Performance Augmentation of Wing through Fluidic Actuation

Talha Ahmed

Regn No. 00000201201

Supervisor

Dr Imran Akhtar

DEPARTMENT OF MECHANICAL ENGINEERING

COLLEGE OF ELECTRICAL & MECHANICAL ENGINEERING

NATIONAL UNIVERSITY OF SCIENCES AND TECHNOLOGY

ISLAMABAD

AUGUST, 2019

Performance Augmentation of Wing through Fluidic Actuation

By

Talha Ahmed

Regn Number. 00000201201

A thesis submitted in partial fulfillment of the requirements for the degree of

MS Mechanical Engineering

Thesis Supervisor:

Dr Imran Akhtar

Thesis Supervisor's Signature:____

DEPARTMENT OF MECHANICAL ENGINEERING

COLLEGE OF ELECTRICAL & MECHANICAL ENGINEERING

NATIONAL UNIVERSITY OF SCIENCES AND TECHNOLOGY, ISLAMABAD

AUGUST, 2019

DECLARATION

I certify that this research work titled "*Performance Augmentation of Wing through Fluidic Actuation*" is my own work. The work has not been presented elsewhere for assessment. The material that has been used from other sources it has been properly acknowledged / referred.

Talha Ahmed
00000201201
MS-ME-16

LANGUAGE CORRECTNESS CERTIFICATE

This thesis has been read by an English expert and is free of typing, syntax, semantic, grammatical and spelling mistakes. Thesis is also according to the format given by the university.

Talha Ahmed
00000201201
MS-ME-16

Signature of Supervisor
(Dr Imran Akhtar)

COPYRIGHT STATEMENT

ACKNOWLEDGEMENTS

In the name of Allah, the Almighty, the most Beneficial and the most Merciful Who has guided me and given me the strength and will to complete this thesis. Indeed, I could have done nothing without Your priceless help and guidance.

I am profusely thankful to my beloved parents for their support and prayers throughoutmy life. I would also like to acknowledge my complete family, specially my wife for encouraging me and motivating me for completing this study.

I am especially thankful to my supervisor Dr Imran Akhtar for his help throughout my thesis. His absolute calmness in understanding my problems and continuous guidance led me to complete this endeavor. I would also like to thank Dr Zafar Abbas Bangash and Dr Tariq Talha for being on my thesis guidance and evaluation committee especially Dr Zafar Abbas Bangash to clear my concepts regarding the basics of aerodynamics.

Thanks to all the faculty members of Department of Mechanical Engineering for providing an ideal academic environment for studies as well as research. Thanks to all M.Sc course mates (Maj Salman, Maj Ovais, Maj Zeeshan and Maj Sajid). We had memorable time.

Finally, I would like to express my gratitude to all the individuals who have rendered valuable assistance to my study.

Dedicated to my caring parents, loving wife and adorable daughters whose prayers and tremendous supportled me to this wonderful accomplishment

ABSTRACT

Wingtip vortices are very significant facts in fluid dynamics and have been studied extensively in aerospace applications. These vortices are strongly associated with induced drag. The focus of this work is to numerically determine the lift and drag characteristics of a rectangular wing planform at various angles of attack, and determine the enhancements by introducing wingtip blowing. By effectively changing the blowing momentum coefficient, such jets can influence the strength, and location of the wingtip vortices, resulting in improved lift, reduced drag, and higher lift to drag ratio.

The objective of this dissertation is to assess the possible aerodynamic benefits from blowing at wingtip of a rectangular 3-D NACA 0012 finite wing. Computational Fluid Dynamics software (Fluent) is used to numerically simulate the flow around a finite wing. Reynolds Averaged Navier-Stokes (RANS) equations in conjunction with a k–ϵ turbulent model were employed in this study. Geometry and mesh are created in ANSYS design modeler and ANSYS mesh, respectively. The jet slot area (A_j) is set as 0.02 m^2 which spans 50% of the chord, and pressurized air is blown through the slot in the wing spanwise direction. Numerical results reveal that the strength of the vorticity is significantly reduced once blowing is done at the wingtip. Steady stream wise blowing near the tip creates inboard vortices which interact with the tip vortex and make it weaker.

Keywords: *Wingtip Vortex, Lift Coefficient, Drag Coefficient, Blowing momentum coefficient, Lift to Drag Ratio.*

TABLE OF CONTENTS

Declaration i
Language Correctness Certificate ii
Copyright Statement iii
Acknowledgements iv
Abstract vi
Table of Contents viii
List of Figures ix
List of Tables xi
CHAPTER 1: INTRODUCTION AND MOTIVATION 1
1.1 Introduction 1
1.2 Wingtip Vortices 1
1.3 Lifting Line Theory 2
1.3.1 Horseshoe Vortex Model 4
1.3.2 Downwash 5
1.4Finite Wing Theory 6
1.5 Research Objective 10
1.6 Research Problem 11
1.7 Significance of the Study 11
1.8 Thesis Structure 11
CHAPTER 2: LITERATURE REVIEW 13
2.1 Wingtip Vortices Reduction Via Active Flow Control 14
2.2 Objectives 22
2.3 Contribution 22
CHAPTER 3: NUMERICAL METHODOLOGY 23
3.1 Fluid Dynamics Model 23
3.2 Circulation Across the Wingtip Vortex 24
3.3 ANSYS Fluent Simulation Scheme 24
3.3.1 Wings Geometry 24
3.3.2 Grid Generation 25
3.3.3 Domain and Mesh 26
3.3.4Spatial and Temporal Discretization 28
3.3.5Pressure Velocity Coupling 28
3.3.6 Boundary Conditions 28
3.3.7 Parameter Selection 30
CHAPTER 4: NUMERICAL SIMULATIONS 31
4.1 Numerical Validation of 3-D Finite Wing 31

4.2Grid Independence 33
4.3 Pressure Contours 34
4.4 Velocity Contours 35
4.5 Streamlines 36
CHAPTER 5: RESULTS AND DISCUSSION 38
5.1Numerical Simulation of 3-D Finite Wing with Blowing from Wingtip 38
5.1.1Effect of Area Parameter 38
5.1.2Effect of Jet Location from Leading Edge..39
5.2 Numerical Simulations of Blowing at Various Angles of Attack 40
5.2.1 Effect of Blowing Momentum Coefficient 41
5.2.2 Pressure Contours 43
5.2.3 Velocity Contours 44
5.2.4 Streamlines 45
5.2.5 Lift-to-Drag Ratio @ Angle of Attack ($\alpha = 2.5^o$) 46
5.2.6 Lift-to-Drag Ratio @ Angle of Attack ($\alpha = 5^o$) 47
5.2.7 Lift-to-Drag Ratio @ Angle of Attack ($\alpha = 7.5^o$) 47
5.2.8 Comparison - Lift to Drag Ratio 48
5.2.9 Comparison - Lift to Drag Ratio % Increase 48
CHAPTER 6: CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK 49
6.1 Conclusions 49
6.2Future Work 50
REFERENCES 51

LIST OF FIGURES

Figure 1.1:Lift distribution over a wing 3
Figure 1.2:Vertical velocity distribution 3
Figure 1.3:Upwash and downwash......3
Figure 1.4:Wing tip vortex shedding behind a flying aircraft 4
Figure 1.5:A simplified horseshoe vortex model 5
Figure 1.6:Horseshoe shaped vortex 5
Figure 1.7:Front view of wing 6
Figure 1.8:Flow pattern 6
Figure 1.9:Examples of wingtip vortices 7
Figure 1.10:Trailing vortex 8
Figure 1.11:Front and top views of the flow pattern on a three-dimensional wing 8
Figure 1.12:Schematic diagram of a three-dimensional wing 9
Figure 1.13:Spanwise load distribution over a straight wing 10
Figure 3.1:Rectangular Wing with NACA 0012 Airfoil Section made in ANSYS DesignModeler 25
Figure 3.2:Different Kinds of Grids 25
Figure 3.3:C - Type Flow Domain made in ANSYS DesignModeler 26
Figure 3.4:Mesh – 3-D Finite Wing along with Domain made in ANSYS 27
Figure 3.5:Boundary Conditions for Flow over 3-D Finite NACA 0012 Wing. 29
Figure 3.6:Rectangular Wing with NACA 0012 Airfoil Section having Rectangular Slot.......30
Figure 4.1:Comparison between Computation Results and Ref Data (C_D vs α). 31
Figure 4.2:Comparison between Computation Results and Ref Data (C_L vs α).......32
Figure 4.3:Grid Independence Report of C_L and C_D 33
Figure 4.4:Pressure Contours at different locations (looking from side) 35
Figure 4.5:Velocity Contours at different locations (looking from side) 36
Figure 4.6: Streamlines at different locations (looking from side)......37
Figure 5.1:Effect of jet slot area on lift-to-drag ratio......38

Figure 5.2:Effect of jet location from L.E on lift-to-drag ratio.………................................……....................39

Figure 5.3:Drag coefficient at various angles of attack with different blowing momentum coefficient …......41

Figure 5.4:Lift Coefficient at various angles of attack with different blowing momentum coefficient............42

Figure 5.5:Lift-to-drag ratio variation at various angles of attack with different blowing momentum coefficient...42

Figure 5.6:Pressure Contours at different locations (looking from side). ....….44

Figure 5.7:Velocity Contours at different locations (looking from side).. ....….45

Figure 5.8:Streamlines at different locations (looking from side) …...46

Figure 5.9:Lift-to-Drag Ratio @ α = 2.5^{o}…..46

Figure 5.10:Lift-to-Drag Ratio @ α = 5^{o}. ....….......47

Figure 5.11:Lift-to-Drag Ratio @ α = 7.5^{o} ……………………………….......……….............…....…47

Figure 5.12:Comparison - Lift-to-Drag Ratio.. . …......48

Figure 5.13:Comparison - Lift-to-Drag Ratio % Increase………………..…........48

LIST OF TABLES

Table 3.1: Boundary Conditions.....29
Table 3.2: Parameter Selection.....30
Table 4.1: Validation Data.....32
Table 4.2: Grid independence study for NACA 0012 wing at R_e = 30,000.....33
Table 5.1: Effect of area parameter.....39
Table 5.2:12 x Cases of Blowing (Rectangular slot).....40

CHAPTER 1: INTRODUCTION AND MOTIVATION

This chapter briefly gives away the importance of wingtip vortices, followed by the basic understanding on presence of wingtip vortex.Moreover, understanding of this important flow is then examined in reference to its impact in the aerodynamic performance. At the end of the chapter significance of study and thesis structure is elaborated.

1.1 Introduction

The basic mechanism of flight was understood and practiced (with gliders) a long time ago. Many theorists became interested in its problems, and a decade before the first powered flight Lanchester (born 1868) in England postulated the type of flow that would be experienced by a *finite wing*. In a paper in 1894 which later led to the publication of his book "Aerodynamics" in 1907 he stated that the high pressure beneath the wings would spill out around the tips into the low-pressure region above the wings, forming vortices which would stream out behind the wing [1]. These vortices would roll up into two main trailing vortices of opposite sign, located one behind each tip, and would be deflected downward. Further, their effect at the wing would tip the resultant force vector back, causing a component of the lift to become *induced drag*. It is clear that Lanchester clearly understood this phenomenon when it is noted that in 1897 he secured a patent covering the use of end plates at the wing tips to minimize the *spillage* there – six years before the Wright brothers' flight. Although Lanchester's work was mathematical in scope, his presentation was not, and it remained to Prandtl (born 1875) to extend the work of Lanchester into the *Prandtl lifting-line theory* presented in 1911 [2].

1.2 Wingtip Vortices

Wingtip vortices also tend to occur at points other than wing tips and therefore at times are also known as the lift-induced vortices or trailing vortices. Induced drag is associated with the wingtip vortices and this happens due to the lift generation that is three dimensional. Wake turbulence is usually caused due to the wingtip vortices. Water tends to freeze or condense in the vortices depending on the humidity in the atmosphere and wing loading of the aircrafts. As the aerodynamic lift in the air is generated, there is lower pressure on the top surface than the bottom surface, therefore, the air flows from below towards the top in a circular manner. A flow pattern that is circulatory is observed and is known as the vortex. Rotary motion of air in the shed wingtip vortices reduces the effective angle of attack of the air. Wingtip vortices are generated due to the 3-D generation of lift and are associated with the downwash and induced drag [3]. In order to reduce the induced drag, it is necessary to choose a careful geometry of the wing and cruise conditions. The concept of the horseshoe vortex reflects the wingtip vortices occurrence. On the three dimensional finite wing, the wings segment lift is not the same as the prediction of the two dimensional analysis. As a matter of fact, the lift is greatly impacted due to the neighboring wings sections lift. It is hard to predict the overall wing lift and the span wise lift is predicted through the lifting line theory.

A high lift to drag ratio is desirable for some aircrafts like long range airliners and gliders having wings with high aspect ratio. However, on the other hand, such wings tend to have structural constraints as well. At some instances, winglets have also been used for the purpose of reducing drag and these can be seen in use of modern airlines. Effective aspect ratio of the wings is enhanced with the help of winglets, thereby, changing the pattern and magnitude of vorticity in the pattern of vortex [3].

1.3 Lifting Line Theory

Lift distribution is predicted by the Prandtl lifting line theory which also known as the Lanchester Prandtl wing theory. The shedding of trailing vortices are described by the lifting line theory and it describes the span wise changes in the lift distribution. Elliptical lift distribution gives minimum induced drag for a given wing span and surface. It predicts the distribution of lift over a three-dimensional wing [4]. The theory is based on the fact that the vortex loses strength along the wingspan as from the trailing edge it is shed as the vortex sheet. The circulation concept is applied in the lifting line theory and makes use of the Kutta-Joukowski theorem:

$$\tilde{L}_{(y)} = \rho V \Gamma_{(y)} \qquad (1\ .1)$$

The unknown is modeled and the local circulation enables to account for the influence of one section over the others [4]. The change in the span-wise lift is equal to the circulation change span-wise. The concept of circulation is reflected in the Figure 1.1.

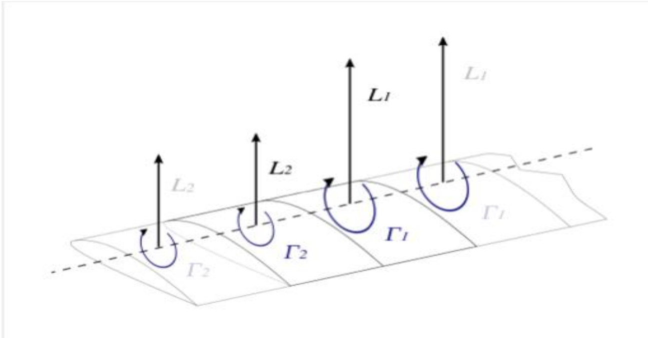


Figure 1.1: Lift distribution over a wing

The vortex filament does not terminate in the air or begin in the air and if there is any change span-wise, then it is modeled through vortex filament shedding down the flow and behind the wing [5].

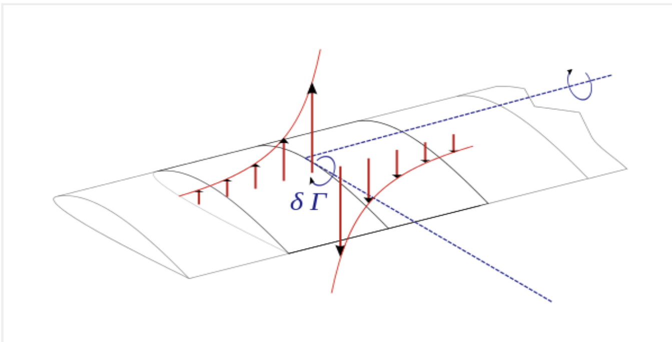


Figure 1.2: Vertical velocity distribution

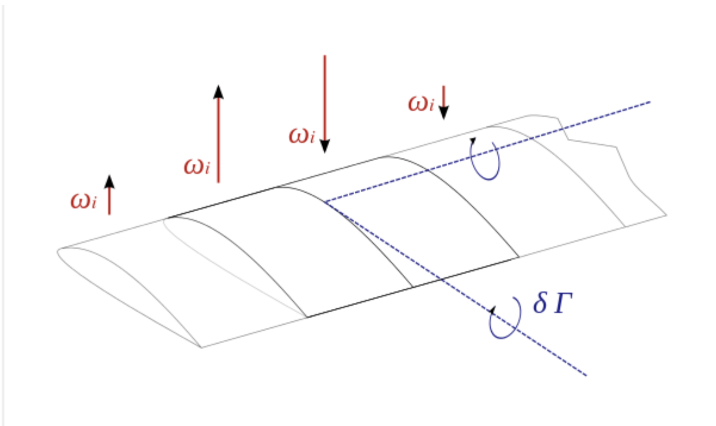


Figure 1.3:Upwash and downwash

The upwash and downwash induced by the shed vortex is usually computed at each neighbor segment and Figure 1.3 represents the phenomenon. Upwash on the outboard and downwash on the inboard is known as the sideways influence and is significant for the lifting line theory. The local induced change in the attack of angle or angle of attack is quantified through the integral sum of the downwash induced due to the wing sections[6]. The desired amount of life is equal to the down washed wing section's lift's integral sum. Based on this, an integro-differential equation is obtained:

$$L_{total} = \rho V_\infty \int_{tip}^{tip} \Gamma_{(y)}\, \mathrm{d}\, y \tag{1.2}$$

1.3.1 Horseshoe Vortex Model

Mathematical model which predictsthe aerodynamic properties for a wing of finite span is known as lifting line theory and named as Lanchester–Prandtl wing theory after Lanchester (1907) and Prandtl (1918). This model is based on the assumption that vortices strength reduce over the whole wing because theycontinuously shed downstream a trail of vortices forming avortex sheet from the entire trailing edge, rather than, justfrom the wing tips.

