4-2014

Flow Separation Control Using Synthetic Jets on a Flat Plate

Karunakaran Saambavi

Embry-Riddle Aeronautical University - Daytona Beach

Follow this and additional works at: https://commons.erau.edu/edt

Part of the Aerospace Engineering Commons

Scholarly Commons Citation
https://commons.erau.edu/edt/180

This Thesis - Open Access is brought to you for free and open access by Scholarly Commons. It has been accepted for inclusion in Dissertations and Theses by an authorized administrator of Scholarly Commons. For more information, please contact commons@erau.edu, wolfe309@erau.edu.
FLOW SEPARATION CONTROL USING SYNTHETIC JETS ON A FLAT PLATE

by

KARUNAKARAN SAAMBAVI

A Thesis Submitted to the College of Engineering, Department of Aerospace Engineering
in Partial Fulfillment of the Requirements for the Degree of
Master of Science in Aerospace Engineering

Embry-Riddle Aeronautical University
Daytona Beach, Florida
April 2014
FLOW SEPARATION CONTROL USING SYNTHETIC JETS ON A FLAT PLATE

by

KARUNAKARAN SAAMBAVI

This thesis was prepared under the direction of the candidate’s Thesis Committee Chair, Dr. Yechiel Crispin and Thesis Committee Members Dr. Reda Mankbadi and Dr. Dongeun Seo and has been approved by the Thesis Committee. It was submitted to the Department of Aerospace Engineering in partial fulfillment of the requirements for the Degree of Master of Science in Aerospace Engineering.

Thesis Review Committee:

Dr. Yechiel Crispin
Committee Chair

Dr. Reda Mankbadi
Committee Member

Dr. Dongeun Seo
Committee Member

Dr. Yi Zhao
AE Graduate Program Coordinator

Dr. Robert Oxley
Associate Vice President of Academics

4/24/2014
Date

4/24/2014
Date

4/24/2014
Date

4/25/2014
Date
ACKNOWLEDGEMENTS

I would like to express my sincere thanks to my advisor, Dr. Yechiel Crispin for his continuous support, suggestions and consistent encouragement. The completion of this thesis would have been impossible without his guidance.

I would like to express my sincere gratitude to the committee members, Dr. Reda Mankbadi and Dr. Dongeun Seo. I would also like to thank my friends who had helped me in the process of this thesis.

I cannot finish without expressing my deepest gratitude to my family and special friends for their continuous support, love, patience and encouragement during my academic years.

Karunakaran Saambavi
Embry-Riddle Aeronautical University
April, 2014
Researcher: Karunakaran Saambavi
Title: Flow Separation Control Using Synthetic Jets on a Flat Plate
Institution: Embry-Riddle Aeronautical University
Degree: Master of Science in Aerospace Engineering
Year: 2014

The primary goal of this thesis is to assess the effect of synthetic jets on flow separation. CFD simulation is conducted for laminar flow over a flat plate using the commercial software ANSYS FLUENT. The oscillating zero mass jet flow is simulated by imposing a harmonically varying boundary condition on the wall surface.

In this work, the effect of synthetic jets for different angles of attack and different frequencies ranging from 200 Hz to 800 Hz was assessed. The application of the synthetic jet actuators is based in their ability to energize the boundary layer, thereby providing significant increase in the lift coefficient.

The performed numerical simulation investigates the flow at $Re = 2 \times 10^6$. The oscillatory injection takes place at one fourth the length of the chord from the leading edge. Streamline fields and the pressure contours obtained for different angles of attack are compared with published data. An increase in the lift coefficient can also be observed due to the pulsating jet flow.
TABLE OF CONTENTS

Thesis Committee Review .......................................................... ii
Acknowledgement ................................................................... iii
Abstract ................................................................................. iv
List of Tables ........................................................................ viii
List of Figures .......................................................................... ix

Chapter

I Introduction .............................................................................. 1

Motivation ................................................................................. 1
Flow Separation ....................................................................... 2
Flow Control ........................................................................... 4
Theoretical Background ............................................................... 4
Dissertation Outline ................................................................. 6

II Synthetic Jet Actuators ......................................................... 8

Principles of Operation ............................................................... 8
Applications of Synthetic Jets ................................................... 9
Literature Review .................................................................... 11
### III Fundamental Equations and Computational Methods

<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fundamental Equations</td>
<td>17</td>
</tr>
<tr>
<td>Conservation of Mass</td>
<td>17</td>
</tr>
<tr>
<td>Conservation of Momentum</td>
<td>18</td>
</tr>
<tr>
<td>CFD: Overview</td>
<td>19</td>
</tr>
<tr>
<td>Definition, Benefits and Applications of CFD</td>
<td>20</td>
</tr>
<tr>
<td>CFD Process</td>
<td>20</td>
</tr>
<tr>
<td>FLUENT/ POINTWISE Description</td>
<td>22</td>
</tr>
</tbody>
</table>

### IV Numerical Simulation on a Flat Plate

<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overview</td>
<td>25</td>
</tr>
<tr>
<td>Grid Generation</td>
<td>25</td>
</tr>
<tr>
<td>Solution Methodology</td>
<td>30</td>
</tr>
<tr>
<td>Boundary Conditions</td>
<td>31</td>
</tr>
<tr>
<td>Problem Definition in Fluent</td>
<td>33</td>
</tr>
<tr>
<td>Definition of Fluid Properties and Equation of State</td>
<td>35</td>
</tr>
<tr>
<td>Definition of Operating Conditions</td>
<td>35</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>-----------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Definition of Boundary Conditions</td>
<td>35</td>
</tr>
<tr>
<td>Solution Execution and Convergence</td>
<td>35</td>
</tr>
<tr>
<td>V Results and Discussions</td>
<td>38</td>
</tr>
<tr>
<td>Flow over a Flat Plate</td>
<td>31</td>
</tr>
<tr>
<td>Synthetic Jets in Quiescent Flow</td>
<td>51</td>
</tr>
<tr>
<td>Interaction of Synthetic Jets with Cross-Flow</td>
<td>53</td>
</tr>
<tr>
<td>Effect of Frequency</td>
<td>58</td>
</tr>
<tr>
<td>VI Conclusions and Future Work</td>
<td>60</td>
</tr>
<tr>
<td>References</td>
<td>61</td>
</tr>
<tr>
<td>Appendices</td>
<td>65</td>
</tr>
<tr>
<td>A Creating the Mesh in POINTWISE</td>
<td>65</td>
</tr>
<tr>
<td>B Setting the Problem in FLUENT 14.5</td>
<td>67</td>
</tr>
<tr>
<td>C The User-Defined Function for Transient Velocity</td>
<td>69</td>
</tr>
</tbody>
</table>
LIST OF TABLES

Table 1: Flow separation control techniques .......................................................... 7
Table 2: Nodes and Cells in the mesh .................................................................... 26
Table 3: Spacing and number of points on the connector for simple grid ........... 26
Table 4: Nodes and Cells in the mesh including the flat plate ........................... 27
Table 5: Spacing and the number of points on the connector ............................. 28
Table 6: Physical properties of the numerical simulation .................................... 30
Table 7: Streamwise and Cross-stream velocities for different angles of attack 31
Table 8: Boundary conditions for modeling synthetic jets in quiescent medium 36
Table 9: Monitored equations and convergence criteria ........................................ 37
Table 10: Coefficient of lift computed for different angles of attack at Re =2 × 10^6 46
LIST OF FIGURES

Figure 1: Schematic of flow separation 3

Figure 2: Schematic of a synthetic jet actuator (not to scale) 9

Figure 3: Two-dimensional grid distribution and boundary conditions used to simulate the synthetic jet 27

Figure 4: Two-dimensional grid distribution and boundary conditions used for synthetic jet flat plate simulations 29

Figure 5: The growth of the separation bubble on the surface of the flat plate for uncontrolled flow for $M_{\infty} = 0.1$ and $Re=2 \times 10^6$ and thin airfoil stall 40

Figure 6: Comparison of flow over a flat plate with (a) experimental and (b) computational results at an angle of attack 3° 41

Figure 7: Comparison of flow over a flat plate with (a) experimental and (b) computational results at an angle of attack 7° 42

Figure 8: Comparison of flow over a flat plate with (a) experimental and (b) computational results at an angle of attack 9° 43

Figure 9: Comparison of flow over a flat plate with (a) experimental and (b) computational results at an angle of attack 15° 44

Figure 10: Comparison of $C_L$ results for a flat plate ($M_{\infty} = 0.1$ and $Re = 2 \times 10^6$) with numerical simulation obtained by Rosas C.R. on NACA 0012 airfoil ($M_{\infty} = 0.3$ and $Re =$
1x10^6) and with experimental data from “Theory of wing sections” by Ira H. A. et al. on NACA 0006 airfoil (Re = 3x10^6)

