ABSTRACT

GREGORY ZAR MCGOWAN. Computational Study of Adaptive Circulation Control Airfoils. (Under the direction of Dr. Ashok Gopalarathnam.)

Current projections for future aircraft concepts call for stringent requirements on high-lift and low cruise-drag. The purpose of this study is to examine the use of circulation control, through trailing edge blowing, to meet both requirements. This study was conducted in two stages: (i) validation of computational fluid dynamic procedures on a general aviation circulation control airfoil and (ii) a study of an adaptive circulation control airfoil for controlling lift coefficients in the low-drag range.

In an effort to validate computational fluid dynamics procedures for calculating flows around circulation control airfoils, the commercial flow solver FLUENT was utilized to study the flow around a general aviation circulation control airfoil. The results were compared to experimental and computational fluid dynamics results conducted at the NASA Langley Research Center. This effort was conducted in three stages: (i) a comparison of the results for free-air conditions to those from previously conducted experiments, (ii) a study of wind-tunnel wall effects, and (iii) a study of the stagnation-point behavior. In general, the trends in the results from the current work agreed well with those from experiments, some differences in magnitude were present between computations and experiments. For the cases examined, FLUENT computations showed no noticeable effect on the results due to the presence of wind-tunnel walls. The study also showed that the leading-edge stagnation point moves in a systematic manner with changes to the jet blowing coefficient and angle of attack, indicating that this location can be sensed for use in closed-loop control of such airfoil flows.

The focus of the second part of the study was to examine the use of adaptive circulation control on a natural laminar flow airfoil for controlling the lift coefficient of the low-drag range. In this effort, adaptive circulation control was achieved through blowing over a small mechanical flap that can be deflected up or down. Such a blown trailing-edge flap allows for control of the jet direction to be independent of the jet momentum coefficient. This study was performed in
two stages. In the first study, a two-dimensional thin-airfoil thin-jet theory and accompanying computer program was developed. With this method, changes to the airfoil ideal lift coefficient were studied for various jet blowing rates and angles showing that the ideal lift coefficient could be adjusted by varying either the blowing rate or the flap angle. In the second stage, a hybrid computational study was conducted. This hybrid method involved the use of the CFL3D Reynolds-averaged Navier-Stokes code in conjunction with an integral boundary layer method. The surface pressure distributions for the airfoil were determined using CFL3D. Using these pressure distributions, the boundary layer transition locations were calculated using the integral boundary layer method. The transition-location data was then used to determine the lift-coefficient range in which extended laminar flow could be achieved for cases with and without blowing. The results of this study confirmed that, in addition to flap angle, blowing across the trailing edge flap can be used to adjust the range of lift coefficients over which extensive laminar flow can be achieved. The blown trailing-edge flap was shown to be more effective at altering the location of the low-drag range than a cruise flap with no blowing. In addition, the blown flap eliminates separation off the flap at high flap angles.
Computational Study of Adaptive Circulation Control Airfoils

by

Gregory McGowan

A thesis submitted to the Graduate Faculty of North Carolina State University in partial fulfillment of the requirements for the Degree of Master of Science

Aerospace Engineering

Raleigh, NC 2004

APPROVED BY:

Dr. Ashok Gopalarathnam
Advisory Committee Chairman

Dr. Harvey Charlton
Advisory Committee Minor Rep.

Dr. William Roberts
Advisory Committee Member
BIOGRAPHY

Gregory Zar McGowan was born in Edenton, NC on June 20, 1979, to Jabie Zar McGowan Jr. and Patricia M. Davis. He was raised in the small eastern NC town of Creswell, NC. After graduating from Creswell High School in 1997 he was accepted at East Carolina University and began the Pre-Engineering curriculum offered there. After being accepted to North Carolina State University in 1998 he transferred into the Mechanical and Aerospace Engineering Department. His graduate career began after completing his Bachelor’s degree in the Summer of 2002. After completion of his Master’s degree Greg plans on pursuing a Doctoral degree in Aerospace Engineering at North Carolina State University.
ACKNOWLEDGEMENTS

I would like to start off by acknowledging Dr. Ashok Gopalarathnam for giving me the opportunity to pursue my Master’s under his guidance. Without his support and encouragement, both emotionally and intellectual, this thesis would likely not have been completed. It has been a true honor to work with him through my graduate career.

I would also like to thank Dr. William Roberts and Dr. Harvey Charlton for being on my committee.

I would like to thank Dr. Gregory Jones for his help and discussions on the topic of circulation control.

I thank NASA Langley Research Center and the National Institute of Aerospace for the funding for this research.

I am grateful to Dr. Hassan Hassan and Dr. Christopher Rumsey for their help in running CFL3D.

Thanks to Xudong Xiao, Jeff Cheng, and Craig Cox for all their help with keeping things running smoothly on my workstation. Without their help I would likely still be reading through Linux discussion boards.

I would also like to thank all my friends, colleagues, and family who have been there for me throughout my endeavor. I would especially like to thank my Mom for her love and support throughout my entire life and my girlfriend Emily for listening to all my problems and helping me to solve each of them to the best of your ability.

Most of all I thank God for giving me strength, drive, and patience. All of which were necessary for being successful in any goal.
# Table of Contents

List of Figures ........................................................................ vi

Nomenclature .......................................................................... viii

Chapter 1  Introduction .............................................................. 1
  1.1  Background ...................................................................... 1
    1.1.1  Brief Introduction to Circulation Control .................. 1
    1.1.2  Brief Introduction to Adaptive Airfoils ................. 2
  1.2  Objectives ........................................................................ 4
  1.3  Outline of Thesis ............................................................. 6

Chapter 2  Fluent Analysis of High-Lift CC Airfoil .................. 7
  2.1  Geometry and grid details .............................................. 7
  2.2  Solver settings ............................................................... 10
  2.3  Boundary conditions ....................................................... 10
  2.4  Discussion of Results ..................................................... 13
    2.4.1  Results for free-air conditions ............................. 13
    2.4.2  Wind-tunnel wall effects ..................................... 15
    2.4.3  Stagnation-point location .................................. 20
  2.5  Summary ........................................................................ 21

Chapter 3  Analytical Study of Adaptive CC Airfoil ............... 24
### List of Figures

