COMPUTATIONAL AND EXPERIMENTAL COMPARISON OF A
POWERED LIFT, UPPER SURFACE BLOWING CONFIGURATION

A Thesis
presented to
the Faculty of California Polytechnic State University,
San Luis Obispo

In Partial Fulfillment
of the Requirements for the Degree
Master of Science in Aerospace Engineering

by
Jay Mark Marcos
November 2013

© 2013

Jay Mark Marcos

ALL RIGHTS RESERVED

COMMITTEE MEMBERSHIP

TITLE: Computational and Experimental Comparison of a
Powered Lift, Upper Surface Blowing Configuration

AUTHOR: Jay Mark Marcos

DATE SUBMITTED: November 2013

COMMITTEE CHAIR: Dr. David D. Marshall
Associate Professor of Aerospace Engineering

COMMITTEE MEMBER: Craig Hange
Engineer at NASA Ames Research Center

COMMITTEE MEMBER: Dr. Kristina Jameson
Propulsion Specialist at Spaces Systems/Loral, LLC

COMMITTEE MEMBER: Dr. Kim Shollenberger
Professor of Mechanical Engineering

ABSTRACT

Computational and Experimental Comparison of a Powered Lift, Upper Surface Blowing Configuration

Jay Mark Marcos

In the past, 2D CFD analysis of Circulation Control technology have shown poor comparison with experimental results. In Circulation Control experiments, typical results show a relationship between lift coefficient, C_L , vs blowing momentum coefficient, C_μ . CFD analysis tend to over-predict values of C_L due to gridding issues and/or turbulence model selection. This thesis attempted to address both issues by performing Richardson's Extrapolation method to determine an acceptable mesh size and by using FLUENT's 2-equation turbulence models. The experimental results and CAD geometry were obtained from Georgia Tech Research Institute for comparison with the CFD analysis.

The study showed that 3D CFD analysis of circulation control showed similar results of over-predicting C_L , which can also be attributed to gridding issues and turbulence model selection. When compared to the experimental results, the $k - \omega$ turbulence model produced the lowest errors in C_L of approximately 15-17%. The other turbulence models produced errors within 5% of $k - \omega$. A fully unstructured volume mesh with prismatic cells on the surfaces was used as the grid. The CCW configuration was analyzed with and without wind tunnel walls present, which produced errors of 20% and 15% in C_L , respectively, when compared to experimental results. Despite the large errors in C_L , CFD was able to capture the trend of increasing C_L as C_μ was increased. Results reported in this thesis can be further calibrated to allow the CFD model to be used as a predictive tool for other CCW applications.

ACKNOWLEDGMENTS

First I would like to thank my advisor Dr. David Marshall for the opportunities he has given me during my time as an undergrad and graduate student at Cal Poly. As an eager and excited undergrad student looking for more CFD experience back in 2008, Dr. Marshall allowed me to be part of his newly funded NASA research project. Under his mentorship, I was exposed to industry level CFD practices and state of the art resources. From then on, it only made sense to pursue higher education in CFD with Dr. Marshall as my advisor. Without him, this work would not have been possible. This work was funded as part of a NASA Research Announcement award under Contract NNL07AA55C with Craig Hange and Joe Posey as the technical monitors.

I wish to thank the excellent work of Robert Englar and his team at Georgia Tech Research institute in their vast contribution to circulation control technology. All their research into CCW provided me with the data and geometry that I needed to perform the tasks in this thesis. Thank you for all that you've contributed as well as the patience you've shown me every time I bugged you for more data to validate with.

A special thanks to all my fellow aero grad students in the aerospace graduate lab for their constant encouragements to finish my thesis, as well as their constant distractions that kept me sane when working late into the nights. Thank you to Bryan Blessing and John Pham for mentoring me when I first became part of the CFD team. You two have taught me most of what I know about CFD.

I'd like to thank my parents for all the sacrifices that they've made to provide me with everything that I needed throughout my six years here at Cal Poly. Not only have you funded this amazing experience, you've also never stopped providing me with the support and trust that I needed to finish strong. I love you both.

Finally, thank you Jennifer. You've always stood by me throughout this stage of my life, always giving me hope and always believing that I can finish this thesis. There were times when even I was close to giving up, but your words of encouragement always got me back on track, always pointed me to the direction of my goals. You're my motivation and my inspiration in many ways, and I dedicate this to you.

TABLE OF CONTENTS

List of Figures	viii
List of Tables	x
Nomenclature	xi
1 INTRODUCTION	1
1.1 Objectives	1
1.2 NASA Research Announcement	3
1.3 Background	4
1.3.1 Circulation Control	4
1.3.2 Computational Fluid Dynamics	6
1.3.3 CFD Studies on 2D Circulation Control	7
2 GEOMETRY DESCRIPTION	10
2.1 Wind Tunnel Experiments	10
2.2 Circulation Control Wings	11
2.3 Coanda Effect	12
2.4 Blowing Parameter, $C\mu$	13
2.5 GTRI Configurations	14
2.5.1 Test Configurations	20
2.5.2 Configuration B	22
2.6 Wind Tunnel	23
3 MESH GENERATION	25
3.1 Gridding Techniques	25
3.1.1 Geometry/CAD	26
3.1.2 Surface Mesh	28
3.1.3 Volume Mesh	31
3.2 Boundary Layer Mesh	35
3.3 Grid Refinement	37
3.3.1 Richardson's Extrapolation Method	38
4 SIMULATION SETUP	40

