals using first order schemes.
Figure 4.5 Residuals using first/second order schemes

Figure 4.6 Residuals using second order schemes
time as specified in the figure 4.3 also plays an important role due to the large mesh (approximately 3 hundred thousand cells) being considered. Hence, the first/second order schemes specifically limited Linear scheme is utilized for divergence schemes, cellMDLimited Gauss linear 0.5 for divergence schemes and Gauss Linear corrected for Laplacian schemes as mentioned in the Appendix 1.4.

4.1.2 Variation of turbulence models

The turbulence is modelled only using Reynolds Averaged Navier Stokes (RANS) or Reynolds Averaged Stress (RAS) models. There is a wide range of this specific class turbulence models available in OpenFOAM as mentioned in the section 3.2.4.

The idea is to model the near wall flow physics better. So, k-ω Shear Stress Transport (2 equations), k-kl-ω (3 equations) were used from the list of available turbulence models for incompressible flows. Though k-ω models are well known for their near wall treatment within RANS turbulence models, it is a good idea to evaluate the performance for the specific cases of this thesis. The parameters used for simulation are mentioned in the table 4.2.

Table 4.2 Computational and Physical Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Values or Schemes</th>
</tr>
</thead>
<tbody>
<tr>
<td>U∞</td>
<td>46 m/s</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>3.1*10^6</td>
</tr>
<tr>
<td>Discretization Schemes</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Linear Solver and parameters</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Time and data Input</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Boundaries</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>k-ω SST, k-kl-ω model</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>0°, 8°, 12°</td>
</tr>
</tbody>
</table>

As can be seen from figure 4.7-4.8 and appendix table 4.2-4.4, the aerodynamic lift coefficient is better predicted using k-ω SST turbulence model than k-kl-ω model. The lift coefficient predicted by k-ω SST model differed from experimental value by 6%, 9.07% and 2.5% but the values predicted by k-kl-ω model differed with experimental values by 12.2%, 22.24% and 24.36 % for AOA 0, 8 and 12 respectively. However the drag coefficient is better predicted by k-kl-ω model than k-ω SST turbulence model. But the drag coefficient predicted by k-kl-ω differed from the experimental data by 9.2%, 25.3% and 5.6% and the values calculated by k-ω SST turbulence model differed from experimental values by 230%, 190% and 186 % respectively for AOA 0, 8 and 12 respectively.

This clearly illustrates that the k-kl-ω model outperforms the k-ω SST turbulence model in the near wall treatment by predicting the drag coefficient much closer to the experimental values. Also, this gives a better evaluation of wall shear stress and there by separation point. This is highly necessary when implementing active flow control as the separation point plays a crucial role in the location of actuation.
Figure 4.7 Lift Coefficient vs AOA for different turbulence models.

Figure 4.8 Drag Coefficient vs AOA for different turbulence models.
The main reason behind this better performance of k-kl-ω model is that it considers the transition to turbulence in an accurate way as compared to the k-ω SST from its third equation and this is the major difference between these two models. Thus the three parameters laminar kinetic energy(kl), turbulent kinetic energy(kt) and specific rate of dissipation(omega) are calculated in k-kl-ω model implementation whereas the k-ω SST considers only turbulent kinetic energy(k) and specific rate of dissipation(omega). Hence, k-kl-ω model will be a better choice for implementing active flow control and resolve near wall flow physics much better when compared to the k-ω SST model which is better only at predicting turbulent flow phenomena.

4.1.3 Variation in turbulence parameters

The turbulent parameters being considered play a vital role in validation with experimental results. As the turbulence levels of wind tunnels are quite different depending on various factors in its construction. Ideally, every wind tunnel is built with an objective to have zero level of turbulence before the fluid is interacted with the specimen being considered. But, when it comes to the real atmospheric environment, it is obvious that some level of turbulence is created before the airfoil or any other specimen being analyzed. In this case, as the validation is being done with the experimental results available in the theory of wing sections airfoil data [57], the conditions that are pertinent to the wind tunnel in which the experiments were conducted are to be used for the computational analysis too. However, due to the limited specific data available with in the book, we have to conduct the study of turbulence parameters and obtain the values in order to validate the results for different angles of attack. The computational and simulation parameters for this set of simulations are as provided in table 4.3.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Values or Schemes</th>
</tr>
</thead>
<tbody>
<tr>
<td>U_∞</td>
<td>46 m/s</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>3.1*10^6</td>
</tr>
<tr>
<td>Discretization Schemes</td>
<td>Appendix 1.4</td>
</tr>
<tr>
<td>Linear Solver and parameters</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Time and data Input</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Boundaries</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>k-ω SST model</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>0°</td>
</tr>
</tbody>
</table>

The two parameters that are to be analyzed include turbulence intensity(I) and viscosity ratio μ_t/μ. Using these two values, the parameters involved in the turbulence models can be initialized. The turbulence intensity is exactly specified as "few hundredths" in [57]. The exact magnitude is not clearly mentioned. This leaves with an assumption of the value for I within the range of 0.01% - 0.09%. However the viscosity ratio is not specified. So, it is required to obtain these parameters for the further validation of Cl and Cd coefficients for different angles of attack. Initially, both I and μ_t/μ are varied within the range of 0.01%-10% and 0.001-100 respectively for AOA 0°. It should be noted that parameters in the turbulence model can be calculated from either
viscosity ratio or turbulent length scale depending on whether the external or internal aerodynamics being considered respectively. The calculation of the turbulence model parameters from the turbulent intensity and viscosity ratio is specified in the section 2.4.

![Lift Coefficient vs Turbulent Intensity](image1.png)

**Figure 4.9** Lift Coefficient vs turbulent Intensity [Data Source: Appendix table 3.3]

![Drag Coefficient vs Turbulent Intensity](image2.png)

**Figure 4.10** Drag Coefficient vs turbulent Intensity [Data Source: Appendix table 3.3]

In order to obtain realistic inlet boundary conditions for the turbulence variables it is sometimes convenient to estimate the $\mu_t/\mu$. The main advantage with using the $\mu_t/\mu$ is that this directly says something about how strong the influence of the $\mu_t$ is compared to the molecular viscosity. $\mu_t/\mu$ is especially convenient to use in low-turbulence cases where it is difficult to guess any characteristic turbulent length scale. Typical examples
are external aerodynamics, like flow around cars, aircrafts and submarines. For internal flows and flows where the origin of the turbulence can be related to some physical features of the problem it is often better to instead estimate the turbulent length scale.

To check the dependency of turbulence intensity on aerodynamic coefficients, it is varied with a constant viscosity ratio of 5. It is clearly observed from the figure 4.9 and 4.10 that as the turbulent intensity (I) changes to higher orders of magnitude, the lift coefficient decreases and the drag coefficient increases. This can be related to the increasing turbulent intensity and thereby increasing the turbulent kinetic energy which affects both the pressure and viscous forces over the airfoil by increasing them. As it is mentioned in the physical conditions of turbulence considered in the experimental setup, it would be better to consider the turbulence intensity levels within few hundredths of percentage.

The physically valid extreme values of I and $\mu_t$ are tested computationally to obtain an overview on its impact on aerodynamic coefficient. From table 4.4, it clearly proves that by increasing viscosity ratio in larger magnitudes at higher values of I will decrease Cl and increase Cd and also are significantly varying from the experimental values. Hence, the viscosity ratio has to maintained low in magnitude.

Table 4.4 Variation of viscosity ratio at higher turbulence intensity

<table>
<thead>
<tr>
<th>Turbulent Intensity(I)</th>
<th>Viscosity Ratio</th>
<th>Cl</th>
<th>Cd</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental</td>
<td></td>
<td>0.24</td>
<td>0.0064</td>
</tr>
<tr>
<td>1</td>
<td>10</td>
<td>5</td>
<td>0.234052</td>
</tr>
<tr>
<td>2</td>
<td>10</td>
<td>10</td>
<td>0.212855</td>
</tr>
</tbody>
</table>

With the conclusion obtained from varying the I and $\mu_t$ over different orders of magnitude, different values within the few hundredths i.e., 0.01-0.09 percent are varied along with the various values of viscosity ratio. Although the lift and drag coefficient does not have a significant change by varying the viscosity ratio consistently, even with a large variation of magnitude in the case 10 from table 4.5. However the stability of the simulations has to be considered as the iterations diverged for the same case with only difference in turbulence parameters. This leads to considering the turbulent intensity of 0.06 for the rest of the simulations as it provides physically valid values for turbulence parameters i.e., turbulent kinetic energy and specific dissipation rate ($\omega$) as shown in the Appendix table 3.4.

The stability of the simulation involving the turbulent models is depended on the values of k and omega whose values are better for the computation if they are close to zero and close to infinity respectively for ideal cases. However, it can be depicted from the mathematical calculation formulas mentioned in section 2.4 that a lower value of turbulent kinetic energy lowers the magnitude of specific rate of dissipation($\omega$) and hence it is required to optimize the values of Turbulent intensity and viscosity ratio which would in turn provide the physically valid k and omega. Hence the case of I = 0.06 and $\mu_t = 0.01$ with physically valid results along with stable simulation results
Table 4.5 Variation of $\mu_P$ with $I$

<table>
<thead>
<tr>
<th>Experimental</th>
<th>Turbulent Intensity</th>
<th>Viscosity ratio</th>
<th>Cl</th>
<th>Cd</th>
</tr>
</thead>
<tbody>
<tr>
<td>-NA-</td>
<td>-NA-</td>
<td></td>
<td>0.26</td>
<td>0.0064</td>
</tr>
<tr>
<td>1</td>
<td>0.01</td>
<td>0.001</td>
<td>0.244107</td>
<td>0.0210803</td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>0.01</td>
<td>-Diverged</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.01</td>
<td>0.1</td>
<td>-Diverged</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.03</td>
<td>0.001</td>
<td>0.244027</td>
<td>0.0210811</td>
</tr>
<tr>
<td>5</td>
<td>0.03</td>
<td>0.01</td>
<td>-Diverged</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.03</td>
<td>0.1</td>
<td>-Diverged</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.06</td>
<td>0.001</td>
<td>0.244582</td>
<td>0.021074</td>
</tr>
<tr>
<td>8</td>
<td>0.06</td>
<td>0.01</td>
<td>0.25394</td>
<td>0.0209722</td>
</tr>
<tr>
<td>9</td>
<td>0.06</td>
<td>0.1</td>
<td>0.244356</td>
<td>0.0211242</td>
</tr>
<tr>
<td>10</td>
<td>0.06</td>
<td>10</td>
<td>0.245207</td>
<td>0.0211407</td>
</tr>
</tbody>
</table>

with lift and drag coefficient more close to experimental values is chosen for further simulations. However, it also should be noted that the difference in the aerodynamic coefficients is not very significant in all these cases. There is a difference in maximum and minimum values of aerodynamic lift and drag coefficients for the analyzed cases are 4.06 percent and 0.8 percent respectively which clearly indicates to consider the physically valid and computationally stable turbulence parameters.