1.1 Details of the Coanda surface for trailing edge from Ref. 8  
1.2 Schematic illustration of the adaptive circulation control airfoil concept  
2.1 General Aviation Circulation Control (GACC) airfoil geometry used in the current research  
2.2 Grid generated for FLUENT comparison to FUN2D  
2.3 Grid generated for FLUENT study of wall effects  
2.4 Comparison of $C_{\mu}^{\text{integrated}}$ with $C_{\mu}^{\text{isentropic}}$ for $\alpha = 0$; the straight line is included to indicate deviation from a perfect correlation  
2.5 Comparison of Langley experimental results with the NCSU FLUENT results  
2.6 CC effects on the flow field at $\alpha = 0$ for various values of $C_{\mu}$  
2.7 CC effects on flow field at $C_{\mu} = 0.047$ for various values of $\alpha$  
2.8 CC effects on flow field at $C_{\mu} = 0.078$ for various values of $\alpha$  
2.9 FLUENT prediction of wind-tunnel wall effects for varying values of $C_{\mu}$ at $\alpha = 0$ deg.  
2.10 FLUENT prediction of wind-tunnel wall effects for varying values of $C_{\mu}$ at $\alpha = -5$ deg.  
2.11 FLUENT prediction of wind-tunnel wall effects for varying values of $C_{\mu}$ at $\alpha = 5$ deg.  
2.12 CC effects on stagnation-point location  
3.1 Schematic of vorticity distribution used in the TATJT code  
3.2 An overview of the structure of the TATJT code  
3.3 Comparison of the $C_l$ predicted for a flat plate with $C_{\mu} = 1.0$ using TATJT with the results of Spence  
3.4 Comparison of the predicted $C_l$-$\alpha$ curves for flat plate with those from Ref. 1 and experiments at a jet-blowing angle of 31.4 deg  
3.5 Effects of blowing angle and blowing coefficient on ideal lift coefficient ($C_{l_{\text{ideal}}}$)  
4.1 Modified NLF-0414 geometry with CC capability  
4.2 Illustration of grid used for hybrid computations  
4.3 Blowing techniques for different flap deflections
4.4 Drag polar and transition curve for the NLF-0414 airfoil (no blowing) from XFOIL for $Re = 6 \text{ million}$. .......................... 37
4.5 Comparison between XFOIL and CFL3D computations for flap deflections of 0 and 5 deg with no blowing. ......................... 38
4.6 Lift curves for adaptive CC airfoil without and with blowing ($C_\mu = 0.04$). ................................................................. 40
4.7 Transition curves for adaptive CC airfoil without and with blowing ($C_\mu = 0.04$) at $Re = 6 \text{ million}$. ............................... 41
4.8 CC effects on flap effectiveness at large flap angles. .............. 42

A.1 Pressure coefficient around GACC airfoil using FLUENT runs at $\alpha = -5 \deg$ at various blowing coefficients .......................... 53
A.2 Pressure coefficient around GACC airfoil using FLUENT runs at $\alpha = 0 \deg$ at various blowing coefficients .......................... 54
A.3 Pressure coefficient around GACC airfoil using FLUENT runs at $\alpha = 5 \deg$ at various blowing coefficients .......................... 55

B.1 Flow-field pressure coefficient on baseline airfoil, no blowing and no flap deflection ....................................................... 57
B.2 Effect of flap deflection on pressure coefficient without blowing ................................................................. 58
B.3 Effect of flap deflection on pressure coefficient with blowing, $C_\mu = 0.04$. ................................................................. 59
Nomenclature

\[ A \quad \text{area} \]
\[ b \quad \text{wing span} \]
\[ c \quad \text{chord} \]
\[ C_d \quad \text{drag coefficient} \]
\[ C_l \quad \text{lift coefficient} \]
\[ C_{l,\text{ideal}} \quad \text{airfoil} \ C_l \text{ at which the stagnation point is located at the leading edge of the airfoil} \]
\[ C_m \quad \text{pitching moment coefficient about quarter chord} \]
\[ C_\mu \quad \text{momentum coefficient} \]
\[ h \quad \text{slot height} \]
\[ \dot{m} \quad \text{mass flow rate} \]
\[ M \quad \text{Mach number} \]
\[ P \quad \text{pressure} \]
\[ q \quad \text{dynamic pressure} \]
\[ R \quad \text{gas constant for air} \]
\( r \) radius of coanda surface

\( Re \) Reynolds number

\( T \) temperature

\( U \) velocity magnitude

\( w \) slot width

\( x_{tr} \) x-coordinate for transition location

\( \alpha \) angle of attack

\( \delta_f \) flap angle in degrees

\( \gamma \) ratio of specific heats

\( \mu \) viscosity

\( \rho \) density

**Subscripts**

\( duct \) stagnation conditions inside plenum

\( fc \) conditions at flow-control boundary

\( \infty \) freestream conditions

\( J \) slot-exit conditions
Chapter 1

Introduction

1.1 Background

1.1.1 Brief Introduction to Circulation Control

The idea of the circulation control (CC) airfoil is by no means new; the concept has been around since the late 1930s. For this research, circulation control refers to changing the circulation of the airfoil using a stream of high-velocity air emanating from a slot near the trailing edge of the airfoil. Circulation control airfoils have historically been viewed as a means to obtain high lift. The majority of research efforts have focused on blowing in a positive, or downward, direction at the trailing edge of the airfoil. Early efforts accomplished this downward inclination using a jet of high-velocity air that is blown straight out of the trailing edge at the desired angle.² This pneumatic-flap concept has been studied theoretically and experimentally by several researchers over the past several decades.¹⁻⁶ As time has progressed, more researchers have begun to take advantage of the Coanda effect⁷⁻¹⁰ by blowing over a round trailing edge, as shown in Fig. 1.1. This Coanda-based circulation control is currently attracting significant interest as a means of achieving high lift.

As is the case with all designs, there are trade-offs to be made for this increased
performance. Issues such as mass-flow requirements and reduced efficiency due to trailing-edge bluntness when operating in conditions at which high lift is not needed have hindered the implementation of these circulation control airfoils on production aircraft. Typically these airfoils become undesirable when in cruise conditions due to the blunt trailing edge of many of the designs.

While it is clear that circulation control has the capability to result in very high lift, there is a need for approaches that will, in addition, help achieve low cruise drag. Current projections by NASA for future Extremely Short Take Off and Landing (ESTOL) vehicles call for lift-coefficient goals of 10 along with cruise $L/D$-ratio goals of 25. Advanced personal air vehicle (PAV) concepts are currently targeting goals such as $C_{L_{\text{max}}}$ of 4 along with $L/D$ of 20. To achieve such goals, it is necessary to develop concepts that integrate the achievement of very high lift with the capability for significant drag reduction at cruise/climb conditions.

1.1.2 Brief Introduction to Adaptive Airfoils

Recent research in the Applied Aerodynamics Group at NCSU has led to the development of an automated cruise-flap system.\textsuperscript{11,12} The cruise flap, introduced by
Pfenninger,\textsuperscript{13,14} is a small trailing-edge flap which can be used to increase the size of the low-drag range of natural-laminar-flow (NLF) airfoils. The automation in Refs. 11 and 12 is achieved by indirectly sensing the leading-edge stagnation-point location using surface pressure measurements and deflecting the flap so that the stagnation-point location is maintained at the optimum location near the leading edge of the airfoil. Maintaining the stagnation point at the optimum location results in favorable pressure gradients on both the upper and lower surfaces of the airfoil. With such a cruise-flap system, the airfoil is automatically adapted for a wide speed range. This automated cruise-flap system was successfully tested in the subsonic wind tunnel at NCSU.\textsuperscript{12}

While the use of a cruise flap on an NLF airfoil results in low drag over a large range of speeds, there is a need for a revolutionary approach that integrates the achievement of significantly lower drag over a large range of operating speeds with the capability for generating very high lift at takeoff and landing conditions. Toward this objective, it is of interest to study an approach that integrates aerodynamic adaptation with the well-established high-lift capability of circulation control (CC) aerodynamics. This aerodynamic adaptation carries with it the possibility for significant skin-friction drag reductions through extensive laminar flow in addition to the high-lift benefits of CC aerodynamics. In a manner similar to a cruise flap, it is believed that by utilizing the stagnation-point sensing scheme, an adaptive CC airfoil can achieve extensive laminar flow over a large lift-coefficient range.