Figure 11: Comparison of CD vs. CL between the present case for a flat plate (M∞ = 0.1 and Re = 2x10^6) and the experimental data obtained from “Theory of Wing Sections” by Ira H. A. et al. on NACA 0006 airfoil (Re = 3x10^6)

Figure 12: Pressure contour plots for a flow over a flat plate measured in Pascals at Re = 2x10^6 and M∞ = 0.1

Figure 13: Pressure contour plots for a flow over a flat plate measured in Pascals at Re = 2x10^6 and M∞ = 0.1

Figure 14: Pressure contour plots for a flow over a flat plate measured in Pascals at Re = 2x10^6 and M∞ = 0.1

Figure 15: Pressure contour plots for the synthetic jet actuation with f = 700 Hz

Figure 16: Velocity vectors colored by static pressure during blowing

Figure 17: Effect of oscillatory flow separation control on C_L on the flat plate at α = 0° and α = 5°, M = 0.1, Re = 2x10^6, f = 800Hz, V_j = 3.4 m/s

Figure 18: Effect of oscillatory flow separation control on C_L on the flat plate at α = 10° and α = 15°, M = 0.1, Re = 2x10^6, f = 800Hz, V_j = 3.4 m/s

Figure 19: Effect of oscillatory flow separation control on C_L on the flat plate at α = 18°, M = 0.1, Re = 2x10^6, f = 800Hz, V_j = 3.4 m/s
Figure 20: $C_L$ versus angle of attack for a flat plate. The controlled numerical simulation has been performed on a NACA 0012 at $Re = 1 \times 10^6$

Figure 21: Comparison of $C_D$ vs. $C_L$ between the present case for a flat plate with and without synthetic jets ($M_{\infty} = 0.1$ and $Re = 2 \times 10^6$) and the experimental data obtained from “Theory of Wing Sections” by Ira H. A. et al. on NACA 0006 airfoil ($Re = 3 \times 10^6$)

Figure 22: Influence of the variation of frequency on CL. Simulations correspond to $M_{\infty} = 0.1$, $\alpha = 10^\circ$ and $Re = 2 \times 10^6$

Figure 23: Response frequency Versus Input frequency
CHAPTER 1
INTRODUCTION

Motivation

Flow control is one of the leading areas of research in fluid mechanics. One of the important applications of flow control is in the aerospace industry, where flow control techniques increase the performance of the aircraft and reduce drag. Flow control can be used to delay transition, reduce turbulence, prevent separation, and to modify the flow-field. It is basically an application-dependent technique. Therefore a particular application must be carefully evaluated through analytical, experimental or numerical means to reach desired goal.

Among the many active flow control devices, one of the most widely investigated devices is the synthetic jet actuator which is also known as the zero-net-mass-flux actuator. The potential applications of this simple device are thrust vectoring of jets, mixing enhancement in shear layers, reduction in separated flow regions, heat transfer, drag reduction in turbulent boundary layers, etc. The most important application among these has been the reduction of separation in flow regimes e.g. on wings at high angles of attack. Synthetic jet actuator is a very versatile device because it generates unsteady forcing which has been proven to be more effective than steady forcing. Also synthetic jet actuator transfers linear momentum to the flow-field without net mass injection. Therefore the need to supply fluid for blowing and suction is eliminated. This also eliminates additional energy supply, complicated piping and reduces the inherent losses
present in the conventional active flow control devices. The work presented in this dissertation mainly focuses on the flow separation control using synthetic jet actuators.

**Flow Separation**

Flow separation is the detachment or breakaway of the fluid from the solid surface and it takes the forms of eddies and vortices. Flow separation occurs when the velocity at the wall is zero or negative and an inflection point exist in the velocity profile. It can also be caused by an adverse pressure gradient in the direction of the flow or due to a geometric discontinuity, that is, corners, sharp turns or higher angles of attack representing sharply decelerating flow where the loss in energy leads to separation. Separation thickens the rotational flow region adjacent to the surface and increases the velocity component normal to the surface.

Separation is always associated with some kind of losses such as increase in pressure drag, loss of lift, stall and pressure recovery losses. Vortex shedding is another undesirable characteristic of separation which causes vibrations in the structure which leads to serious failures when the resonance frequency is reached.

At the beginning of the twentieth century, Ludwig Prandtl explained the physical phenomenon of flow separation. Figure 1 shows the velocity profile in a two-dimensional boundary layer in the vicinity of the separation point. Upstream of the separation point, within the boundary layer of thickness $\delta$, a strong velocity gradient $du/dy$ is produced by viscosity which prevails near the wall. At the wall, the no-slip condition causes the velocity to vanish; increasing rapidly with the vertical distance until it gradually approaches the freestream velocity $U_\infty$. Compared to the freestream, the flow in the
boundary layer suffers a greater deceleration. This slowing-down process becomes very noticeable near the surface, that the successive velocity profiles in the streamwise direction change. The energy associated in the flow close to the surface is small; therefore the ability of the flow to overcome the adverse pressure gradient becomes limited. The shear stress opposes the outer-flow field prior to separation because the velocity gradient near the wall is positive. After separation, the velocity gradient at the surface is negative. Therefore the separation point occurs at a point where the velocity gradient vanishes \((du/dy)_{wall} = 0\). Downstream of the separation point, the flow adjacent to the surface reverses in direction so that a circulatory movement in a plane normal to the surface takes place.

![Figure 1: Schematic of flow separation](image)
Flow control

The modern use of flow control was initiated by Ludwig Prandtl at the beginning of the twentieth century, although the idea has been around for centuries. Since then, the ultimate goal of this extensive research has been to develop techniques to manipulate the fluid flow to achieve a variety of desired outcomes in industrial applications. Performance improvement and efficiency maximization in an application involving fluid flow are the desired goals in achieving flow control. This chapter includes the general idea of flow control and its advantages, control techniques and a historical perspective.

Theoretical Background

Flow control refers to an attempt of favorably altering the characteristics of the flow-field. The subject has received significant attention by engineers and scientists since a desired change in the fluid behavior can be generated by actively or passively controlling the flow-field. Some of the important advantages of flow control are the benefits it brings to an industrial application involving fluid flow such as performance improvement, noise reduction, lift enhancement, prevention of separation, drag reduction, maximization of efficiency, fuel savings of vehicles, etc.

Prandtl introduced the boundary layer theory and the mechanics of steady separation in 1904, which is now the pioneer of the modern idea of flow control. In his study, he used active control by applying suction to delay the boundary layer separation from the surface of a cylinder. No significant advances were made until the 1940s. Laminar flow control over a wing was the focus shortly before and during the Second World War because the military required the development of fast and efficient aircrafts, ships and missiles, in
which laminar flow control could play a critical role for success. These studies explored the feasibility of utilizing full-scale boundary layer control over a large aircraft. A successful example of such studies includes the flight-test program of the X-21, in which suction was used to delay the transition on a swept wing, which proved the ability to achieve laminar flow over approximately 75% of the wing surface [1]. Later, in the early 1970s, the oil crisis brought interest in flow control in the transport sector. Studies including drag reduction for commercial aircraft and other sea/land vehicles were also investigated to conserve energy. In addition, methods for drag reduction in oil pipelines and other industrial applications were emphasized by the government agencies and private corporations. In the 1990s, the flow control studies shifted towards the need to reduce emissions of greenhouse gases and the construction of super-maneuverable fighter planes and hypersonic vehicles [2]. Nowadays, the numerical simulations of complex flows are possible due to the availability of high speed large capacity computers. Many studies attempt to manipulate coherent structures in transitional and turbulent shear flows; other studies seek the development of micro-electromechanical system (MEMS) that can be applied for flow control diagnosis, cooling of electronic components, medical applications etc.

Flow controls can be active or passive depending on the energy expenditure and the control loop involved. Passive control methods modify the flow without any auxiliary power and without a control loop. Some of these techniques include the use of fixed mechanical vortex generators for separation control; geometric shaping to manipulate the pressure gradient; and the placement of riblets on the surface for drag reduction. On the
other hand, active control methods involve energy and auxiliary power into the flow. The summary of flow control techniques is given in the Table 1 below.

Among the active flow control devices, actuators have received a great deal of attention during the last decade. There are many types of actuators used in active flow control; some of the most popular include fluidic, thermal, acoustic, piezoelectric, electrodynamic, electromagnetic and shape-memory alloy actuators. Out of these, the synthetic jet actuators (also known as zero-net-mass-flux electro-dynamic actuators) will be the primary focus of this study. Their principles of operation, development and applications are presented in Chapter 2.