4.1	Governing Equations	40
4.1.1	Continuity Equation	40
4.1.2	Momentum Equation	41
4.1.3	Energy Equation	41
4.2	Turbulence Modelling	43
4.2.1	Standard $k - \epsilon$ Model	44
4.2.2	Realizable $k - \epsilon$ Model	45
4.2.3	RNG $k - \epsilon$ Model	47
4.2.4	Standard $k - \omega$ Model	49
4.2.5	Shear Stress Transport $k - \omega$ Model	50
4.2.6	Near-Wall Treatment	52
4.2.7	Wall Functions	54
4.2.8	Near-Wall Treatment	55
4.3	Operating Conditions	56
4.3.1	Boundary Conditions	57
4.3.2	Wind Tunnel Inlet and Outlet	57
4.3.3	Engine Boundary Condition	58
4.3.4	Slot Boundary Condition	60
4.4	Solver Settings	63
5	RESULTS	65
5.1	Case Listings	65
5.2	Grid Convergence Study	69
5.2.1	Mesh Description	72
5.3	Turbulence Model Sensitivity	73
5.4	Free Air Analysis	78
5.5	Modeling the Plenum	80
5.6	Wall y^+	82
5.7	Turbulence Intensity	84
6	FINAL REMARKS	85
6.1	Conclusions	85
6.2	Future Work	86
	Bibliography	88

LIST OF FIGURES

1.1	Experimental lift coefficient as a function of C_μ at different values of C_T	2
1.2	Experimental drag coefficient as a function of C_μ at different values of C_T	3
1.3	CCW diagram of a Buccaneer with the blowing slots on the leading edge. ¹	5
1.4	X-Wing ²	6
1.5	Comparison of experimental and Reynolds Average Navier Stokes (RANS) CFD lift performance data for a typical CC airfoil. ³	9
2.1	Airfoil with circulation control located at the leading and trailing edges ⁴	11
2.2	Coanda Effect ⁵	12
2.3	Trailing edge of a circulation control airfoil with high-pressure jet blown from a slot ⁴	13
2.4	Circulation control wing with a rounded trailing edge and an upper surface blowing engine configuration. ⁶	15
2.5	Circulation control wing with a dual radius flap and an over the wing engine configuration. ⁶	15
2.6	Rear view of the pneumatic wing, CCW flaps, and engine nozzles. ⁶ .	18
2.7	Front view of the Over the Wing engine simulator. ⁶	18
2.8	60° leading-edge Kueger flap and CCW trailing-edge flaps. ⁶	19
2.9	15° hood on the engine nacelle. ⁶	19
2.10	CCW/OTW Configuration A model installed in GTRI's MTF tunnel. ⁶	20
2.11	Engine nozzle ($x/C=0.75$, $z/D=0.23$ and flap (0° deflection) ⁶	20
2.12	CCW/OTW bottom view with flap at 0° . ⁶	20
2.13	Configuration B with engine exhaust at $x/C = 0.03$ and $z/D = 0.37$. ⁶	23
2.14	Configuration B with its different parameters.	23
2.15	2D CC Airfoil installed in GTRI Model Test Facility ⁷	24
3.1	Initial geometry in SolidWorks.	27
3.2	Cleaned up geometry in ICEM CFD.	27
3.3	Wind-tunnel model represented and parted out in ICEM CFD.	28
3.4	Unstructured triangular surface mesh on the trailing edge.	30
3.5	Unstructured triangular surface mesh on the trailing edge.	30

3.6	Generation of the first layer of triangular elements using the Advancing Front method. This method searches for the optimum point C with respect to edge AB that results in the highest quality element. ⁸	32
3.7	A Delaunay triangulation with their circumcircles.	33
3.8	Progression (from left to right) of the Octree (Robust) meshing method. ⁹	34
3.9	Near-body unstructured volume mesh using Octree.	35
3.10	Close up of the boundary layer mesh on the engine nacelle.	37
4.1	Subdivisions of the near-wall region ¹⁰ .	53
4.2	Near-wall treatments used in FLUENT ¹⁰ .	54
4.3	Velocity inlet and pressure outlet boundary conditions.	58
4.4	Two approaches to model the circulation control slot boundary.	61
5.1	Experimental lift coefficient as a function of C_μ at different values of C_T	66
5.2	Experimental drag coefficient as a function of C_μ at different values of C_T	66
5.3	Mesh refinement on Configuration B.	70
5.4	Convergence of lift coefficient towards the extrapolated solution as mesh increases proportionally in size.	71
5.5	Convergence of drag coefficient towards the extrapolated solution as mesh increases proportionally in size.	72
5.6	Lift coefficient as a function of C_μ with $C_T = 0$	74
5.7	Lift coefficient as a function of C_μ with $C_T = 2.1$	75
5.8	Drag coefficient as a function of C_T with $C_\mu = 0$	76
5.9	Drag coefficient as a function of C_T with $C_\mu = 0.5$	77
5.10	Lift coefficient as a function of C_μ , with and without wind tunnel, with $C_T = 2.1$, and with $k - \omega$ turbulence model	79
5.11	Mach streamlines plotted on the leading edge. Top: air flow is more uniform on the wing. Bottom: the interaction of the wing with the side walls causes less uniform flow near the wall	80
5.12	Velocity profile at CCW slot face	82
5.13	Contours of wall y^+ on the wind tunnel walls.	83
5.14	Contours of wall y^+ for Case 112 (engine off) and Case 212 (engine on).	83