4.1.4 Mesh independent study with yPlus

The meshing of geometry which plays a crucial role in the computation has to be checked for its independence on the results majorly by two parameters. The first one is the mesh dependency with respect to yPlus and the other one being the number of cells in the lateral direction of the airfoil. The definition of yPlus and its calculation is as defined in the section 2.5. A detailed computational study is conducted and the residuals of all parameters involved along with the aerodynamic coefficients are analyzed in detail for the different values of yPlus. Once the grid has provided the results independent of yPlus, the study on the number of cells in the lateral direction is conducted.

As the present thesis requires clearly resolved boundary layer phenomena, this prohibits the usage of wall functions. The theory that lies behind wall functions is the universal character of the law-of-the-wall. By utilizing the wall functions, it is asserted that under many flow conditions between the wall and the outer edge of the logarithmic layer is invariant when appropriate scaling is used. This appropriate scaling is the difference between computations with wall functions and without them. Hence, to study the effectiveness of active flow control, it is absolutely necessary to resolve the boundary layer without any approximations using the law of the wall theory.

As specified in the section 2.5, the boundary layer is divided into three layers which include, viscous sub layer, buffer and log layers depending on the conditions pertinent on dimensionless wall distance($y+$) and dimensionless velocity ($u+$). In theory, it is necessary for any computation without wall functions to have first layer of mesh nearest
to the wall by satisfying the condition of y+ less than 1. This physically means that the meshing is done in such a way that velocity, pressure and other parameters are calculated from the beginning of the viscous sub layer and also it is necessary to make sure that at least a total 40 layers of mesh lie within the outer edge of log layer to resolve the boundary layer properly. This requires a normal to the wall stretching of 1.15 assuming the y+ is less than 1 [45].

In this case, the computations are performed with initial yPlus of 2 and varied the mesh normal to the wall first layer distance in such a way that the value of the yPlus at the end of final iteration is less than 1 and specifically it is maintained to be even less than 0.3. As shown in the table 4.6, the meshing is done to have three different values of final iteration y+ less than 0.3.

**Table 4.6 Variation of yPlus**

<table>
<thead>
<tr>
<th>Wall Normal Distance</th>
<th>Y+</th>
<th>Total number of cells</th>
<th>Cl</th>
<th>Cd</th>
<th>Computational time(sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental</td>
<td>-NA-</td>
<td>-NA-</td>
<td>0.26</td>
<td>0.0064</td>
<td>-NA-</td>
</tr>
<tr>
<td>1</td>
<td>8e-6</td>
<td>0.33</td>
<td>158588</td>
<td>0.2807</td>
<td>0.00569</td>
</tr>
<tr>
<td>2</td>
<td>6e-6</td>
<td>0.24</td>
<td>201928</td>
<td>0.281</td>
<td>0.00566</td>
</tr>
<tr>
<td>3</td>
<td>4e-6</td>
<td>0.16</td>
<td>259388</td>
<td>0.2812</td>
<td>0.00562</td>
</tr>
</tbody>
</table>

Although it is mentioned that the study is by varying the value of yPlus, it has to be understood that the aspect ratio of the cells increases as the first layer distance normal to the wall decreases by a huge amount. For example, in this case, by moving wall normal distance (y) from $8e^{-6}$ to $6e^{-6}$ and keeping the number of cells in the lateral direction over the airfoil, aspect ratio moves up by 100%. So, it should be made sure to keep the aspect ratio of the mesh in control by adding more number of cells or nodes to mesh especially on wall i.e., airfoil. But it should be assumed that constraints here are aspect ratio and all other mesh quality parameters with the variable being yPlus.

It can be concluded from the above data obtained from the computations that by refining mesh with respect to better yPlus value, the difference between the maximum and minimum lift and drag coefficients is 0.17% and 1.2% and all the values are in line with experimental results considering the best data obtained during validation simulations with experimental results. This difference with experimental results persisted in the simulations due to the reasons mentioned in section 4.2.

However, the computational time required to complete the simulation on 8 computational cores for each of the simulation has increased with the decrease in wall normal distance or better yPlus. As it can be seen the computational time is significantly high for the third case with yPlus as 0.16. Hence there should be compromise with the accuracy having computational time as a constraint. As the results are quite accurate and the changes in the values of aerodynamic coefficients are less, the wall normal
distance of $6e^{-6}$ and $yPlus$ is chosen for further computations with the available computational resources in consideration. Also the chosen case is optimized with respect to computational time.

### 4.2 Validation with Experimental results

The validation of aerodynamic coefficients mainly $Cl$ and $Cd$ provides clearly the accurateness of computational study with respect to experimental results. This is generally an initial phase of CFD involved research projects to set the right parameters for simulation. The different parameters to be played around include boundary conditions, initial values, discretization schemes, linear solvers, meshing techniques, turbulence models, turbulence parameters, accuracy of the CFD code utilized and geometry of the specimen in consideration.

The impact of each of the aforementioned CFD study parameters can have a significant effect on the computational results and its comparison with the experimental data. For example, Baseline Validation of unstructured grid Reynolds-Averaged Navier-Stokes toward Flow Control performed by Ronald D.J. and Sally A.V [58] has illustrated that a small change in the geometry of the airfoil for incorporating actuator for AFC implementation has impacted the validation results with experiment. The same computational setup is used for the three geometries considered with small changes near leading and trailing edge as shown in the figure 4.2.1.

![Figure 4.11](image)

**Figure 4.11** Leading and Trailing edge geometries of NACA 0015, TAU 0015 and TAU 0015m airfoils.

As shown in the figure 4.2.2, the computed $Cl$ with variation in $\alpha$ for the NACA0015, TAU0015, and TAU0015m airfoils compared with the experimental data. For the NACA0015, the maximum $Cl$ and stall angle are 30% and 4 degrees higher than the experimental data. For the TAU0015m airfoil, the maximum $Cl$ and stall angle are 23% and 2 degrees higher than the experiments. In closer agreement, the Computed results for the TAU0015 airfoil approach the experimental results, overestimating the stall angle by 2 degrees and the maximum $Cl$ by 9%. So the subtle differences in geometry for the
NACA0015, TAU0015m and TAU0015 airfoils lead to large differences in the computed stall $\alpha$ and maximum Cl.

![Figure 4.12 Lift Coefficient vs AOA for all three airfoils with validation](image)

In the present thesis, the NACA 2412 airfoil is considered for implementing active flow control. The experimental data is obtained from the airfoil data available in [57]. It should be noted that the tolerance of the experimental values is around $\pm 0.01$ for lift coefficient and $\pm 0.001$ for drag coefficient due to the availability of graphical data only. Also, the experimental data is available only in two graphs with Cl vs AOA and Cl vs Cd. Hence the Cl vs AOA graph is done for AOA $0^\circ$ to $20^\circ$ whereas Cd vs AOA is done for AOA $0^\circ$ to $15^\circ$. As the Cl decreases after AOA 15 for this airfoil whereas the Cd keeps on increasing with AOA, the Cl vs Cd graph from which the Cd values are obtained cannot provide the drag coefficients for AOA greater than $15^\circ$.

The advantage of parabolic domain is utilized here by keeping the same mesh with the only change in the trailing edge tail part of the mesh. Depending on the angle of attack, the inlet velocity angle is changed in such a way to make an angle with the airfoil chord equivalent to the angle of attack being considered. This can be simple implemented in OpenFOAM by changing the initial value of velocity in the inlet and internal fields of the boundary. Also, the corresponding change is made in rotating the Cartesian coordinate system axes by the same angle for the calculation of aerodynamic coefficients.

It can be observed from the Appendix table 3.5 that computational iterations either diverge or have wild oscillations for both the turbulence models for angles of attack higher than 17 and 14 for $k-\omega$ SST and $k-k\ell-\omega$ turbulence models respectively. This is clearly due to the higher instability physical phenomena due to the high turbulence involved at higher angles of attack. So, the steady state simulations cannot provide
converging solutions for high AOA and it is necessary to use transient or unsteady solvers to model the turbulence involved and fluid flow.

The figure 4.2.3 shows that both the turbulence models have provided similar results with respect to Cl till the AOA $10^0$ with the k Omega SST case 1 providing closer results to experimental data. However by the addition of turbulence at airfoil in addition to inlet and internal field in k Omega SST case 2, the Cl increased significantly and provided results which differentiate from the experimental case very much. Also the k-kl-$\omega$ model provided computed similar results as k-$\omega$ SST case 2 till AOA $10^0$.