**Dissertation Outline**

The physical understanding of the behavior of synthetic jet actuators on a flat plate for active flow control is the main topic of this study. It is accomplished by a computational approach. The goal of the study is to understand the effect of frequencies and angles of attack. The studies are performed on a flat plate to evaluate the effectiveness of the synthetic jet actuator as a flow control device in altering the properties of the boundary layer. This dissertation is organized into six chapters. An introduction to flow separation and control is given in the first chapter. Chapter 2 provides a detailed description of the synthetic jet actuators, their operation and applications and a literature review. The governing equations and computational methods used are discussed in chapter 3. The next chapter is about setting up the problem in the CFD tool which is followed by the
results and discussions in Chapter 5. Conclusions based on the research performed in this work are presented in Chapter 6. The final chapter also includes the ideas for future work.

Table 1: Flow separation control techniques

<table>
<thead>
<tr>
<th>FLOW SEPARATION CONTROL</th>
<th>Modification of velocity profile in the boundary layer</th>
<th>Steady suction</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Moving boundaries</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Tangential steady blowing</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Oscillatory blowing and suction</td>
<td></td>
</tr>
<tr>
<td>Reduction of steepness of adverse pressure gradient</td>
<td>Surface streamlining</td>
<td></td>
</tr>
<tr>
<td>Control of fluid’s viscosity near the wall</td>
<td>Heat transfer to/from the fluid</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Injection of secondary fluid with higher/lower viscosity</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Cavitation</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Chemical reaction</td>
<td></td>
</tr>
<tr>
<td>Enhancement of mixing in shear layer</td>
<td>Vortex generators, turbulators, etc.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Normal steady blowing</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Pulsed jets</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Oscillatory blowing and suction</td>
<td></td>
</tr>
<tr>
<td>Additional (active) control methods</td>
<td>Acoustic excitations</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Oscillating flap or wire</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Oscillatory surface heating</td>
<td></td>
</tr>
</tbody>
</table>
CHAPTER 2

SYNTHETIC JET ACTUATORS

Active flow control using synthetic jets has received people’s attention in recent years. It deals with suction and blowing into the boundary layer. The addition of energy into the flow allows the “new” boundary layer to overcome the adverse pressure gradient and therefore delay separation. The drag coefficient can be significantly decreased by shifting the transition point in the boundary layer in the downstream direction by using suction. On the other hand, additional energy is supplied to the fluid particles in the boundary layer by blowing which enhances the mixing of blowing fluid and oncoming flow within the boundary layer. The use of oscillatory blowing and suction has been found to be more effective than just steady blowing or steady suction alone.

The first description of a device similar to “synthetic jet actuator” was given by K.U. Ingard in 1953 [3]; however, in the recent years there have been significant advances in the development of “synthetic jet” or “zero-net-mass-flux” actuators which are widely used for a variety of flow control applications. These actuators require low energy for operation although they are small in size, low weight and low cost. They can be easily integrated into the surface of the object as needed e.g. into an airplane wing.

Principles of Operation

The schematic of a synthetic jet actuator is shown in Figure 2. A typical actuator consists of a cavity open at the top by a small slit through which fluid is free to flow. The flow
inside the cavity is driven by a moving surface, either an oscillating piston or a vibrating diaphragm. The oscillation of the diaphragm produces a fluctuation of the pressure field in the cavity and at the exit slit, causing it to periodically act as a source and a sink. This behavior results in a jet originating from the slit. A non-zero momentum is imparted to the external flow even though there is no net mass injected during a cycle. The slit is the only communication between the cavity of the actuator and the external flow. Ambient fluid from the external flow enters the cavity and exits the cavity in a periodic manner. Upward motion of the diaphragm generates flow which separates at the sharp edges of the slit and rolls into a pair of vortices generated at the two edges of the slit. These vortices then move away from the slit at their own induced velocity [4].

![Figure 2: Schematic of a synthetic jet actuator (not to scale)](image)

**Applications of synthetic jets**

Some of the applications of synthetic jets include improving heat transfer, enhancing mixing, and jet vectoring and controlling a turbulent boundary layer for drag reduction.

A. Separation control over Airfoils

The main aim of flow control is to increase lift and decrease drag. This is usually achieved by controlling the boundary layer flow in order to minimize separation. The
purpose of this research is to improve aerodynamic performance. The ability to increase lift at higher angles of attack by preventing separation and stall has important application in the aerospace industry.

B. Thrust Vectoring

Thrust vectoring is the capability to change the direction of the thrust of an aircraft engine in a desired direction thereby increasing the maneuverability of the aircraft without depending completely on the conventional control surfaces. Generally thrust vectoring is achieved by varying the nozzle geometry; however, the mechanical complexity of such a variable geometry nozzle results in as much as 30% of the weight of the engines. One of the most important advantages of thrust vectoring using Active Flow Control (AFC) is the elimination of the complex movable surfaces that significantly add weight. Fluidic control (using synthetic jets) of exhaust jet form the engine allows for a change in the thrust vector for a fixed geometry nozzle. Some other advantages are that the frequency, amplitude and the phase of the excitation can be controlled; they operate in harsh thermal environments; they are not susceptible to electromagnetic interference, they have no moving parts and are easy to integrate into a working device.

C. Forebody Vortex Control (FVC)

The system of vortices that forms and separates from the forebody of an aircraft or a missile at higher angles of attack affects the aerodynamic loads and moments acting on the vehicle. These vortex configurations depending on the angle of attack may be symmetric or asymmetric. The asymmetry may produce strong yawing moments that cause stability and control problems. Therefore, a method to control the strength and
configuration of the separating vortices is of high importance. Forebody Vortex Control is a technique to manage the loads and moments acting on the vehicle by introducing controlled perturbations near the forebody nose, where the vortices originate. By flow control, the asymmetric state of forebody vortices can be made symmetric, thus the side forces of the vehicle can be eliminated.

D. Control of Flow – Induced Cavity Oscillations

Understanding the flow over open cavities is of great importance for a wide range of engineering applications including aircraft landing gears, car sunroofs, etc. Self-sustained oscillations inside the cavity generate intense pressure fluctuations that can lead to structural damage or failure of critical components; thus suppression of these oscillations becomes an important flow control problem. In compressible flow, cavity oscillations arise from a flow-acoustic resonance mechanism involving a feedback process impinging near the downstream corner of the cavity. This generates acoustic waves that propagate back upstream and interact with the shear layer to excite further instabilities. It has been shown that it is possible to suppress these cavity oscillations and flow induced cavity resonance by employing synthetic jets upstream of the cavity leading edge.

Literature Review

Some of the earlier active control methods employed the acoustic excitation or a steady blowing/suction to alter the attached or separated turbulent boundary layer flow on aerodynamic surfaces. In 1987, Zaman et al. [5] studied the effect of acoustic excitation on flow separation over airfoils over a large angle-of-attack range. Significant
improvement in lift was obtained post stall due to large amplitude acoustic excitations. Most effective results were produced for frequencies that resulted in large transverse velocity fluctuations rather than large-amplitude pressure fluctuations. Experiments by Chang et al. [6] demonstrated the influence of frequency on separation control at post stall angles of attack. Their results showed that the flow separation was reduced at angles of attack lower than the stall angle by using small amplitude excitation frequency close to the shear layer instability frequency. Maximum lift increment (about 50% in lift coefficient) was found at an angle of attack 22°. Their data also showed that the effective forcing frequency reduced separation over a wider range of angles of attack.

The study by Seifert et al. [7] applied a combination of steady and oscillatory blowing to the surface of a NACA 0015 airfoil in the tangential direction. They proved that larger increments in lift could be obtained by using a less powerful excitation device if an oscillatory jet was used instead of a steady jet. A combination of oscillatory blowing with a small amount of steady blowing proved to be the most efficient method for active flow control of separation. As an extension of their previous work, Seifert et al. [8] examined the several parameters including the location of the blowing slot, the steady and the oscillatory momentum coefficients of the jet, the frequency of the imposed oscillations, and the shape of the airfoil on reducing separation. They concluded that the most effective location for the excitation was nearest to the separation location. Since the earlier experimental work of Seifert, Glezer and Wygnanski among others, the use of zero-net-mass-flux (ZNMF) has become very prominent for active flow control during the past decade. These actuators produce oscillatory jets which impart zero net mass into the flow field during blowing and suction cycles but impart momentum to the flow.
During the past two decades, a number of experiments have been performed to evaluate the potential of active flow control actuators. James et al. [9] investigated the evolution of a synthetic round turbulent jet formed by a submerged oscillating diaphragm that is flush mounted in a flat plate. An isolated synthetic jet is produced by the interactions of a train of vortices that are typically formed by altering momentary ejection and suction of fluid across an orifice such that the net mass flux is zero. The time-averaged structure of the synthetic jet was found similar to convectional round turbulent jet. Smith et al. [10] also tested synthetic jet actuators at higher frequencies on a 24% thick airfoil in which flow reattachment was achieved at angles of attack up to 18°. Their results suggested that there is a threshold jet momentum below which the excitation had negligible effect on the flow-field and this threshold jet momentum decreased as the excitation location approached the separation point.