LIST OF TABLES

2.1	Aerodynamic Configurations Evaluated.	21
4.1	CFD Operating Conditions.	57
4.2	Calculated engine mass flow rates.	60
4.3	Calculated CCW mass flow rates.	63
4.4	FLUENT Solver Settings.	64
5.1	Case listing: Wind tunnel included, CCW slot modeled as mass-flow inlet, $C_T = 0$	67
5.2	Case listing: Wind tunnel included, CCW slot modeled as mass-flow inlet, $C_T = 2.1$ ($\dot{m}_{engine}=0.3830$ kg/sec)	68
5.3	Case listing: Wind tunnel not included, CCW slot modeled as mass-flow inlet, $C_T = 2.1$ ($\dot{m}_{engine}=0.3830$ kg/sec)	68
5.4	Case listing: Wind tunnel included, CCW plenum modeled, $C_T = 2.1$ ($\dot{m}_{engine}=0.3830$ kg/sec)	69
5.5	Calculation of GCI and discretization error using Richardson's Extrapolation method.	71
5.6	Mesh Description	73
5.7	Comparison between calculated and targeted values of C_μ and C_T . . .	78
5.8	Case listing: Wind tunnel included, CCW plenum modeled, $C_T = 2.1$ ($\dot{m}_{engine}=0.3830$ kg/sec)	81

NOMENCLATURE

Roman Symbols

$\dot{m}$	Mass Flow Rate
A	Area
C_μ	Blowing Momentum Coefficient
C_D	3D Drag Coefficient
C_L	3D Lift Coefficient
C_l	2D Lift Coefficient
C_T	Thrust Coefficient
M	Mach Number
P	Pressure
q	Dynamic Pressure
R	Universal Gas Constant
S	Wing Reference Area
T	Temperature
U	Velocity
y^+	Dimensionless Wall Distance

Abbreviations

CCW	Circulation Control Wing
CFD	Computational Fluid Dynamics
GCI	Grid Convergence Index
OTW	Over The Wing

Greek Symbols

α	Angle of Attack
δ_{jet}	Thrust Deflection
γ	Ratio of Specific Heat
ρ	Density

Subscripts

∞	Freestream Conditions
eng	Engine Conditions
o	Total Conditions
$slot$	Slot Conditions

Chapter 1

INTRODUCTION

1.1 Objectives

The purpose of this study is to further investigate the ability of computational fluid dynamics to reproduce experimental data and use experimental data to validate current CFD methods. CFD can be a very powerful tool, saving a considerable amount of time and money, when it comes to solving fluid mechanics and heat transfer problems. However, CFD can also provide misleading and non-physical results if not applied correctly for each unique situation. Therefore, validation of CFD solvers and techniques is necessary for CFD to have real credibility in solving real world problems. CFD practices are often validated against empirical formulas from experimental results.

The CFD validation effort in this study used experimental data from a circulation control wing configuration designed and performed by Georgia Tech Research Institute. For as long as circulation control technology has been in development, countless of experimental data have been gathered to gain more understanding of its complexities and gain higher reliability in its use. By coupling circulation control wings with an over-the-wing propulsive system to further increase powered lift, the

complexity of the problem increases even more. Georgia Tech has gained a thorough understanding of the CCW/OTW coupling through wind tunnel experiments of their CCW configurations. One of their CCW configurations consists of a 2-D airfoil cross section coupled with a 3-D over-the-wing engine, which is the inspiration for this work.

In order to validate my CFD methods, experimental results from GTRI's Configuration B⁶ was used as the baseline data. A more detailed discussion about the wind tunnel model is presented in Chapter 2.5. The wind tunnel evaluation of Configuration B consisted of analyzing a sweep of blowing momentum coefficients, C_{μ} , for different values of engine thrust coefficient, C_T . Results from this particular test are shown in Figure 1.1 and Figure 1.2. The main goal was to match lift and drag coefficients for a given C_{μ} and C_T .

Figure 1.1 Experimental lift coefficient as a function of C_{μ} at different values of C_T

Figure 1.2 Experimental drag coefficient as a function of C_{μ} at different values of C_T

1.2 NASA Research Announcement

The scope of this study was inspired by a NASA Research Announcement project awarded to Cal Poly San Luis Obispo, along with Georgia Tech Research Institute and DHC Engineering. The goal of the project was to develop and validate predictive codes for a cruise efficient, short take-off and landing (CESTOL) subsonic aircraft and develop an open collection of experimental data for this type of aircraft. In the course of more than 3 years, the project went through several different phases in designing the aircraft. In each phase, a different methodology of the design process was more closely examined. During Phase 1, the focus was on selecting and further refining the CESTOL concept by choosing one design as well as completing a preliminary design for a large-scale wind tunnel model and test. During Phase 2, the chosen CESTOL design was analyzed computationally and experimentally until it was finalized. Anal-

ysis of the design included aerodynamic studies of the powered high lift technology to better understand the flow physics of the technology. The large-scale wind tunnel model was manufactured during Phase 2 and the wind tunnel experiments were planned out and scheduled at NASA Ames. The third phase of the project focused on completing the experiments and analyzing the results.

1.3 Background

1.3.1 Circulation Control

Powered high lift technology was incorporated in the NASA CESTOL design in the form of circulation control wings (CCW), also known as a blown flap, in order to achieve the short take off capabilities of the aircraft. Circulation control is a form of high lift device used on the wing of an aircraft to increase lift. Typically, it's used to improve low-speed lift during take off and landing. CCW works by blowing high velocity air over the the leading edge and/or trailing edge of the wing through a series of slots on the wing surface. The wing typically has a rounded trailing edge to deflect the air downward by way of the Coanda effect, thus creating lift. The Coanda effect is the tendency of a fluid jet to attach to a curved surface. Lift is increased not only by way of the downward deflection of the flow due to the Coanda effect, but also by the conventional lift production caused by the higher velocity air on the top surface of the wing. The higher pressure gradient between the top and bottom surfaces of the wing provides an additional increase in lift. Examples of some aircraft that utilizes circulation control technology are the X-Wing and and Blackburn Buccaneer.