A similar concept was conceived in the late 50’s by Küchemann,\textsuperscript{15} in which he aimed at carrying the roof-top of the pressure distribution all the way to the trailing edge of the airfoil. Through utilizing this type pressure distribution, he believed that it may be possible to, in the context of transonic flow, to carry the supersonic region rearward, and possibly position the terminating shockwave at
the trailing edge.

### 1.2 Objectives

The current long-term research effort aims to explore the use of an adaptive circulation control airfoil that not only retains the high-lift capability of CC systems, but also enables the achievement of extended runs of laminar flow on both the upper and lower surfaces of the airfoil for achieving reduced skin-friction drag over a wide range of lift coefficients. The main idea behind the current concept is that extended laminar flow and associated skin-friction drag reduction can be achieved by combining a blown trailing edge with airfoil shaping. The idea is to design airfoils with extensive favorable pressure gradients so that the pressure-recovery region is positioned close to the jet slot. As a result of the high-speed jet flow from the slot, the concept avoids the turbulent boundary-layer separation that would otherwise occur if there was no jet. In addition, by using a small mechanical flap to control the angle of the jet, the concept allows for control of the lift-coefficient range over which the extended laminar flow is achieved. The overall idea behind this adaptive CC airfoil concept is illustrated schematically in Fig. 1.2.

![Figure 1.2: Schematic illustration of the adaptive circulation control airfoil concept.](image)

As a first step toward the long-term goal of studying an adaptive CC airfoil, an effort was undertaken for establishing and validating computational fluid dynamics
(CFD) analysis procedures for blown-trailing-edge airfoils. The CFD package used for this work was the FLUENT flow solver. The results are compared to CFD and experimental data obtained from a recent study by Jones et al.\textsuperscript{8} of a General Aviation CC (GACC) airfoil conducted at the NASA Langley Research Center. Since previous CFD studies on this airfoil did not include tunnel walls, the CFD study also includes an investigation of the effect of tunnel walls on the solution. In order to provide a foundation for the adaptive CC airfoil concept, the effects of CC on the leading-edge stagnation-point location were also examined.

This thesis describes the results of the second part of this long-term exploratory research effort. In the second part of the study, the focus has been on using analytical and computational techniques to determine if an adaptive CC natural-laminar-flow (NLF) airfoil can be used to adjust the lift-coefficient range over which laminar flow is achieved on both the upper and lower surfaces. In other words, the current work focuses on the use of CC for airfoil adaptation. An important part of this research is to compare an adaptive CC airfoil capability to that of a conventional cruise-flap system. A conventional cruise-flap system is one in which a small trailing-edge flap is used with no blowing to adapt an airfoil. These cruise flaps were introduced by Pfenninger\textsuperscript{13,14} and have been proven to be beneficial to the performance of NLF airfoils.\textsuperscript{11,16–20}

The exploratory study of adaptive CC airfoil was conducted using an analytical approach as well as a hybrid computational approach. In Chapter 3, the analytical approach, using a discrete-vortex implementation of a thin-airfoil thin-jet theory, is described. This approach was used for exploring the use of a pneumatic flap for airfoil adaptation by determining if a pneumatic flap can be used for adjusting the ideal lift coefficient at which the stagnation point is located at the leading edge of the thin airfoil. In Sec. 4.1, the hybrid computational approach is described, in which a laminar-flow airfoil is analyzed at several conditions using the CFL3D.
Reynolds-averaged Navier-Stokes code and an integral boundary-layer code to determine the transition locations. Using this hybrid computation approach, a laminar airfoil was analyzed with and without blowing at several flap angles to obtain the surface pressure distributions on the airfoil at these conditions. The pressure distribution for each condition was then used as input to an integral boundary layer method to determine the growth of the laminar boundary-layer transition amplification factor, \( n \), on the upper and lower surfaces of the airfoil. The extents of laminar flow for each condition were then obtained by determining the location on each surface at which the \( n \) reached a user-specified value of the critical transition amplification factor, \( n_{\text{crit}} \). Thus the focus of the computational modeling was only to determine the effect of the adaptive CC on the extents of laminar flow at different conditions. No effort was made in this research to compute the resulting drag.

1.3 Outline of Thesis

The verification of FLUENT CFD method is presented in Chapter 2. This chapter begins with details of the GACC airfoil used in the FLUENT computations. The results are shown for simulations with and without tunnel walls present. In addition the effect of CC on leading-edge stagnation point location is shown. Chapter 3 details the modeling using the Thin-Airfoil Thin-Jet theory. This includes details of assumptions made and iteration method used to arrive at a final solution. Results are validated using results from Spence. Chapter 4 describes the hybrid CFD approach used in predicting transition locations on an adaptive CC airfoil. The results of this approach are also presented in this chapter. Conclusions are given in Chapter 5 along with suggestions for future research considerations.
Chapter 2

Fluent Analysis of High-Lift CC Airfoil

The commercial flow-solver code FLUENT version 6.1 was used in the initial stages of this research. Grid generation was performed using GAMBIT, which is the preprocessor packaged with the FLUENT code. These codes were used to study two separate cases. The first case involves the examination of the GACC airfoil in free air with the objective of comparing the FLUENT results to CFD and wind-tunnel results presented in Ref. 8. The second case involves simulations of the GACC airfoil in the Basic Aerodynamic Research Tunnel (BART) to examine the influence of tunnel walls on this particular airfoil. Results from FLUENT were obtained for a matrix of 15 data points for both the cases.

2.1 Geometry and grid details

The geometry chosen for the current research was the General Aviation Circulation Control (GACC) airfoil, designed by Jones.8 The GACC airfoil was derived from a 17% GAW(1) airfoil by modifying the trailing edge to incorporate a 2% r/c coanda surface and is shown in Fig. 2.1.

For the first study, a circular computational domain (Fig. 2.2) was generated that extends to approximately 20 chord lengths in all directions and is comprised
Figure 2.1: General Aviation Circulation Control (GACC) airfoil geometry used in the current research.

of 132,762 cells. For the study of wall effects, a second grid was generated to include the wind-tunnel geometry and is shown in Fig. 2.3. The experiments were conducted by Jones et al.\textsuperscript{8} in the BART wind-tunnel which is located at the NASA Langley Research Center in Hampton, Va. The BART tunnel has a physical test-section size of 28” × 40” × 120”. The GACC model chord length was 9.4” with angle of attack changes made about the half-chord location. Further details of the experimental setup are given in Ref. 9. For the computation with walls, a separate grid was generated for each angle of attack, each of which is comprised of 123,602 cells and extends to 20 chord lengths upstream and downstream of the airfoil.