In 2003, Lee et al. [11] investigated the effects of piezoelectric synthetic jet actuator on an adverse pressure gradient flow. Hot-wire anemometer was used to measure mean value of the boundary layer velocity profile. Their results showed that the actuators must have sufficient velocity output to produce strong enough vortices for effective flow control. They observed that the excitation frequency played a major role in flow control rather than the amplitude of oscillation.

Experimental investigations by Smith et al. [12,13] on thrust vectoring using synthetic jets showed the static pressure near the primary jet flow can be altered by the synthetic jets adjacent to the primary jet fluid which resulted in the deflection of the primary jet towards the synthetic jet thus resulting in vectoring of the primary jet. In 2004, in a workshop held by NASA Langley Research Center, computational methods were
compared against the experimental data for three different cases that involved synthetic jets.

A number of numerical simulations of synthetic jet flow-field have also been reported in the literature since the late 1990s. In 1997, two-dimensional incompressible calculations for both laminar and turbulent synthetic jets were reported by Kral et al. [2]. The harmonic motion of the actuator was simulated with blowing and suction boundary condition at the orifice exit (the flow within the cavity was not taken into account). The turbulent solutions were computed using the unsteady Reynolds-averaged Navier-Stokes (URANS) equations. URANS simulations were conducted on a NACA 0015 airfoil with a oscillatory jet located at the leading edge in the tangential direction to the surface by Donovan et al. [14]. The lift coefficients were significantly increased due to the actuator-like effect. Although regions of separated flow existed the results were in good agreement with the experimental result of Seifert et al. [8]. Compressible URANS equations were used by Wu et al. [15] to perform numerical computations on post-stall flow over a NACA 0012 airfoil. Effect of periodic blowing and suction near the leading edge was showed in his study though the mesh was not able to capture the details of the jet. An increase in lift was obtained when the periodic excitation was activated.

Majority of the simulations reported in the literature so far have been performed for two dimensional configurations. Rizzetta et al. [16] investigated the flow-field of both two- and three- dimensional high-aspect ratio synthetic jets using direct numerical simulations (DNS) of unsteady compressible Navier-Stokes equations. In 2001, Mittal et al. [17] conducted a numerical simulation that included an accurate model of the jet cavity. Their boundary conditions in the two-dimensional Cartesian grid included moving boundaries.
The investigations included synthetic jets in both quiescent and boundary layer flows. In the quiescent medium the formation of the vortex rings were observed at the orifice exit. The operational and geometrical parameters of the jet were the key factors that indicated if the vortices were expelled or ingested back into the cavity. The two-dimensional DNS simulations conducted by Lee et al. [18] studied the behavior of an array of synthetic jets pulsing into an initially quiescent medium. It was observed that the jet formation was highly sensitive to the Reynolds Number.

Two-dimensional incompressible URANS computations were performed by Guo et al. [19] to simulate the effect on vectoring angle of a single synthetic jet of various frequencies, amplitudes and angles located at different distances from the primary jet. These results compared well with the experimental results of Smith et al. [13] when the effect of actuator cavity was considered.

Interaction of the synthetic jet with the cross-flow is a key area of investigation in active flow control. Cui et al. [20] performed two-dimensional simulations using the incompressible Reynolds-averaged Navier Stokes equations. Flow interaction between the synthetic jet and the external flow for various amplitudes, frequencies and phase differences was investigated for cases with and without cavity. In 2004, Ravi et al. [21] employed DNS to study the effect of slot aspect ratio on the formation of three-dimensional synthetic jets in quiescent and external boundary layer flow. More recently three-dimensional simulations were reported by Kotapati et al. [22] in 2005 and 2006 for a test run at NASA Langley Research Center where the actual actuator cavity in the experiment was approximated as an equivalent rectangular cavity. Their results showed
that the URANS calculations are capable of predicting the overall features of the oscillatory synthetic jet flow-field.
CHAPTER 3
FUNDAMENTAL EQUATIONS AND COMPUTATIONAL METHODS

Fundamental Equations

Computational Fluid Dynamics method comprises of solution of Navier Stokes equations at required points to get the properties of the fluid flow at those points. This technique exists since the advancement in complex mathematical algorithms in 1930. Simple CFD problems were solved analytically, but with the increase in fluid flow complexity, mathematical complexity increases exponentially. With 3D interactive capability and powerful graphics, use of CFD has gone beyond research and into industry as a design tool.

The governing equations for computational fluid dynamics (CFD) are based on conservation of mass, momentum and energy. Fluent uses a finite volume method (FVM) to solve the governing equations. The FVM involves discretization and integration of the governing equation over the control volume.

The basic equations for unsteady-state incompressible laminar flow are conservation of mass and momentum. When heat transfer or compressibility is involved the energy equation is also required. The governing equations are:

**Conservation of Mass**

For a chemically non-reacting fluid, the law of conservation of mass states that “the rate of change in mass inside the control volume must be equal to the decrease of mass out of
the control surface”. Thus, the differential form of the continuity equation can be written as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

**Conservation of Momentum**

The principle of conservation of momentum is basically an application of Newton’s second law to an element of fluid. Therefore, when considering a given mass of fluid in a lagrangian frame of reference, conservation of momentum states “the rate at which the momentum of the fluid mass in a control volume changes is equal to the net external force acting on the mass”. The external forces which act on a mass of the fluid may be classified as body forces (i.e. gravitational or electromagnetic forces) or surface forces (i.e. pressure and viscous stresses). The equation in the differential form can be written as follows:

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + u \frac{\partial}{\partial x} + v \frac{\partial}{\partial y}$$

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2}$$

**X - Momentum**

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \mu \nabla^2 u$$
Written out in full:

$$
\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = -\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)
$$

Y - Momentum

$$
\rho \frac{Dv}{Dt} = - \frac{\partial p}{\partial y} + \mu \nabla^2 v
$$

Written out in full:

$$
\rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) = -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)
$$

CFD: Overview

The governing equations of fluid flow have been known for over a century. The Navier-Stokes equation is a highly non-linear equation whose solution for problems of practical interest has been possible only after the advent of high-speed large memory computers only a couple of decades ago. Until 1980s, experimental fluid dynamics was the only real way of understanding and quantifying the fluid behavior in complex configurations, sometimes aided by simple analytical/computational models. These experiments are generally performed on small scale models of the full configurations. In general it is not feasible to perform experiments on full-scale configurations such as aircrafts, automobiles, power plants, etc. Furthermore, experimental measurements are quite expensive and require considerable time to complete and therefore can be performed on a limited number of models and a limited number of flow configurations. Since 1970s there
have been extraordinary advances in the development of both the numerical algorithms for the solution of the Navier-Stokes equations as well as in computing hardware that it is now feasible to compute the flow-fields of complete configurations such as an aircraft or an automobile. A large number of codes such as FLUENT, ANSYS, STARCCM+, CFX, CFD++ etc. have been developed for a variety of industrial applications. Turbulence modelling still needs to be investigated as it introduces errors in simulations. However, substantial progress has been made in the last four decades in the development of turbulence models to calculate a wide variety of complex flow-fields reasonably accurately. Computational Fluid Dynamics (CFD) is now considered as an important branch of fluid dynamics complementing the experimental fluid dynamics.

**Definition, Benefits and Applications of CFD**

A standard definition of CFD is given in reference which states that it is the science of determining a numerical solution to the governing equations of fluid flow while advancing the solution through space or time to obtain a numerical description of the complete flow-field of interest”. CFD can be applied to solve problems in fluid flow, heat transfer, mass transfer, acoustics, chemical reactions and related phenomena.

**CFD Process**

A brief outline of the process for performing a CFD analysis is given in this section. Several steps are required for modelling fluid flow using CFD. Essentially, there are three main stages in every simulation process: preprocessing, flow simulation and post-processing. Preprocessing is the first step in building and preparing a CFD model for the flow simulation step. This included the problem specification and construction of the
computer model via computer-aided design software. Once the computer model is constructed, a suitable grid or mesh is created in the computational domain. Then the flow conditions including the material properties of the fluid such as density, viscosity, thermal conductivity, etc. are specified.

The computational domain should be chosen in such a way that an acceptably accurate answer is obtained without excessive computations being required. The nature of the flow-field and the geometry generally provides a guide for a suitable mesh construction as to its structure (structured, unstructured, zonal or hybrid) and topology (c-, o-, h- or hybrid). There is also an option to adapt the mesh according to the solution. It can automatically cluster in regions of higher flow gradients by sensing the solution as it evolves. The good mesh should display qualities such as orthogonality, lack of skewness and gradual spacing to obtain accurate solutions.