The Buccaneer, shown in Figure 1.3, is a British strike aircraft which served the Royal Navy. Its small wing was capable of high speed at low altitudes, however, it was not sufficient to generate the lift necessary for aircraft carrier landings and take-offs. Compressed gas from the engine was "blown" through surface vents of the horizontal stabilizers and the wing, also known as Boundary Layer Control. Blowing compressed air on the surfaces of the wing and stabilizers smoothed and reattached the boundary layer, which reduced airflow separation at the back of the wing, increasing the effectiveness of the trailing edge control surfaces to generate more lift.

Figure 1.3 CCW diagram of a Buccaneer with the blowing slots on the leading edge.¹

The X-Wing, shown in Figure 1.4 was an experimental hybrid helicopter and fixed wing aircraft. Conventional helicopters control lift by controlling the twist of the blades, however, the X-Wing also used compressed air bled from the engine to blow on the surfaces of its blades, increasing the virtual surface areas of the blade thus creating lift.

Figure 1.4 X-Wing²

1.3.2 Computational Fluid Dynamics

Computational fluid dynamics is the study of fluid flows and heat transfer using numerical methods and algorithms. Fluid dynamics is encountered in many areas of industry. For example, the automotive sector relies heavily on fluid dynamics and heat transfer when designing their automobiles, from the car's cooling system to its external aerodynamics. Understanding fluid dynamics and heat transfer is important in improving the overall design of the car. CFD allows their engineers to build a prototype of a design they wish to analyze and obtain data, allowing them to predict the performance of their design.

It wasn't until the onset of computers has CFD become such a powerful tool in solving fluid mechanics problems. Commercial CFD codes became available and started to be accepted by major companies as recently as the 1980's. Although

far from being perfect, the high-speed super computers of today can provide better solutions and can be achieved at a much faster rate. The use of CFD to solve fluid mechanics problems can be a very advantageous alternative to a more time consuming and very expensive wind tunnel experiments.

CFD played an important role during the design process of the NRA project. During Phase 1, it was used to determine which hybrid wing body design was to be chosen. Cal Poly was responsible for evaluating available CFD tools in order to model the complicated flow field associated with the coupling of the propulsion system and the aerodynamics of the aircraft. Since the large-scale wind tunnel test was still in its preliminary stages, representative configurations were analyzed using CFD to further refine the computational tools that will be used in the later stages of the design. CFD was used on similar configurations to develop techniques to model the propulsion system that would be used on the large-scale wind tunnel model. It was also used to investigate different turbulence models and determine any necessary modifications to improve the models for the specific flow field of the project.

1.3.3 CFD Studies on 2D Circulation Control

NASA and the Air Force Research Laboratory have showed a lot of interest in further investigating circulation control technology as part of the Cruise Efficient Short Take Off and Landing initiative¹¹. They've expressed the need to improve powered-lift systems to meet take-off and landing goals. Extensive research and experiments are necessary in order to prove if integration of the fairly new circulation control

technology is worth the significant modifications needed to be done on an existing aircraft.

Computational fluid dynamics has been in the forefront of the development of circulation control technology. The added complexities in the flow provided by circulation control make it even more difficult for empirical methods to provide validation. By coupling empirical methods with CFD, engineers are able to develop more accurate trade studies in designing wing geometries and blowing systems.

To further drive this effort, NASA has held Circulation Control workshops in order to develop a detailed experimental database that can be used by the public for CFD validation. Results from the 2004 NASA/ONR Circulation Control (CC) workshop^{12,3} have shown that when applied to CC airfoils, CFD does a poor and inconsistent job in predicting some of the capabilities of circulation control. An example is shown in Figure 1.5, where CFD over-predicts airfoil lift coefficient, C_l , as the blowing momentum coefficient, C_{μ} , is increased. The cause of the problem for this particular example was linked to turbulence models and grid issues. As was done at the Circulation Control workshop, this study attempted to validate circulation control experimental data by applying CFD tools to 3-dimensional circulation control wing configurations.

Figure 1.5 Comparison of experimental and Reynolds Average Navier Stokes (RANS) CFD lift performance data for a typical CC airfoil.³

Chapter 2

GEOMETRY DESCRIPTION

2.1 Wind Tunnel Experiments

The NASA Research Announcement project awarded to Cal Poly, along with Georgia Tech Research Institute and DHC Engineering, allowed for the in-depth investigation into the design of a cruise efficient, short take-off and landing (CESTOL) subsonic aircraft. The goal of the NRA project was to develop and validate the predictive capabilities for the design and performance of this particular aircraft. To achieve its short, take-off and landing capabilities, integration of an advanced high lift device on the aircraft was necessary. With Georgia Tech's experience in circulation control technology, this was the high lift device of choice. From the preliminary to the final design of a large-scale wind tunnel model, experiments performed by Georgia Tech on circulation control technology were integral in every phases of the project. The extensive amount of experimental data that Georgia Tech had generated for different wind tunnel configurations that employ circulation control allowed the CFD team at Cal Poly to validate their methods.