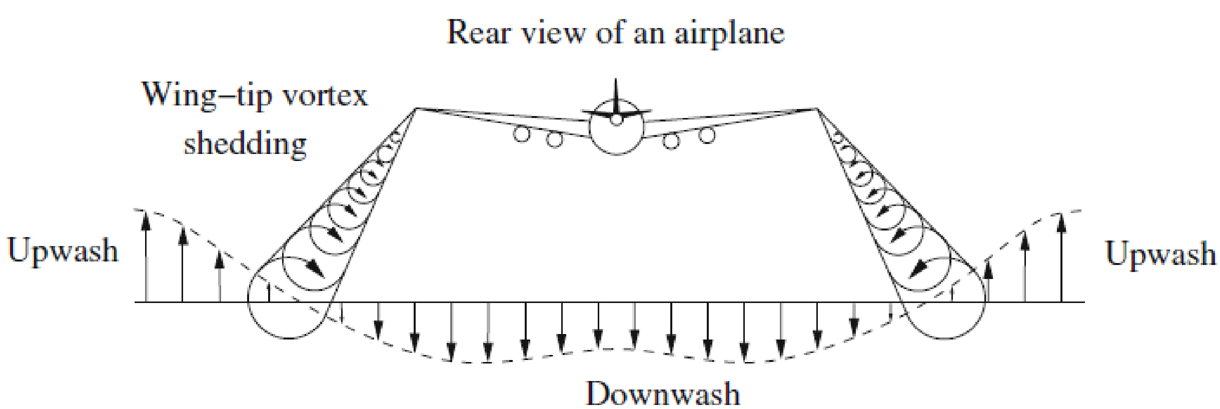


Figure 1.4: Wing tip vortex sheddingbehind a flying aircraft

Mathematical modeling is carried out by distributing the complete span of a wing into horseshoe vortices and simulating it into finite wing. Lifting line comprises of bound vortices which pass through the centre of each airfoil section of the wing. Lifting line also has circulation variable over the span. Trailing vortices formed induce the downwash at the lifting line thus modifying the angle of attack.

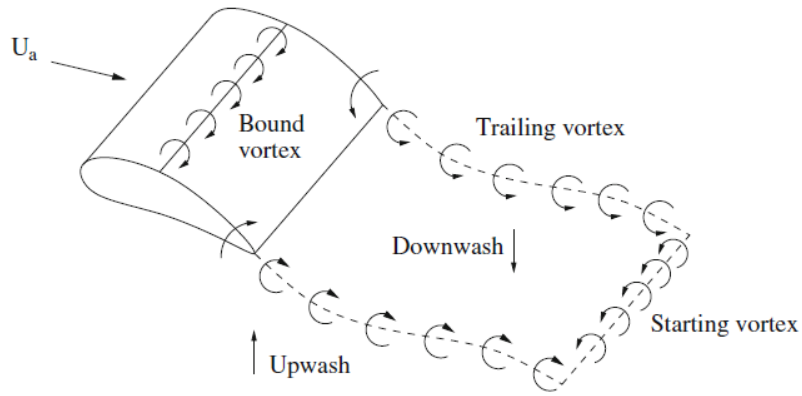


Figure1.5: A simplified horseshoe vortex model

1.3.2 Downwash

Downwash significantly changes the airfoil section's effective angle of attack [7]. As a result, the lift curve is changed. Interference effects are also produced which are integral for the stability analysis and for control purposes. The downwash magnitude is then estimated with the help of Biot-Savart law. Infinite downwash is predicted with two discrete trailing vortices. However, this has not been successful. The wing is represented through several horseshoe shaped vortex as shown in Figure 1.6.

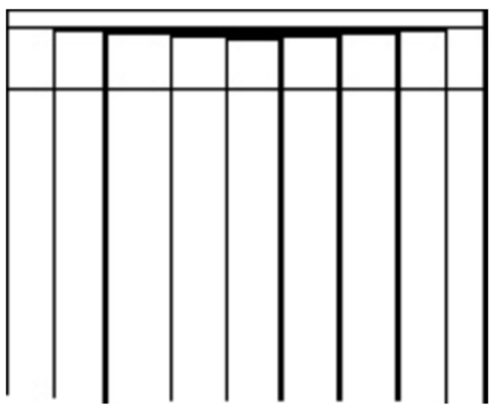


Figure 1.6: Horseshoe shaped vortex

In this manner there is variation in circulation from the root to the tip [6]. The relation of the circulation and trailing vortex is strength is then given by:

$$\Gamma_{wake} = \Delta\breve{\Gamma}_{wing} \qquad (1.3)$$

Whenever there is a change in the circulation, a vortex is shed from the wing [7]. The trailing wake becomes sheet of vorticity when number of horseshoe vortices reaches infinity.

The derivative of wing's total circulation is the trailing vortex strength per unit (vorticity). Basic relations for the finite wings are obtained in this manner [7]. The vorticity strength is give as follows in the trailing vortex sheet:

$$\gamma = d\,\Gamma/dy \quad (1.4)$$

1.4 Finite Wing Theory

In 2-D analysis the sections of the finite wing behave as given in Figure 1.7. The distribution lift in this case varies with the chord distribution. However, it cannot work in this manner as there is air leak around the tips due to the high pressure on the lower surface and upper surface's lower pressure. This causes pressure difference.

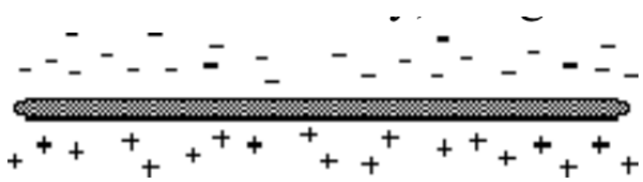


Figure 1.7: Front view of wing

In 3-D analysis or flow, air is pushed towards down. However the downward velocity tends to move outward and this air is squeezed outboard of the wing with the flow pattern as given in Figure 1.8.

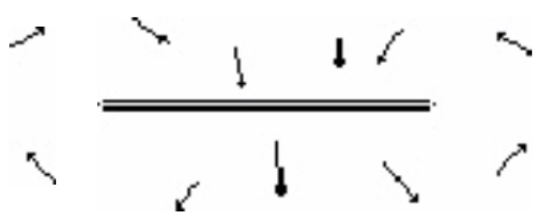
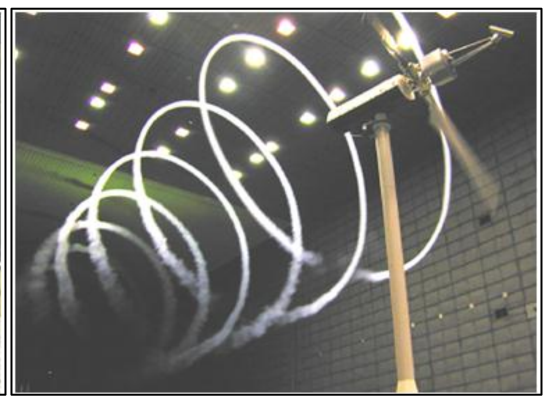


Figure 1.8: Flow pattern

The effects of wingtip vortices as depicted in Figure 1.9 are particularly important for a collection of situations which include vortices in the vicinity of airports and contrails in the sky, vortices from tips of airplane propellers and vortices from wind turbine blades.

(a) Wingtip vortices from an Airbus A300 (jetphotos.net)

(b) Contrails behind a Boeing 747 engines entrained by wingtip vortices (air-liners.net)



(c) Vortices from the tips of propeller blades of a Thunder Mustang (Thunder Mustang blogspot)

(d) Smoke visualization of wingtip vortices from a Wind Turbine experiment (Chattot, 2007)

Figure 1.9:Examples of wingtip vortices

A sheet of smoke has been used by NASA to visualize the trailing vortex and is shown in Figure 1.10.

Figure 1.10: Trailing vortex

An airfoil section is a two dimensional object having the same properties as of a wing of infinite span. But aircraft wing is a three dimensional object having a finite span and produces lift due to difference in pressure between its lower and upper surfaces. Flow over the aircraft wings is different from airfoil sections due to spanwise component of the flow in wings.

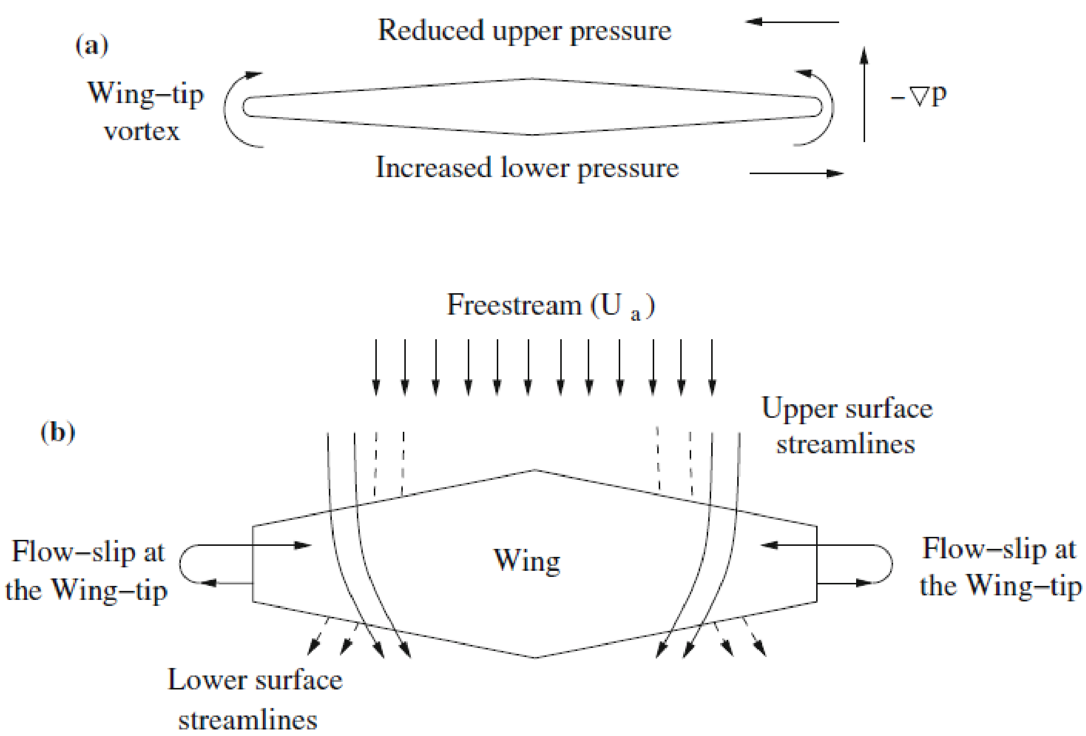


Figure 1.11: Front and top views of the flow pattern on a three-dimensional wing

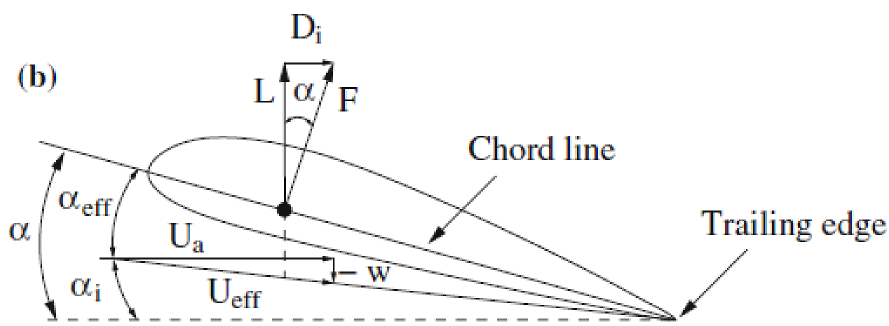


Figure 1.12: Schematic diagram of a three-dimensional wing

Angle of attack (α) is the angle between free stream direction and chord line. Localwind speed is inclined below the freestream velocity (U_a) bythe angle α_i, referred to as the inducedangle of attack. If thedownwash induced is w_i and the freestream velocity is U_a,then the induced angle of attack (α_i) is written as:-

$$\alpha_i = \tan^{-1}\left(\frac{-w_i}{U_a}\right) \tag{1.5}$$

Downwash is opposite to that of lift hence represented with negative sign. If the angle is small then induced angle of attack (α_i) can be written as:-

$$\alpha_i \approx \left(\frac{-w_i}{U_a}\right) \tag{1.6}$$

Effective angle of attack (α_{eff}) for a three-dimensional wing can be written as:-

$$\alpha_{eff} = \alpha - \alpha_i \tag{1.7}$$

Also, the effective freestream velocity (U_{eff}) will nowbecome

$$U_{eff} = \left(U_a^2 + w^2\right)^{1/2} \tag{1.8}$$

Lift generated in three dimensional wing is less due to less angle of attack (effective) even if the downwash is negligible. Hence finite wing has to operate on greater angle of attack (α). Moreover drag will be associated due to downwash. This additionaldrag is known as induced drag (D_i). This induced drag and lift perunit span produced by the wing is:-

$$D'_i = \rho U_a \Gamma\,(y) \sin \alpha_i \tag{1.9}$$

$$L' = \rho U_a \Gamma(y) \cos \alpha_i \tag{1.10}$$

Assuming small angle:-

$\cos \alpha_i \approx 1$, $\sin \alpha_i \approx \alpha_i$

The drag and lift per unit span becomes

$$D_i' = -\rho U_a \left(\frac{w_i}{U_a} \right) \Gamma(y) = L' \alpha_i \tag{1.11}$$

$$L' = \rho U_a \Gamma(y) \tag{1.12}$$

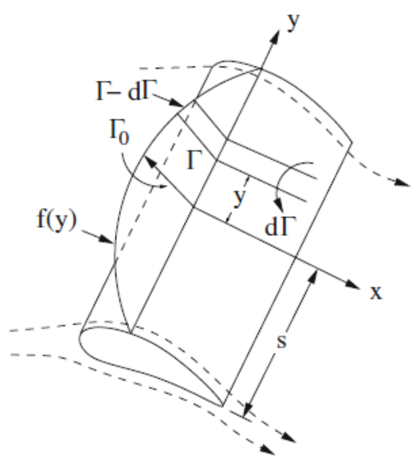


Figure 1.13: Spanwise load distribution over a straight wing

The trailing vorticity is strongest at the tips as there is a change in the wing circulation quickly near the tips. This is the reason, top vortices are seen and not the complete sheet of vortex. The wing lift and the coefficient for induced drag is given below:

$$C_L = \frac{L}{\frac{1}{2}\rho V^2 S},$$
$$C_{D,i} = \frac{D_i}{\frac{1}{2}\rho V^2 S}. \tag{1.16}$$

1.5 Research Objective

The basic objective of this research study is to assess the potential aerodynamic benefits from blowing at the tip of a rectangular wing using a 3D finite NACA 0012 wing.

1.6 Research Problem

When there is a pressure difference between the bottom and top surface of the wing, then lift force is generated. The air near the further end of the wing which is known as the wingtip tends to curl around when the air flows from a higher-pressure region to the lower pressure region. This is known as the wingtip vortex. This refers to the three dimensional lift generation and is an unavoidable side effect. Such effect is not favored as it gives rise to drag and original flow direction on the swing is altered. Induced drag is followed as the flow direction change leads towards a downwash. Moreover, this also results in instability. Vortices can also make the aircrafts go out of control. Keeping this is mind, regulation has been made to maintain distance between the aircrafts during the takeoff and landing periods. Vortex reduction is then carried out to enhance the aircraft efficiency and handling efficiency of the aircrafts. A conventional method largely followed involves reducing the vortices strength through the winglet deployment. However, newer approaches involve flow control at the wingtip as the weight penalty in this case is lesser than the winglet.