The grids for all of the analyses are hybrid unstructured grids. The domains consist of an unstructured grid far from the airfoil in order to reduce the number of cells and structured grid near the airfoil in order to maintain good resolution through the boundary and shear layers.
Figure 2.2: Grid generated for FLUENT comparison to FUN2D.
2.2 Solver settings

For this study, the steady, coupled implicit solver with node-based discretization scheme was selected. The coupled solver was chosen for two reasons. First compressibility effects need to be modeled, as the Mach number at the slot exit can often approach the sonic condition as the blowing rate is increased. Secondly, the FUN2D code utilizes a compressible solver and because the results from the current study were compared with FUN2D results, a compressible solver was also used for the FLUENT analysis. There was an attempt to run these problems with the segregated (decoupled) solver using very low relaxation factors, however it was found that for the cases with larger blowing rates the solution became unstable and did not converge. In addition the Spalart-Allmaras turbulence model was chosen in order to compare with the FUN2D results.

2.3 Boundary conditions

FLUENT does not allow the user to input freestream Mach number and Reynolds number directly. Instead, the freestream velocity and operating pressure were calculated using Eqs. 2.1–2.3 and provided as inputs for the analyses. The Mach and Reynolds numbers were set to 0.1 and 533,000, respectively, to match those used in Ref. 8

\[ U_\infty = M_\infty \sqrt{\gamma R T_\infty} \]  

(2.1)
\[ \rho_\infty = \frac{Re\mu_\infty}{U_\infty c} \quad (2.2) \]

\[ P_\infty = \rho_\infty RT_\infty \quad (2.3) \]

An approximate method was developed to estimate the required velocity at the flow control boundary \((U_{fc})\) to achieve a desired \(C_\mu; C_{\mu_{desired}}\). This method assumes incompressible flow throughout the duct, and was derived by solving the continuity equation. The equation for \(U_{fc}\) from this approximate method is given in Eq. 2.4.

\[ U_{fc} = U_\infty \sqrt{\frac{C_\mu A_{fc}b}{2A_{fc}^2}} \quad (2.4) \]

Once FLUENT converged, an integration was performed across the slot exit as shown in Eq. 2.5 to obtain the actual \(C_\mu\) of the jet at the slot. This \(C_\mu\), however, is different from \(C_{\mu_{desired}}\) because the \(U_{fc}\) for the latter is set using an approximate method.

\[ C_{\mu_{integrated}} = \frac{\int_{\text{slot}} \rho V^2 \, dy}{\frac{1}{2} \rho_\infty V_\infty^2 cb} \quad (2.5) \]

Furthermore, in order to be consistent with the methods used for calculating \(C_\mu\) in Ref. 8, all of the \(C_\mu\) values presented in this paper were calculated using isentropic flow relations. The equations for this procedure are given in Eqs. 2.6–2.8. In order to determine how close the isentropic \(C_\mu\) is to the integrated \(C_\mu\), the two values are compared in Fig. 2.4 for several cases. The \(C_\mu\) values indicated along the x-axis are values calculated using the isentropic relations. Values for \(C_\mu\) on the y-axis were computed by integrating the flow across the slot exit. The solid line in Fig. 2.4 indicates where the data points would lie if the two methods generated the same values for \(C_\mu\). The symbols are representative of the
actual values calculated using FLUENT and isentropic relations. Although the differences are very small, care must be taken to ensure consistency in the CFD solutions and experiments.

\[ \dot{m} = \rho J U J A J \quad (2.6) \]

\[ U_J = \sqrt{\frac{2 \gamma R T_duct}{\gamma - 1} \left( 1 - \left( \frac{P_{\infty}}{P_{duct}} \right)^{\frac{\gamma - 1}{\gamma}} \right)} \quad (2.7) \]

\[ C_\mu = \frac{\dot{m} U_J}{q_{\infty} c b} = 2 \left( \frac{h w}{c b} \right) \left( \frac{\rho J}{\rho_{\infty}} \right) \left( \frac{U_J}{U_{\infty}} \right)^2 \quad (2.8) \]

Figure 2.4: Comparison of \( C_{\mu integrated} \) with \( C_{\mu isentropic} \) for \( \alpha = 0 \); the straight line is included to indicate deviation from a perfect correlation.
2.4 Discussion of Results

The results from FLUENT predictions for the GACC airfoil are presented in three parts. In the first part, the prediction for the GACC airfoil in free-air conditions are compared with the results presented in Ref. 8. In the second part, the predicted results for the GACC airfoil with tunnel walls are presented and compared with the free-air results. In the third part, the effect of $\alpha$ and $C_\mu$ on the leading-edge stagnation-point location are presented and discussed.

2.4.1 Results for free-air conditions

In this part of the study, FLUENT results for free-air conditions are compared with CFD and experimental results from Ref. 8. The comparison is illustrated using $C_l$-$\alpha$ curves in Fig. 2.5. The results from FLUENT analyses consist of a matrix of 15 data points for $\alpha = -5, 0,$ and 5 deg and $C_\mu = 0, 0.008, 0.024, 0.047,$ and 0.078 and are presented in Fig. 2.5 using red dashed lines and square markers. The wind-tunnel results from Ref. 8 are presented as blue markers with best-fit curves in Fig. 2.5 for several angles of attack and for $C_\mu = 0, 0.007, 0.015, 0.025, 0.041,$ and 0.060. The values of $C_\mu$ for the FLUENT results differ from those for the results of Ref. 8 because of the difference between the actual $C_\mu$ and the desired $C_\mu$ when using the approximate method in Eq. 2.4 for estimating the $U_{fc}$ using incompressible-flow equations.
Although the values of $C_\mu$ for the FLUENT results do not match those for the results of Ref. 8, it is clear that the trends and most of the predictions for the $C_l$ are close to those from Ref. 8. In particular, the FLUENT predictions for $C_\mu = 0$, 0.008, and 0.047 agree quite well with the results for similar values of $C_\mu$ from Ref. 8. Two discrepancies between the FLUENT predictions and those from Ref. 8 are apparent: (i) for the $C_\mu = 0.024$ and (ii) for $C_\mu = 0.078$. The reason for the first discrepancy in the results is attributed to the incorrect prediction of the jet-separation location on the Coanda surface for $C_\mu = 0.024$. The apparent discrepancy in the results for $C_\mu = 0.078$ is attributed to nonlinear effects at the high blowing rates and the fact that the highest blowing rate in the results of Ref. 8 is for $C_\mu = 0.060$.

The flow-field data for the FLUENT results are presented in two separate
parts. In the first part, the effects of increasing $C_\mu$ for a constant angle of attack is presented. The second part examines the effect of angle-of-attack changes and their influence on the CC airfoil for a constant $C_\mu$. The flow-field data is presented as pressure contours and streamline plots; these aid in the understanding of the effects of CC on the flow over the airfoil.