2.2 Circulation Control Wings

A circulation control wing is a form of high-lift device that has been in development phase for over sixty years¹³. It is used on the main wing of an aircraft to increase lift, normally during take-off and landing. The physics behind circulation control technology is fairly straightforward. Circulation control wings increase lift by increasing the velocity of the airflow over the leading and trailing edge. By blowing high-pressure jet air through a series of slots on the aircraft wing, the velocity of the airflow over the leading and trailing edge increases. Figure 2.1 shows an example of circulation control airfoil and the location of the possible slots. By blowing high velocity air from the leading edge, the velocity of the airflow downstream of the leading edge increases, thus generating lift through conventional airfoil lift production. The source of this high-pressure jet air typically comes from, but not restricted to, the engine bleed air. Although the increased air velocity over the wing increases the lift through conventional lift production, this isn't the only way lift is increased when using circulation control wings. Circulation control wings typically have rounded trailing edges to redirect the high-pressured air along the trailing edge by way of the Coanda effect.

Figure 2.1 Airfoil with circulation control located at the leading and trailing edges⁴

2.3 Coanda Effect

In its simplest definition, the Coanda effect is the tendency of a fluid jet to attach to a nearby surface. The fluid jet attaches itself to the surface and will remain attached even if the surface curves away from the initial jet direction. Figure 2.2 shows a simple demonstration of the Coanda effect⁵.

Figure 2.2 Coanda Effect⁵

As the curved surface gently approaches the stream of water, the water follows the surface of the curved shape. The stream of water stays attached even though the surface curves away from the initial direction of the stream. On a circulation control wing, the rounded trailing edge tangentially redirects the flow downward, thus causing a further increase in lift as shown by Figure 2.3.

Figure 2.3 Trailing edge of a circulation control airfoil with high-pressure jet blown from a slot⁴

2.4 Blowing Parameter, C_μ

An important parameter associated with circulation control is its blowing momentum coefficient, as defined by C_μ as

$$C_\mu = \frac{\dot{m}_{slot} U_{slot}}{q_{slot} S} \quad (2.1)$$

where $\dot{m}_{slot}$ is the mass flow through the slot, U_{slot} is the jet velocity at the slot, and S is the wing reference area. The momentum term $\dot{m}_{slot} U_{slot}$ can also be thought of as the jet thrust provided by the circulation control. The mass flow is usually measured experimentally using appropriate flow meters but can also be calculated isentropically using compressible flow relationships. For 3-dimensional compressible flow, C_μ can be defined as

$$C_\mu = \frac{2\rho_{slot} A_{slot} U_{slot}^2}{\rho_\infty U_\infty^2 S} \quad (2.2)$$

where the subscript “slot” implies condition at the circulation control slot and A is the area of the slot.

Typically, the jet velocity is calculated from isentropic relations as

$$U_{slot}^2 = \frac{2\gamma RT_{o,slot}}{\gamma - 1} \left[1 - \frac{P_{\infty}}{P_{o,slot}}^{\frac{\gamma-1}{\gamma}} \right] \quad (2.3)$$

where the subscript “o” implies the total temperature and pressure in the plenum duct, the subscript “ ∞ ” implies freestream conditions, R is the gas constant, and $\gamma=1.4$ for air.

2.5 GTRI Configurations

Extensive research in powered, high-lift systems by Georgia Tech Research Institute have shown that the combination of a circulation control wing combined with either an Upper Surface Blowing or Over the Wing propulsion system can generate very high lift at low blowing parameters^{4,14}. Examples of these configurations are shown in Figures 2.4- 2.5.

Figure 2.4 Circulation control wing with a rounded trailing edge and an upper surface blowing engine configuration.⁶

Figure 2.5 Circulation control wing with a dual radius flap and an over the wing engine configuration.⁶

As previously mentioned, the entrainment of the CCW flow can highly augment the wing lift. However, when combined with an engine mounted above the wing, either USB or OTW, the CCW flow deflects the engine thrust for even greater lift. The entrainment of thrust onto the curved surface of the CCW causes an increase in velocity over the flap surface. This is where most of the lift contributed by the

engine thrust comes from. However, the engine thrust also contributes to the lift generated through the deflected thrust coefficient component, $C_T \sin(\alpha + \delta_{jet})$. As the component of the thrust pointing opposite of lift increases, the reactant force acts in the lift direction, causing additional increase in lift. Associated with the deflection of the thrust are not only an increase in lift, but also an increase in drag. The ability for the system to deflect the thrust for greater drag is ideal for approach. This is accomplished by optimizing the geometry of the flap to have a greater flap angle on a very small flap. However, the system must also be able to recover thrust during short take-offs and steep climb. This is accomplished by varying the blowing momentum coefficient of the circulation control wing and the thrust coefficient of the OTW/USB engine, C_T . The concern lies with whether the small and high angled flap that's ideal for thrust turning can retain its ability to deflect the high velocity exhaust. Optimizing both the geometric and aerodynamic performance of the CCW/OTW system is important in achieving a successful STOL system. GTRI have accomplished this through their low-speed experimental efforts.