1.7 Significance of the Study

The research study is significant as the significance of wingtip blowing is reflected in the study. In this manner, the performance of the wing can be enhanced and further research and development can also be carried out to enhance the performance of wing. Blowing can be used by different organizations to enhance wing efficiency and organizations operating in the field who adopt it at an early stage can be more beneficial than the others as it has various advantages associated with it. Furthermore, the same work can be further expanded and researched upon to obtain more accurate results. The research also suggests the jet slots which has better aerodynamic performance and the angle of attack is also discussed at which angle of attack is suitable for aerodynamic performance.

1.8 Thesis Structure

The first chapter is the introductory chapter in which the background, objectives of the study, problem of the research, study significance are discussed. The second chapter carries out a detailed review of the literature. The third and fourth chapter focuses on the numerical methodology adopted and how the numerical simulations are carried out. The last chapter discusses the conclusion of the research work, future recommendations and discussion based on the results obtained.

CHAPTER 2: LITERATURE REVIEW

Aerodynamic efficiency is significant factor in the commercial transport of the Aircraft. The efficiency of the jets can be enhanced with the help of enhancing the lift to drag ratio at the cruise flight condition. There is domination of other sources of the aircraft as the engines become quieter. There are mainly three types of blowing configurations associated with the wings. These include externally blown flap, internally blown flap and wing tip jet. An external jet blows the flap or wing in the externally blown flap. The external jet can be the engine exhaust nozzle. This has been studied widely for the lift augmentation of a vehicle of short take-off and landing. The internally blown flap has been used for air blown over the flap from the slit in the wing's trailing edge and air blown from a slit in the flap's trailing edge. Internally blown flap has also been used for the lift augmentation of the vehicles of short take-off and landing. Air. Is blown in the steam wise direction in externally blown flap and internally blown flaps. On the other hand, wing tip jets have air blown in them in cross stream direction.

Aerodynamic efficiency is also significant for the commercial aircrafts. If there is a reduction of 10% of the drag on large military transport aircraft then it can save up to 13 million gallons of fuel along the lifetime [8]. Reducing the wingtip vortices lowers the operating costs and sheds vorticity and induced drag. Moreover, it also may lead to reducing the global warming as lower fuel consumption is made. As cited in [8], commercial jet aircrafts generate carbon dioxide which is more than 600 million tons.

The strength of the wingtip vortices are related in a proportional manner to the airplane's weight and it is related in an inverse manner to the wingspan of the airplane and to the speed of the airplane. The slower and heavier the airplane, the grater is the angle of attack required for cruising. This also leads to stronger vortices of the wingtip. Therefore, strongest vortices are induced during the time of takeoff, landing and climb. Such vortices can be generated by the larger aircrafts that are more than the vortices of an entire small airplanes. Moreover, if the small airplane is preceded by larger aircrafts then an uncontrollable and sudden variations may arise in the altitude and violent slamming also may occur [8].

Devices that have been suggested for the vortex control problem include tip reshaping, winglets usage and non planer wings. In cases tip devices has also been added for the purpose of reducing the noise. Winglets are usually added to the wing for the purpose of control of vortices and this is a method of passive control. On the other hand, the optimization of the passive control is only for condition of one flight and in other flight times it tends to be ineffective. Various levels of controls can be applied using the active control and it enables to have effective control over the flight. Wing is the most aerodynamic structure of an airplane as it enables the airplane to fly and work in an efficient manner. Heavy frictional and drag forces are faced by the airfoils. Active means have also been used. Active flow control can be done by using the existing airplane parts for example ailerons. On the other hand, using the existing parts can also lead towards structural problems of the aero plane. Such problems can also be avoided with the use of the tip blowing or suction on wing.

2.1 Wingtip Vortices Reduction Via Active Flow Control

Research efforts have been made to reduce the strength of the vortices, however, these research works have been successful modestly as the nature of the vortices taken into account has been stable. Due to the solid body rotation, the vortices tend to re-laminarize. There are mainly two philosophies for the control of the vortex i.e. instability excitation and turbulence injection. Crow instability is excited for the instability excitation and this is done through the vortex displacement which results in instability growth. Vorticity is reduced through the turbulence injection and the mixing of the vortex is enhanced within the core. However, it is still controversial that which of these methods is superior than the other. The results from previous study show that a measurable impact was not produced due to the low aspect ratio jets and the jets with high aspect ratios were significant only when the Mach numbers were low. It was also concluded in study that spanwise blowing does not impact the reduction in the drag [9].

Vortex can account for 40% of the plane drag and it was obvious that it needs to be minimized [10]. In the first attempt to reduce the vortices, it was concluded that the distribution of the lift shall be elliptical for the purpose of minimum induced drag. Later on, the tip shape was emphasized. Tip devices have also been considered for the reduction of noise and for the problem of the wake vortex [10]. Span wise blowing is used commonly as it enhances the effective wing span.

A thorough review of the literature is discussed in this chapter. Goodarzi, Rahimi and Forwidouni [11] carried out their research study on the active flow control investigation using a blowing jet. Blowing jet of width 2.5%is investigated with six different angles of attack. The attack angle from 12 degrees to 17 degrees is investigated. More than 200 numerical simulations are conducted in the research work. For the purpose of solve flow equations, FLUENT has been used and for the modeling of Spalart-Allmaras, a viscous model has been used. The results of the study reflect that drag is reduced and amount of life is increased due to the blowing. Moreover, high attack angles lead to improved performance of the airfoil. The results also show that when the jet velocity is increased then there is an amplifying impact on the mechanism of lift generation.

Tavella, Wood and Harrits [12] explored the aerodynamics of a rectangular wing in a spanwise direction. Aerodynamic forces are induced for roll and lateral control of the aircraft by changing the wing span. The research work takes into account different aspect ratios, attack angles and jet intensities. The results reflect scaling laws for lift gain and generation of moment due to blowing. It was also concluded in the research work that an infinitely small aspect ratio results in unbounded lift gain. The experiments also conclude in the research work that wing loading changes can be produced through tip blowing and leads towards rolling moments and lift augmentation.

Research work of Lim [13] focused on the trailing vortices generated due to the lifting surfaces. The research studied the trailing vortices in detail. Axial blowing and spanwise blowing is also discussed in the research work. The research is done using Navier-Stokes equations and the equations are solved using implicit and finite difference scheme making use of LU-ADI factorization technique. Actuator plane concept is used in the research study for simulation of wing tip blowing. In order to simplify the parametric study procedure for the blowing of wing tip, the chimera solution blanking feature is used. The results of the study suggest that axial wing tip blowing, the flow field near-field behavior, do not dissipate the strength of the circulation of the trailing vortex further downstream.

Mineck [14] carried out his research work on the benefits of the aerodynamics from span wise blowing at Wingtip. Spanwise blowing in the semi span wing was tested in the research work using Langley 7 using 10 foot high speed tunnel. Tests were conducted with and without blowing with 0.30 Mach number and the range of angle of attack used was -2 degrees to 11 degrees. Additional testing was also conducted in the research work. A Mach number of 0.72 was taken with 0.2 to 0.5 blowing. Various wingtip blowing configurations were also tested in the research work for the investigation of vertical locations, chordwise length and exhaust direction. The results of the study suggest that jet blowing did not have much impact on the drag or lift, however, blowing from the jets which had longer lengths of chord enhanced the lift near the wing tip and the drag was also reduced at lower number of Mach. For simulation of the jet blowing at the wingtip, Navier-Stokes solver was used and modified according to the need. The aerodynamic characteristics were predicted through the modified solver and Mach numbers 0.30 – 0.72 were used. The results indicated that as the jet momentum coefficient was increased, the drag and the lift increased as well. The research argues that aerodynamic efficiency cannot be improved through spanwise blowing at the wingtip as the momentum of jet is more than the wing drag reduction and wing lift increase due to spanwise blowing is also small.

Boyd [15] carried out research work on the Navier-Stokes computations of a wing flap model. For all the cases in the research study, a dual vortex structure is revealed through the vorticity contours. At the mid-flap chord location, the dual vortex merges into single vortex. The merged vortex strength is reduced due to the upper surface blowing. The lower shear layer is thickened due to the lower surface blowing and the merged vortex is also weakened. The lower surface vortex is forced farther outboard due to the side surface blowing and this increased the aerodynamic span of the flap. It is concluded from the research work that the blowing configurations examined do not have much benefit or global aerodynamic penalty.

Edstrand and Cattafesta [16] made an experimental investigation of the steady circulation control blowing. A rounded wingtip along the suction side of airfoil has been examined in the study. Wingtip vortices are known to be adverse by product of the generation of lift of airfoil. The study makes use of semi span NACA 0012 airfoil model which is mounted in an open jet wind tunnel. Stereo particle image velocimetry is used for the measurement of the wake of the vortex. There is reduction in the maximum swirl velocity for the uniform blowing and the reduction is by30%. Moreover, as the momentum coefficient increases, there is increase in the circulation and this in turn increases the vortex strength. This modifies the lift on the wing of the aircraft. It was also concluded in the study that circulation was not much impacted due to the segmented steady blowing configuration when the momentum coefficient values were low and this reduced the time averaged swirl velocity by 20 percent for 0.0023 coefficient of momentum.

The blowing was studied numerically and the geometry of suction slot was studied in research work of Yousefi, Saleh and Zahedi [10]. The research analyzed valuable results as jet angle was changed, jet width was changed, and other parameters were changed. NACA 0012 airfoil was used in the research work. Menter's shear stress turbulent model was used along with the RANS equations. The airfoil upper surface was applied with perpendicular and tangential blowing at the trailing edge and leading edge's perpendicular suction. The chord length's jet widths were varied from 1.5 – 4%. The value of jet velocity taken was free stream velocity's 0.3 and 0.5. The result of the study concluded that when the blowing jet width increases then the lift to drag ratio also increases in a continuous manner in tangential blowing. However, the lift to drag ratio decreases in perpendicular blowing in a quasi linear manner. For tangential blowing, the most effective widths of chord length were found which turns out to be 3.5% and 4%. For perpendicular blowing, the smaller jet widths were more effective. As the suction jet width enhances, there is improvement in the lift to drag ratio as well and the maximum value is reached at a chord length of 2.5%. The research also recommended that more research is needed for investigation of blowing parameters, slots, arrangement of the slots and jet parameters.

As there is pressure difference between the upper and lower surfaces then the lift force is produced. This leads to formation of the vortices. Baljit et al [17] investigates the impact of suction and jet blowing in boundary layer separation control and the research makes use of NACA 0012 airfoil. Reynolds number 1.2×10^5 was used in the study chord length of 25% position was used. The findings show that NACA 0012 airfoil's aerodynamic performance is impacted due to suction and jet blowing. It also shows that it can be an effective means for separation control of the boundary layer in subsonic flow. The results indicate that decent results are produced with the help of subsonic wind tunnel. Positive results in the lift have also been produced through the suction system and jet blowing.

[18] investigated the aerodynamic characteristics with low aspect ratio 50^0 sweptback wing when air in the form of a thin sheet is blown out from the wing tips in a span wise direction shows that at a 0.138 momentum discharge rate coefficient, the stalling angle increases, C_l enhances for all angles of attack, C_{lmax} increases by 36%.

Duraisamy and Baedar [19] investigated effect of span wise blowing on tip vortex formation. A solution is developed for the compressible Reynolds averaged Navier-Stokes equations and the solution is of high-resolution computational methodology. Span wise blowing is studied with the help of this methodology. High order accurate schemes are used for the purpose of reducing the numerical errors. Multiple overset grids are also used for solving the equations and this ensures adequate resolution in a proper manner. Simple corrections are added to the Spalart-Allmaras turbulence model for the validation of mean flowfield for baseline and control configurations. The RANS equations are closed using the Spalart-Allmaras turbulence model. The aerospace computations are done largely using this model as it very efficient and the model was formulated specifically for the aerospace applications. Moreover, it is an linear eddy viscosity model therefore the model is also simple to execute and is not very expensive and it also tends to be numerical stable. On the other hand, such models are not much useful for the flow rotations, stream line curvature and other related phenomenon. The RANS simulations usually have to face diffusion errors and turbulence modeling may also occur.

Many applications are impacted in an adverse manner due to the trailing vortices and they are harmful for the control efforts made for the jets. The study of Edstrand and Cattafesta [20] makes use of stereo particle image velocimetry for observing different control level impacts on the wing's surface. Pressurized air is blown in the spanwise direction with 1mm slot at the airfoil's suction side which spans 70% chord. The results of the study indicate that control jet appears to behave in an akin manner to a jet in crossflow. Moreover, it does not oppose the motion of the vortex and it rolls up the vortex and bend towards the back. Relative to the baseline case, a more diffuse vortex is obtained. Wake hazard is reduced with the help of the diffused vortex by 40% in the 1st five chords downstream. The research study concludes that the uniform blowing configuration outperforms the configuration of the segmented blowing for the metric of wake-hazard. On the other hand, there was 15% reduction for the configuration of segment blowing. The results of the research work are good enough for enhancing the safety of landing in the aircraft. Moreover, it was also concluded that wake-detection metric had better performance than the wake-hazard metric. Moreover, in the wakehazard metric there was 20% reduction due to the configuration of segmented blowingwhereas , on the other hand there was only 10% reduction using the uniform configuration.

Yousefi, Saleh and Zahedi [10] study the NACA 0012 airfoil trailing edge. The steady blowing, tangential as well as perpendicular is studied in the research work. The research work is conducted for studying the impacts of blowing amplitude and coefficient and how they impacts the characteristics of the airfoil dynamics. Turbulent flow was used in the research study with a Reynolds number of 5×10^5. Menter's shear stress model was used for the modeling of the turbulent model. Tangential and perpendicular blowing was done for the airfoil modeling and the blowing jet length is taken chord lengths 3.5%. Previous studies suggest that there are two distances in which the blowing jet acts in an optimum manner. One distance is 40% and the other is 80% of the length of the chord from the leading edge. In this research work, therefore, 80% of the chord length distance has been considered. The blowing velocity is considered to be 0.1 – 0.5 of the freestream velocity. The results of the study indicate that the coefficients of lift and drag are not considerable by increasing the blowing amplitude. The maximum lift to drag ration is obtained with 0.5 amplitude, however, a lower blowing amplitude is favorable for the perpendicular blowing. Moreover, the stall angle is not impacted due to the tangential blowing and it even cases gradual stall of the airfoil NACA 0012. Lastly, the study also concludes that stall angle slightly improves from 14 degrees to 16 degrees through the perpendicular blowing. The researcher further argues that slot arrangements, slot entrance, blowing, jet parameters are not yet been examined to the full extent. Laboratory studies are also limited which are carried out on suction and blowing parameters.

Margaris and Gursl [9] study the impact of steady blowing on the flowfield of the wing tip. The research was carried out to control the trailing vortex. A PIV system was used and taken into account and velocity measurements were taken. The rectangular wing was considered equipped with blowing slots. The results of the study indicate that blowing impact is sensitive towards the blowing configurations. A diffused trailing vortex may be obtained due to the strength and location of the vortices and the interaction with the tip vortex. A diffused trailing vortex is obtained due to the span wise blowing and a much stronger vortex may be generated if it is applied too close to the suction surface. Moreover, multiple co-rotating vortices are produced through blowing in the vertical direction and the coefficient of blowing increases as well. Stronger vortices can also be produced depending on the vertical blowing direction. Differences can be seen in the time averaged measurements in the PIV measurements and differing phenomenon of the flow field are also obtained.

Margaris and Gursul [8] carried out their research work on vortex topology of the wing tip blowing. A parametric study was carried out for examining the continuous blowing impact from the high aspect ratio jets on the vortex's tip. The research concluded counter rotating vortices were produced of unequal strength by the high aspect ratio jet. Single or multiple vortex are produced due to the interaction with the tip vortex. The topology of the flow depends on various parameters and these include the configuration of the blowing, coefficient of the blowing and wing incidence. The research study examined the spanwise blowing and the vertical blowing. This was examined from both the longitudinal and the vertical positions. Diffused vortices are obtained due to the blowing from the pressure surface in the near wake. The shape of the tip was varied in the research study. Coanda effect was promoted through the round tip and this modified the interaction of the vortex. The tip vortex as well as the jet generated vortices were impacted due to the wing incidence. The complexity of the phenomenon is reflected in the findings of the research study. The results also reflect that there are a number of parameters on which the vortices are dependent on. The research work also suggested that more studies are needed to study the matter in detail and see the potential for further applications.

Samal and Dash [21] explored the wingtip vortex reduction using the process of wingtip suction. The study was conducted to reduce the induced drag. An inswept and untwisted rectangular wing was considered in the research study for the experimental purposes. NACA 0012 is used and 10 degree of angle of attack is used. The numerical results are then compared in the study with the low speed wind tunnel experiments that have been carried out by NASA. The results proved to be good and in accordance with the experimental results of NASA. CFD code was used to obtain the results. This was done to validate the results that are obtained. Other configurations were also evaluated computationally to see how they impact the wing tip vortex. The results of the research study reflect that when the suction is applied, the vorticity strength is greatly reduced on the wing tip and it is also reduced along the bottom surface slot near to the trailing edge.