All the above mentioned steps constitute the preprocessing prior to the second step of flow simulation, in which the boundary conditions and initial conditions of the problem must be specified. For the second step, the mesh coordinates the boundary conditions in the computational domain and the material properties of the fluids are imported into a flow solver such as FLUENT. The flow solver then executes the solution process for solving the governing equations of the fluid flow. The governing equations are solved employing a suitable numerical algorithm which is coded in the flow solver. As the solution process proceeds, the solution is monitored for convergence implying that there is little change in the solution from one solution step to the next. Once the solution within a specified error tolerance is obtained, the post-processing step begins.
This step involves the display of converged flow variables in graphical and animated forms. The post-processing can be conducted using various software like FLUENT, TECPLOT, etc. Computed flow properties are then compared with the experimental data (if available) or with the computations of other investigators to validate the solutions. The simulation is complete at this point although, a sensitivity analysis of the results is recommended to understand the possible differences in the accuracy of the results with variations in the mesh size and the other parameters used in the algorithms.

**FLUENT / POINTWISE Description**

FLUENT is the world's largest provider of commercial CFD software and services. FLUENT software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants. Special models that give the software the ability to model in-cylinder combustion, aeroacoustics, turbomachinery and multiphase systems have served to broaden its reach. FLUENT can be used to solve complex flows ranging from incompressible (subsonic) to highly compressible (supersonic or hypersonic) including the transonic regime. FLUENT provides mesh flexibility, including two-dimensional triangular/quadrilateral meshes and three-dimensional tetrahedral, hexahedral or hybrid meshes. The grid can also be refined or coarsened based on the flow solution using the grid adaptation capability. Furthermore, it provides multiple solver options, which can be modified to improve both the rate of convergence of the simulation and the accuracy of the computed result. The software code is based on the finite volume method and has a
wide range of physical models which allow the user to accurately predict laminar and turbulent flows, chemical reactions, heat transfer, multiphase flows and other related phenomena.

Any geometry can be created in CAD software like CATIA or Pro-Engineer and imported in mesh generating software like POINTWISE, GRIDGEN, GAMBIT etc. For this research since the geometry is not complicated it is created and meshed in POINTWISE.

POINTWISE is flexible, robust and reliable software for mesh generation. POINTWISE provides high mesh quality to obtain converged and accurate CFD solutions especially for viscous flows over complex geometry.

Mesh generation, also known as grid is the process of forming nodes across the geometry. Nodes are the points at which the Navier Stokes equations will be solved for the fluid properties. When these nodes are connected, a mesh is formed and the domain or the control volume is called discretized.

There are several aspects to be considered for mesh generation. First is the type of the mesh. There are two main type of mesh: Structured and Unstructured. A structured mesh has all the nodes arranged such that the cells formed by joining adjacent nodes are rectangular in shape. This helps in easy reference of each cell making it numerically simple to deal with. An Unstructured mesh (Figure) has nodes distributed randomly, hence the mesh cells can be tetrahedral, octahedral and pyramid in shape for 3D mesh and triangular in shape for 2D mesh. This random arrangement of nodes require a mapping file to keep the track of the nodes, increasing the file size of unstructured mesh
compared to structured mesh. Unstructured mesh is useful for meshing complex and curved geometries. Since we are dealing with a 2D flat plate structured mesh is used. Once the model is meshed, the boundary conditions are specified in POINTWISE.

Details about meshing the geometry using POINTWISE is mentioned in Appendix A.
CHAPTER 4

NUMERICAL SIMULATION ON A FLAT PLATE:

(Methodology and Solution approach)

Overview

Numerical simulations are a valuable tool, especially when experiments become complex, expensive and time consuming. The validated computational code can be employed to conduct a large number of parametric studies for similar configurations quickly and inexpensively for extensive analysis. Thus, CFD simulations constitute a crucial part of this study. This chapter describes the methodology and solution approach used in the numerical simulations. A description of the grid generation procedure is given first, followed by specifying the boundary conditions. Finally, a detailed step by step set-up description of a solution using FLUENT is presented.

Grid Generation

The first major step when conducting a CFD analysis is the construction of the geometry and a suitable mesh. The geometry was both created and meshed using POINTWISE since the geometry (flat plate) was relatively simple. The synthetic jet was modeled as an oscillating diaphragm (providing suction and blowing). The first step in setting up the mesh is the creation of a control area followed by the creation of nodes (points where the grid lines of the mesh connect) on the edges of the geometry. This process was accomplished by specifying a gradually increasing or decreasing spacing, which provided a non-uniform mesh with finer resolution in certain areas of the computational domain for e.g. near the oscillating diaphragm. Once the nodes were created, actual mesh was generated. A variety of options for mesh generation are available for mesh generation are
available in POINTWISE, including both structured and unstructured elements. Synthetic jet actuation is initially examined in a quiescent flow which helps verifying the assumed jet model. Therefore a simple symmetric mesh is generated in POINTWISE. After several trials, a structured mesh with 242,756 quadrilateral cells was created. The length of the oscillating diaphragm was 5 cm.

\begin{table}[h]
\centering
\caption{Nodes and Cells in the mesh}
\begin{tabular}{|c|c|}
\hline
Triangle & 0 \\
Quadrilateral & 242,756 \\
Total Cells & 242,756 \\
Total Points & 243,876 \\
\hline
\end{tabular}
\end{table}

\begin{table}[h]
\centering
\caption{Spacing and number of points on the connector for simple grid}
\begin{tabular}{|c|c|c|}
\hline
Boundary & No. of points & Spacing \\
\hline
Oscillating diaphragm & 200 & 0.25 \\
Jet exit & 800 & 0.25 to 15.61 \\
Pressure Inlet & 300 & 0.21 to 17.42 \\
\hline
\end{tabular}
\end{table}
The flat plate of length 1m and thickness 5 mm was considered for simulations. A structured mesh was generated in POINTWISE with 167,351 quadrilateral cells. The oscillating diaphragm was 5 cm in length.

**Table 4: Nodes and Cells in the mesh including the flat plate**

<table>
<thead>
<tr>
<th>Type</th>
<th>Count</th>
</tr>
</thead>
<tbody>
<tr>
<td>Triangle</td>
<td>0</td>
</tr>
<tr>
<td>Quadrilateral</td>
<td>167,351</td>
</tr>
<tr>
<td>Total Cells</td>
<td>167,351</td>
</tr>
<tr>
<td>Total Points</td>
<td>168,800</td>
</tr>
</tbody>
</table>
The table 5 below shows the number of points and spacing considered on each boundary. Spacing and the count of nodes on each connector is an important parameter in capturing the flow field accurately.

*Table 5: Spacing and the number of points on the connector*

<table>
<thead>
<tr>
<th>Boundary</th>
<th>No. of points</th>
<th>Spacing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Oscillating diaphragm</td>
<td>80</td>
<td>0.63</td>
</tr>
<tr>
<td>Jet Inflow</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Streamwise Direction</td>
<td>490</td>
<td>Bottom-middle-top:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>66.24-0.01-36.85</td>
</tr>
<tr>
<td>Cross-stream</td>
<td>280</td>
<td></td>
</tr>
<tr>
<td>direction</td>
<td></td>
<td>Left-diaphragm-right</td>
</tr>
<tr>
<td></td>
<td></td>
<td>7.25-6.71-8.47</td>
</tr>
<tr>
<td>Jet Exit</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Streamwise Direction</td>
<td>490</td>
<td>Bottom-middle-top:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>66.24-0.01-36.85</td>
</tr>
<tr>
<td>Cross-stream</td>
<td>370</td>
<td></td>
</tr>
<tr>
<td>direction</td>
<td></td>
<td>Left-diaphragm-right</td>
</tr>
<tr>
<td></td>
<td></td>
<td>7.25-0.63-8.47</td>
</tr>
<tr>
<td>Flat plate</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Upper surface</td>
<td>240</td>
<td>Left-diaphragm-right</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4.23-0.63-7.07</td>
</tr>
<tr>
<td>Lower surface</td>
<td>150</td>
<td>6.71</td>
</tr>
</tbody>
</table>
Figure 4: Two-dimensional grid distribution and boundary conditions used for synthetic jet flat plate simulations
Nomenclature

\( f: \) Synthetic jet frequency

\( x: \) Streamwise direction tangential to the surface

\( y: \) Cross-Stream direction normal to the surface

\( u: \) Streamwise velocity

\( v: \) Cross-Stream velocity

\( V_j: \) Amplitude of the synthetic jet

\( U_{\infty}: \) Free stream velocity

Solution Methodology

*Table 6: Physical properties of the numerical simulation*

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>101325 Pa (1 atm)</td>
</tr>
<tr>
<td>Temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Speed of sound</td>
<td>340 m/s</td>
</tr>
<tr>
<td>Free stream velocity</td>
<td>34 m/s</td>
</tr>
<tr>
<td>Kinematic viscosity</td>
<td>1.79(10^{-5}) kg/ms</td>
</tr>
<tr>
<td>Density</td>
<td>1.23 kg/m^3</td>
</tr>
<tr>
<td>MACH</td>
<td>0.1</td>
</tr>
<tr>
<td>Chord</td>
<td>1 m</td>
</tr>
</tbody>
</table>
Reynolds Number: \( Re = \frac{\rho \cdot v_c}{\mu} = 2 \times 10^6 \)

The streamwise velocity \((u)\) and the cross-stream velocity \((v)\) were computed for different angles of attacks.