For the angles of attack higher than 100, which involve high instabilities, the flow cannot be modeled by the RANS turbulence models within the steady state as depicted in the figure 4.2.3. Both the turbulent models cannot clearly model the vorticity involved which has an effect on Cl. Hence the stall angle occurs $3^0$ before the experimental stall angle and 10 before for k-kl-$\omega$ and k-$\omega$ SST models respectively. This is one of the reasons in addition to those mentioned in for the variation of lift coefficient with the experimental data. Also, due to the incorporation of additional turbulence at airfoil in k Omega SST case 2, the stall phenomena is accurately predicted but the same case differentiates from the experimental data by a huge numerical value in the case of drag coefficient which is discussed below.

![Lift Coefficient vs Angle of Attack](image)

**Figure 4.13** Lift Coefficient vs AOA [Data Source: Appendix table 3.5]

The drag coefficient also varies from the experimental values in both the turbulence models as shown in the figure 4.2.4. The k-kl-$\omega$ model provides better Cd compared to k-$\omega$ SST due to its third equation which models the transition to turbulence clearly. This physical phenomenon cannot be accounted in the k-$\omega$ SST model which results in the higher Cd values. Also, the introduction of turbulence at wall in k Omega SST case
2 has clearly impacted the Cd value by increasing it which is the nature of turbulence. However, k-kl-ω model without any additional turbulence at airfoil modeled the viscous forces and pressure induced drag accurately. This also helps to conclude that there is no necessity to add additional turbulence at wall (airfoil) for the further simulations involving active flow control.

![Drag Coefficient vs Angle of Attack](image)

**Figure 4.14** Drag Coefficient vs AOA [Data Source: Appendix table 3.6]

However, both the turbulent models were not able to provide the Cd and Cl equivalent to experimental data. This could be related to multiple reasons like

- The airfoil coordinates provided in the experimental data have only 20 coordinates of (x,y). However a curve could not be fitted as a curve in the ICEM meshing software instead the straight lines appeared clearly in the airfoil geometry due to the limited number of points available to join the curve. This introduced wild oscillations in pressure, wall shear stress and other post processed parameters in paraview. So, the coordinates were interpolated and increased to 1000 points to remove this error. This induces a minor change in the airfoil curve which might be one of the reasons for the non equivalence of computational results with experimental data.

- The turbulence models used can model the turbulence phenomena only to a certain extent and as the angle of attack increases, the RANS turbulence model generally fail to model the flow physics accurately. 

- The turbulent parameters like I and \( \mu_r/\mu \) of the wind tunnel in which the experiments were carried are not provided clearly. This can definitely have an impact as shown in the section 4.1.3.
- Although it is mentioned that the surface is smooth and no external roughness is added, the practical or manufacturing errors introduce a certain degree of roughness to the airfoil.

- Also the discretization schemes used provide second order accuracy which can introduce numerical errors in the aerodynamic coefficients.

- The steady state cases analyzed can only provide accurate results to a certain extent and time evolution has to be incorporated in the computations to model the instability using time discretization.

- Also for higher angles of attack, due to the instabilities involved, the cd and cl follow a periodic oscillations with a variation of 0.05 for Cl and a variation of 0.005 for Cd. Hence a mean value is averaged which can introduce errors of the similar magnitude in few cases.
Chapter 5
ACTIVE FLOW CONTROL

5.1 Baseline cases for different locations

The meshing techniques for active flow control are different from the previous simulations as the slot is incorporated in the suction side of the airfoil within the same geometry of airfoil by slicing the airfoil curve according to the slot width and location used for actuation. Hence, as mentioned in the section 2.3, the node distribution on the airfoil is manipulated to have a smooth transition before and after slot along with the more number of elements for the slot.

This change in the meshing might have an impact on the simulation results which requires the baseline case study without implementing any active flow control for this mesh to see the impact of the same. The figure 5.1 below provides the aerodynamic coefficients values along with the separation point location. Any improvement in the flow physics will be compared with this data.

The computational and physical parameter utilized for this simulations are provided in the table 5.1

<table>
<thead>
<tr>
<th>Table 5.1 Computational and Physical Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
</tr>
<tr>
<td>U_∞</td>
</tr>
<tr>
<td>Reynolds number</td>
</tr>
<tr>
<td>Discretization Schemes</td>
</tr>
<tr>
<td>Linear Solver and parameters</td>
</tr>
<tr>
<td>Time and data Input</td>
</tr>
<tr>
<td>Boundaries</td>
</tr>
<tr>
<td>Turbulence model</td>
</tr>
<tr>
<td>Angle of Attack</td>
</tr>
</tbody>
</table>

* Note: The discontinuous lines refer to the diverged or oscillating simulations. However for the detailed information on particular case it is suggestible to refer to the data source tables in the appendix.

It is clear that in except the slot location at 60 % chord length from the leading edge; all other cases exhibit a change in the separation point with the progression of iterations although the range of separation is clear in the range of 79%C to 84%C. To understand this phenomenon better, transient simulations of the two different meshing
techniques with and without slot are launched. The separation points in both cases are monitored over time and are presented in the table 5.2 with the case 1 being without any slot in the geometry and case 2 with slot and both the cases are without active flow control.

Table 5.2 Variation of Separation Point with time

<table>
<thead>
<tr>
<th>Time (seconds)</th>
<th>Point of separation</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.001</td>
<td>0.847266</td>
</tr>
<tr>
<td>0.002</td>
<td>0.847266</td>
</tr>
<tr>
<td>0.003</td>
<td>0.847266</td>
</tr>
<tr>
<td>0.004</td>
<td>0.844778</td>
</tr>
<tr>
<td>0.005</td>
<td>0.849753</td>
</tr>
<tr>
<td>0.006</td>
<td>0.844778</td>
</tr>
<tr>
<td>0.0065</td>
<td>0.844778, 0.854726</td>
</tr>
<tr>
<td>0.007</td>
<td>0.832339, 0.847266, 0.824872</td>
</tr>
<tr>
<td>0.008</td>
<td>0.824872, 0.849753, 0.872123</td>
</tr>
<tr>
<td>0.009</td>
<td>0.817403, 0.837316, 0.839804, 0.844778</td>
</tr>
<tr>
<td>0.001</td>
<td>0.812422, 0.837316, 0.844778</td>
</tr>
</tbody>
</table>

The node distribution over airfoil before and after slot are presented in the table 5.4

The movement of separation point over airfoil with the increase in steady state number of total iterations is attributed to two main reasons.

- As can be seen from the Tables 5.2 and 5.3 and the wall shear stress plot in the figure 5.2 and 5.3 respectively, the separation point (where wall shear stress value tends to zero) remained the same for the duration of 0.003 seconds in case
active flow control 48

figure 5.2 wall shear stress vs time

<table>
<thead>
<tr>
<th>Time (seconds)</th>
<th>Point of separation</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0015</td>
<td>0.816688</td>
</tr>
<tr>
<td>0.003</td>
<td>0.813591</td>
</tr>
<tr>
<td>0.0045</td>
<td>0.813591</td>
</tr>
<tr>
<td>0.006</td>
<td>0.819784</td>
</tr>
<tr>
<td>0.0075</td>
<td>0.791901, 0.7981, 0.816688</td>
</tr>
<tr>
<td>0.009</td>
<td>0.779499, 0.785701, 0.819784, 0.832167</td>
</tr>
</tbody>
</table>

table 5.4 node distribution around slot

<table>
<thead>
<tr>
<th>location</th>
<th>before slot</th>
<th>slot</th>
<th>after slot</th>
</tr>
</thead>
<tbody>
<tr>
<td>40%</td>
<td>220</td>
<td>15</td>
<td>280</td>
</tr>
<tr>
<td>50%</td>
<td>220</td>
<td>15</td>
<td>280</td>
</tr>
<tr>
<td>60%</td>
<td>250</td>
<td>15</td>
<td>250</td>
</tr>
<tr>
<td>65%</td>
<td>260</td>
<td>15</td>
<td>240</td>
</tr>
<tr>
<td>70%</td>
<td>300</td>
<td>15</td>
<td>200</td>
</tr>
<tr>
<td>75%</td>
<td>270</td>
<td>15</td>
<td>230</td>
</tr>
</tbody>
</table>
1 and 0.006 seconds in case 2. As the time increased, the flow becomes clearly unsteady which can be seen in the wall shear plots clearly and the flow separates and reattaches over the length of the airfoil with more than one separation point after a duration of 0.006 seconds in case 1 and 0.0075 seconds in case 2. So when the flow is modeled in the steady state, the solver iterates and models the flow for one specific time from the transient case which results in the shifting of the separation point to one of these locations. So, it is logical to use the range for separation point and not a single point with respect to the turbulent flow physics involved.

- Also, the steady solver might be able to converge the solution and the separation point when the exact node distribution is done on airfoil to capture the aerodynamic properties like in the case of the 60% chord distance location from the leading edge which provides a converged point of separation and also Cd and Cl. The node distribution is as provided in the table 5.4. Although the number of elements are maintained the same over the airfoil by shifting the points using linear interpolation with respect to slot location, none of the other cases could resolve the converged separation point for increase in iterations. Also, the number of nodes on the airfoil was increased by a value of 100; this did not improve the result.

Hence, the non-capturing of exact separation point can be attributed to both of these reasons. Although, the separation point is approximately predicted in the baseline cases, this would not be a major concern. This is because the implementation of active flow control is expected to reduce the turbulence and delay the separation by significant value of chord length on the airfoil. This will be studied in the following sections. However, the baseline case for the reference of active flow control is the slot
location at 60% chord length from the leading edge which has provided a better steady state solution with converged Cd and Cl. The other cases when the range in which separation point and aerodynamic coefficients are varying is averaged, it comes very close to this case and hence it is utilized as a baseline case for all the simulations of active flow control.

It is also clear that the present thesis is more focused on substantial improvements in the separation delay and aerodynamic coefficients. So, the tolerance of 1-2 centimeters in the point of separation and and 2-3 % in the aerodynamic coefficients due to the convergence errors like averaging the parameters or the computational errors is acceptable. However it is expected that the AFC cases in generally provide better converged residuals and also aerodynamic coefficients due to the reduction in turbulence phenomenon and hence this tolerance is not required in the AFC cases.