Yousefi and Saleh [22] carried out their research study on control of the suction flow and optimization of the suction jet. The experiment was carried out on NACA 0012 wing. The study was carried out for the investigation of rectangular wing characteristics. The Reynolds averaged Navier-Stokes equations were used in the research work along with the k-ωSST turbulent model. Two different types of slot distributions were considered in the research which includes center suction and tip suction. On the upper surface of the wing perpendicular suction was applied. The lengths of the suction jets were varied by 0.25-2 of the length of the chord. The jet velocity was 0.5 times the freestream velocity. The results of the research work indicated that lift to drag ratio enhanced as there was increase in the length of the suction jet. The results also indicated that aerodynamic characteristics enhanced with the center suction. On the other hand, through center suction the vortexes moved downstream or abated in an frequent manner. More vortexes were removed through the center suction process than the suction of the tip. Moreover, the tip suction turns out to be better when the jet length is less than the half of the wingspan. There was a larger impact on the flow field due to the longer suction jet.

Yousefi and Saleh [10] found the impact of trailing edge blowing on the characteristics of aerodynamic and the parameters of amplitude of the jet and blowing coefficient was tested in the research study. Reynolds number 5×10^5 was considered in the research work and turbulent flow was taken into account. The model of Menter's shear stress turbulent was applied. The upper surface of the airfoil was applied with the perpendicular as well as tangential blowing. The width of the jet was varied from 1.5 – 4% of the length of the chord. The amplitude of the jet was taken to be 0.1, 0.3 and 0.5. Results indicate that in the tangential blowing as there is an increase in the amplitude, the lift to drag ratio enhances as well by 15%. It was also concluded that lesser amplitude of the blowing shows good results for the perpendicular blowing. In the tangential blowing, as the width enhances the lift to drag ratio also enhances and it decreases in the perpendicular blowing. The optimal value of width of blowing jet was 3.5 – 4 % of the length of chord.

Yousefi, Saleh and Zahedi [23] further explored the suction slot geometry. NACA 0012 airfoil was used. The model of Menter's shear stress was used and the RANS equations were employed. At the trailing edge, perpendicular and tangential blowing was applied and at the leading edge, perpendicular suction was applied. The width of the jets were varied from 1.5 – 4% of the length of the chord and the jet velocity taken into account was 0.3, 0.5 of the freestream velocity. The results of the study concluded that lift to drag ratio enhances as the width of the blowing jet increases. This result was for the tangential blowing. For the perpendicular blowing, the lift to drag ratio decreases in an quasi linear manner in the perpendicular blowing. For the tangential blowing, the most effective amounts turned out to be 3.5% and 4% jet widths. The results also indicated that as the there is an improvement in the lift to drag ratio, the width of the suction jet also enhances and the maximum value of 2.5% of the length of chord is obtained.

From the literature, it is obvious that varying results have been obtained. Therefore, it is necessary that more research is carried out to test different parameters as results tend to vary with different parameters. Therefore, wingtip blowing is carried out in the research for the performance augmentation of the wing.

2.2 Objectives

The key objective of this thesis work is to conduct a parametric study to see the effects on aerodynamic characteristics of NACA 0012 wing at Reynolds number (R_e) of 30,000. In order to achieve this objective, we have performed numerical simulations for performance augmentation of wing through wingtip blowing from a rectangular slot starting in ANSYS Fluent.

2.3 Contribution

Performance augmentation of 3-D NACA 0012 finite wing by wingtip blowing to achieve improved aerodynamic characteristic in terms of lift to drag ratio from 4.2% - 28.2% at 2.5^{o} angle of attackby changing various parameters.

CHAPTER 3: NUMERICAL METHODOLOGY

This chapter describes the numerical methodology adopted to investigate the aerodynamic characteristics of a 3-D finite NACA 0012 wing. Fluid flow has been modeled as 3-D, unsteady, turbulent and incompressible flow.

3.1 Fluid Dynamics Model

Fluid mechanics is nonlinear multi-degree of freedom phenomena. The assumption for continuum modeling of fluid ignores the molecular motions where the entire space is engaged by fluid molecules and not depending upon any particular coordinate system. The accuracy of such models is extremely well for macroscopic length scales that are much greater than the inter-atomic distances. So the constant sub-division of the fluid into infinitesimal elements, having bulk material's properties, is allowed in continuum mechanics. Derivation of differential equations from fundamental physical laws may be applicable for describing the reaction of fluid in terms of expected properties like pressure, velocity, vorticity as well as their space and time derivatives.

The three basic laws of physics (conservation of mass, momentum and energy) set the mathematical representation of fluid dynamics. Computational fluid dynamics of 3-D finite wing is external flow problem and Navier-Stokes equations for continuity and momentum are governing equations for the flow. They can be represented as follows:

Continuity Equation

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{3.1}$$

Momentum Equation

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} \tag{3.2}$$

3.2 Circulation Across the Wingtip Vortex

The circulation Γ is depicted in equation 3.3 as (Anderson Jr., 2010)

$$\Gamma = -\oint \mathrm{V.ds} = -\iint (\nabla \times V).ds \tag{3.3}$$

Positive-counter clockwise sense of integral is shown by minus sign in above equation.

3.3 ANSYS Fluent Simulation Scheme

As already described, 3-D finite wing is external flow problem and the Navier-Stokes equations are the governing equations of the flow filed. ANSYS Fluent has been used for numerical simulation in present study. k–ϵ turbulent model is used as a turbulence model in present study.

3.3.1 Wings Geometry

The study is conducted on a NACA 0012 rectangular wing (in rectangular wing, root chord length is equal to the tip chord length) having a chord length of 1 m and span of 2 m, as shown in Figure. 3.1.Aspect ratio of wing is one of the important geometric property of a finite wing. Therefore an aspect ratio of 2 is used in the present study.

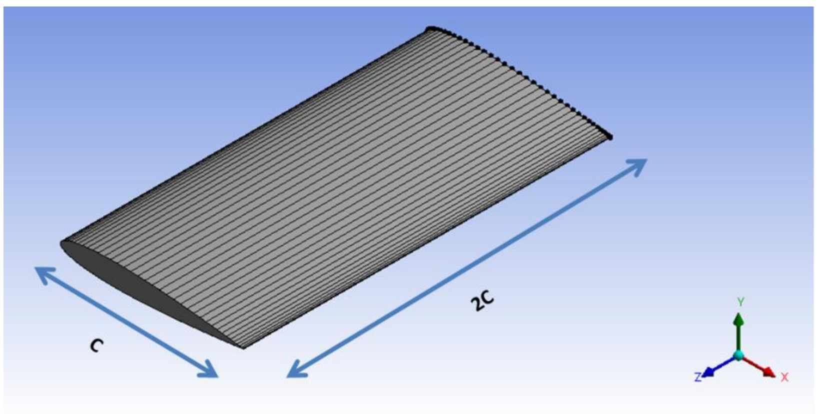


Figure. 3.1: Rectangular Wing with NACA 0012 Airfoil Section made in ANSYS DesignModeler.

3.3.2 Grid Generation

Subdivision of whole physical domain into small non-overlapping control volumes is basic requirement of continuum modeling of fluid. For this purpose, 3D geometries and mesh are created in ANSYS. Simulation time and the accuracy of the results are dependent on grid generation or meshing. Classification of different kinds of grids is as under:

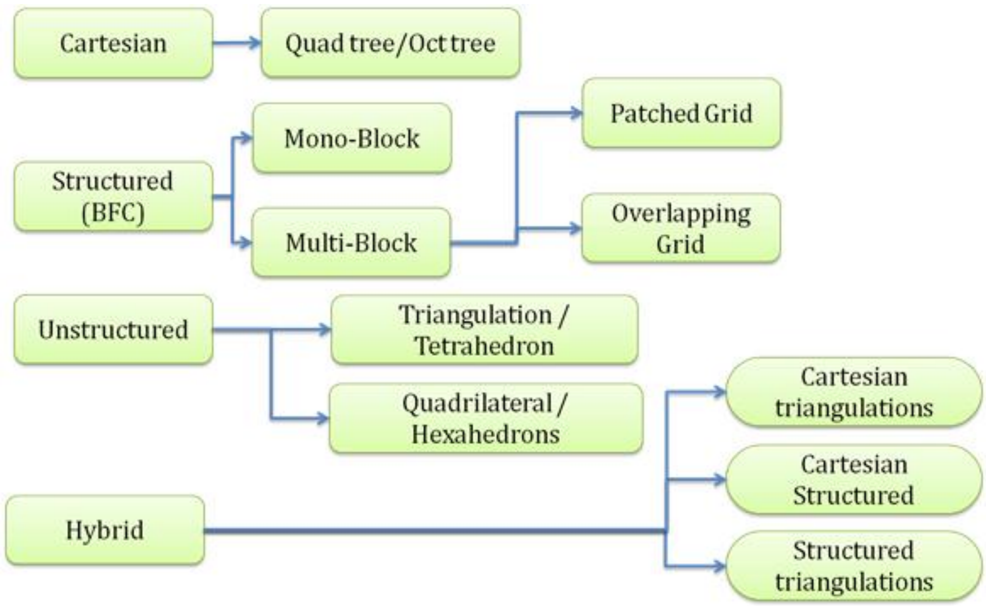
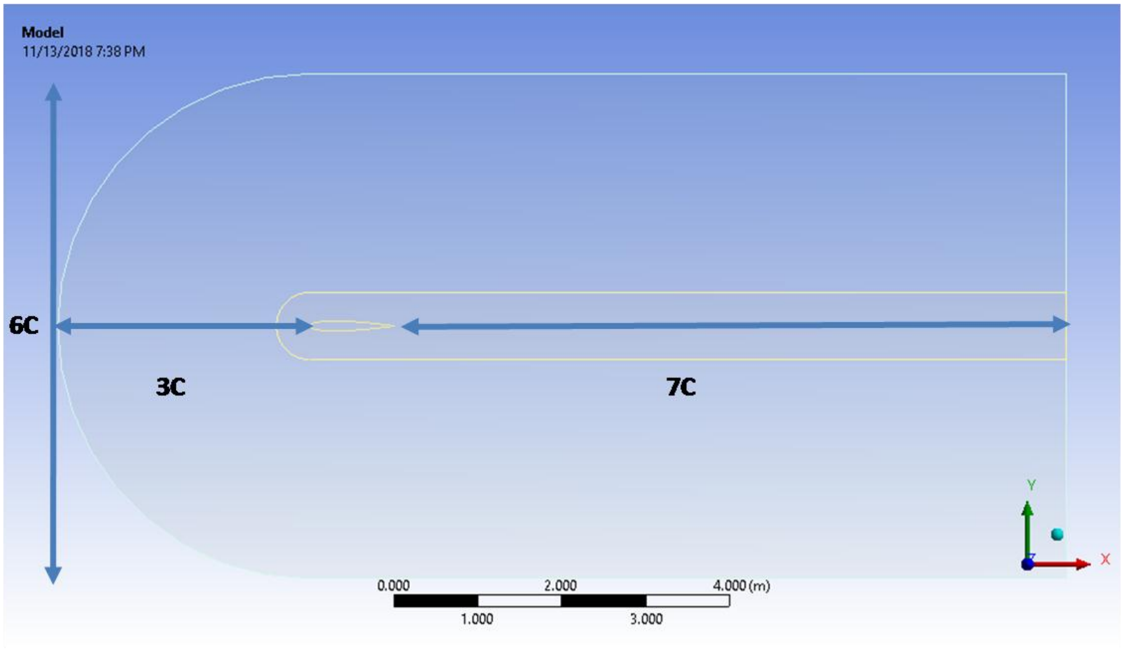


Figure. 3.2: Different Kinds of Grids

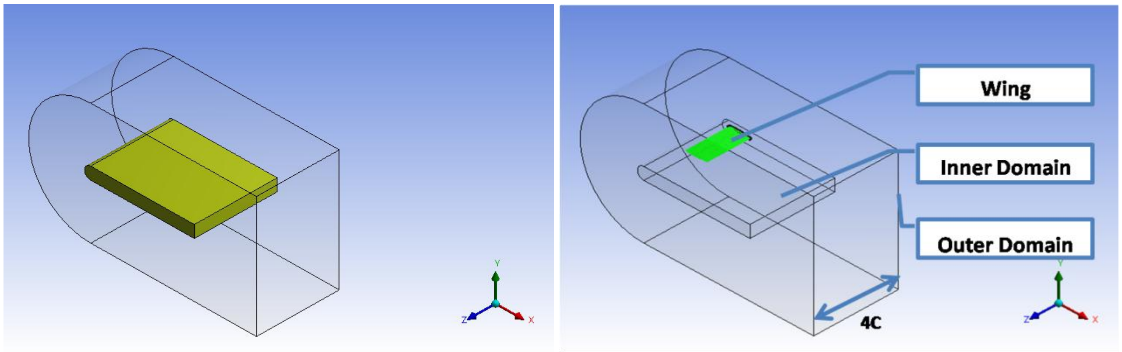
3.3.3 Domain and Mesh

A C-type domain having inner and outer blocks is generated in ANSYS DesignModeler as shown in Figure. 3.3. The domain is large enough to avoid the outer boundary from affecting the flow near the NACA 0012 wing. The grid extends from 3C upstream to 7C downstream, while the upper and lower boundaries extend 3C in each direction.



(a) C- Type Flow Domain - Side View

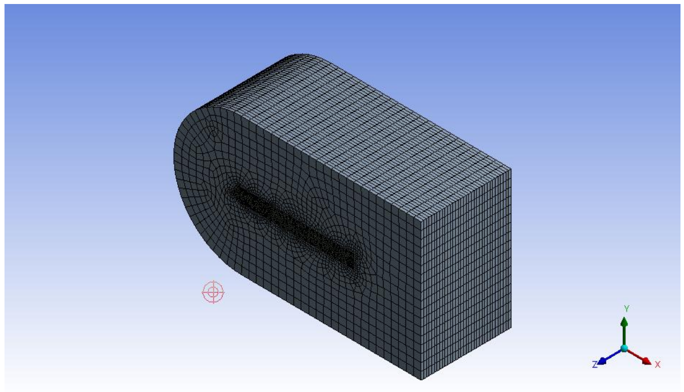
Moreover, the grid extends 4C in the span wise direction. For the purpose of fine grid, another block called inner domain is created around the NACA 0012 wing which extends 5C downstream.



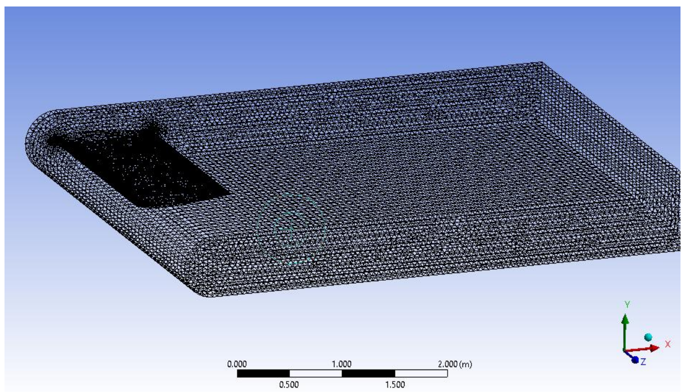
(b) C - Type Flow Domain - Isometric View

Figure. 3.3: C - Type Flow Domain made in ANSYS DesignModeler.

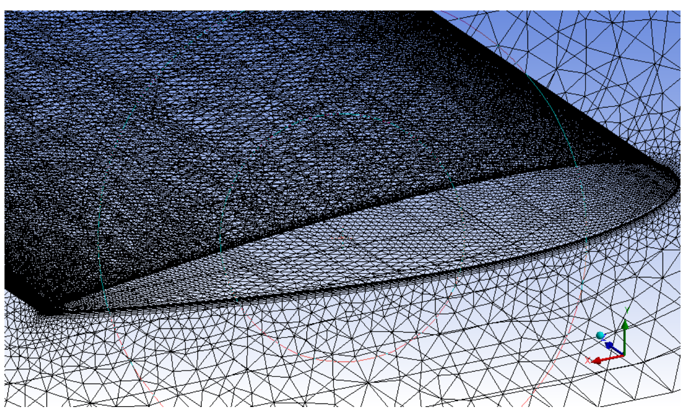
A structured along with unstructured mesh as shown in Figure. 3.4 is used in this computational study with a total of 2,359,109 elements. A structured grid is used in outer domain, whereas unstructured grid is used in inner domain.



(a) Mesh - Outer Domain



b) Mesh - Inner Domain with Wing



(c) Mesh - Wing

Figure. 3.4: Mesh – 3-D Finite Wing along with Domain made in ANSYS

3.3.4 Spatial and Temporal Discretization

In present simulations, a second order accurate upwind scheme is used for spatial discretization. Since the implicit scheme is unconditionally stable for any time step size, a second order accurate implicit scheme is used here for temporal discretization.