Table 7: Streamwise and Cross-stream velocities for different angles of attack

<table>
<thead>
<tr>
<th>Angle-of-attack</th>
<th>( u \ (m/s) )</th>
<th>( v \ (m/s) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>34</td>
<td>-</td>
</tr>
<tr>
<td>2°</td>
<td>33.979</td>
<td>1.187</td>
</tr>
<tr>
<td>3°</td>
<td>33.953</td>
<td>1.779</td>
</tr>
<tr>
<td>4°</td>
<td>33.917</td>
<td>2.093</td>
</tr>
<tr>
<td>5°</td>
<td>33.870</td>
<td>2.963</td>
</tr>
<tr>
<td>7°</td>
<td>33.747</td>
<td>4.144</td>
</tr>
<tr>
<td>9°</td>
<td>33.581</td>
<td>5.319</td>
</tr>
<tr>
<td>10°</td>
<td>33.483</td>
<td>5.904</td>
</tr>
<tr>
<td>15°</td>
<td>32.841</td>
<td>8.799</td>
</tr>
<tr>
<td>18°</td>
<td>32.33</td>
<td>10.507</td>
</tr>
<tr>
<td>20°</td>
<td>31.950</td>
<td>11.629</td>
</tr>
</tbody>
</table>

Boundary Conditions

The correct specification of boundary conditions on the boundaries of the computational domain is very critical in CFD modeling for obtaining the accurate solutions. Defining these boundary condition involves boundary location identification (i.e. inlets, outlets,
walls, axis, etc.) and flow variables specification. These boundary condition locations for computational domain are shown in Figure 4. No-slip condition is applied on all the walls of the flat plate except the diaphragm. In two dimensions, the no-slip condition can be written as:

\[ u(x, y) = 0 \]

and

\[ v(x, y) = 0 \]

where \( x \) and \( y \) describe the coordinates of a point on the fixed boundary at which the velocity is zero. No-slip condition requires a finer grid near the walls as shown in Figure 4 because of resolution required for boundary layer near the walls.

Other boundary conditions are velocity inlet, pressure outlet, wall and symmetry as shown in figure. The symmetry condition indicates that the geometry is identical with respect to a specified boundary. This condition allows the computations to be performed only for half of the actual model in 2D, resulting in saving a considerable amount of computational time. In Figure 3 left half of the model about the line of symmetry is shown. Outflow conditions were specified at the jet exit (pressure outlet), where a constant exit pressure of one atmospheric pressure was imposed. The inflow conditions correspond to the oscillating diaphragm, wherein a sinusoidal velocity inlet boundary condition, normal to the boundary was specified. This velocity inlet boundary condition was imposed through a User Defined Function (UDF), which is a function written by the user in C (programming language) and hooked into the FLUENT solver. The velocity inlet condition is time periodic simulating the sinusoidal motion of the oscillating
diaphragm. Although FLUENT includes a moving wall boundary condition feature, previous studies by Tang et al. [23] have shown that the use of a velocity inlet condition instead of a moving boundary condition has significant effect on the jet exit velocity and resulted in a substantial saving of the computational time. In two dimensions, the velocity inlet condition at the oscillating diaphragm can be written as:

\[ u(x, y = const, t) = 0 \]

\[ v(x, y = const, t) = V_j \sin(\omega t) \]

where \( V_j \) is the velocity amplitude of the diaphragm, which was determined such that it was 10% of the freestream velocity. In equation \( \omega \) related to the frequency \( f \) by the relation,

\[ \omega = 2\pi f \]

**Problem definition in FLUENT**

Once the grid file was imported into FLUENT, the first step in the solution process involved checking the mesh for errors. The quality of the grid plays a key role in the accuracy of the result. Smoothness, skewness and node point distribution are the attributes associated with the mesh quality. The accuracy of solution of any problem depends on the density and distribution of the nodes in the mesh. Large truncation errors may be resulted when there are abrupt changes in the volume between adjacent cells; therefore the smoothness of the mesh is very important. The cell shapes also have a significant impact on the accuracy of the result. Skewness is defines as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume.
Highly skewed cells decrease the accuracy and also destabilize the solution. Another important parameter that needs to be considered when checking the grid is the cell volume. The cell volume must be positive; a negative volume is an indication of improper cell connectivity. Once the grid is verified for its quality, certain modifications to the geometry can be made (i.e. scaling, translation, rotation, etc.) before proceeding to the problem definition for initiating the numerical solution process in FLUENT.

The computation of a solution in FLUENT requires specification of a number of parameters associated with the dimensionality of domain, properties of the fluid, choice of numerical solution method, the turbulence model, the convergence criteria, whether the flow is steady or unsteady, etc. The linearization process may take an implicit or explicit form with respect to the system of dependent variables. In the implicit form, the unknown value in each cell is computed using a relation that includes both the existing and unknown values from the neighboring cells. Therefore, each unknown appears in more than one equation of the system; these equations are therefore solved simultaneously. In the explicit form the unknown value in each cell is computed using a relation that includes only the known values, therefore the equation can be solved in a non-coupled manner one at a time. The dimensionality of the problem needs to be specified as well; in this case a two-dimensional solution was employed. Due to the unsteady nature of the flow-field, a time-dependent, first-order accurate implicit time stepping scheme was employed for the two-dimensional calculations. Under the problem definition, another specification requires the selection of the flow as inviscid, laminar or turbulent. Several turbulence models are available for example: the Reynolds stress model (RSM), k-ω and k-ε. In this study we are dealing only with laminar flow-field.
**Definition of Fluid Properties and Equation of State**

Another input in the problem initialization is the definition of fluid properties. In our calculations, air was selected as the working fluid from the FLUENT’s database. The material of the flat plate was considered to be Aluminum. For all calculations, an incompressible solution method was selected.

**Definition of Operating Conditions**

Operating conditions include pressure and gravity. Atmospheric pressure was set as the operating pressure for all test configurations. Gravitational acceleration can also be specified for any problem. In this case the gravitational effects have been excluded.

**Definition of Boundary Conditions**

A summary of the boundary conditions for the test configuration for synthetic jets in quiescent medium is given in Table 8. Once the problem definition, operating conditions, fluid properties and the boundary conditions are specified, the solution process can be initiated.

**Solution Execution and Convergence**

Each test configuration has to be initialized before the code can be executed. This initialization provides an initial guess for the first iteration of the solution. During this process the user must specify which part of the computational domain will be provided with initial conditions. For the synthetic jet actuator flow-field in quiescent medium, the velocity inlet was selected since the sinusoidal motion was applied at this boundary. Once
Table 8: Boundary conditions for modeling synthetic jets in quiescent medium

<table>
<thead>
<tr>
<th>ZONE</th>
<th>TYPE</th>
<th>BOUNDARY CONDITION</th>
</tr>
</thead>
</table>
| Oscillating      | Velocity Inlet| Velocity magnitude and direction:  
| membrane         |               | \( v(t) = V_j \sin(\omega t) \)  
|                  |               | - Initial velocity is specified by UDF  
|                  |               | - \( x \)-component: 0  
|                  |               | - \( y \)-component: 1 (unit vector direction)  |
| Flat Plate       | Wall          | No-slip condition                                                                  |
| Inflow           | Pressure Inlet| Inlet Pressure: 101325 Pa (atmospheric)                                           |
| Jet Exit         | Pressure Outlet| Outlet Pressure: 101325 Pa (atmospheric)                                          |
| Vertical Axis    | Symmetry      | Physical geometry and expected flow solution have mirror symmetry with respect to the vertical axis |

The flow field is initialized; the number of iterations must be specified by the user. This number was selected depending on the frequency of the motion. Each unsteady case was run for about three seconds before the converged solution was obtained. After three seconds, the solution became periodic and started to repeat itself in the next cycle. In some cases more than three seconds were required for the solution to converge.

Several flow properties were monitored and checked for convergence. The employed convergence criterion required the scaled residuals to decrease to \( 10^{-6} \) for all the
governing equations. Also the under relaxation parameters were varied to achieve convergence. Table shows a list of variables and their respective convergence criterion.

Table 9: Monitored equations and convergence criteria

<table>
<thead>
<tr>
<th>Equation</th>
<th>Convergence Criteria</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>Residual &lt; 10^-6</td>
</tr>
<tr>
<td>x-momentum</td>
<td>Residual &lt; 10^-6</td>
</tr>
<tr>
<td>y-momentum</td>
<td>Residual &lt; 10^-6</td>
</tr>
</tbody>
</table>

The user-defined function was written in ‘C’ programming language. A detailed description of hooking the UDF into FLUENT is explained in Appendix B and C.