5.2 Suction

The active flow control is initially implemented using boundary layer suction technique which is predominantly believed to delay the separation and reduce the turbulence over an airfoil. The steady or constant suction is simulated by the parameters which include location of actuation/suction ($L_s$), velocity ratio ($U_s/U_\infty$), angle of suction ($\beta$), slot width (b). As the case considered is steady, the frequency of actuation is not involved.

5.2.1 Variation of location of actuation/ suction

The slot location is varied by keeping the velocity ratio ($U_s/U_\infty$), slot width (b) and angle of suction ($\beta$) as constant. As mentioned in the section 3.3.2, the actuator is modeled using a boundary condition. There is no change in the geometry, but however a slot with width of approximately 1.5 mm with a tolerance of 0.05 mm is sliced from the airfoil geometry and mesh blocking is done as mentioned in section 3.3.1. The baseline cases for each of this location are simulated and the aerodynamic lift and drag coefficient along with point of separation are provided in the section 5.1.

The physical and computational parameters utilized for performing the Active Flow Control simulations are as provided in the table 5.5. It should be noted that as the simulation is in progress, parameters mentioned in section 2.2.6 are varied. Except them, the other parameters throughout the suction cases are as mentioned in the table 5.5 unless it is specifically mentioned in the case.

* Note: The discontinuous lines refer to the diverged or oscillating simulations. However for the detailed information on particular case it is suggestible to refer to the data source tables in the appendix.

Case 1:

It can be observed from the figures 5.4- 5.6 that the active flow control has some impact on the separation point at the $U_s/U_\infty = 0.3$. There is a delay in the point of
Table 5.5 Computational and Physical Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Values or Schemes</th>
</tr>
</thead>
<tbody>
<tr>
<td>( U_\infty )</td>
<td>46 m/s</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>( 3.1 \times 10^6 )</td>
</tr>
<tr>
<td>Discretization Schemes</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Linear Solver and parameters</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Time and data Input</td>
<td>Appendix 1.2</td>
</tr>
<tr>
<td>Boundaries</td>
<td>Appendix 1.2</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>k-(kl)-(\omega) model</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>12°</td>
</tr>
</tbody>
</table>

Figure 5.4 Separation point vs Slot Location for different cases of velocity ratio * [Data Source: Appendix table 2.2 - 2.5]

Table 5.6 Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
</tr>
<tr>
<td>( U_s/U_\infty )</td>
</tr>
<tr>
<td>( B )</td>
</tr>
<tr>
<td>( \beta )</td>
</tr>
<tr>
<td>Varied</td>
</tr>
<tr>
<td>( L_s )</td>
</tr>
</tbody>
</table>
Figure 5.5 Lift Coefficient vs Slot Location for different cases of velocity ratio *[Data Source: Appendix table 2.2-2.5]*

Figure 5.6 Drag Coefficient vs Slot Location for different cases of velocity ratio *[Data Source: Appendix table 2.2-2.5]*
separation and also increase in Cl with a decrease in Cd. Except the slot at 75%C has a higher drag than baseline case. For an overview on the impact of the AFC in this case, the best location with respect to improvements in aerodynamic coefficients is provided in table 5.7

The diverged case of slot location at 65% is attributed to the unsuitability of the $U_s/U_\infty$ ratio at the specific location. Also, it might be the node distribution as mentioned in the section 5.1, steady state simulations could not resolve the instabilities existing at the lower $U_s/U_\infty$ of 0.3.

**Table 5.7** Best case with respect to separation delay

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Slot Location</td>
<td>60%C</td>
</tr>
<tr>
<td>Separation delay</td>
<td>13.93%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>6.5%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-6.41%</td>
</tr>
</tbody>
</table>

**Case 2:**

**Table 5.8** Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td></td>
</tr>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.5</td>
</tr>
<tr>
<td>B</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$\beta$</td>
<td>90°</td>
</tr>
<tr>
<td>Varied</td>
<td></td>
</tr>
<tr>
<td>Ls</td>
<td>40%, 50%, 60%, 65%, 70% and 75% C</td>
</tr>
</tbody>
</table>

From the figures 5.4- 5.6, it can be depicted that all the slot locations being considered performed better than the baseline case except the 75%C location. The simulation with slot at 75% chord location did not properly predict the instabilities involved although it is clearly seen that there is a separation delay even with a lower $U_s/U_\infty$ ratio of 0.3 from the figure 5.1 and hence the range of separation points is provided in the Appendix table 2.3. This resulted in the oscillatory cd and cl even by increasing the iterations.

The delay in separation is clear along with the lift increment and drag decrement as compared to the baseline case. The constant suction with $U_s/U_\infty = 0.5$ has worked better as the slot is moved away from the separation point obtained in the baseline case. With respect to separation point and Cl this phenomena is until slot location of 60%C and the impact of AFC in this parameters reduces as the Ls is moved towards leading edge. However, drag keeps on decreasing until $L_s = 40$%C which can be attributed to the reduction in viscous forces as the boundary layer detaches earlier.
as compared to the slot at $L_s = 60\% C$. This can also be supported by the data from appendix table 2.3 in which drag reduction is higher compared to baseline case at $L_s = 60\% C$.

The best result in this case is as provided in the table 5.9. It should be noted that $Cl/Cd$ is best in the case of $L_s = 40\% C$ (111.05) as compared to $L_s = 60\% C$ (109.09) in reference to the baseline case (96.36). However, due to the main focus being on the separation delay, 60\% C is considered better. Also due to the simulation being two dimensional and doesn't account for span wise vorticity, focusing on the $Cl/Cd$ with a notion of efficiency is not logical. And, as mentioned in the objectives of the thesis, this study is being done for understanding the concrete boundary layer physics which would be similar in higher angles of attack and hence the separation delay plays an important role.

**Table 5.9 Best case with respect to separation delay**

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Slot Location</td>
<td>60% C</td>
</tr>
<tr>
<td>Separation delay</td>
<td>15.19%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>7.16%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-5.34%</td>
</tr>
</tbody>
</table>

**Case 3:**

**Table 5.10 Brief overview on parameters varied**

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td></td>
</tr>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.7</td>
</tr>
<tr>
<td>$B$</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$\beta$</td>
<td>90°</td>
</tr>
<tr>
<td>Varied</td>
<td></td>
</tr>
<tr>
<td>$L_s$</td>
<td>50%, 60%, 65%, 70% and 75% C</td>
</tr>
</tbody>
</table>

With $U_s/U_\infty = 0.7$, the active flow control phenomena has worked in a similar way as the case 2 above using $U_s/U_\infty = 0.5$. Except the $L_s = 75\% C$ also delayed the separation significantly compared to the lower velocity ratios of suction but the $Cd$ is higher than the baseline case which can be attributed to the combination of pressure and viscous drag forces. Along with the $L_s = 50\% C$ has diverged which arises the question of AFC with suction far away from the separation point in controlling the instabilities due to turbulence.
Also from the figures 5.4- 5.6, it can be observed that the Cl is almost the same with $U_s/U_{\infty} = 0.5$ and 0.7 for $L_s = 60\%C$ and 65\%C which can be noted as the saturation in the improvement due to the limited amount of instabilities at the considered AOA of 12° as compared to the higher AOAs like post stall angles of 16° – 18° for NACA 2412 airfoil. It can be predicted that any noticeable significant improvements in Cl at this velocity ratios will be observed in such Angles of Attack due to the presence of high pressure drag forces. In short aerodynamic parameters will be more sensitive to the AFC parameters at higher angles of attack with the boundary layer phenomena being observed here.

Also, reduction in the drag decrements from the table 5.7 to table 5.9 to table 5.11 is due to the dominance of viscous forces as compared to the pressure drag forces at this AOA. This can be countered and made to work positively only by increasing the AOA which has the higher pressure drag than the viscous drag forces. The figures 5.4- 5.6 provide more graphical illustration.

Table 5.11 Best case with respect to separation delay

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Slot Location</td>
<td>60%C</td>
</tr>
<tr>
<td>Separation delay</td>
<td>16.32%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>7.9%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-2.83%</td>
</tr>
</tbody>
</table>

Case 4:

Table 5.12 Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td></td>
</tr>
<tr>
<td>$U_s/U_{\infty}$</td>
<td>0.9</td>
</tr>
<tr>
<td>B</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$\beta$</td>
<td>90°</td>
</tr>
<tr>
<td>Varied</td>
<td></td>
</tr>
<tr>
<td>$L_s$</td>
<td>50%, 60%, 70% and 75% C</td>
</tr>
</tbody>
</table>

By using higher velocity ratio and varying the slot location, the boundary layer phenomena can be clearly understood from the figures 5.4- 5.6. As observed in the previous cases with lower velocity ratios, the trends in the separation delay and lift increment are similar but the cd values being higher than or close to the base line case conveys the domination of viscous forces with the over attachment of the boundary layer which in this case is above 95\%C as seen in figure 5.6 for $L_s = 50\%$, 60\% and 70\%. This is also supported by the continuing trend in the reduction in drag decrements from tables 5.7, 5.9, 5.11 and 5.13 as mentioned in the case 3.
This case also supports that active flow control through suction close to the separation point is not enhancing aerodynamic performance or the separation delay better than the slots located away from the point of separation. Also the divergence of iterations for $L_s=40\%$C and decreasing trend in Separation delay along with lift coefficient with $L_s = 50\%$C as compared to $L_s = 60\%$C supports that suction does not work better as the slot is moved far away from the point of separation. This leads to a critical point for boundary layer suction which in this case is at $L_s = 60\%$C that provides better results than the slot location before or after it.