The first part of the flow-field data is shown in Figs. 2.6(a)–(c). It can be seen that as the blowing rate is increased the streamlines become more curved—an indication of increased circulation. The second part of the flow-field data is shown in Figs. 2.7(a)–(c) and Figs. 2.8(a)–(c) to illustrate the effects of changing the angle of attack while holding blowing rates constant. The results are presented for two blowing rates: the mild blowing case $C_\mu = 0.047$ and the highest blowing rate $C_\mu = 0.078$. The results show that changes to $C_\mu$ have a significant effect on the jet-separation location and the resulting $C_l$. In comparison, changes to $\alpha$ have a much smaller effect on the jet-separation location.

### 2.4.2 Wind-tunnel wall effects

In this sub-section, the FLUENT results for the GACC airfoil with the effect of wind-tunnel walls are presented. Figures 2.9–2.11 show the influence of the wall on the CFD solution. These figures present the predicted $C_l$ as a function of $C_\mu$ for $\alpha = 0, 5, \text{ and } -5 \text{ deg}$ respectively. Figure 2.9 also includes a comparison to results for the FUN2D study$^8$ for $\alpha = 0$ deg, the only angle of attack for which the FUN2D results were presented in Ref. 8. Figures 2.9–2.11 indicate that the presence of walls has very little influence on the CFD solution. The solution, including tunnel walls, consistently show that for low blowing coefficients, the $C_l$ values are predicted to be lower than those without walls. However, at the largest blowing coefficients, the trend reverses and $C_l$ values with walls are predicted to be higher than those without walls.
Figure 2.6: CC effects on the flow field at $\alpha = 0$ for various values of $C_\mu$. 

(a) $C_\mu = 0.000$

(b) $C_\mu = 0.047$

(c) $C_\mu = 0.078$
Figure 2.7: CC effects on flow field at $C_\mu = 0.047$ for various values of $\alpha$. 

(a) $\alpha = -5$ deg 

(b) $\alpha = 0$ deg 

(c) $\alpha = 5$ deg
Figure 2.8: CC effects on flow field at $C_\mu = 0.078$ for various values of $\alpha$. 

(a) $\alpha = -5$ deg

(b) $\alpha = 0$ deg

(c) $\alpha = 5$ deg
Figure 2.9: FLUENT prediction of wind-tunnel wall effects for varying values of $C_\mu$ at $\alpha = 0$ deg.
2.4.3 Stagnation-point location

The motivation for examining the stagnation-point behavior is that the stagnation-point location was used successfully in earlier research\textsuperscript{11,12} for closed-loop control of a trailing-edge flap. It was, therefore, desirable to examine the CFD solutions for the CC airfoils to see if there was any evidence that would suggest that a similar approach could be extended for use with CC airfoils.

Stagnation-point location, measured as an arc length from the jet exit around the upper surface of the airfoil, as a function of $C_l$ is presented in Fig. 2.12. Each curve in Fig. 2.12 represents a different blowing rate and for each blowing coefficient there are three points that correspond to three different angles of attack($-5, 0, \text{ and } 5 \text{ deg}$). From Fig. 2.12 it can be seen that the stagnation point moves in a predictable manner, both with angle of attack and with changing blow-
ing rate. This behavior provides an indication that the stagnation-point location can be used as a means to develop closed-loop control of the jet $C_\mu$ on CC airfoils.

2.5 Summary

In this chapter the use of FLUENT for validation of CFD methods for circulation control has been described. The results were compared to experimental and computational results. Three studies were performed on the given geometry: (i) CFD computations on free-air configuration, (ii) study of wall effects on CFD solution, and (iii) a study of leading-edge stagnation point behavior.

It was found that the computations from FLUENT capture trends very well for cases with blowing. It was noticed that the computations carried out with the wind-tunnel walls present exhibited very little difference from solutions in free-air. A systematic movement in leading-edge stagnation point was examined, indicating that this type of airfoil could be used for aerodynamic adaptation.
Figure 2.10: FLUENT prediction of wind-tunnel wall effects for varying values of $C_\mu$ at $\alpha = 5$ deg.
Figure 2.12: CC effects on stagnation-point location.
Chapter 3
Analytical Study of Adaptive CC Airfoil

The objective of the analytical study was to determine the effect of jet angle and $C_\mu$ on the ideal lift coefficient of an airfoil. A simple and rapid analysis method was sought for this study. As shown in past studies, it is possible to represent the jet sheet using a distribution of vortices. It is then possible to extend the well-known thin airfoil theory for the study of the jet-flapped circulation control airfoil. The assumptions that were made for this approach include: inviscid, incompressible, irrotational flow, thin airfoils, and shallow jet shapes. In spite of these seemingly drastic simplifications, these methods correlate well for jet deflections up to 60 degrees.

3.1 Boundary Conditions

As shown in Fig. 3.1, Thin-Airfoil Thin-Jet Theory (TATJT) uses a discrete vortex method to solve for the flow around the thin airfoil (represented as the camberline) and the jet streamline. To accomplish this modeling successfully, boundary conditions must be applied properly. First, the flow must be constrained to be tangential to the camberline of the airfoil and the streamline that represents the jet. This condition is satisfied by imposing a zero-normal-velocity boundary condition.
at each collocation point along the camberline and jet streamline.

![Diagram of vorticity distribution](image)

Figure 3.1: Schematic of vorticity distribution used in the TATJT code.

### 3.2 TATJT Algorithm

Since the shape of the jet streamline is not known, it needed to be determined as a part of the solution procedure. Under the assumptions of small deflection angles and small angles of attack, Spence\(^1\) has shown that in the context of thin airfoils and thin jets, the circulation distribution along the jet can be described as a simple function of the jet curvature (Eq. 3.1), where \(C_\mu\) in Eq. 3.1 is the jet momentum coefficient, and is given in Eq. 3.2. From Eq. 3.1 it can be seen that the unknowns are jet curvature \(y''(x)\), and circulation distribution \(\gamma_j(x)\).

\[
\gamma_j(x) = -\frac{1}{2}U_\infty C_\mu \ y''(x) \tag{3.1}
\]

\[
C_\mu = \frac{\rho_J U_J^2 h}{\frac{1}{2} \rho_\infty U_\infty^2 c} \tag{3.2}
\]
The result of thin airfoil theory gives the circulation distribution along the
given shape. So in order to determine the actual jet shape a Newton iteration
(Eq. 3.3) was implemented to iterate on the jet shape until the condition in Eq. 3.1
was satisfied. The flowchart in Fig. 3.2 gives an overview of this method. The
Jacobian matrix, residual vector, and correction vector for the Newton iteration
are given in Eqns. 3.4, 3.5, and 3.6.