The software utilized for the post-processing of the results were ANSYS FLUENT and TECPLOT. This software was used to generate the scalar and vector fields, animations, plots of grids and x-y graphs.
CHAPTER 5
RESULTS AND DISCUSSIONS

Flow over a flat plate

Although the aerodynamic characteristics of airfoil sections are dependent on the shape of the airfoil and the Reynolds number, the angle-of-attack plays a key role when computing the lift. A very thin flat plate (5 mm) with a chord length of 1m was considered for the numerical simulations. Because of the sharp leading edge, flow separates from the upper surface at the leading edge at an angle of attack as low as 3° -5° and reattaches further downstream on the surface leaving a separation bubble. As the angle of attack increases, the reattachment point moves aft and the bubble grows. The separation bubble covers almost the complete chord and $C_{L\text{max}}$ is reached. This type of stall is called the thin airfoil stall or long bubble stall. Near the stall, the relationship between $C_L$ and $\alpha$ of the airfoil sections and their stall characteristics are dependent on the thickness chord ratio, the shape of upper surface near the leading edge, and the Reynolds Number.

The streamlines and the pressure contours for various angles of attack at Mach 0.1 ($U_\infty = 34$ m/s) is shown below. Also some validation studies have been performed. The validation is achieved by comparing the results obtained with experimental data and numerical data obtained from other authors. The quantities that have been compared are the lift coefficient, drag coefficient and the streamlines. The lift and the drag coefficients are computed using the equations. [24]
Numerical results obtained are compared with NACA 0006 wing section (experimental data) obtained from “Theory of Wing Sections” by Ira H. Abbott and Albert E. Von Doenhoff [25]. Also the results have been compared with the numerical results obtained by Celerino Resendiz Rosas [26] on NACA 0012 airfoil.

The results from the numerical simulation showed that the $\text{CL}_{\text{max}}$ is reached for an angle of attack of $11^\circ$. At an angle of attack of $11^\circ$ the separation bubble covers almost the complete chord. The streamlines below clearly shows the growth of the separation bubble.
Figure 5: The growth of the separation bubble on the surface of the flat plate for uncontrolled flow for $M_\infty = 0.1$ and $Re=2 \times 10^6$ and thin airfoil stall.

The streamlines obtained from the numerical simulation for a flat plate of 0.5% thickness/chord ratio has been compared with the flow around a flat plate of 2% thickness/chord ratio obtained from “Visualized flow” compiled by the Japanese Society of Mechanical Engineers [28] as shown in Figure 6 - 9.
Figure 6: Comparison of flow over a flat plate with (a) experimental and (b) computational results at an angle of attack $3^\circ$. 
Figure 7: Comparison of flow over a flat plate with (a) experimental and (b) computational results at an angle of attack $7^\circ$. 
Figure 8: Comparison of flow over a flat plate with (a) experimental and (b) computational results at an angle of attack $9^\circ$. 
Figure 9: Comparison of flow over a flat plate with (a) experimental and (b) computational results at an angle of attack 15°
Coefficient of lift values obtained from the computation was plotted against the angle of attack to view the pattern as shown in Figure 10. To validate the obtained results they have been compared with the experimental results of NACA 0006 and numerical results obtained by Rosas [26] for NACA 0012. Also the Coefficient of Lift ($C_L$) versus the coefficient of Drag ($C_D$) has been plotted and validated with the experimental results as shown in Figure 11.

The normal, axial, lift and drag coefficients for an aerodynamic body can be obtained by integrating the pressure and skin friction coefficients over the body surface from the leading to the trailing edge. For a two-dimensional body,

$$C_N = \frac{1}{c} \left[ \int_0^c (C_{p,l} - C_{p,u}) dx + \int_0^c \left( C_{f,u} \frac{dy_u}{dx} + C_{f,l} \frac{dy_l}{dx} \right) dx \right]$$

$$C_A = \frac{1}{c} \left[ \int_0^c \left( C_{p,u} \frac{dy_u}{dx} - C_{p,l} \frac{dy_l}{dx} \right) dx + \int_0^c (C_{f,u} + C_{f,l}) dx \right]$$

$$C_L = C_N \cos \alpha - C_A \sin \alpha$$

$$C_D = C_N \cos \alpha - C_A \sin \alpha$$

where

$C_{p,u}$ – Pressure Coefficient on upper surface

$C_{p,l}$ – Pressure Coefficient on the lower surface

$C_{f,u}$ – Friction Coefficient on the upper surface

$C_{f,l}$ – Friction Coefficient on the lower surface
Table 10: Coefficient of lift computed for different angles of attack at $Re = 2 \times 10^6$

<table>
<thead>
<tr>
<th>Angle of attack ($\alpha^\circ$)</th>
<th>Coefficient of lift ($C_L$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>0.1006</td>
</tr>
<tr>
<td>2</td>
<td>0.2381</td>
</tr>
<tr>
<td>3</td>
<td>0.3523</td>
</tr>
<tr>
<td>4</td>
<td>0.4386</td>
</tr>
<tr>
<td>5</td>
<td>0.5483</td>
</tr>
<tr>
<td>7</td>
<td>0.7521</td>
</tr>
<tr>
<td>9</td>
<td>0.9651</td>
</tr>
<tr>
<td>10</td>
<td>1.0982</td>
</tr>
<tr>
<td>11</td>
<td>1.1012</td>
</tr>
<tr>
<td>12</td>
<td>0.8960</td>
</tr>
<tr>
<td>15</td>
<td>0.7023</td>
</tr>
<tr>
<td>18</td>
<td>0.6999</td>
</tr>
</tbody>
</table>
Figure 10: Comparison of $C_L$ results for a flat plate ($M_\infty = 0.1$ and $Re = 2 \times 10^6$) with numerical simulation obtained by Rosas C.R. on NACA 0012 airfoil ($M_\infty = 0.3$ and $Re = 1 \times 10^6$) and with experimental data from “Theory of wing sections” by Ira H. A. et al. on NACA 0006 airfoil ($Re = 3 \times 10^6$).

Figure 11: Comparison of $CD$ vs. $CL$ between the present case for a flat plate ($M_\infty = 0.1$ and $Re = 2 \times 10^6$) and the experimental data obtained from “Theory of Wing Sections” by Ira H. A. et al. on NACA 0006 airfoil ($Re = 3 \times 10^6$).
The pressure contour plots for flow over a flat plate at different angles of attack are shown in Figures 12 -14. The pressure measured is the gauge pressure in Pa.

**Figure 12:** Pressure contour plots for a flow over a flat plate measured in Pascals at $Re = 2 \times 10^6$ and $M_\infty = 0.1$
Angle of Attack 4°

Angle of Attack 7°

Angle of Attack 9°

Angle of Attack 10°

Figure 13: Pressure contour plots for a flow over a flat plate measured in Pascals at Re $2 \times 10^6$ and $M_\infty = 0.1$
Figure 14: Pressure contour plots for a flow over a flat plate measured in Pascals at \( \text{Re} = 2 \times 10^6 \) and \( M_\infty = 0.1 \)
Synthetic jets in Quiescent flow

Before investigating the interaction of synthetic jets with cross flow, synthetic jet actuation is examined in a quiescent flow. It helps verifying the assumed jet model and assessing the formulation of synthetic jets. The flow is assumed to be 2-dimensional, incompressible and laminar.

Boundary conditions are carefully examined to obtain the accurate result. Initially the boundary conditions of the model are investigated in quiescent medium and then the verified boundary conditions are applied to the flow separation simulations.

The boundary layer simulations are performed for a free stream velocity: \(U_{\infty} = 34\text{m/s}\). The frequency of the jet was set to be 700 Hz. (The specific value was chosen so that the result can be compared with the already obtained value by Kihwan Kim [27])

Synthetic jets in a quiescent flow result from the interactions of a series of vortices that are created by periodically moving diaphragm. The exiting flow separates at both edges of the diaphragm and rolls into a pair of vortices during the blowing period as shown in Figure 15 (a). During the suction period, the flow in the vicinity of the slot comes into the slot and the created pair of vortices departs from the slot at a self-induced velocity as shown in Figure 15 (b).

A series of vortex pairs are symmetric with respect to the centerline of the jets. Typically the moving mechanisms of synthetic jet actuators, e.g. acoustic waves or the motion of the diaphragm or a piston, induce the pressure drop which alternates periodically across the exit slot.
Although the simulations in this research do not take into account the high-fidelity modeling for the synthetic jet actuation consisting of cavity, orifice and inner moving boundary, the result validate that the assumed velocity condition contains the essential conditions of synthetic jets.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{synthetic_jet_actuation}
\caption{Pressure contour plots for the synthetic jet actuation with $f = 700$ Hz}
\end{figure}
Figure 16: Velocity vectors colored by static pressure during blowing

Figure 16 shows the direction of the velocity vector during blowing. The separation of the flow into a pair of vortices is showed by the arrows and the length of the arrow indicates the magnitude of velocity.