This leads to the point of varying the velocity ratios, angles of suction and slot width at this location specifically to understand the critical point of suction phenomena. Also, it should be noted that few simulations with $L_s = 20\%$ and $30\%$C has diverged while performing computations at this present velocity ratio. However, due to the limited grid dependency study conducted at $L_s = 20\%$C and $30\%$C, the results are not documented here.

**Table 5.13** Best case with respect to separation delay

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Slot Location</td>
<td>60%C</td>
</tr>
<tr>
<td>Separation delay</td>
<td>17.45%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>8.87%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-0.85%</td>
</tr>
</tbody>
</table>

### 5.2.2 Variation of velocity magnitude

With a clear ideology on the impact of the location of the slot, the effect of the velocity magnitude is studied in this section by keeping the location of the actuator ($L_s$), slot width (b) and angle of suction ($\beta$) as constant. The magnitude of the suction velocity is changed in terms of velocity ratio of slot to the inlet velocity as in the previous section. The angle of suction is maintained constant at $90^\circ$ to the tangent of the slot in the negative direction which is suitable for boundary layer suction. However the effect of the angle of suction will be studied in the next section. The slot width is approximately 1.5 mm with a tolerance of 0.05 mm in all the cases.

It should be noted that some of the cases here can also be found in the section 5.2.1 mainly the velocity ratios of 0.5, 0.7 and 0.9 but the lower and higher extremes of velocity ratio are studied here. Also, by tabulating the results of variation of velocity with a specific location provides a clear platform to understand the aerodynamic performance and separation delay in detail.

* Note: The discontinuous lines refer to the diverged or oscillating simulations. However for the detailed information on particular case it is suggestible to refer to the data source tables in the appendix.
**Figure 5.7** Separation point vs Velocity ratio for different cases of Slot Location *[Data Source: Appendix table 2.6 - 2.11]*

**Figure 5.8** Lift Coefficient (C_l) vs Velocity Ratio for different cases of Slot Location *[Data Source: Appendix table 2.6 - 2.11]*

**Table 5.14** Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Constant</strong></td>
<td></td>
</tr>
<tr>
<td>( L_s )</td>
<td>75% C</td>
</tr>
<tr>
<td>( B )</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>( \beta )</td>
<td>90°</td>
</tr>
<tr>
<td><strong>Varied</strong></td>
<td></td>
</tr>
<tr>
<td>( U_s/U_\infty )</td>
<td>0.1, 0.2, 0.3, 0.5, 0.7, 0.9</td>
</tr>
</tbody>
</table>
Case 1:

As predicted in the section 5.2.1, location of slot close to the point of separation is very sensitive to active flow control. But here it can be seen from figures 5.7- 5.9 that it is sensitive to the velocity ratio also. For the lower value of $U_s/U_\infty$ especially 0.1 it performs worse than the baseline case and by increasing the ratio, the AFC works but the instabilities involved due to the turbulence are very clear visible in the oscillation of the separation points indicated in the appendix table 2.6.

These oscillations results in the unpredictability of the AFC phenomenon in the separation delay and also Cd and Cl through steady state simulations. Hence, to understand it clearly transient simulations have to be performed. As the objective of the thesis is to find the impact of varying the parameters and hence the detailed flow physics involved in the suction close to point of separation is not studied by transient cases. Also, the convergence with $U_s/U_\infty = 0.3$ can be credited to the unexpected reduction in the turbulence along with the favorable grid resolution with the node distribution for this specific level of turbulence as explained in the baseline cases within section 5.1.

It can be observed that although there are oscillations in the separation point which can be attributed to transient phenomena explained in the section 5.1, the trend is little clear that by increasing the velocity magnitude, the separation can be delayed in the slots close to the point of separation but the trade off involved is the magnitude of velocity which in turn relates to high power consumption by actuators for less performance improvement. It should be noticed that for the higher value of $U_s/U_\infty = 0.7$, with the attachment of boundary layer, the lift is enhanced and also the reduction in drag which is the significance of AFC with suction. But the lower values of Separation delay and improvements in Cl and Cd are related to the slot location.
Table 5.15 Best case with respect to separation delay

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.7</td>
</tr>
<tr>
<td>Separation delay</td>
<td>9.99%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>6.82%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-1.06%</td>
</tr>
</tbody>
</table>

Case 2:

Table 5.16 Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td></td>
</tr>
<tr>
<td>$L_s$</td>
<td>70% C</td>
</tr>
<tr>
<td>B</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$\beta$</td>
<td>90°</td>
</tr>
<tr>
<td>Varied</td>
<td></td>
</tr>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.1, 0.2, 0.3, 0.5, 0.7, 0.9</td>
</tr>
</tbody>
</table>

As predicted, varying the velocity magnitude provides an even better understanding about active flow control through suction phenomenon which is clearly depicted in the figures 5.7-5.9. The AFC works at the slot location of 70%C from $U_s/U_\infty = 0.2$ as the oscillations in separation point and cd and cl with $U_s/U_\infty = 0.1$ are related to the presence of clear instabilities in the suction side of the airfoil that signifies the less impact of AFC. However the separation delay keeps improving by increasing the velocity ratio from 0.2 to 0.9 as shown in the figure 5.7. Also the lift increment is also continuous which attributes the reduction in the pressure drag forces. However, the viscous forces have a little impact on the lift coefficient calculation as compared to pressure drag and the impact of AFC on the viscous forces is not observed.

The steep increments in the lift coefficient and also separation delay from 0.2-0.3 and 0.3-0.5 as compared to the change from 0.5-0.7 and 0.7-0.9 clearly illustrates that the instabilities and the turbulence involved for this specific angle of attack are reduced with a lower magnitudes of $U_s/U_\infty$, i.e. 0.5 (in this case) significantly and any further increment in the $U_s/U_\infty$ physically relates to the performance improvement with high power usage or less efficiency in a qualitative description. However, for higher AOAs than $12^{0}$, there would be a significant aerodynamic performance improvement only at higher velocity ratios as the instabilities or turbulence involved is much higher (which generally results in stall).

This phenomenon is even more clearly supported by the figure 5.9 in which it is clearly observed that Cd decreases as compared to the baseline case until the $U_s/U_\infty$ of 0.5. And the sudden increase in the value of Cd at $U_s/U_\infty$ from 0.5-0.7 is credited to the dominance of viscous drag forces once the pressure drag forces are reduced.
to the best possible extent by boundary layer suction. Hence any further separation delay or lift increment by increasing the velocity is also accompanied by the increase in the drag coefficient. This conveys that any practical implementation of AFC through suction needs the optimization of the parameters involved and results in negative consequences when not optimized. This phenomenon might occur at higher $U_s/U_\infty$ values for higher AOAs.

As aforementioned, the objective of the thesis is to delay the separation by reducing the turbulence and instabilities involved and any calculation of lift to drag coefficients and efficiency are not practically applicable. It is due to the present 2D computations excluding the span wise vorticity that can be observed only in 3D simulations. Hence, the best output for this case is the one that delays the separation farther and hence it is mentioned in the table 5.17 accordingly.

**Table 5.17** Best case with respect to separation delay

<table>
<thead>
<tr>
<th><strong>Best Output</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.9</td>
</tr>
<tr>
<td>Separation delay</td>
<td>16.23%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>8.5%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>+1.28%</td>
</tr>
</tbody>
</table>

**Case 3:**

**Table 5.18** Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td></td>
</tr>
<tr>
<td>$L_s$</td>
<td>65% C</td>
</tr>
<tr>
<td>$B$</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$\beta$</td>
<td>90°</td>
</tr>
<tr>
<td>Varied</td>
<td></td>
</tr>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.1, 0.2, 0.3, 0.5, 0.7, 0.9</td>
</tr>
</tbody>
</table>

For the slot location at 65% C, the figures 5.7- 5.9 should provide the identical results as the above case 2 (section 5.2.2) and they can be observed only for the $U_s/U_\infty$ of 0.5 and 0.7 in all the parameters being analyzed which include point of separation, Cd and Cl. However, the iterations either diverge or provide oscillating results which can be related to the two reasons.

- The node distribution of the pre and post slot location and its impact as described in the section section 5.1 respectively. As the $U_s/U_\infty$ of 0.9 has also provided diverging results inspite of the assured reduction in turbulence and instabilities by AFC through suction.
As the slot location moves away from the point of separation, the velocity magnitude required for the suction of the boundary layer will be relatively high and this might have provided oscillatory or diverging results for $U_s/U_\infty$ of 0.1, 0.2 and 0.3. It can be a combination of both these reasons.

Hence, the best output is calculated just from the available results

**Table 5.19 Best case with respect to separation delay**

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.7</td>
</tr>
<tr>
<td>Separation delay</td>
<td>15.77%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>7.88%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-2.13%</td>
</tr>
</tbody>
</table>

**Case 4:**

**Table 5.20 Brief overview on parameters varied**

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td></td>
</tr>
<tr>
<td>$L_s$</td>
<td>60% C</td>
</tr>
<tr>
<td>B</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$\beta$</td>
<td>90°</td>
</tr>
<tr>
<td>Varied</td>
<td></td>
</tr>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.3, 0.5, 0.7, 0.9, 1.5</td>
</tr>
</tbody>
</table>

For the slot location of 60% C at which the AFC through suction has provided better results and also was observed to be a critical location with respect to aerodynamic performance, the sensitivity of AFC parameters on the separation delay along with $C_d$ and $C_l$ is depicted in the figures 5.7- 5.9. Although the pattern of the Separation delay and lift coefficient variation by changing the velocity ratio is similar to the slot location of 70% C as described in case 2, the steep increments are not noticed in both of these parameters in figures 5.7 and 5.9. This is due to the present study being done with the $U_s/U_\infty$ starting at 0.3 with which AFC is clearly working with good impact. However the noticeable thing is the change of $U_s/U_\infty$ between 0.9-1.5 which can be observed in the appendix table 2.9. The Separation delay and lift increment is very low and supports the physical phenomenon (explained in the case 2 of the present section) of less pressure drag forces at AOA 12 relative to higher AOA's which indeed are reduced with low values of $U_s/U_\infty$. Again, at higher AOAs the improvement of separation delay and $C_l$ will be more observable with high steep (or slope).