\[ \mathbf{J} \cdot \delta \mathbf{y} = -\mathbf{F} \]  \hspace{1cm} (3.3)

\[ \mathbf{J} = \begin{pmatrix} \frac{dF_1}{dy_1} & \cdots & \frac{dF_1}{dy_n} \\ \vdots & \ddots & \vdots \\ \frac{dF_n}{dy_1} & \cdots & \frac{dF_n}{dy_n} \end{pmatrix} \]  \hspace{1cm} (3.4)

\[ \mathbf{F} = (\gamma J)_{\text{Eq. 3.1}} - (\gamma J)_{\text{from TATJT}} \]  \hspace{1cm} (3.5)

\[ \delta \mathbf{y} = \begin{pmatrix} \delta y_1 \\ \vdots \\ \delta y_j \\ \vdots \\ \delta y_n \end{pmatrix} \]  \hspace{1cm} (3.6)

At each step of the iteration, the jet shape is updated using the correction
vector, \( \delta y \), in Eq. 3.6. Once the residuals are close to zero, the jet shape is found
and the vorticity distribution is determined using Eq. 3.1.
### Validation of TATJT Code

To determine how well the TATJT code was performing, results from Spence\(^1\) were compared to TATJT computations. The plot in Fig. 3.3 shows comparison of lift coefficient \((C_l)\) for various flap deflections with Spence’s results for a flat plate with \(C_\mu = 1.0\). At low values of flap deflections it can be seen that the comparisons are very good. However, as deflections become large, TATJT predictions are greater than the values predicted by Spence. The reason for this discrepancy is that Spence uses a linearized boundary condition, in which the distributed vortices are assumed to lie on the x-axis. In TATJT, on the other hand, the vorticity and the zero-normal-flow boundary conditions are applied at points on the actual airfoil camberline and jet streamline (Fig. 3.1).

In addition, the \(C_l\) versus angle of attack curves were also compared with
Figure 3.3: Comparison of the $C_l$ predicted for a flat plate with $C_\mu = 1.0$ using TATJT with the results of Spence.

Spence’s results for a flat plate. The plots in Fig. 3.4 show that these methods agree closely. It must also be noted that the experimental results presented in Fig. 3.4 are from tests conducted at the National Gas Turbine Establishment in 1953-1954 and were extracted from the work of Spence.\footnote{Spence, T. (1953). The influence of blowing on the characteristics of a flat plate boundary layer. National Gas Turbine Establishment. Report No. G.T.E.312.}
Figure 3.4: Comparison of the predicted $C_\mu$-$\alpha$ curves for flat plate with those from Ref. 1 and experiments at a jet-blowing angle of 31.4 deg.
3.4 Discussion of Results

It has been shown that, in the context of thin airfoil theory, it is possible to calculate an ideal lift coefficient ($C_{\text{ideal}}$). This value corresponds to the ideal angle of attack which is the angle of attack at which the stagnation point lies exactly on the leading edge of the thin airfoil. At this condition, there is no suction peak on either the upper or lower surface at the leading edge of the airfoil: for a thin airfoil at the ideal condition, the vorticity at the leading edge is zero. For this reason, the $C_{\text{ideal}}$ for a natural laminar flow (NLF) airfoil also corresponds to the $C_l$ for the middle of the low-drag range (or drag bucket) of the NLF airfoil. If it can be shown that it is possible to shift this $C_{\text{ideal}}$ through the use of the pneumatic-flap, then it follows that the pneumatic-flap can be used as a cruise flap.

In order to determine this $C_{\text{ideal}}$, the condition of zero vorticity on the first panel of the airfoil, i.e. the leading edge, was satisfied. The ideal angle of attack for a pneumatic-flapped airfoil was determined using a Newton iteration and TATJT to determine the condition at which the vorticity at the leading-edge is zero. The results in Fig. 3.5 (a) show that it is indeed possible to shift the $C_{\text{ideal}}$ using a deflection of the pneumatic-flap. The results shown are for a constant $C_{\mu}$ of 1 and are given for two airfoils, one symmetric and one with a 1% parabolic camber. Figure 3.5 (b) presents the dependence of $C_{\text{ideal}}$ on $C_{\mu}$ for various flap deflections for the 1% parabolic cambered airfoil. These results show that the $C_{\text{ideal}}$ can also be changed using various blowing rates, independent of the flap deflection angle.

3.5 Summary

In this chapter a simple and rapid analysis method has been presented. This method integrates the well known thin-airfoil theory with thin-jet theory devel-
oped by Spence.\textsuperscript{1} A simple MATLAB program was written to perform this anal-
ysis. Given an airfoil camberline and jet deflection angle, this program utilizes
a multi-dimensional Newton iteration to determine the jet shape. Results from
this method compared very well to results from Spence.\textsuperscript{1} This code was then
utilized to examine the influence that both jet angle and jet blowing rate had on
the airfoil’s ideal lift coefficient. It was shown that the ideal lift coefficient could
be systematically changed using a circulation control airfoil.
Figure 3.5: Effects of blowing angle and blowing coefficient on ideal lift coefficient ($C_{l\text{ideal}}$).
4.1 Geometry and Grid Details

In order to get a better understanding of the detailed flow physics of the problem, a circulation control airfoil was designed to be analyzed using a Navier-Stokes flow solver. This airfoil, presented in Fig. 4.1, is a modified version of the NLF-0414 airfoil\textsuperscript{16} designed at NASA Langley Research Center. The only modification made to the airfoil is a change in the trailing edge shape to accommodate for blowing across a movable flap on both the upper and lower surface. The flap hinge line was located at 90\% of the chord with a slot height-to-chord ratio of 0.001. The leading edge of the flap was designed with a radius-to-chord ratio of 0.008, as shown in Fig. 4.1(b); to utilize the Coanda effect.\textsuperscript{8} This was done to keep the flow attached at larger flap deflections.

The flow solver CFL3D\textsuperscript{21} (version 5.0 modified), developed at NASA Langley Research Center, was used in the current research. CFL3D requires structured grids for computations, these grids were generated using Gridgen (version 15). Figure 4.2 illustrates the details of one of the grids. Minimum wall spacing was set to $1 \times 10^{-6}$ resulting in $y^+$ values less than one, guaranteeing sufficient resolution.
through the boundary layer. Each of the grids were multiblock with 3 blocks and total of 55,936 cells.

(a) Entire airfoil

(b) Close-up of trailing edge

Figure 4.1: Modified NLF-0414 geometry with CC capability

(a) Grid details near airfoil

(b) Close-up of trailing edge

Figure 4.2: Illustration of grid used for hybrid computations.

4.2 Solver Settings

The NASA Langley Reynolds averaged Navier-Stokes code CFL3D (Computational Fluid Dynamics Laboratory – 3D) was chosen for this work. The flow is assumed to be steady – a reasonable assumption for small flap angles. However, as flap angle increases in the zero blowing case, flow separation occurs, leading to
unsteady flow. However, as is explained in the following sections with the introduction of blowing, the separated region becomes much smaller thus making the steady flow assumption reasonable again. To simulate typical flight conditions for low-speed aircraft the Mach number and Reynolds number were set to 0.25 and 6 million, respectively. All computations were carried out as fully turbulent using the $k-\omega$ turbulence model.

4.3 Boundary Conditions

Blowing rates are changed by specifying properties at the flow control boundary inside the duct. The density and velocity components were specified in order to achieve desired $C_\mu$. As is noted from Fig. 4.1, the flow control boundaries have different heights. As a result of these differences in order to keep the same $C_\mu$ for both upper and lower slots the boundary conditions must change.