Georgia Tech Research Institute has conducted experimental evaluations on pneumatic powered circulation control wings with engines located over the wing to improve powered high-lift and cruise performance of a cruise efficient short take-off and landing (CESTOL) aircraft configuration. The integration of these systems provides very high lift for short take-off and landing, low thrust recovery for approach, and high thrust recovery for take-off and climb while minimizing mechanical complexity. The purpose

of these evaluations was to develop the most effective pneumatic aerodynamic and propulsive geometries with minimal noise production. The aerodynamic wind tunnel and acoustic models that were evaluated are shown in Figures 2.6 - 2.8. High-pressure air was supplied to the model to power its engine simulator, while low-pressure air was used to power the CCW flap system. To perform the necessary aerodynamic, propulsive, and acoustic tests, different parameters of the model were varied. Varying the engine's horizontal location, x/C , where x is x-location of the engine and C is the chord of the wing, as well as its vertical location, z/D , where z is the z-location of the engine and D is its exhaust diameter, were possible. The wing has a chord length of $C = 8.6$ inches (with flap at 0 degree deflection) and the nacelle exhaust has a diameter of $D = 1.63$ inches. Different hood angles were possible by installing an engine hood, shown in Figure 2.9 to mechanically deflect the engine thrust downward. The airfoil wing is shown in Figure 2.8 along with its different trailing edges deflected at different angles and a 60° leading-edge Krueger flap.

Figure 2.6 Rear view of the pneumatic wing, CCW flaps, and engine nozzles.⁶

Figure 2.7 Front view of the Over the Wing engine simulator.⁶

Figure 2.8 60° leading-edge Kueger flap and CCW trailing-edge flaps.⁶

Figure 2.9 15° hood on the engine nacelle.⁶

Initial aerodynamic and propulsive testing of each powered-lift configuration was performed in GTRI's Model Test Facility (MTF) while acoustic testing was performed in GTRI's Acoustics Flight Simulation Facility. Figures 2.10- 2.12 shows a typical configuration installed in the MTF's six component floor balance. The wing of the model extends from the floor to the ceiling to simulate a 2D airfoil and eliminate wing tip effects while still having effects of a 3D engine. These figures show the engine nacelle located at $x/C = 0.75$, with 0° flap deflection, no leading edge device, and no hood angle. The configuration that exhibits the best aerodynamic, propulsive, and acoustic performance was to be incorporated with a large scale, 3D powered-lift,

and OTW configuration and was to be tested in a large-scale wind tunnel.

Figure 2.10 CCW/OTW Configuration A model installed in GTRI's MTF tunnel.⁶

Figure 2.11 Engine nozzle ($x/C=0.75$, $z/D=0.23$ and flap (0° deflection))⁶

Figure 2.12 CCW/OTW bottom view with flap at 0° .⁶

2.5.1 Test Configurations

With all the different CCW/OTW parameters that can be varied, GTRI investigated a total of 37 configurations in order to evaluate the aerodynamic and propulsive performance of the CCW system. These configurations were varied geometrically by varying engine locations, leading edge and trailing edge flap angles, slot heights,

and engine hood angles. The configurations also included variations in momentum blowing coefficient (C_μ) at a constant thrust coefficient (C_T), variations in C_T at a constant C_μ , as well as variations in angles of attack, α . Table 2.1 shows 12 out of the 37 configurations that were tested by GTRI. The configurations shown in Table 2.1 represent a typical low drag take-off configuration, which meant that there were no leading-edge devices and the CCW flap had a 0° deflection. The main purpose for testing this group of configurations was to determine the effects of engine location relative to the wing surface. For each engine location, the effects of varying thrust coefficient (C_T) and blowing momentum coefficient (C_μ) were also determined. These configurations influenced the use of CFD, thus this thesis, to replicate the results that GTRI obtained from their testing. Configuration B in particular was used as the baseline model for validating the CFD methods discussed throughout this study.

Table 2.1 Aerodynamic Configurations Evaluated.

Configuration	$\delta_{flap}, ^\circ$	$\delta_{LE}, ^\circ$	x/C	z/D	$\delta_{hood}, ^\circ$	h_{CCW}, in
A	0	0	0.75	0.23	off	0.02
B	0	0	0.03	0.37	off	0.02
C	0	0	0.03	0.37	15	0.02
D	0	0	0.03	0.37	30	0.02
E	0	0	0.03	1.20	off	0.02
F	0	0	0.03	1.20	30	0.02
G	0	0	0.75	1.20	off	0.02
H	0	0	0.75	1.20	30	0.02
I	0	0	0.25	1.20	off	0.02
J	0	0	0.25	1.20	30	0.02
K	0	0	0.25	0.31	off	0.02
L	0	0	0.25	0.31	30	0.02

2.5.2 Configuration B

Configuration B, shown in Figure 2.13 as installed in the wind tunnel, has the engine located higher off the upper surface of the wing and more forward as compared to Configuration A. The purpose of this configuration was to determine if the additional distance of the engine from the wing upper surface would still increase powered lift. This thesis focuses on CFD's ability to replicate and validate GTRI's experimental results. As previously mentioned, Configuration B represents a typical low drag-take off configuration, which meant there were no leading edge device, 0° CCW trailing edge flap, and 0° no hood angles. This geometry was chosen out of all of GTRI's 37 configurations simply to perfect the CFD methods used to replicate the experimental data. The high velocity exhaust of the engine and the CCW can offer a lot of instability in the numerical solution just on their own. Simplifying the rest of the flow field by eliminating leading edge devices and trailing edge flap angles allowed the focus to be more directed at the physics of circulation control. A summary of Configuration B's different parameters is shown in Figure 2.14.

Figure 2.13 Configuration B with engine exhaust at $x/C = 0.03$ and $z/D = 0.37$.⁶

Configuration B: $\frac{x}{C} = 0.03$, $\frac{x_{te}^*}{D} = 5.25$, $\frac{z}{D} = 0.37$

Figure 2.14 Configuration B with its different parameters.