Another observable finding is the $C_d$ increase from the starting ratios of $U_s/U_\infty$ i.e. 0.3-0.5 as against the $L_s$ at 70% C (case 2 of the present section) which again supports that at this specific location, reduction of pressure drag occurs at $U_s/U_\infty$ which immediately is followed by an increase in the viscous drag forces in a very sensitive
manner with respect to the velocity ratio. Also, it proves that at this location low velocity ratios might improve aerodynamic performance significantly. For higher AOs, the separation occurs earlier than the present AOA and hence this critical location might move toward the leading edge. As aforementioned due to the objective of the present thesis, the best output is with the $U_s/U_\infty$ of 1.5

**Table 5.21** Best case with respect to separation delay

<table>
<thead>
<tr>
<th>Best Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_s/U_\infty$</td>
</tr>
<tr>
<td>Separation delay</td>
</tr>
<tr>
<td>Lift Enhancement</td>
</tr>
<tr>
<td>Drag Decrement</td>
</tr>
</tbody>
</table>

**Case 5:**

**Table 5.22** Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
</tr>
<tr>
<td>$L_s$</td>
</tr>
<tr>
<td>$B$</td>
</tr>
<tr>
<td>$\beta$</td>
</tr>
<tr>
<td>Varied</td>
</tr>
<tr>
<td>$U_s/U_\infty$</td>
</tr>
</tbody>
</table>

From the above cases, it can be said that the $U_s/U_\infty$ of 0.5, 0.7, 0.9 generally work better in most of the present cases unless the slot location is close to the separation point or there is a sensitivity with respect to node distribution. Hence, the impact of these velocity ratios in the slot locations farther away from the point of separation is studied in this case. The results are illustrated in the figures 5.7-5.9.

The trend in separation point is similar to the previous cases with higher $U_s/U_\infty$ values delaying the separation better and also increasing the CI. The oscillations in CI and Cd for $U_s/U_\infty$ of 0.9 for $L_s$ at 40% C supports that the suction does not work better for farther slot locations from point of separation. Also the divergence of iterations for $U_s/U_\infty$ of 0.7 for $L_s$ at 50% C indicates the instabilities not being properly reduced. Also the data shows improvement in Separation point and CI is higher for $L_s = 50$% C than $L_s = 40$% C.

**5.2.3 Variation of angle of suction**

The impact of the angle of suction with respect to the tangent of the slot is simulated computationally to understand the variation in the aerodynamic coefficient and delay of separation by keeping the suction velocity magnitude ($U_s/U_\infty$), slot location ($L_s$) and
Table 5.23 Best case with respect to separation delay

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.9 at $L_s = 50%$C</td>
</tr>
<tr>
<td>Separation delay</td>
<td>17.21%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>8.4%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-1.06%</td>
</tr>
</tbody>
</table>

Table 5.24 Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td></td>
</tr>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.7, 0.9, 1</td>
</tr>
<tr>
<td>$b$</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$L_s$</td>
<td>70% C</td>
</tr>
<tr>
<td>$\beta$</td>
<td>90\degree</td>
</tr>
<tr>
<td>Varied</td>
<td></td>
</tr>
<tr>
<td>$\alpha$</td>
<td>12\degree, 13\degree</td>
</tr>
</tbody>
</table>

Slot width ($b$) constant. The angles of suction considered are 30\degree, 45\degree, 90\degree, 150\degree for the two slot locations 70\% and 75\% chord length from the leading edge as case 1 and case 2 respectively. As it can be observed from the case 1 in section 5.2.2, $L_s = 75\%$ did not have a significant improvement with respect to $C_d$ and $C_l$ along with point of separation. So, it is considered specifically by varying angles to see any specific impact of the angle of suction. Also, the $L_s = 70\%$ is one of the standard test cases with good improvements of $C_d$, $C_l$ and point of separation in reference to the slot location.

The suction velocity magnitude is kept constant by a value of $U_s/U_\infty = 0.3$. As it can be observed from the section 5.2.1 and 5.2.2, this specific velocity ratio is the one at which active flow control has started to provide improvements with respect to aerodynamic performance. Hence, any effect of the angle of suction can be clearly noticed with this suction velocity. The slot width is 1.5 mm with a tolerance of 0.05 mm.

Case 1:

Table 5.25 Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td></td>
</tr>
<tr>
<td>$U_s/U_\infty$</td>
<td>0.3</td>
</tr>
<tr>
<td>$b$</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$L_s$</td>
<td>75% C</td>
</tr>
<tr>
<td>Varied</td>
<td></td>
</tr>
<tr>
<td>$\beta$</td>
<td>30\degree, 45\degree, 90\degree, 150\degree</td>
</tr>
</tbody>
</table>
It is clearly evident from the graphs 5.10-5.12 that by changing the angle of suction at the most sensitive slot location for suction in this present thesis, the aerodynamic performance is not impacted. Although the separation seems to be delayed with respect to baseline case, there are oscillations with increase in the number of iterations in all other angles of suction except 90°. Also, these oscillations pertained in the case of lift and drag coefficients too. Though the reason for this phenomenon is not clearly known from the steady state simulations, it might be due to sensitivity of the slot location or the unsupportive flow physics through AFC suction for separation delay. To understand this clearly, the transient cases has to be simulated. However as this understanding is not part of the objective of the thesis, it is not considered in a serious manner.

**Table 5.26 Best case with respect to separation delay**

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle of suction</td>
<td>90°</td>
</tr>
<tr>
<td>Separation delay</td>
<td>6.4%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>4.26%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>+13.04%</td>
</tr>
</tbody>
</table>

![Figure 5.10 Separation Point vs Angle of Suction](Data Appendix table 2.12)

**Figure 5.10 Separation Point vs Angle of Suction**

**Case 2:**

The impact on the aerodynamic performance by varying the angle of suction at $L_s = 70\%C$ is presented in the figures 5.13-5.15. It clearly illustrates no other angle has worked for AFC suction better than 90°. This can be seen in the separation delay, Cd and Cl. Although the iterations converged with an angle of suction 30°, the impact is much lesser than 90°. There might be a possibility of improved performance by increasing the velocity ratio to higher $U_s/U_{\infty}$ values but as the 90° worked better at lower $U_s/U_{\infty}$ values, the angle of suction is not varied with any other parameter. However to
Table 5.27 Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Constant</strong></td>
<td></td>
</tr>
<tr>
<td>$U_s / U_\infty$</td>
<td>0.3</td>
</tr>
<tr>
<td>$b$</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$L_s$</td>
<td>70% C</td>
</tr>
<tr>
<td><strong>Varied</strong></td>
<td></td>
</tr>
<tr>
<td>$\beta$</td>
<td>$30^\circ, 45^\circ, 90^\circ, 150^\circ$</td>
</tr>
</tbody>
</table>
understand the strange phenomenon of the performance of 150° with which the separations occurred much earlier than the slot, it is required to study the transient cases which is not the objective of the present thesis. So, it can be concluded that angle of suction of 90° works better for suction.

Table 5.28 Best case with respect to separation delay

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle of suction</td>
<td>90°</td>
</tr>
<tr>
<td>Separation delay</td>
<td>6.46%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>4.86%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-3.9%</td>
</tr>
</tbody>
</table>

Figure 5.13 Separation Point (%C) vs Angle of Suction [Data Source: Appendix table 2.13]

5.2.4 Variation of Slot width

Table 5.29 Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
</tr>
<tr>
<td>$U_s/U_\infty$</td>
</tr>
<tr>
<td>$L_s$</td>
</tr>
<tr>
<td>$\beta$</td>
</tr>
<tr>
<td>Varied</td>
</tr>
<tr>
<td>$b$</td>
</tr>
</tbody>
</table>
**Figure 5.14** Lift Coefficient (Cl) vs Angle of Suction [Data Source: Appendix table 2.13]

**Figure 5.15** Drag Coefficient (Cd) vs Angle of Suction [Data Source: Appendix table 2.13]
From the figures 5.16 to 5.18 it can be observed by increasing the slot width the aerodynamic performance significantly increases especially lift coefficient (Cl) and delay in the point of separation whereas this is accompanied by the increase in Cd too which is due to the phenomenon being seen in the previous suction cases wherein the pressure drag reduction is accompanied by the increase in the viscous drag forces due to the attachment of the boundary layer and this is again proved in this case.

The point of separation significantly increased and in turn cl also has improved in similar magnitudes and also the increase in cd is led to its rise above the baseline case. However as aforementioned the main focus is on the separation delay and hence the best case here is the slot width of 4.6 mm.

Also as observed from the table 5.30 the magnitudes of improvements of the best case of 4.6 mm is much higher than all other suction cases at this velocity ratio and the slot location especially separation delay and lift enhancement. Also the drag is also increased by a very high value. However as mentioned in the previous sections and also proved in the section 5.2.5, the cd reduces with increase in AOA where the pressure drag forces dominates very much as compared to viscous drag forces.

**Table 5.30 Best case with respect to separation delay**

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>slot width</td>
<td>4.6 mm</td>
</tr>
<tr>
<td>Separation delay</td>
<td>19.36%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>10.4%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>+4.5%</td>
</tr>
</tbody>
</table>

**Figure 5.16** Separation Point (%C) vs slot width (b) [Data Source: Appendix table 2.14]
Figure 5.17 Lift Coefficient (Cl) vs slot width (b) [Data Source: Appendix table 2.14]

Figure 5.18 Drag Coefficient (Cd) vs slot width (b) [Data Source: Appendix table 2.14]
5.2.5 Variation of Angle of Attack

From the detailed observations made on the impact of the active flow control parameters using constant suction on the separation delay, lift and drag coefficients, the next step would be to check the performance of these parameters at higher angles of attack. As the previous cases were performed at an AOA of $12^\circ$, the present simulations were made at AOA $13^\circ$ and the comparison of the $cd$ and $cl$ with the baseline simulations of this AOA is done in the below tables and figures.