Angle of attack sweeps were computed, both with and without blowing, for seven different flap settings, ($\delta_f = -10, -5, 0, 5, 10, 15, \text{ and } 20 \text{ deg}$). In order to simplify the problem for this research dual blowing was not examined; i.e. for positive flap deflections blowing takes place over the upper surface (Fig. 4.3(a)) of the flap and for negative flap blowing takes place over the lower surface of the flap (Fig. 4.3(b)). Since the basic geometry of the airfoil changes with flap deflections a different grid had to be used for each flap setting.
(a) Positive flap deflections with upper-surface blowing
(b) Negative flap deflections with lower-surface blowing

Figure 4.3: Blowing techniques for different flap deflections.

4.4 Methodology

Generally in order to study the airfoil aerodynamic adaptation to different flight conditions, a plot of airfoil lift versus drag, Fig. 4.4(a), would be examined. The low-drag range, or drag bucket, for airfoils that are capable of achieving laminar flow will be well defined, and can be shifted by using a cruise flap. This phenomenon is illustrated in Fig. 4.4(a). However, the current research does not address the issue of drag for two reasons: (i) CFD computations are carried out under the assumption of fully-turbulent flow and (ii) determining drag for the blown cases from the CFD solution requires decomposition of the drag and thrust components from the net streamwise force. This first reason is due to the fact that the transition locations are unknown and prediction of transition as a part of the CFD solution is still an area of research. Likewise, decomposition of thrust and drag for blown systems is also an area of ongoing research. Because of these difficulties in calculating drag on CC airfoils, it is desirable to consider alternate methods for examining the effectiveness of an adaptive CC airfoil.

Because it is known that for NLF airfoils the amount of achievable laminar flow
Figure 4.4: Drag polar and transition curve for the NLF-0414 airfoil (no blowing) from XFOIL for $Re = 6$ million.

is directly related to the low-drag range of the airfoil, the objective of the current CFD effort was not to examine resulting drag changes. Instead the objective was to examine changes to the lift-coefficient-range over which extended laminar flow is achieved. An example is shown in Fig. 4.4 where the relationship between the drag bucket and laminar flow occurrence for the NLF-0414 airfoil is illustrated. Specifically, the low-drag range indicated by the $C_l$ versus $C_d$ curves in Fig. 4.4(a) is manifested in the transition curves shown in Fig. 4.4(b). Using this knowledge, the transition curves are examined to determine the effectiveness of the adaptive CC airfoil concept. These curves were examined for several flap deflections both with and without blowing.

Since the CFD solution was calculated assuming a fully turbulent flow, the transition curves cannot be determined directly from CFL3D. However, knowing the pressure distribution around the airfoil it was possible to use an integral boundary layer method to arrive at an estimate for transition location. This calculation was performed using a subroutine in the PROFOIL$^{24}$ code, in which
the airfoil surface-pressure distribution was supplied to the code and transition locations on the upper and lower surfaces were readily obtained. The results for the zero-blowing cases, presented in Fig. 4.5(b), match very closely with those predicted by XFOIL.

### 4.5 Validation of Hybrid Approach for Zero Blowing

Validation of CFD procedures was accomplished by utilizing XFOIL, which is a fully-coupled viscous/inviscid panel method that solves the flow around 2-D airfoils. The results for these runs are shown in Fig. 4.5. It can be seen in Fig. 4.5(a) that the lift curves from the CFD agree well with those from XFOIL except for small deviations at high angles of attack. The CFD solution, as mentioned in Sec. 4.2, is assumed fully turbulent using the $k-\omega$ turbulence model; whereas the results from XFOIL are transitional, using the $e^N$ method. These comparisons show favorable results and give confidence in the CFD procedures implemented.

![Figure 4.5: Comparison between XFOIL and CFL3D computations for flap deflections of 0 and 5 deg with no blowing.](image-url)
4.6 Discussion of Results

In this subsection, results from the hybrid computation approach are presented. The objective of this part of the study was to evaluate the effectiveness of the adaptive CC concept in adjusting the $C_l$ range over which extensive laminar flow is achieved. All the results in this subsection are for the NLF airfoil shown earlier in Fig. 4.1 operating at $Re = 6$ million. For all the cases with blowing, the $C_\mu$ was set to 0.04. Figure 4.6 presents the $C_l$-$\alpha$ curves for several flap angles for both blown and unblown conditions. As mentioned in the previous sections, for positive flap deflections blowing takes place over the upper surface of the flap, and for negative flap deflections, blowing takes place over the lower surface. The bold lines in Figs. 4.6(a) and 4.6(b) represent the 0-deg flap deflection without blowing and serve as a demarcation line between positive and negative flap deflections.

The results of Fig. 4.6 show clearly that trailing-edge blowing increases the flap effectiveness when compared to the no-blowing cases. That is, for a given flap angle and given $\alpha$, there is a greater increase in $C_l$ (relative to the unblown zero-deg flap case) with blowing than without.

As mentioned in Sec. 4.1, the $C_l$-shift in the low-drag range of an airfoil due to flap can be determined by instead examining the $C_l$-shift in the transition curves. Using this idea, the transition curves for each flap deflection from CFL3D are presented in Fig. 4.7 for both blown and unblown cases. For each case, the airfoil geometry is shown in the lower part of each plot. The thin solid line represents the adaptive CC airfoil with blowing, whereas the dashed line represents the adaptive CC airfoil without blowing. The thicker line in each subfigure of Fig. 4.7 represents the case with zero flap deflection and no blowing and is common to all figures. Similar to the transition results shown earlier from XFOIL in Fig. 4.4(b), there is a definite shift in the transition curves using just the flap alone. Examining the
conditions without blowing it can be seen that as the flap angle is changed, the transition curves shift, down for upward flap deflection and up for downward flap deflections. When blowing is introduced it is shown that the magnitude of the shift is increased for all flap deflections, which is a clear indication that the $C_l$ range over which extended laminar flow is achievable can be shifted through the implementation of adaptive CC.