3.3.5 Pressure Velocity Coupling

In present simulations, pressure based solver with double precision is used. Continuity and momentum equations both contribute to formulation of pressure correction equation. Nonlinear quantities are part of momentum equations convective terms. The momentum and continuity equations are coupled due to the presence of velocity component in each equation. Clearly, there is no transport equation for the pressure and its gradient is not known in advance. So, in the solution of the flow field, a restriction is introduced to couple the pressure with velocity that if the correct pressure field is applied in the momentum equations, the resulting velocity field should satisfy continuity. In order to link pressure with velocity, guess and correct based iterative solution strategies like SIMPLE, SIMPLEC, PISO etc. are available in the solver. All the algorithms are just improved versions of SIMPLE first introduced by Patankar and Spalding (1972). In these algorithms, initial guesses for the velocity and pressure fields are used to start iteration process. In our work we use SIMPLE scheme. In ANSYS Fluent, a second order accurate up-wind scheme is used to solve the momentum equation and pressure correction equation is solved by a second order accurate method. For solution control, the default under relaxation factor is used as in the solver.

3.3.6 Boundary Conditions

Computational fluid dynamics problems are defined in terms of appropriate initial boundary conditions. The present external flow scenario include 3-D Finite NACA 0012 wing having a span of 2 m. The definitions of geometry of region of interest along with boundary conditions are given in Figure 3.5. It is in account that the location of the outlet is selected far away from the wing so that a fully developed flow reaches with no change in flow direction. Flow direction is taken from left to right. For unsteady case, the initial values of all the flow variables are required to be defined at all solution points with in the floe domain. It is initialized with an inflow velocity where the velocity component in cross flow direction is taken as zero.

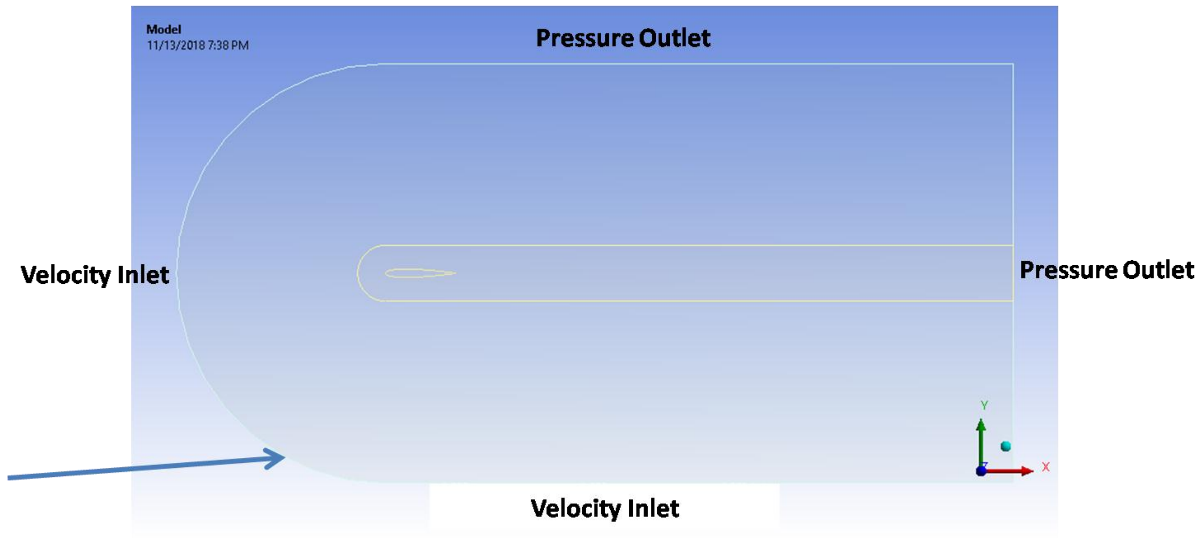


Figure. 3.5: Boundary Conditions for Flow over 3-D Finite NACA 0012 Wing.

Gauge pressure is also set as zero Pascal because of the input requirement of static gauge pressure for pressure boundary conditions. No slip shear condition is imposed on the surface of NACA 0012 finite wing that means fluid attached to the structure has same velocity as the wall and forms a thin viscous boundary layer when fluid passed through it. The problem is considered as a three dimensional domain for CFD calculations. Software uses chord length as 1 m and span as 2 m. Fluid viscosity is used to set the Reynolds number to 30,000. Table 1 illustrates the set of boundary conditions being used in the present computational study.

Table 3.1: Boundary Conditions

Position	**Condition**
Inlet	Velocity magnitude = 1 m/s
Outlet	Pressure outlet
NACA 0012 wing	No slip stationary wall boundary condition
Lateral directions	Symmetry
Angle of attack	2.5^0, 5^0 and 7.5^0
Reynolds number	30,000

3.3.7 Parameter Selection

Figure. 3.6. shows the NACA 0012 rectangular wing with blowing jet location, jet length and jet width from leading edge. The effects of blowing jet form tip of NACA 0012 rectangular wing is described by an important parameter called as jet momentum coefficient.

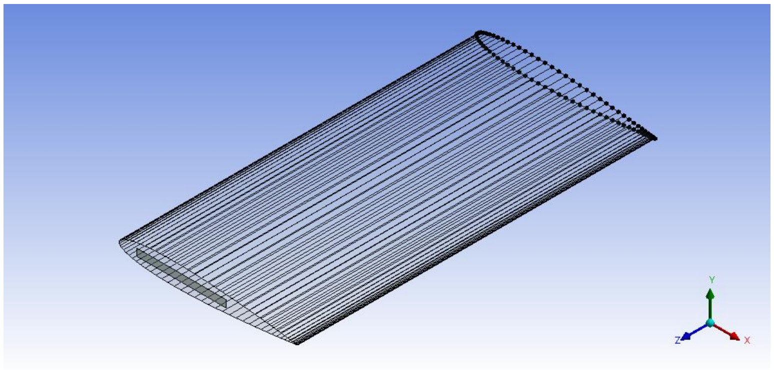


Figure. 3.6: Rectangular Wing with NACA 0012 Airfoil Section having Rectangular Slot.

Table 3.2 depicts the different parameters over the range of jet momentum coefficient. Jet rectangular slot starts at 0.1 m from leading edge.

Table 3.2: Parameter Selection

Parameters	Different Values	Unit
Jet slot length (L_{jet})	0.5	m
Jet slot height (h)	0.04	m
Jet slot area (A_{jet})	0.02	m^2
Jet Velocity (V_{jet})	1, 1.5, 2 and 2.5	m/s
Jet momentum coefficient (C_μ)	0.02, 0.045, 0.08 and 0.125	-

CHAPTER 4: NUMERICAL SIMULATIONS

This chapter presents the validation of the setup used in current simulation to prove its potential to capture computational results related to finite wing accurately. We perform simulations for 3-D finite NACA 0012 wing at low Reynolds number for validation purpose. Results are compared with experimental published results and a good agreement is found.

4.1 Numerical Validation of 3-D Finite Wing

Most of the commercial CFD codes use finite-volume technique. In present work, ANSYS Fluent is used for numerical simulations that also adopts finite-volume strategy to obtain numerical solutions. Pressure-based solver is used along with pressure-velocity coupling solution method. In the simulation, second-order upwind method is used using SIMPLE scheme until O(3) convergence criterion for all residuals is satisfied.

The computation results of finite NACA 0012 wing without blowing are compared with Aranda et al. [21] and Yousefi et al. [18]. The computational results agreed well with the previous numerical data as shown in Figure. 4.1 and Figure. 4.2.

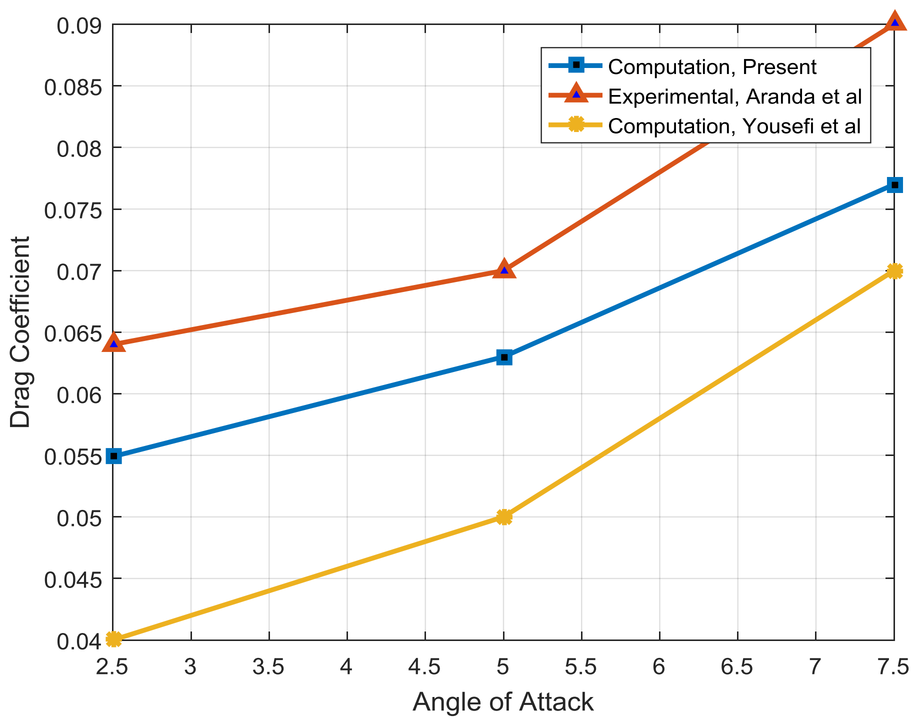


Figure. 4.1: Comparison between Computation Results and Ref Data (C_D vs α).

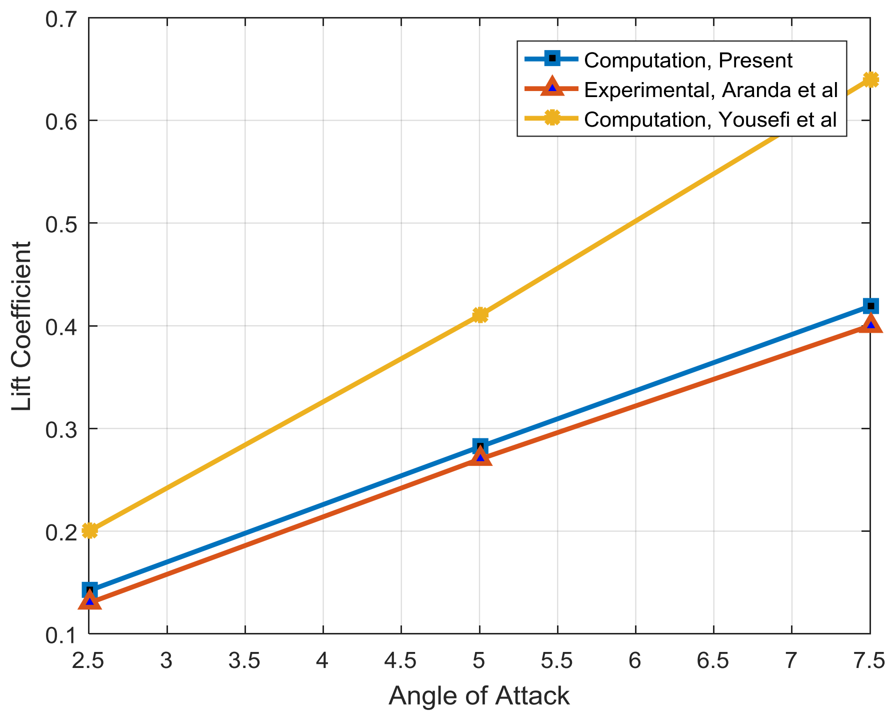


Figure. 4.2: Comparison between Computation Results and Ref Data (C_L vs α).

Table 4.1 shows the %Error from the ref data and simulation conducted on 3-D NACA 0012 wing at Reynold number = 30,000.

Table 4.1: Validation Data

Angle of Attack α	**Present Value C_D**	**Ref Value C_D**	**% Error**	**Present Value C_L**	**Ref Value C_L**	**% Error**
2.5°	0.0549	0.064	**14%**	0.142	0.130	**9%**
5°	0.063	0.070	**10%**	0.282	0.270	**4.44%**
7.5°	0.077	0.090	**14%**	0.419	0.400	**4.75%**

4.2 Grid Independence

Grid independence study carried out with three different mesh cells number is shown in Table 4.2 and Figure 4.3.

Table 4.2: Grid independence study for NACA 0012 wing at R_e = 30,000

Number of Cells	**Angle of Attack 5 ° Ref Values**		**Angle of Attack 5 ° Present Values**		**% Error**	
	Lift Coefficient	**Drag Coefficient**	**Lift Coefficient**	**Drag Coefficient**	**Lift Coefficient**	**Drag Coefficient**
1,391,559	0.270	0.070	0.322	0.031	**19.25%**	**55%**
1,658,357	0.270	0.070	0.310	0.035	**14.81%**	**50%**
2,359,109	0.270	0.070	0.282	0.063	**4.44%**	**10%**

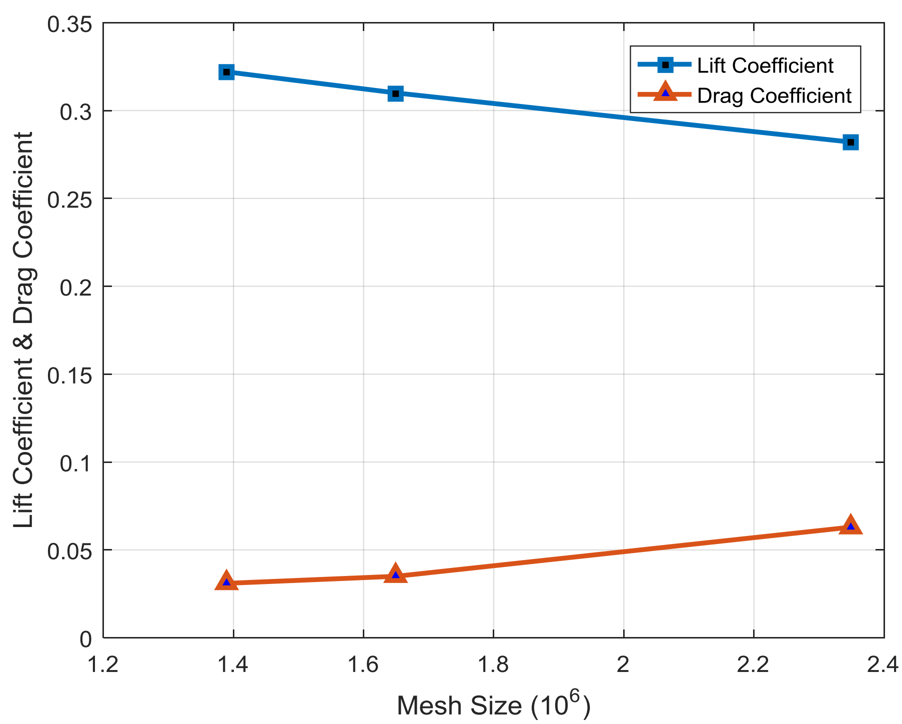
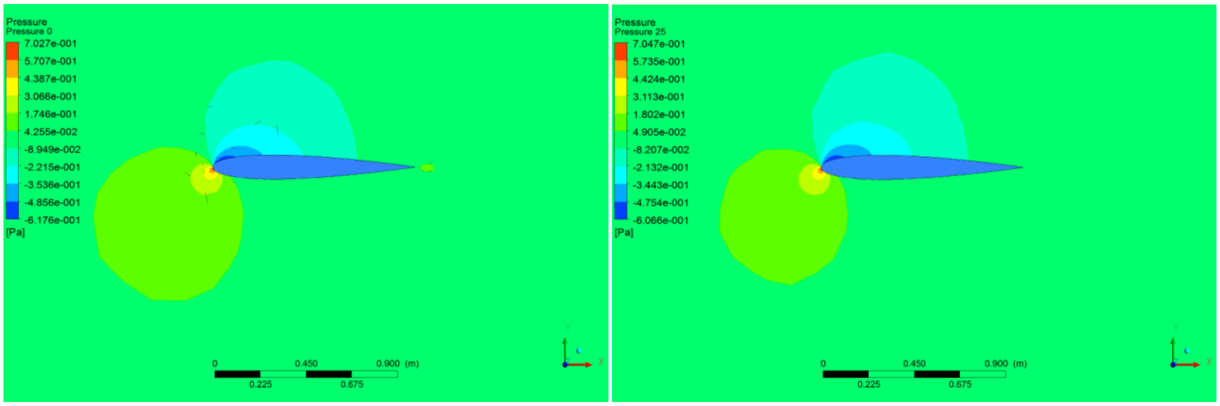


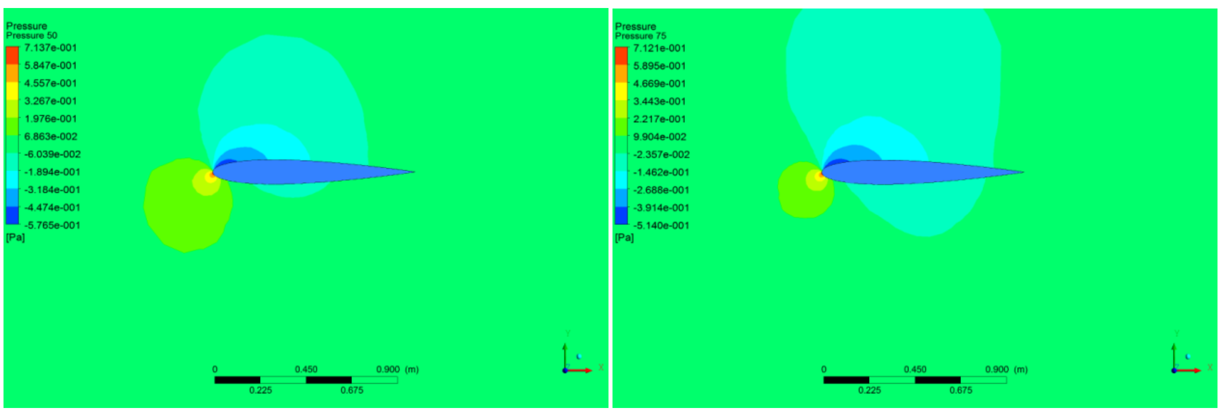
Fig. 4.3: Grid Independence Report of C_L and C_D .