Although the AFC computations with slot location far away from the point of separation has worked better with respect to aerodynamic performance, this present case studies the AFC implementation with slot location at 70% by varying the velocity ratio at higher values. This set of computations are performed with an objective to check the improvements at higher angles of attack and in a way to cross check the pattern of the obtained results in the sections 5.2.1-5.2.4. Also, to qualitatively compare the effectiveness when the flow is sucked with a velocity equivalent to inlet velocity. Hence, only limited cases are evaluated at an AOA of $13^\circ$.

From the figure 5.19, it can be observed that active flow control clearly works for AOA $13^\circ$. The separation is delayed as compared to the baseline case. The major observation here is the effectiveness of AFC in delaying the separation by 21.04% with a velocity ratio of 1 for AOA $13^\circ$. However the separation is delayed by only 16.23% with a velocity ratio of 0.9 for AOA $12^\circ$. It should be noted that in the case of AOA $12^\circ$ even though the computation is performed with a velocity ratio 0.1 less in magnitude, a difference in 0.1 of velocity ratio will not make a significant difference. As the separation is delayed by 14.8% with a velocity ratio of 0.7 and the difference is just 1.43% for an increase of velocity ratio by 0.2.

Also the lift coefficient is increased by 11.88% for AOA 13 with the suction velocity ratio of 1 as compared to 8.5% with a suction velocity ratio of 0.9 for AOA $12^\circ$. The drag coefficient is increased by 0.12% with a suction velocity ratio of 1 AOA 130 as compared to the increased by 1.28% for AOA $12^\circ$ with a suction velocity ratio of 0.9. It can be observed that although the pressure drag forces are reduced which resulted in a significant increase in the $Cl$ for AOA $12^\circ$ as compared to the $12^\circ$, the viscous drag forces still persist resulting in little increase in the $Cd$ as compared to the baseline case for both angles of attack.

The velocity fields for the best case attached in this section for AOA 12 with velocity ratio of 0.9 and slot location at 60%C with slot width of 1.5 mm are attached in the figures 5.22 and 5.23 for without and with Active Flow control respectively. The wall shear stress magnitude on the suction side of the airfoil whose zero value providing the point of separation are provided in the figures 5.24 and figures 5.25 for without and with Active Flow control respectively for the same case.

Similarly, velocity fields for the best case attached in this section for AOA 13 with velocity ratio of 1 and slot location at 70%C with slot width of 1.5 mm are attached in
**Figure 5.19** Separation Point (%C) vs slot width (b) [Data Source: Appendix table 2.14]

**Figure 5.20** Lift Coefficient (Cl) vs slot width (b) [Data Source: Appendix table 2.14]

**Figure 5.21** Drag Coefficient (Cd) vs slot width (b) [Data Source: Appendix table 2.14]
**Figure 5.22** Velocity magnitude for AOA 12 without Active flow Control

**Figure 5.23** Velocity magnitude for AOA 12 with Active flow Control through suction
**Figure 5.24** Wall shear stress indicating the Point of Separation (blue) (No AFC)

**Figure 5.25** Wall shear stress indicating the Point of Separation (blue) (AFC)
Figure 5.26 Velocity magnitude for AOA 13 without Active flow Control

Figure 5.27 Velocity magnitude for AOA 13 with Active flow Control through suction
Figure 5.28 Wall shear stress indicating the Point of Separation (blue) (No AFC)

Figure 5.29 Wall shear stress indicating the Point of Separation (blue) (AFC)
the figures 5.26 and 5.27 for without and with Active Flow control respectively. The wall shear stress magnitude on the suction side of the airfoil are provided in the figures 5.28 and 5.29 for without and with Active Flow control respectively for the same case.

5.3 Blowing

The constant blowing case is studied by varying the parameters in the similar pattern as done with the suction case. However, with an overview on the magnitude of parameters obtained from the constant suction implementation as part of AFC simulations, the initial testing of blowing is done using the similar magnitudes of velocity ratios, slot locations and angles of suction. However slot width is kept constant in this case with a value of 1.5 mm all through the constant blowing implementation.

Also it should be noted that the symbols utilized here for the same parameters are a little different to distinguish suction parameters from blowing. The velocity ratio \( U_b/U_\infty \), slot location \( L_b \), blowing angle \( \beta \), slot width \( b \) are considered. The computational parameters are used as mentioned in table 5.31

**Table 5.31 Computational and Physical Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Values or Schemes</th>
</tr>
</thead>
<tbody>
<tr>
<td>( U_\infty )</td>
<td>46 m/s</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>( 3.1 \times 10^6 )</td>
</tr>
<tr>
<td>Discretization Schemes</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Linear Solver and parameters</td>
<td>Appendix 1.1</td>
</tr>
<tr>
<td>Time and data Input</td>
<td>Appendix 1.2</td>
</tr>
<tr>
<td>Boundaries</td>
<td>Appendix 1.2</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>k-( \omega )-( \omega ) model</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>12°</td>
</tr>
</tbody>
</table>

5.3.1 Variation with location

**Table 5.32 Brief overview on parameters varied**

<table>
<thead>
<tr>
<th>AFC parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Constant</strong></td>
</tr>
<tr>
<td>( U_{sb}/U_\infty )</td>
</tr>
<tr>
<td>B</td>
</tr>
<tr>
<td>( \beta )</td>
</tr>
<tr>
<td><strong>Varied</strong></td>
</tr>
<tr>
<td>Ls</td>
</tr>
<tr>
<td>60%, 65%, 70%, 75%C and 80% C</td>
</tr>
</tbody>
</table>
With the observations made in the AFC using constant suction, the parameters are varied in a similar way as aforementioned. By keeping the velocity ratio \( (U_b/U_\infty) \) as 1 and angle of blowing as 30\(^\circ\), the slot location is varied along the suction side of airfoil and the results obtained are provided in the figures 5.30-5.32.

From the figure 5.30, it can be clearly depicted that as the slot location is close to the point of separation, the AFC with constant blowing has worked better which is contradictory to the constant suction in which the slot location away from the point of the separation has worked better. Also it can be observed from all the figures that as the slot location is moved away from the point of separation the impact of the blowing on the aerodynamic performance is reduced very much.

Also, the lift coefficient has increased above the baseline value for only slot locations close to the separation point. However, the increase in the lift coefficient is not significant with respect to the numerical value and it has just increased by 1.55% for slot location at 80%C which is very less as compared to the enhancements obtained in the suction case which are generally around 5-8% and this magnitude of \( U_b/U_\infty \), it is 8.8% as mentioned in the case 4 of section 5.2.1.

Similarly, the drag coefficient is also lower in magnitude for the slot location close to the point of the separation which signifies a better aerodynamic performance than the baseline case. The decrement in the Cd value is 3.74% for the slot location of 80%C. This again is in contradiction to the constant suction however this is desirable. For the suction case as mentioned in the section 5.2.1, the drag has increased at higher velocity ratios due to the increase in viscous drag accompanied by reduction in pressure drag.

This leads to the conclusion that constant suction and blowing do not go on par with each other and in this case the consequences are vice versa. The main concern here is although the separation point has significantly moved downstream for the slot location close to the point of separation; the enhancement in Cl is not exactly matching the consequences of delaying the separation

Table 5.33 Best case with respect to separation delay

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Slot Location</td>
<td>80%C</td>
</tr>
<tr>
<td>Separation delay</td>
<td>17.5%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>1.55%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-3.74%</td>
</tr>
</tbody>
</table>

5.3.2 Variation of velocity

It can be observed from the figures 5.33-5.35 that the impact of changing the velocity ratios from lower magnitudes to higher values has the similar effect as in the case of suction. The iterations are diverged with the velocity ratio of 0.1 which indicates that the magnitude is not enough to activate the effects of active flow control through blowing.
**Figure 5.30** Separation Point (%C) vs Slot location [Data Source: Appendix table 3.1]

**Figure 5.31** Lift Coefficient (Cl) vs Slot location [Data Source: Appendix table 3.1]
Figure 5.32 Drag Coefficient (Cd) vs Slot location [Data Source: Appendix table 3.1]

Table 5.34 Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td></td>
</tr>
<tr>
<td>$L_s$</td>
<td>80% C</td>
</tr>
<tr>
<td>$b$</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$\beta$</td>
<td>30°</td>
</tr>
<tr>
<td>Varied</td>
<td></td>
</tr>
<tr>
<td>$U_b/U_\infty$</td>
<td>0.1, 0.5, 1</td>
</tr>
</tbody>
</table>

The point of separation has delayed and moved downstream by a significant magnitude as the magnitude of blowing velocity is increased from 0.5 to 1. The impact was similar in the case of aerodynamic coefficients too and both the drag and lift coefficients increased with the velocity ratio increment. This phenomenon by varying the velocity magnitude is similar to the suction cases in which velocity is varied. The best output is the one as mentioned in the previous case.