Examining the effects of CC on the flow field gives additional insight on the nature of the drag reduction. Figure 4.8 presents the effects of CC on the streamlines. This particular figure is representative of a situation in which the flow becomes separated off the surface of the flap. By introducing blowing it can be seen that the flow is no longer separated off the flap. Prevention of the separation will yield a significant reduction in the associated pressure drag that is not seen by examining the shift in the transition curves. Thus it is deduced that blowing not only results in a greater $C_l$-shift of the transition curves with flap, but also is successful in eliminating the separation off the flap surface at large flap angles, resulting in decreased pressure drag.
Figure 4.7: Transition curves for adaptive CC airfoil without and with blowing ($C_\mu = 0.04$) at $Re = 6$ million.
Figure 4.8: CC effects on flap effectiveness at large flap angles.
4.7 Summary

In this chapter a hybrid computational analysis method has been described. The hybrid method consisted of the use of CFL3D to solve for flow properties around the airfoil and an integral boundary layer method to determine where transition would occur based off of the CFL3D solution. The transition data was utilized to determine what effect circulation control has on the amount of laminar flow that an airfoil can achieve. It was seen that with the presence of blowing the amount of achievable laminar flow has been extended. The addition of circulation control has also been shown to eliminate regions of separated flow when flap deflections and or angles of attack are high.
Chapter 5
Concluding Remarks

5.1 Summary of Results

Current projections for future aircraft technologies call for challenging goals for both high lift for takeoff/landing conditions and low drag at cruise/climb conditions. Revolutionary approaches are needed to satisfy the demanding requirements. One approach is to explore the use of concepts that synergistically integrate aerodynamics and propulsion for achieving high efficiency at multiple operating conditions. The overall objective of this research effort is to explore the use of adaptive circulation control airfoils to achieve low drag at cruise and climb conditions while retaining the well-known very-high-lift capability of traditional circulation-control airfoils. In this work, circulation control is achieved by blowing a jet of high-velocity air through slots on the upper and lower surfaces over a small adjustable mechanical flap. The focus is on application to low-speed aircraft with natural-laminar-flow wings, although variations of the concept could be used for transonic applications in future. The idea is to combine airfoil shaping and circulation control, so that extensive laminar flow is achieved over a significant portion of the chord and turbulent separation in the recovery region of the airfoil is avoided by use of the jet blowing. Additionally, the extensive laminar flow can be achieved over a large lift coefficient range by adjusting the angle of the me-
chanical flap which also deflects the jet. The focus of this paper is to explore one specific aspect of this larger research effort: the capability of circulation control to adapt an airfoil to suit different flight conditions.

The specific objective of the research presented was to determine if an adaptive circulation control system is effective at achieving laminar flow over a large lift coefficient range and whether such a system is more effective than a conventional unblown cruise-flap system. To explore this capability, a three-pronged approach was used: (i) a validation of computational methods using FLUENT on an existing general aviation circulation control airfoil, (ii) an analytical approach using a thin-airfoil thin-jet method to study the changes to the ideal lift coefficient of an airfoil with change to the jet angle of a pneumatic flap, and (iii) a hybrid computational approach in which a laminar-flow airfoil with adaptive circulation control was analyzed using CFL3D, a Reynolds-averaged Navier-Stokes (RANS) code, and the resulting pressure distribution was used as input to a boundary-layer code to determine the extents of laminar flow. No drag predictions were made using the RANS solution for two reasons: (a) the determination of transition locations as a part of a RANS solution is still an area of research and (b) the decomposition of thrust and drag for blown systems is still being studied. For these reasons, the effectiveness of the concept was determined using the hybrid computational approach simply by examining the extents of laminar flow at different lift coefficients.

Results of the first study show that while the FLUENT predictions do not match the CFD and experimental results of Ref. 8 exactly, the overall trends are followed very closely. Throughout the range of blowing coefficients, FLUENT consistently predicted a slightly lower overall lift coefficient.

In addition to the validation, a study was performed on the influence of wind tunnel walls on the CFD solution. For low blowing coefficients, it was found
that the lift is predicted to be lower for the cases with walls. The trends are reversed for the higher blowing coefficients, for which the cases with walls yield a higher predicted lift. Although the solutions are different, the differences are small, and could as well be attributed to differences in the grids rather than the actual presence of walls.

The influence of circulation control on the leading-edge stagnation point location was also examined. It was shown that changes to the blowing rate and angle of attack result in systematic changes to the stagnation-point location. This observation indicated that it is possible to use a closed-loop control system by sensing the stagnation-point location.

The results of the analytical effort, using thin airfoil thin jet theory, show that the ideal lift coefficient of an airfoil can be adjusted using either the jet angle or the jet momentum coefficient, and provided the impetus to explore the concept using the higher-fidelity computational approach. The results of the hybrid computational approach confirm that the lift-coefficient range over which extensive laminar flow is achieved can be altered using the flap angle. Furthermore, the results show that the lift-coefficient change with blowing is greater than that achieved without blowing, thus confirming that an adaptive circulation control system is more effective than a traditional cruise-flap system. The results also show that separation off the flap is avoided at large flap angles when blowing is used.

The current research confirms that circulation control offers the potential for achieving laminar flow and significant drag reduction over a large range of flight conditions while still retaining the very-high-lift capability for takeoff and landing. As a result of these findings, it is believed that adaptive circulation control shows promise for achieving the demanding goals for future aircraft concepts.
5.2 Recommendations for Future Work

In the current work the adaptive circulation control airfoil was designed through a simple modification of the trailing edge of an existing natural-laminar-flow airfoil. It is of interest to examine the benefits on an airfoil that is specifically designed for this application. In addition it would also be beneficial to examine the quantitative effects of circulation control on drag. It is of interest to examine the benefits of differential blowing simultaneously from both upper and lower surface slots, this technique may lead to even greater benefits. Further research is also needed at a systems level to determine the impacts on the propulsion system and on the entire aircraft.
Chapter 6

References


Appendix A

Supplemental Figures for Fluent Computations

The following figures are given as a reference for the reader. These figures represent the gamut of simulations that were completed for the work with FLUENT. Figures A.1, A.2, and A.3 represent the pressure coefficient around the GACC airfoil at $\alpha = -5$, 0, and 5 deg respectively.
Figure A.1: Pressure coefficient around GACC airfoil using FLUENT runs at \( \alpha = -5 \) deg at various blowing coefficients
Figure A.2: Pressure coefficient around GACC airfoil using FLUENT runs at \( \alpha = 0 \) deg at various blowing coefficients

(a) \( C_\mu = 0.000 \)
(b) \( C_\mu = 0.008 \)
(c) \( C_\mu = 0.024 \)
(d) \( C_\mu = 0.047 \)
(e) \( C_\mu = 0.078 \)
Figure A.3: Pressure coefficient around GACC airfoil using FLUENT runs at \( \alpha = 5 \) deg at various blowing coefficients
Appendix B

Supplemental Figures for Hybrid Computations

The figures presented in this appendix are supplemental figures from the hybrid computational study. These figures are from the set of simulations with $\alpha = 0$ deg. Figure B.1 shows the reference baseline case with no blowing and no flap deflection. Figures B.2 and B.3 are flow field plots for the adaptive CC foil without ($C_\mu = 0.00$) and with blowing ($C_\mu = 0.04$), respectively.
Figure B.1: Flow-field pressure coefficient on baseline airfoil, no blowing and no flap deflection.
Figure B.2: Effect of flap deflection on pressure coefficient without blowing.
(a) $\delta f = -10 \text{ deg}$

(b) $\delta f = -5 \text{ deg}$

(c) $\delta f = 5 \text{ deg}$

(d) $\delta f = 10 \text{ deg}$

(e) $\delta f = 15 \text{ deg}$

(f) $\delta f = 20 \text{ deg}$

Figure B.3: Effect of flap deflection on pressure coefficient with blowing, $C_\mu = 0.04$. 

59