2.6 Wind Tunnel

In order to accurately reproduce the experimental results gathered by GTRI, the wind tunnel geometry was also included in the CFD model. Aerodynamic and

propulsive experiments were performed in GTRI's Model Test Facility (MTF) wind tunnel as shown in Figure 2.15. For this particular experiment, the 6-component strain gage balance and an angle-of-attack turn table is visible on the floor, as well as the air lines that feed the circulation control blowing slots and engine simulator. The wall boundary layer near the test section floor was eliminated by use of tangential floor blowing. The actual test section measures 30" x 43" x 90"⁷.

Figure 2.15 2D CC Airfoil installed in GTRI Model Test Facility⁷.

Chapter 3

MESH GENERATION

3.1 Gridding Techniques

Generating a high-quality surface mesh was important in the pre-processing of the CFD geometry. Not only did the mesh had to be well defined in regions where complex flows were expected, having a 3-dimensional model made it even more difficult and time consuming to create a good mesh. The intricate shapes of the 3D model needed to be accurately defined by the surface mesh, while the high velocity flows from the circulation control slot must be captured by the volume mesh in order to reduce instability in the flow caused by the mesh. This made generating the 3D mesh a more difficult and time consuming task.

All of the meshes for this work were created using ANSYS ICEM CFD. ICEM CFD is a proprietary software package used for CAD and mesh generation. It has the ability to generate structured, unstructured, multi-block, and hybrid grids with different cell geometries. A CAD model of the wind tunnel configuration and the wind tunnel itself were created using Solidworks. This section will explore the method in which the meshes were created.

3.1.1 Geometry/CAD

Georgia Tech Research Institute provided the CAD models for the wind tunnel configurations. Although these CAD models were great for manufacturing of the actual wind tunnel models, they were unsatisfactory for CFD analysis. The CAD model, as shown in Figure 3.1, contained everything that the real wind tunnel model had, including holes on the surfaces where the screws were placed and all the internal supports and mechanisms. The inner geometries were deleted since the CFD analysis was only focusing on the external flow of the model and the holes for the screws were smoothed out to make the surfaces much easier to mesh. It was important to simplify the geometry enough to make it more suitable for meshing, but not to the extent where the defining features of the geometry was taken away. The initial CAD geometry was ‘cleaned’ up using ICEM and only the external components of the model were kept, which resulted in the final ICEM model shown in Figure 3.2.

Figure 3.1 Initial geometry in SolidWorks.

Figure 3.2 Cleaned up geometry in ICEM CFD.

Next, the different parts of the geometry were grouped together to make it simpler to assign different mesh sizes on different surfaces of the model. This allowed for more direct control in defining different surfaces of the geometry with different cell sizes. The more complicated features of the geometry, such as the leading and trailing edges and the engine components, needed smaller cells to accurately capture those features. However, the less complicated features, such as the large surfaces on the wing, was defined by using larger cells to represent those surfaces. It was important to develop

an intuitive approach on how to ‘part’ out different geometries. ‘Parting’ out the surfaces lead to higher quality meshes and lower cell counts. Figure 3.3 shows the model ‘parted’ out in ICEM, as represented by the different surface colors. Each surface with a different color was then assigned it’s own cell size independently from the rest.

Figure 3.3 Wind-tunnel model represented and parted out in ICEM CFD.

3.1.2 Surface Mesh

Generating a high quality surface mesh on the model was very important when considering the overall quality of the mesh and the accuracy of the solution. The quality of the mesh and accuracy of the solution could propagate from the unstable flow regime near and on the model to the rest of the flow field. Therefore, it was important to choose a mesh that was sufficiently fine to provide an accurate solution and definition of the geometry features of the model. With the application of circulation control combined with an over the wing engine, the model was expected to have very large temperature and pressure gradients. Generating the type of mesh that could capture these instabilities began with choosing and generating a good quality

surface mesh. Two types of surface meshes are available in ICEM, structured and unstructured, with advantages and disadvantages for each type. For the unstructured surface mesh, octagonal, triangular, and quadrilateral meshes were also available in ICEM.

The structured surface mesh, generated with quadrilateral elements, is good at drastically reducing the total number of cells and reducing numerical errors in the solution. However, generating it for a complicated 3D geometry is very difficult and time consuming, especially when using ICEM. Using a structured mesh means that all of the ‘parted out’ surfaces of the geometry must be manually mapped. With over 20 ‘parted out’ surfaces that defined the geometry, mapping each one can be very time consuming. When mapping the surfaces with a specified number of nodes and spacing, each surface will then be ‘linked’ or associated with the other surfaces surrounding it. This makes it hard to directly control the amount of skewness and refinement for each surface because the mapping of each surfaces essentially depends on the mapping of the surfaces around it. For these reasons, the structured surface mesh was not used to define the geometry.

The unstructured triangular surface mesh is great at handling complex 3D geometries. It’s robust enough that it can mesh difficult areas of the geometry in a short amount of time. Generating it on any individual surface is quick, allowing for more time to be spent on improving the quality of the entire surface mesh for a more accurate solution. Having a triangular surface mesh is ideal for generating an un-

structured tetrahedral volume mesh, which is the more common type of unstructured volume mesh used. For these reasons, an unstructured triangular surface mesh was chosen to define the geometry. The resulting surface mesh can be seen in Figures 3.4 and 3.5.

Figure 3.4 Unstructured triangular surface mesh on the trailing edge.

Figure 3.5 Unstructured triangular surface mesh on the trailing edge.