Interaction of synthetic jets with Cross-flow

When a synthetic jet interacts with a cross-flow, it creates a complex flow-field structure in the interaction region, influencing the pressure, velocity and other flow variables. A continuous deformation is experienced by the jet due to the cross-flow depending on the momentum causing the streamlines to deflect. As shown in the previous section a pair of vortex is formed during the blowing stage. Due to the cross-flow, the vortex pair is deflected to the right. Due to the low velocity of the synthetic jet, the vortices generated do not penetrate into the free-stream boundary layer. The counter-clockwise vortex generated at the left is reduced in strength due the clockwise vorticity of the boundary
layer. The clockwise vortex at the right gains strength moves downstream along with cross-flow. The boundary layer is energized due to the high momentum fluid being entrained into the boundary layer from the blowing part of the synthetic jet. Further away from the jet this effect is reduced; however, once the flow continuous downstream, it creates a strong separation bubble immediately downstream of the jet in the vicinity of the wall. If the mixing of the synthetic jet and cross-flow is strong enough the low momentum flow close to the wall is energized by the high momentum flow, causing flow reattachment and thus reducing the or eliminating the separated region.

Important parameters in flow separation control are the excitation frequency, Reynolds number, the shape of the geometry and the injection point of the synthetic jet. The main idea in this research was to obtain an increase in the coefficient of lift by introducing the synthetic jet and to observe the effects of the jet frequency.

Figures 17 to 19 show the lift coefficient, $C_L$, versus the flow time, $t$. The coefficient of lift obtained for both controlled and uncontrolled case has been plotted on the same graph for different angles of attack. Figure 20 shows the mean converged $C_L$, corresponding to the controlled simulations against the corresponding angle of attack. This figure also includes the computational results obtained for the uncontrolled flow and the computational results obtained by Rosas [26]. Figure 20 clearly shows the benefits of synthetic jet actuation as an increase in the mean lift coefficient of the controlled case with respect to the uncontrolled case is observed. The result also follows a pattern close to that obtained by Rosas [26] for a symmetric airfoil. The variation in the result is possibly due to the geometry. Other probable reason for the discrepancy maybe due to the laminar model considered.
Figure 17: Effect of oscillatory flow separation control on $C_L$ on the flat plate at $\alpha = 0^\circ$ and $\alpha = 5^\circ$, $M = 0.1$, $Re = 2 \times 10^6$, $f = 800\text{Hz}$, $V_j = 3.4 \text{ m/s}$
Figure 18: Effect of oscillatory flow separation control on $C_L$ on the flat plate at $\alpha = 10^\circ$ and $\alpha = 15^\circ$, $M = 0.1$, $Re = 2 \times 10^6$, $f = 800$Hz, $V_j = 3.4$ m/s
Figure 19: Effect of oscillatory flow separation control on $C_L$ on the flat plate at $\alpha = 18^\circ$, $M = 0.1$, $Re = 2 \times 10^6$, $f = 800Hz$, $V_j = 3.4 \text{ m/s}$

Figure 20: $C_L$ versus angle of attack for a flat plate. The controlled numerical simulation has been performed on a NACA 0012 at $Re = 1 \times 10^6$
Reduction in drag is obtained due to the presence of synthetic jets as observed in Figure 21.

**Figure 21:** Comparison of $C_D$ vs. $C_L$ between the present case for a flat plate with and without synthetic jets ($M_\infty = 0.1$ and $Re = 2 \times 10^6$) and the experimental data obtained from “Theory of Wing Sections” by Ira H. A. et al. on NACA 0006 airfoil ($Re = 3 \times 10^6$)

**Effect of Frequency**

In these simulations, the frequency of the oscillating jet was set to 200, 400, 600 and 800 Hz. The results of these simulations are presented in the figure.

Figure shows that for the two employed frequencies at an angle of attack of $10^\circ$ the average lift coefficient remains unchanged ($1.1812$ and $1.1817$ for 400 and 800 Hz respectively). This may be because the tested frequencies were not large enough. Frequencies at a higher order of magnitude may produce noticeable effects.
Figure 22: Influence of the variation of frequency on $C_L$. Simulations corresponds to $M_\infty = 0.1$, $\alpha = 10^\circ$ and $Re = 2 \times 10^6$

For simulations at Mach number of 0.1 and $Re = 2 \times 10^6$ the response frequency is much lower as compared to the input frequency. As the input frequency increases a significant increase in the response frequency is observed.

Figure 23: Response frequency Versus Input frequency
CHAPTER 6
CONCLUSIONS AND FUTURE WORK

This chapter summarizes the work presented in this dissertation. Previous chapters have presented the results of the numerical investigation of a synthetic jet actuator. It allowed for the identification of critical parameters which influence the performance of the actuator. The primary idea was to increase the coefficient of lift using the oscillatory jet.

The simulation was conducted on a flat plate for a two-dimensional case. The numerical tool employed for the simulation was ANSYS FLUENT. The results are validated for steady simulations over the flat plate. And also agrees with the experimental and numerical results obtained by various authors. A significant increase in the coefficient of lift is observed.

The phenomenon of flow separation control by synthetic jets delays separation by amplifying the disturbances which convect downstream along the flat plate. There is not only a significant increase in lift but also reduction in drag. Finally, and in the view of the results obtained, synthetic jet actuator is a good device to control the flow.

This research dealt with flow for over a flat plate for a laminar flow model. The results can be further improved and developed considering the additional turbulence models and for airfoils with camber. Also the effect of Reynolds number can be studied.
REFERENCES


APPENDIX A

Creating the mesh in POINTWISE

1. Creating the geometry
   - Create > Draw Curves > line
     A new dialog box appears in which connector is selected as the entity type.

2. The number of points and the spacing is adjusted according to the need of the computation (more points required near the oscillating diaphragm to capture the vortices)

3. Creating domains
   - Grid > Set Type > Structured
   - Create > Assemble Special > domain
   - The connectors are selected to form a completed loop.
   - When all the domains are created click OK

4. Initializing the grid
   - Grid > Solve > Initialize

5. Selecting the solver type and dimension
   - CAE > Select Solver, select ANSYS FLUENT
   - CAE > Set dimension, 2-D

6. Creating Boundary Conditions
   - CAE > Set Boundary Conditions
   - Create new boundaries named Flat Plate, Diaphragm, Velocity Inlet X & Y, Pressure Outlet X & Y and select appropriate boundary in the geometry to assign the boundary condition.
7. Exporting the generated mesh

- Select the Domain.
- File > Export > CAE
- Save the file as .cas to work in FLUENT 14.5, save the file as .pw file for future modifications.
APPENDIX B

Setting the problem in FLUENT 14.5

This setup is for the flow over a flat plate with a oscillating diaphragm on the upper surface of the flat plate.

1. Start FLUENT 14.5 and import the .cas file. A dialog box appears in which select the following
   - Dimension >2D
   - Check double precision
   - Check display mesh after reading
   - Click OK
   - File > Read > Case

2. Initial solution setup steps are followed as follows:
   - The mesh is scaled from mm to m.
   - Then mesh check is performed the quality is checked. Orthogonal quality ranges from 0 to 1 where values close to 0 correspond to low quality.

3. Then the pressure based solver is selected for a transient flow.

4. The user defined function (UDF) is interpreted for the sinusoidal velocity of the diaphragm.
   - Define > Function > Interpret
   - Browse > Interpret > Close

5. Viscous laminar model is chosen for the simulation
6. The next tab allows us to choose the material being used. Air is selected for which the properties are already defined in the FLUENT database.

7. Then boundary conditions are verified (table []) and the zone which contains the oscillating diaphragm is chosen to modify the velocity specification method to components so that the UDF can be interpreted.

8. Reference values are computed from the velocity inlet boundary.

9. Standard pressure and first order upwind momentum spatial discretization is chosen as solution methods.

10. The under-Relaxation factors are set at default values.

11. Depending on the requirement of the particular case the monitors are created. The convergence criteria for the residuals were set at $10^6$ for both continuity and momentum equations.

12. Then the solution is initialized.

13. The time step size (s) and the number of time steps are chosen depending on the frequency of the periodic motion. Then the solution is calculated.

14. Once the converged solutions are obtained post processor results were analyzed in TECPLOT.

15. The pressure, vorticity and velocity contour plots were obtained and the velocity stream lines were plotted.
APPENDIX C

The User Defined function for transient velocity

/**********************************************************************
inlet_velocity_profile.c
UDF for specifying a transient velocity profile boundary condition
***********************************************************************/

#include "udf.h"

DEFINE_PROFILE(inlet_velocity, thread, position)
{
    face_t f;
    real t = CURRENT_TIME;

    begin_f_loop(f, thread)
    {
        F_PROFILE(f, thread, position) = Vj*sin(ω*t);
    }
    end_f_loop(f, thread)