4.3 Pressure Contours

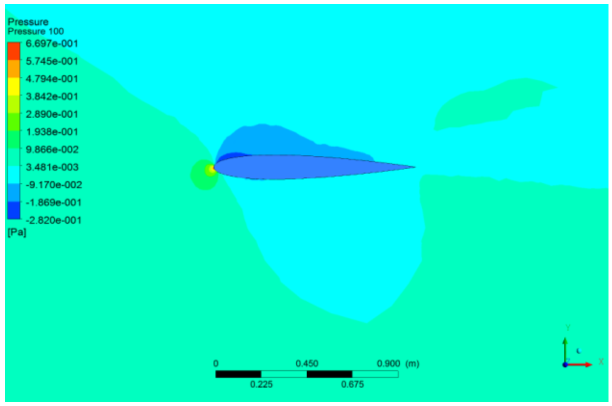
Pressure contours for different locations (z/c) are computed at 5^{o} angle of attack and R_e = 30,000. Figure 4.4 displays at various planes (z/c = 0, 0.5, 1, 1.5 and 2) computed by Fluent code. According to Bernoulli's theorem the upper surface will experience lower pressure than the lower surface, thus the wing is pushed upward effectively into the incoming flow stream.



(a)Pressure Contours at z/c= 0 (b)Pressure Contours at z/c= 0.5



(c) Pressure Contours at z/c = 1 (d) Pressure Contours at z/c= 1.5

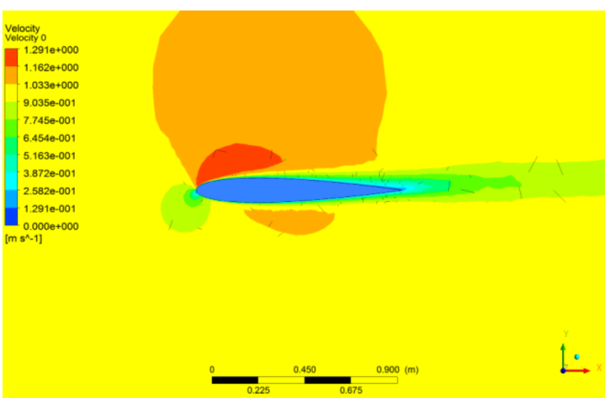


(e) Pressure Contours at z/c= 2

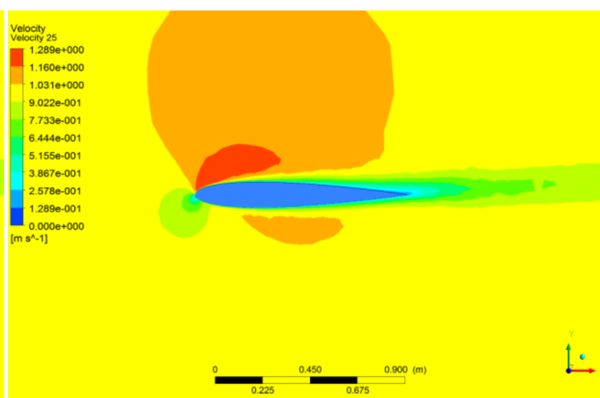
Figure 4.4: Pressure Contours at different locations (looking from side)

4.4 Velocity Contours

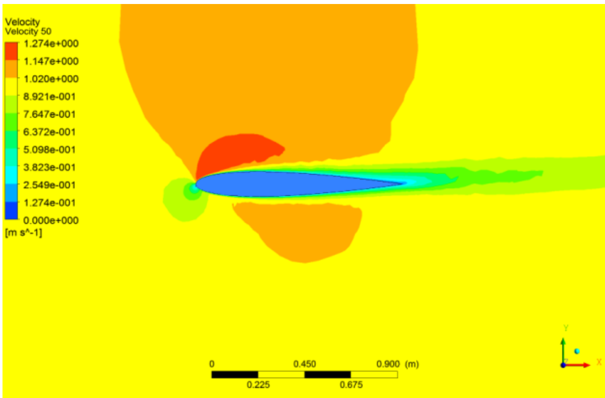
Velocitycontours for different locations (z/c) are computed at 5^{o} angle of attack and R_e = 30,000. Figure 4.5 displays at various planes (z/c = 0, 0.5, 1, 1.5 and 2) computed by Fluent code. The flow accelerates on the upper side of the wing and the velocity of the flow decreases along the lower side. At the leading edge, we can notice the stagnation point where the flow velocity is almost zero.



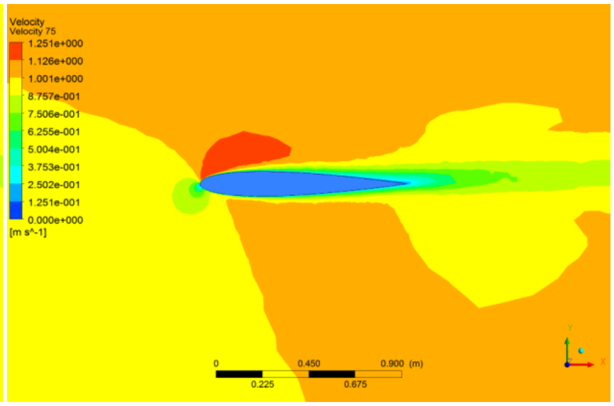
(a) Velocity Contours at z/c = 0



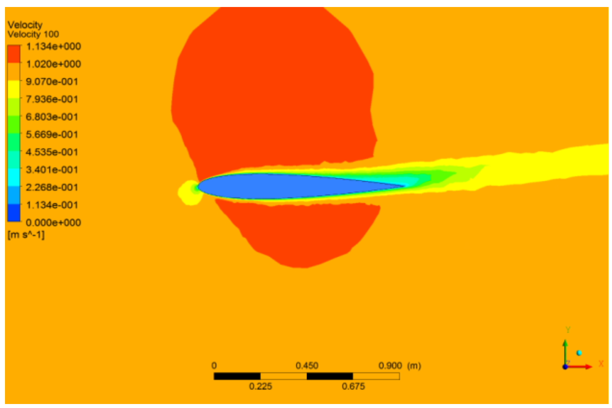
(b) Velocity Contours at z/c = 0.5



(c) Velocity Contours at z/c = 1



(d) Velocity Contours at z/c = 1.5

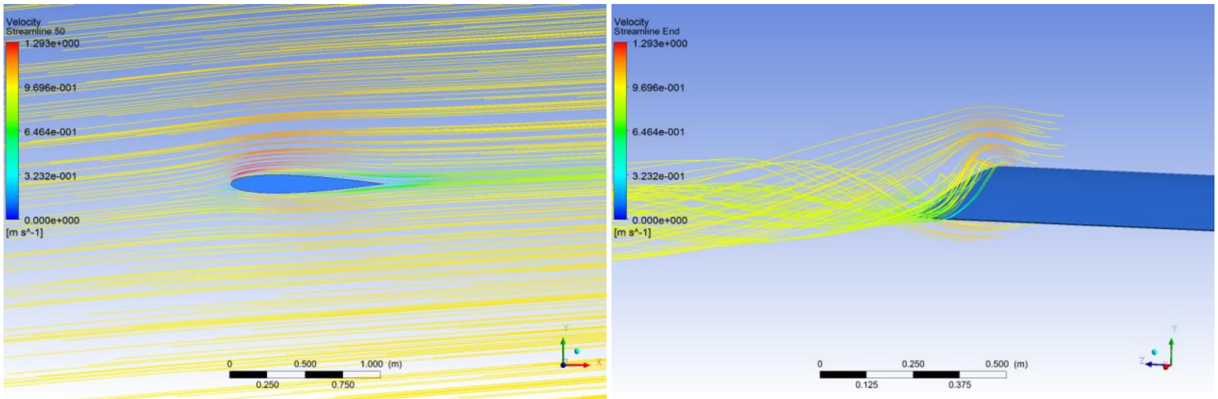


(e) Velocity Contours at z/c = 2

Figure 4.5: Velocity Contours at different locations (looking from side)

4.4 Streamlines

Using blowing as a flow control method is an effective way to postponed separation over the upper side of wing. Delaying separation upon the suction side is an effective way to increase lift and decrease drag, because separation causes a significant loss of energy. Streamlines at mid span and wingtip is shown in Figure. 4.6.



(a) Streamlines at z/c = 1 (b) Streamlines at z/c (Wingtip) = 2

Figure 4.6: Streamlines at different locations (looking from side)

CHAPTER 5: RESULTS AND DISCUSSION

This chapter presents the effect of blowing from wing tip on performance augmentation of 3-D NACA 0012 wing. The parametric study is conducted by changing various parameters to see their effect on lift to drag ratio. We perform simulations by changing various variables to see the effect of best possible configuration which can be adopted to reduce the wingtip vortices for enhanced aerodynamic characteristics. Results are compared with simple case without blowing to see the variations in pressure, velocity, vorticity and pressure coefficient. The results of numerical investigation are present in this chapter.

5.1 Numerical Simulation of 3-D Finite Wing with Blowing from Wingtip

5.1.1 Effect of Area Parameter

Effect of jet slot area is seen in this section by running two cases. Both the cases have rectangular slot area with dimensions of 0.04 x 0.5 m^2 and 0.04 x 0.5 m^2. In both cases, blowing momentum coefficient (C_μ) is fixed at 0.02 and 0.137. Figure. 5.1 shows the dimension of rectangular slot area for the purpose of blowing. The jet slot width is fixed at 4% of chord length, whereas, jet slot length is varied between 50% and 60% of chord length. In both the cases rectangular slot starts from 0.1c. Results of rectangular slot area is shown in Table 5.1. The result carried out at $\alpha = 5^o$ shows that there is not much difference in lift to drag ratio for both cases. For ease of study, we will take our rectangular slot area as 0.04 x 0.5 m^2 for all further simulations.

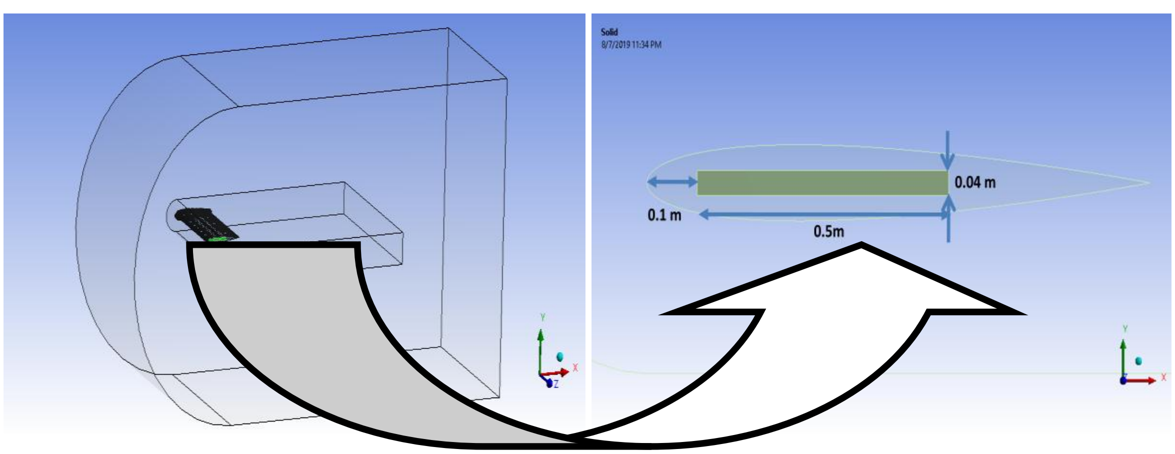


Figure 5.1: Effect of jet slot area on lift-to-drag ratio.

Table 5.1: Effect of area parameter

Cases	A_{jet}	V_{jet}	C_{μ}	C_D	C_L	L/D	Diff
Case 1	0.02	1	0.02	0.065	0.296	4.553	1.72 inc
		2.62	0.137	0.067	0.359	5.358	19.70 inc
Case 2	0.024	1	0.02	0.066	0.300	4.545	1.54 inc
		2.39	0.137	0.068	0.363	5.338	19.25 inc

5.1.2 Effect of Jet Location from Leading Edge

Effect of jet location from leading edge is seen in this section by running three cases. Rectangular slot starts from 0.1c, 0.2c and 0.25 c. Results have been shown in Figure. 5.2. Results shows that as location of rectangular slot moves from leading edge to trailing edge, there is decline in lift to drag ratio due to the reason that the momentum of jet is not effectively reducing the wingtip vortices.

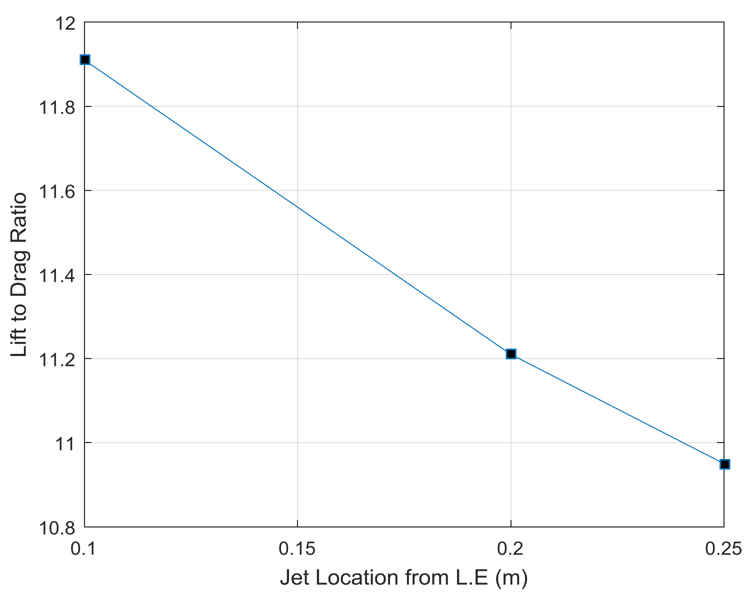


Figure 5.2: Effect of jet location from L.E on lift-to-drag ratio.

5.2 Numerical Simulations of Blowing at Various Angles of Attack

Numerical simulations of 12 x different cases have been carried out in this section. Angle of attack has been varied between 2.5^o to 7.5^o with an increment of 2.5^o. Effects on aerodynamic characteristics with the variation of four different values of blowing momentum coefficient have been investigated for each angle of attack described above. Details of parameters used in these cases are listed in Table 5.2.