Table 5.35 Best case with respect to separation delay

<table>
<thead>
<tr>
<th>Best Output</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_s/U_\infty$</td>
<td>1</td>
</tr>
<tr>
<td>Separation delay</td>
<td>17.55%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>1.55%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-3.74%</td>
</tr>
</tbody>
</table>
Figure 5.33 Separation Point (%C) vs Velocity Ratio \((U_b/U_\infty)\) [Data Source: Appendix table 3.2]

Figure 5.34 Lift Coefficient (Cl) vs Velocity Ratio \((U_b/U_\infty)\) [Data Source: Appendix table 3.2]
Figure 5.35 Drag Coefficient (Cd) vs Velocity Ratio ($U_b/U_\infty$)[Data Source: Appendix table 3.2]

5.3.3 Variation of angle

Table 5.36 Brief overview on parameters varied

<table>
<thead>
<tr>
<th>AFC parameters</th>
<th>Constant</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_b/U_\infty$</td>
<td>1</td>
</tr>
<tr>
<td>b</td>
<td>1.5 mm</td>
</tr>
<tr>
<td>$L_b$</td>
<td>80% C</td>
</tr>
</tbody>
</table>

Varied

| $\beta$ | $15^\circ, 30^\circ, 45^\circ, 90^\circ, 120^\circ$ |

With an understanding of the impact of the slot location and velocity ratio on the aerodynamic coefficients and point of separation, the blowing angle is varied accordingly using the best slot location i.e. 80% C and a velocity ratio of 1.

Table 5.37 Best case with respect to separation delay

<table>
<thead>
<tr>
<th>Best Output</th>
<th>30°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle of suction</td>
<td>30°</td>
</tr>
<tr>
<td>Separation delay</td>
<td>17.5%</td>
</tr>
<tr>
<td>Lift Enhancement</td>
<td>1.55%</td>
</tr>
<tr>
<td>Drag Decrement</td>
<td>-3.74%</td>
</tr>
</tbody>
</table>

From the figures 5.36-5.38, it is clearly evident that $30^\circ$ is a critical angle within the evaluated cases. Any angle higher than $30^\circ$, did not work and iterations diverged. However the $15^\circ$ has provided results very similar to the $30^\circ$ angle but the magnitude
**Figure 5.36** Separation Point (%C) vs Blowing angle [Data Source: Appendix table 3.3]

**Figure 5.37** Lift Coefficient (Cl) vs Blowing angle [Data Source: Appendix table 3.3]

**Figure 5.38** Drag Coefficient (Cd) vs Blowing angle [Data Source: Appendix table 3.3]
of \( \text{cl}, \text{cd} \) and separation delay is lower as compared to the latter. So, \(30^\circ\) is considered to be worked best in the variation of blowing angles
Chapter 6

CONCLUSION

6.1 Achievements

The testing of parameters have provided an the optimal choice of discretization schemes, turbulence models, turbulent parameters, mesh refinement required in terms of yPlus and lateral number of nodes on the airfoil. The validation of computational results with k Omega SST and k-kl-Omega turbulence models have given an overview on the performance of both in comparison with experimental results. As the k-kl-Omega provided very close C<sub>d</sub> values and little higher C<sub>l</sub> values as compared to k Omega SST, it was chosen for further implementation of active flow control.

The impact of the active flow control parameters of constant suction and constant blowing on the aerodynamic performance mainly point of separation, lift and drag coefficients is studied in detail by comparing them with baseline case i.e. no active flow control. The parameters considered mainly include Slot location (suction and blowing), actuation velocity magnitude (suction and blowing), angle of actuation (suction and blowing) and slot width (suction). The computations are performed with a wide range of parameters in the case of constant suction and the magnitudes of parameters that have provided the better results in terms of aerodynamic coefficients and separation delay are utilized for evaluating the performance with respect to the constant blowing.

Active flow control through constant suction is implemented at different slot locations away from the point of separation of baseline case. It is observed that the constant suction of boundary layer works better as the slot is moved away from the point of separation towards the leading edge until it reaches a critical location where the separation delay and increase in C<sub>l</sub> are maximum. As the slot location is moved away from this critical point towards leading edge, there is a reduction in separation delay along with C<sub>l</sub> as compared to the same at the critical point. However, C<sub>d</sub> does not follow this pattern of critical point as it decreases continuously as the slot location is moved from the point of separation of baseline case towards the leading edge. This trend is graphically illustrated in the cases with velocity ratio of 0.5 and 0.9 with stable simulations all through in the section 5.2.1. Hence, using a critical slot location will provide the best separation delay and C<sub>l</sub> for the suction valid specific velocity ratios.

By keeping the slot location constant and increasing the velocity ratio of suction (U_3/U_∞), the lift coefficient increases and separation point moves downstream on the airfoil. But the drag coefficient decreases till a critical value of velocity and it increases above this critical value. Also as the slot location is close to the critical slot location, the magnitude of critical velocity is reduced. Hence, using a critical velocity magnitude at a specific slot location will provide the least drag coefficient.
Also the Lift to Drag ratio (Cl/Cd) calculated from the 2D simulations is the highest at critical slot location with critical velocity magnitude. This is also supported by the less values of Cl/Cd at critical velocities in other slot locations. After varying different angles of section ranging from to 150°, the perpendicular suction with respect to the tangent of slot has provided better aerodynamic performance than any other angle of suction. By increasing the slot width from 1.5 mm to 4.6 mm, it is found that separation delay increases. Also, both the Cl and Cd increases with an increase in slot width but with a decrease in Cl/Cd ratio. Although the Lift to Drag ratios calculated from the 2D simulations might not be practically applicable, this trend can be subjected to computation in 3D simulations to obtain valid results.

By increasing Angle of Attack from 12° to 13° and using similar suction parameters, there is an increase in Separation delay, Cl and Cd by 21.04%, 11.88% and +0.12% as compared to 16.23%, 8.5% and +1.28% with respect to baseline case. The increase in Cd being less shows an improved aerodynamic performance. This has proved that the active flow control works better with higher AOAs as the pressure forces over the suction side of the airfoil decreases and also dominates the viscous force. Also, with respect to AFC parameters, for AOA 12 slot location of 60%, Velocity ratio ($U_s/U_\infty$) of 0.5, slot width of 1.5 mm and angle of suction 90° was the best with respect to Cl/Cd ratio. However, simulations for AOA 13° were conducted for checking the phenomenon only and the similar analogy can be used and the optimal values can be obtained.

The constant blowing phenomenon is analyzed using the optimal magnitudes of the active flow control parameters obtained from the constant suction simulations. There is an increase in the separation delay and lift coefficient which also is accompanied by reduction in the drag coefficient as compared to the baseline case with the slot location close to the point of separation of baseline case than that are away from it. Although the improvements in separation delay and reduction in Cd are on par, the highest increase in lift is very less in magnitude (1.55%) as compared to the improvements obtained from suction (4.6%- 10.5%).

However with an increase in the blowing velocity magnitude, there is an increase in separation delay, Cd and Cl. Also, constant blowing is very sensitive to angle of actuation. Within the performed simulations, the better aerodynamic performance as compared to the baseline case is observed only with angles less than 30° to the tangent at the slot. Any increase in blowing angle from 30°, i.e, 45° and 60° did not provide any converging or better results. But by increasing the blowing angle from 15° to 30°, the separation delay, Cl and Cd increased. However, the increase in Cl in all the blowing cases performed by variation of slot location, velocity magnitude and angle of actuation is not very significant compared to constant suction.

The computational setup is clearly described with all the files of OpenFOAM provided in the Appendix which would be beneficial for anyone to use an OpenSource CFD software for implementing active flow control.
6.2 Future work

The viability of the Active flow control technology with respect to performance aspects is demonstrated in this Master Thesis. The results obtained are being consolidated and ready to be submitted to scientific journals. This thesis provides a solid base for understanding the impact of AFC parameters on suction and blowing. Being an emerging research area with significant contributions only from past few decades, the prototypes are still in Research and Development phase of aerospace and automobile companies.

Hence, the future work of this Master Thesis can be performed with in the below areas.

1. Issues unresolved in this Master Thesis.
   - Checking the grid dependency with respect to the lateral node distribution on airfoil with slot, as this resulted in unstabilization of baseline cases for different slot positions.
   - Studying the boundary layer phenomenon in the case of constant blowing which delayed separation significantly but with less increase in Cl as compared to suction.
   - Possibility of reduction of the computational domain (mesh size) downstream of the airfoil. It can be done in two ways which include using convectional boundary condition or performing the study for different chord lengths downstream.
   - Variation of Reynolds number simulations can be performed.

2. Time dependency
   - Although some transient simulations were performed with and without AFC, they are to be studied for longer duration of physical time for implementing periodic actuation which involves frequency values close to Strouhal number.
   - Also, transient simulations will provide a gateway for implementing AFC at post stall angles of attack which in steady cases could not be converged.

3. Turbulence Models
   - The present thesis utilizes only RANS turbulence modelling and to obtain more accurate and practically implementable results, there is a need to model Unsteady RANS, Large Eddy Simulation (LES), Detached Eddy Simulation (DES) and using Direct Numerical Simulation (DNS) cases.

4. Active Flow control technology
   - In addition to constant suction and constant blowing, there are more AFC technologies that can be analyzed which include Periodic Suction and blowing, zero net mass flow (ZNMF) actuation, sweeping jet actuation and Plasma jets.
5. Fluid flow conditions

- The two dimensional computational study can be extended to 3D. The compressible cases can be studied in addition to incompressible fluid computations.

6. Optimization

- More data can be obtained for different angles of attack with different AFC technologies mainly Periodic suction and blowing and ZNMF with high end turbulence modelling. The data obtained can be optimized to obtain high efficiency with respect to AFC parameters at different angles of attack.

7. Experimental testing

- To support the computational results, experimental testing can be done for any commercial scale implementation.

However using the results obtained from this Master Thesis, an application for the computing time at Barcelona Supercomputing Center is submitted and the application for industrial doctorate Position is in process. Using these resources and as a continuation of this thesis, our proposed future work includes the optimization of actuator groove position, velocity magnitude, frequency, amplitude and angle of perturbation for different post stall angles of attack and Reynolds numbers for zero net mass flux actuation by performing

- 2D compressible flow simulations using LES
- 3D incompressible flow simulation URANS and LES
- 3D compressible flow AFC using URANS and LES
- Fluid - Fluid interaction modeling near actuator location using DNS.
Bibliography


[40] Antti Hellsten. Some improvements in menter’s k-omega sst turbulence model.