Table 5.2: 12 x Cases of Blowing (Rectangular slot)

Angle of Attack = 2.5^o (Cd=0.0549 , Cl=0.142)												
ρ_{jet}	**Slot Area**	**A_{jet}**	**V_{jet}**	**ρ_∞**	**V_∞**	**S**	**C_μ**	**C_D**	**C_L**	**L/D**	**Ref L/D**	**% Increase**
1.225	0.04 x 0.5	0.02	1	1.225	1	2	**0.02**	0.056	0.151	2.696	2.586	**4.270**
			1.5				**0.045**	0.057	0.163	2.860	2.586	**10.582**
			2				**0.08**	0.057	0.177	3.105	2.586	**20.080**
			2.5				**0.125**	0.057	0.189	3.316	2.586	**28.221**
Angle of Attack = 5^o (Cd=0.063, Cl=0.282)												
ρ_{jet}	**Slot Area**	**A_{jet}**	**V_{jet}**	**ρ_∞**	**V_∞**	**S**	**C_μ**	**C_d**	**C_l**	**L/D**	**Ref L/D**	**% Increase**
1.225	0.04 x 0.5	0.02	1	1.225	1	2	**0.02**	0.065	0.296	4.554	4.476	**1.739**
			1.5				**0.045**	0.066	0.315	4.773	4.476	**6.629**
			2				**0.08**	0.067	0.335	5.000	4.476	**11.707**
			2.5				**0.125**	0.067	0.355	5.299	4.476	**18.376**
Angle of Attack = 7.5^o (Cd=0.077 , Cl=0.419)												
ρ_{jet}	**Slot Area**	**A_{jet}**	**V_{jet}**	**ρ_∞**	**V_∞**	**S**	**C_μ**	**C_d**	**C_l**	**L/D**	**Ref L/D**	**% Increase**
1.225	0.04 x 0.5	0.02	1	1.225	1	2	**0.02**	0.08	0.436	5.450	5.441	**0.165**
			1.5				**0.045**	0.083	0.456	5.494	5.441	**0.974**
			2				**0.08**	0.083	0.482	5.807	5.441	**6.731**
			2.5				**0.125**	0.084	0.506	6.024	5.441	**10.711**

5.2.1 Effect of Blowing Momentum Coefficient

The effect of steady blowing from a jet slot of area 0.04 x 0.5 m^2 starting at a location of 0.1c for different blowing momentum coefficient (C_μ) on drag coefficient, lift coefficient and lift to drag ratio is presented from Figure.5.3to Figure.5.5. Figure. 5.3 shows that the drag coefficient increases with different values of blowing momentum coefficient and is highest with C_μ = 0.125 at $\alpha = 5^o$. It can also be seen that there is no much difference in drag coefficient at $\alpha = 2.5^o$ and $\alpha = 7.5^o$.Figure. 5.4 shows the best performance with C_μ = 0.125 at all angle of attacks. It can be seen that a linear increase in lift coefficient is present. Figure. 5.5 shows the L/D which is maximum of 6.024 at 7.5^o angle of attack.

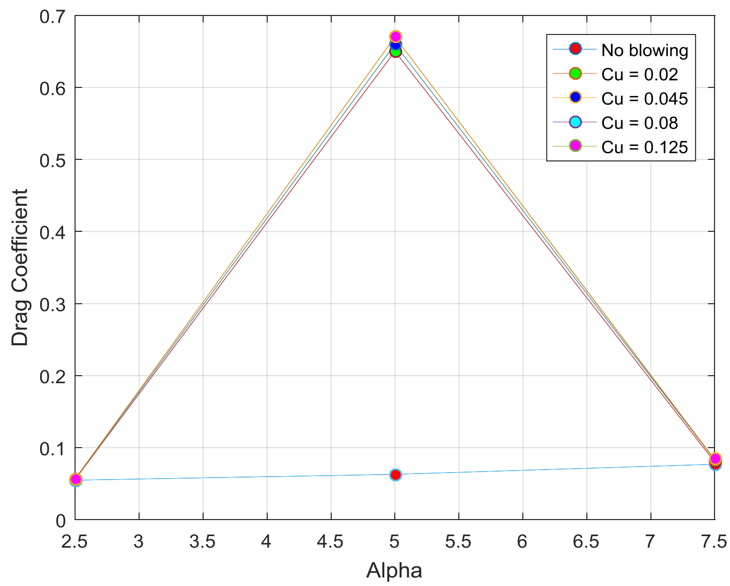


Figure 5.3: Drag coefficient at various angles of attack with different blowing momentum coefficient.

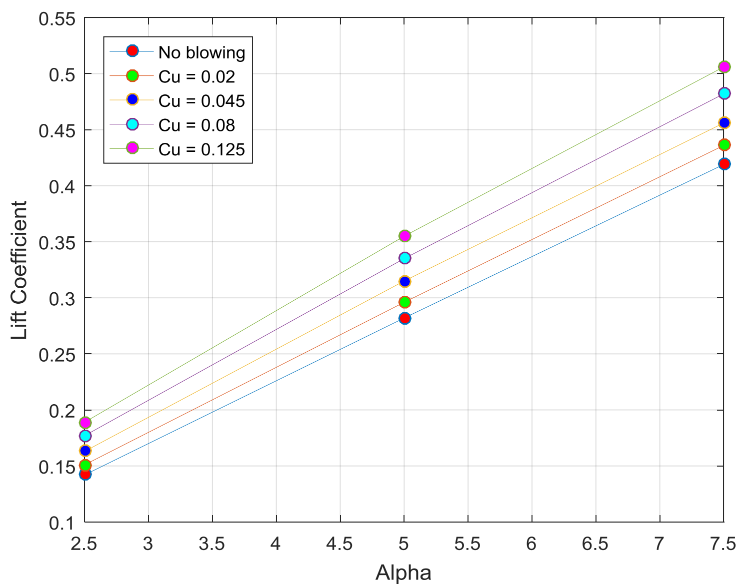


Figure 5.4: Lift Coefficient at various angles of attack with different blowing momentum coefficient.

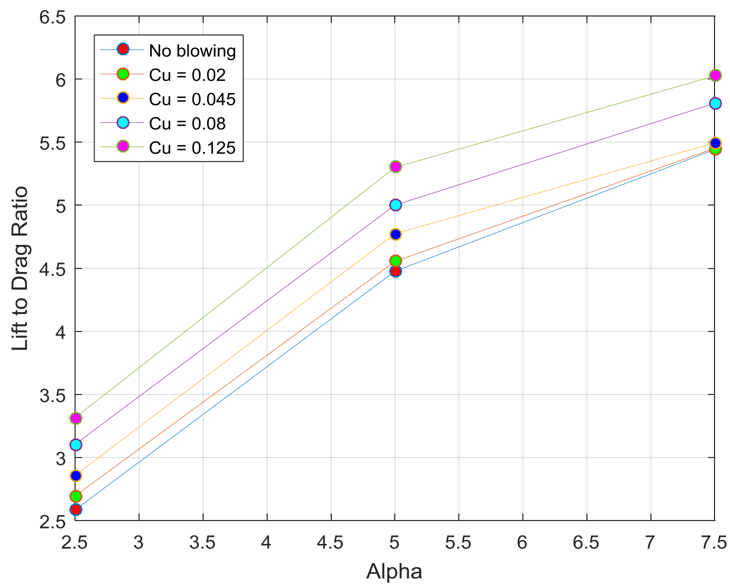
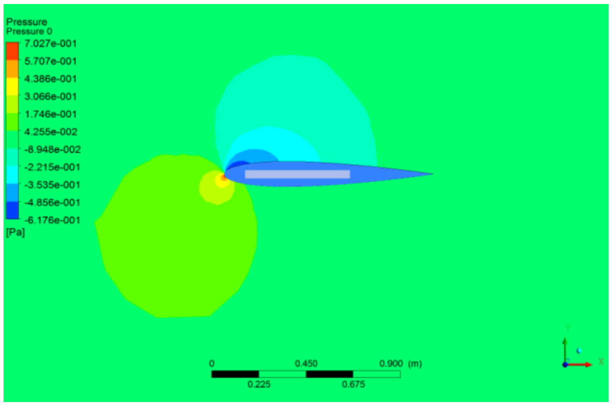


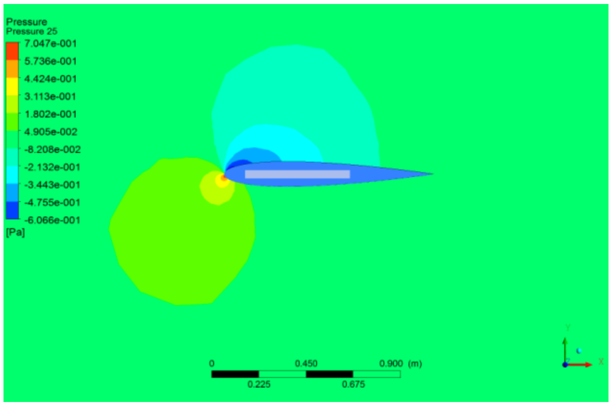
Figure 5.5: Lift-to-drag ratio variation at various angles of attack with different blowing momentum coefficient.

5.2.2 Pressure Contours

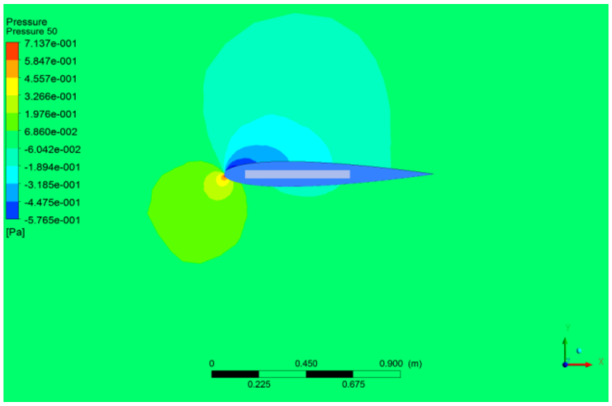
Pressure contours for different locations (z/c) are computed at 5^{o} angle of attack and R_e = 30,000. Figure 5.6 displays at various planes (z/c = 0, 0.5, 1, 1.5 and 2) computed by Fluent code. According to Bernoulli's theorem the upper surface will experience lower pressure than the lower surface, thus the wing is pushed upward effectively into the incoming flow stream.



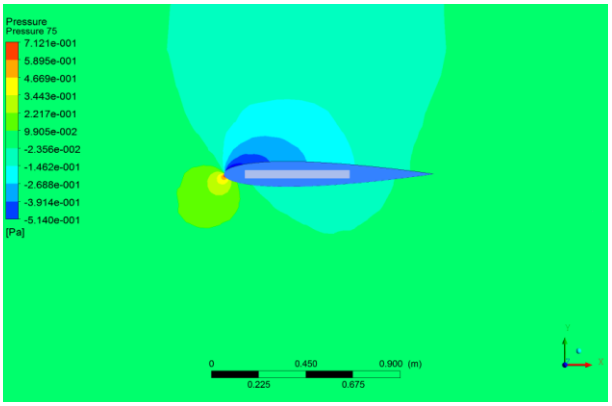
(a) Pressure Contours at z/c = 0



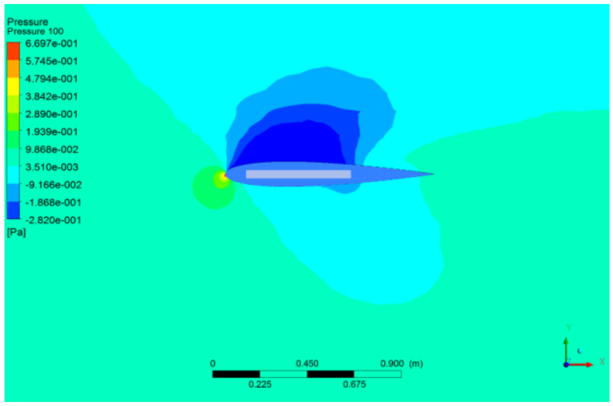
(b) Pressure Contours at z/c = 0.5



(c) Pressure Contours at z/c = 1



(d) Pressure Contours at z/c= 1.5

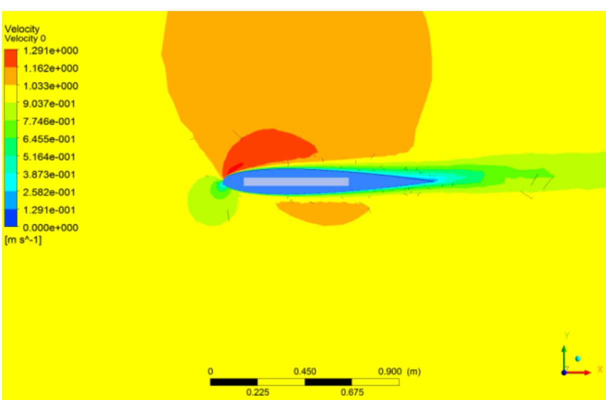


(e) Pressure Contours at z/c= 2

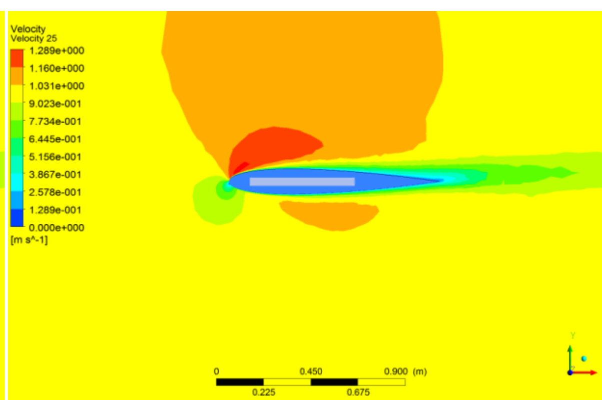
Figure 5.6: Pressure Contours at different locations (looking from side).

5.2.3 Velocity Contours

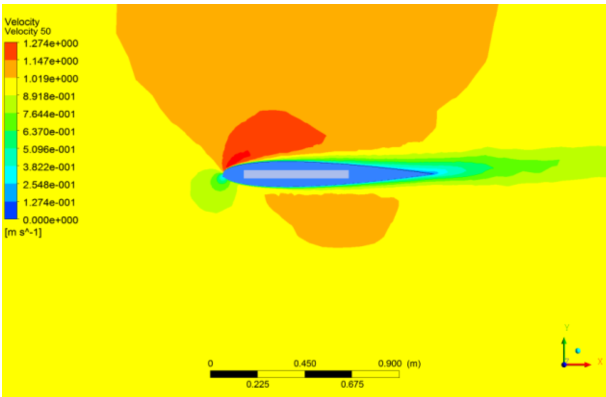
Velocitycontours for different locations (z/c) are computed at 5^{o} angle of attack and R_e = 30,000. Figure 5.7 displays at various planes (z/c = 0, 0.5, 1, 1.5 and 2) computed by Fluent code. The flow accelerates on the upper side of the wing and the velocity of the flow decreases along the lower side. At the leading edge, we can notice the stagnation point where the flow velocity is almost zero.



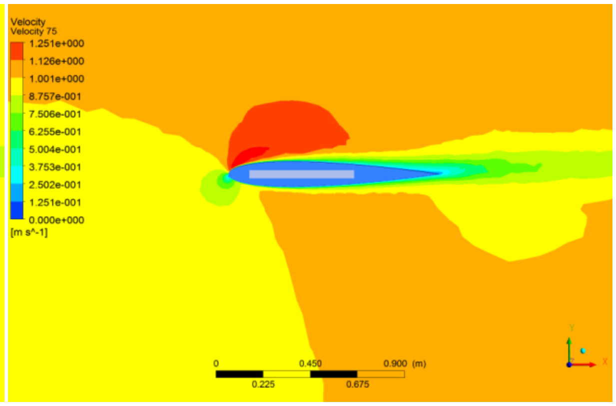
(a) Velocity Contours at z/c = 0



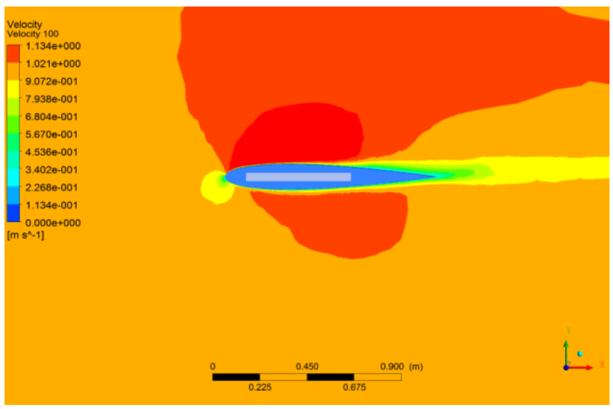
(b) Velocity Contours at z/c = 0.5



(c) Velocity Contours at z/c = 1



(d) Velocity Contours at z/c = 1.5

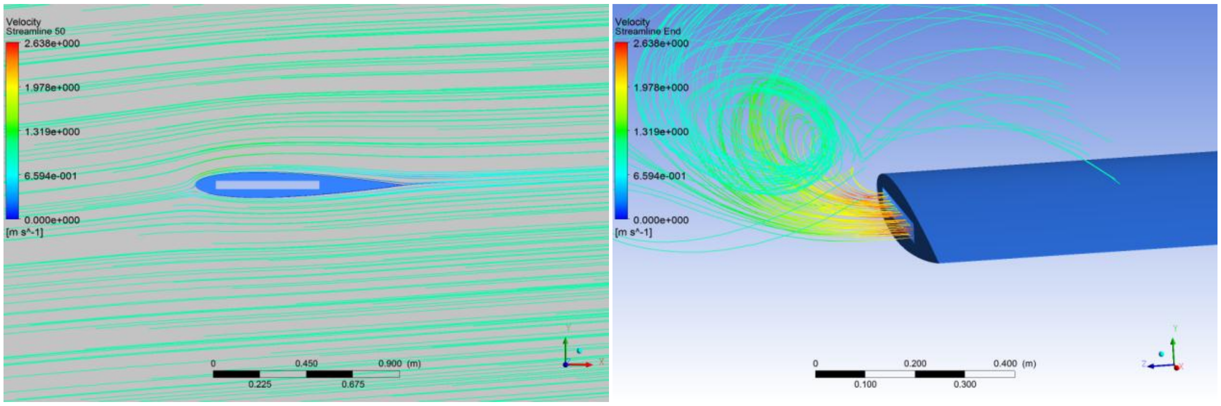


(e) Velocity Contours at z/c = 2

Figure 5.7: Velocity Contours at different locations (looking from side).

5.2.4 Streamlines

Using blowing as a flow control method is an effective way to postponed separation over the upper side of wing. Delaying separation upon the suction side is an effective way to increase lift and decrease drag, because separation causes a significant loss of energy. Streamlines at mid span and wingtip is shown in Figure. 5.8.



(a) Streamlines at z/c = 1 (b) Streamlines at z/c (Wingtip) = 2

Figure 5.8: Streamlines at different locations (looking from side)

5.2.5 Lift-to-Drag Ratio @ Angle of Attack ($\alpha = 2.5^o$)

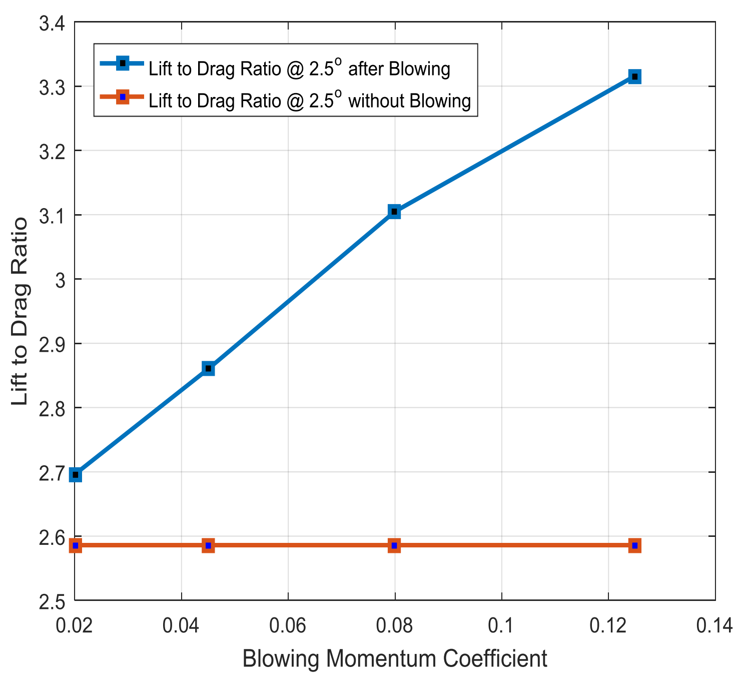


Figure 5.9: Lift-to-Drag Ratio @ $\alpha = 2.5^o$

5.2.6 Lift-to-Drag Ratio @ Angle of Attack ($\alpha = 5^o$)

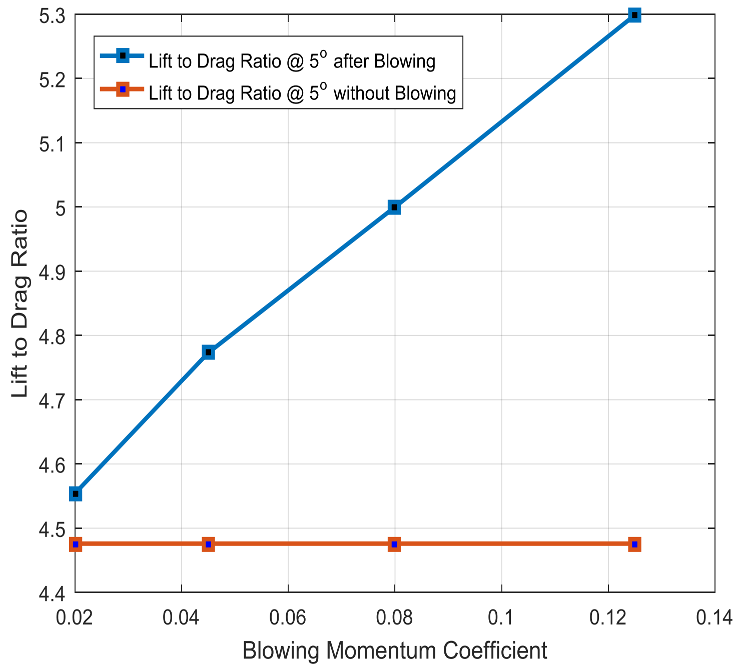


Figure 5.10: Lift-to-Drag Ratio @ $\alpha = 5^o$

5.2.7 Lift-to-Drag Ratio @ Angle of Attack ($\alpha = 7.5^o$)

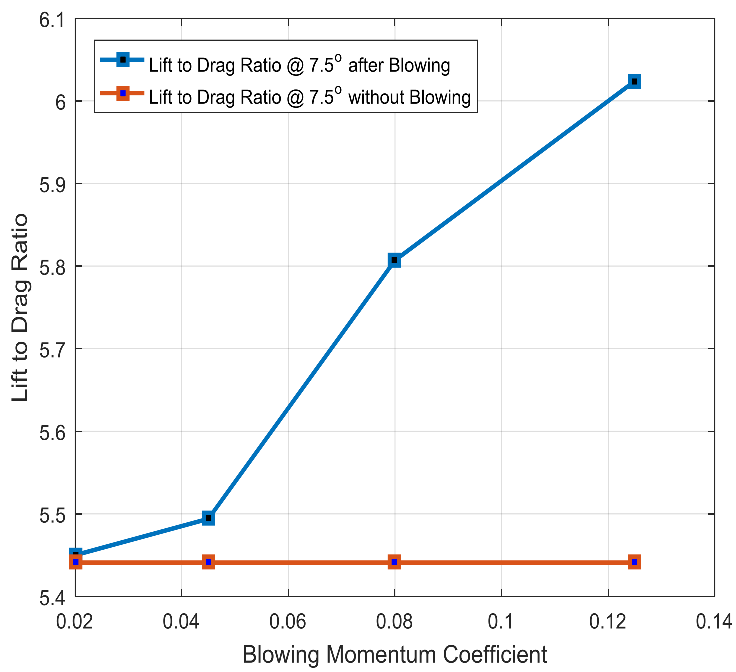


Figure 5.11: Lift-to-Drag Ratio @ $\alpha = 7.5^o$

5.2.8 Comparison - Lift to Drag Ratio

Lift-to-drag ratio comparison between various angles of attack is shown in Figure. 5.12. It can be seen that there is a significant increase between $\alpha = 2.5^o$ and $\alpha = 5^o$. However, difference between $\alpha = 5^o$ and $\alpha = 7.5^o$ is not much.

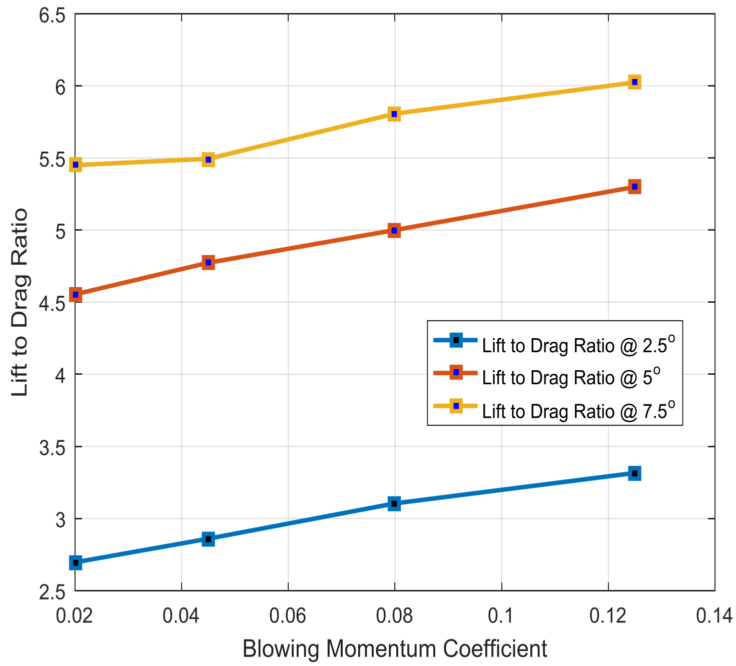


Figure 5.12: Comparison - Lift-to-Drag Ratio.

5.2.9 Comparison - Lift to Drag Ratio % Increase

Lift-to-drag ratio comparison in terms of % increase between various angles of attack is shown in Figure. 5.13. It can be seen that there is a significant increase at $\alpha = 2.5^o$ and in lift-to-drag ratio from 4.270 % to 28.221 %. Whereas at $\alpha = 5^o$, increase in lift-to-drag ratio is from 1.739 % to 18.376 %. Moreover, at $\alpha = 7.5^o$ there is an increase of 0.165 % to 10.711 % in lift-to-drag ratio.

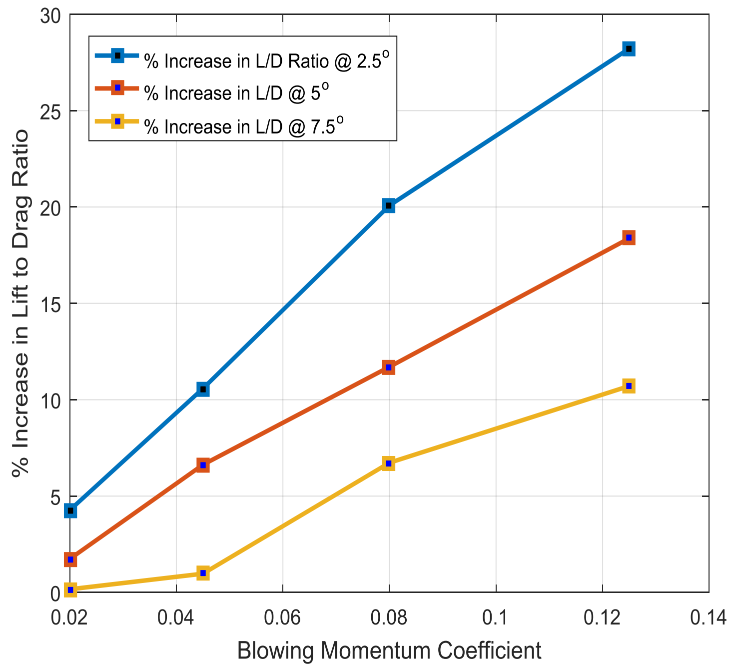


Figure 5.13: Comparison - Lift-to-Drag Ratio % Increase.

CHAPTER 6: CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

In this study, the effect of blowing momentum coefficient (C_μ) having same blowing slot area and location has been studied on a 3-D finite wing and a finite wing with blowing on wingtip. Blowing momentum coefficient has been varied with C_μ = 0.02, 0.045, 0.08 and 0.125 at various angles of attack ranging from 2.5^o to 7.5^o in increments of 2.5^o. Here we give some important conclusions of the study. Recommendations for future work are also included.

6.1 Conclusions

Based on the above study, some of the important conclusions are appended below:

- Blowing momentum coefficient is most effective at C_μ = 0.125. Value of $C_\mu$mainly depends on jet velocity (V_j= 2.5 m/s in thi scase) for all angle of attacks. It improves the aerodynamic efficiency from 10.71% to 28.22%.
- Aerodynamic performance was found to be more effective at lower angle of attack with same slot area and blowing coefficient as compared with higher angle of attack of 5^o and 7.5^o.
- The optimized jet location is found to be started from 10% of the chord from leading edge. Beyond 0.1c, the influence of blowing is less significant at 0.25c and 0.25c irrespective of same C_{μ}.
- The pressure and velocity contours observed at different span wise locations shows that the improvement in aerodynamic characteristics is due to the addition of momentum through blowing jet, which delays the flow separation by virtually enhancing the span of finite wing. The same may be visualized by thinking it as a fluid extension of the wing itself.
- Computations reveals that the strength of the vorticity is significantly reduced when blowing is applied on the wing tip.

6.2 Future Work

A recommendation for future work is to perform present numerical study to see the effects on aerodynamic performance by:

- Variation of jet slot area. In present study, width of jet slot was fixed at 4% of chord length. However, in the light of literature review, same can be varied between 2.5% to 4% of chord length.
- Varying the location of jet slot area with respect to upper and lower surface of wingtip.
- Numerical simulation of circular blowing jet slot.

REFERENCES

[1] F.W. Lanchester, “Aerodynamics,” Constable& Co., Ltd., London, 1918.

[2] L. Prandtl, Applications of Modern Hydrodynamics to Aeronautics, *TR* 116, 1921.

[3] L.J. Clancy, *Aerodynamics*, Pitman Publishing Limited, 1975. London.

[4] E.L.Houghton and P.W. Carpenter, Butterworth Heinmann, *Aerodynamics for Engineering Students* (5th ed.), 2003.

[5] Anderson, John D. (2001), *Fundamental of aerodynamics*, McGraw-Hill, Boston.

[6] A. Pope, *Basic wing and airfoil theory*. Mineola, N.Y.: Dover Publications, 2009.

[7] G. Kanesan, S. Mansor and A. Abdul-Latif, "Validation of UAV Wing Structural Model for Finite Element Analysis", *JurnalTeknologi*, vol. 71, no. 2, 2014.

[8] P. Margaris and I. Gursul, "Vortex topology of wing tip blowing", *Aerospace Science and Technology*, vol. 14, no. 3, pp. 143-160, 2010.

[9] P. Margaris and I. Gursul, "Effect of steady blowing on wing tip flowfield", Published by the American Institute of Aeronautics and Astronautics, Inc., 2004.

[10] K. Yousefi, R. Saleh and P. Zahedi, "Numerical study of blowing and suction slot geometry optimization on NACA 0012 airfoil", *Journal of Mechanical Science and Technology*, vol. 28, no. 4, pp. 1297-1310, 2014.

[11] M.Goodarzi, M. Rahimi and R. Fereidouni, "Investigation of Active Flow Control over NACA0015 Airfoil Via Blowing", *International Journal of Aerospace Sciences*, vol. 1, no. 4, pp. 57-63, 2012.

[12] D.Tavella, N. Wood and P. Harrits, "Measurements on Wing-Tip Blowing", Stanford university, 1985.

[13] H. Lim, "Numerical Study of the Trailing Vortex of a Wing with Wing-Tip By Blowing", Stanford University, 1994.

[14] R.Mineck, "Study of Potential Aerodynamic benefits in Spanwise Blowing at Wingtip", Langley Research Center , Virginia, 1995.

[15] D. Boyd, "Navier-Stokes Computations of a Wing-Flap Model With Blowing Normal to the Flap Surface", NASA, Langley Research Center, Hampton, Virginia, 2005.

[16] A.Edstrand and L. Cattfesta, "Experimental Investigation of Steady Circulation Control Blowing Over A Rounded Wingtip", *International symposium of turbulence and shear flow phenomenon, France*, 2013.

[17] S S Baljit, M R Saad, A Z Nasib, A Sani, M R A Rahman and A C Idris, "Suction and blowing flow control on airfoil for drag reduction in subsonic flow", International Conference on Materials Physics and Mechanics, 2017.

[18] Qilde, "Aerodynamic characteristics of a swept wing with spanwise blowing", College of aeronautics, Cranfield, 1956.

[19] K. Duraisamy and J. Baeder, "Numerical Simulation of the Effects of Spanwise Blowing on Wing-tip Vortex Formation and Evolution", *Journal of Aircraft*, vol. 43, no. 4, pp. 996-1006, 2006.

[20] A. Edstrand and L. Cattafesta, "Topology of a Trailing Vortex Flow Field with Steady Circulation Control Blowing", *53rd AIAA Aerospace Sciences Meeting*, 2015.

[21] S. Samal and P. Dash, "Reduction of Wingtip Vortex from Suction at Wingtip", *Mechanical Engineering Research*, vol. 3, no. 1, p. 152, 2013.

[22] K. Yousefi and R. Saleh, "Three-dimensional suction flow control and suction jet length optimization of NACA 0012 wing", *Meccanica*, vol. 50, no. 6, pp. 1481-1494, 2015.

[23] K.Yousefi and R. Saleh, "The Effects of Trailing Edge Blowing on Aerodynamic Characteristics of the NACA 0012 Airfoil and Optimization of the Blowing Slot Geometry", *Journal of Theoretical and Applied Mechanics*, vol. 52, no. 1, pp.165-179, 2014.