3.1.3 Volume Mesh

A fully structured or a fully unstructured mesh is available in ICEM and there are advantages and disadvantages for picking either mesh. Since the geometry was defined using an unstructured surface mesh, an unstructured tetrahedral mesh was used for the volume.

Choosing a fully structured mesh to capture the flow field is more convenient computationally because the elements are mapped in a structured manner. It results in a lower total cell count, which means the solver can generate solutions much simpler and faster. However, a structured volume mesh near the model would also require a structured surface mesh. A structured surface mesh is much more complicated to generate than the automatic built-in functions for unstructured meshes. Because of the complicated geometry of the model, cell orthogonality on the surface would become much harder to maintain.

Just like it's unstructured surface mesh, the unstructured volume mesh is much simpler to generate due to ICEM's built in algorithms for unstructured meshing. The tetrahedral elements of this mesh are more suitable for representing the complicated geometry of the model. Since there were already triangular elements on the surface of the geometry, an unstructured volume mesh was chosen. Once the cell sizes had been determined for the surfaces, the mesh was then generated using those predetermined sizes. ICEM has a few different volume meshing algorithms to generate this tetrahedral mesh: Octree method, Delaunay method, and Advancing Front.

The Advancing Front uses a bottom-up approach in meshing, in which it grows the volume elements from an existing surface mesh. It generates the volume mesh from its boundaries which means it requires an existing triangular surface mesh to create a tetrahedral volume mesh⁸. The existing elements on the boundaries are used as the initial front of the mesh, using its parameters to grow the next layer of elements. The parameters include element size, element stretching and stretching direction. It repeats this process for each layer until the volume is fully meshed. It has been shown to be very effective in generating 2-dimensional unstructured meshes; however, can be very time consuming and computationally expensive when used in 3-dimensions¹⁵. Figure 3.6 shows the first cell being generated on the first front, which is defined by boundary AB, by using the Advancing Front method.

Figure 3.6 Generation of the first layer of triangular elements using the Advancing Front method. This method searches for the optimum point C with respect to edge AB that results in the highest quality element.⁸

The Delaunay (Quick) meshing method generates a tetrahedral mesh using a bottom-up approach, in which it grows the tetrahedral elements from an existing surface mesh outward into the rest of the domain. This requires an existing mesh to be generated or loaded into ICEM. Once a surface mesh has been defined, the mesh is created based upon the Delaunay triangulation. This method ensures that for a given set of points on the surface, none of those points are contained inside the circumcircle of any triangle^{16 17}, as shown in Figure 3.7

Figure 3.7 A Delaunay triangulation with their circumcircles.

The Octree (Robust) meshing method generates a tetrahedral volume mesh using a top-down meshing approach. This means that the mesh starts away from the surfaces and grows until it reaches the defining surfaces and curves of the geometry. The mesher begins by enclosing the entire geometry with a full tetrahedral mesh. The mesher then conforms the mesh, which guarantees that each pair of adjacent cells share an entire face. To match the given geometry, the mesher adjusts the nodes

to match the specified points, curves, and surfaces of the geometry. The elements contained within the geometry and not touching any user-defined material point, such as any specified volume, are then cut and deleted^{18,19}. Figure 3.8 shows the progression of how the Octree method generates the tetrahedral mesh. Depending on the smoothing options, the final mesh is then smoothed by moving and merging nodes, swapping edges, and deleting bad elements. The result is an unstructured volume and surface mesh.

Figure 3.8 Progression (from left to right) of the Octree (Robust) meshing method.⁹

ICEM uses the Octree method to generate the unstructured triangular mesh on the surface. During surface mesh generation, the entire volume is meshed first and then deleted, leaving only the mesh created on the surface. The Octree method was used to generate the volume mesh since ICEM already used this method to generate the surface mesh. The final unstructured volume mesh near the model using the Octree method is shown in Figure 3.9. The boundary layer mesh has not been generated on the surfaces of the geometry as it requires an existing volume mesh.

Figure 3.9 Near-body unstructured volume mesh using Octree.

3.2 Boundary Layer Mesh

Generating a boundary layer mesh on the surfaces of the geometry was important in order to capture the near-wall flow features of the model as the unstructured tetrahedral mesh was inadequate for capturing these features. A prismatic boundary layer mesh was generated on the surfaces to efficiently capture the effects of boundary layer while still maintaining the benefits of a tetrahedral mesh.

The primary concern in growing the prism mesh was specifying its initial height to be able to capture a certain y^+ , the dimensionless wall distance, which is commonly used in boundary layer theory and in defining the law the wall. The law of the wall states that the average velocity of a turbulent flow is proportional to the logarithmic distance away from wall. Depending on the value of y^+ , the average velocity can be approximated with different methods. The value of the y^+ that is appropriate highly depends on the turbulence model to be used. Once an appropriate y^+ have been

chosen with the turbulence model in mind, the initial height of the prism layer mesh can then be calculated.

A mesh with $y^+ > 60$ would result in having the first layer of the prism to fall within the turbulent log-law region of the boundary layer. Having at least one prism element within the turbulent log-law region would require the flow solver to use semi-empirical formulas, or wall functions, to interpolate the solution variables on the near-wall cells. A $y^+ < 5$ would result in finer prism cells that fall within the viscous sublayer region of the boundary layer. Refining the region near the walls with more prism elements, also known as near-wall treatment, eliminates the need to use wall-functions because the mesh is fine enough to be able to resolve all the way to the laminar sublayer. Typically, prisms with an initial height that falls within the viscous sublayer, with a $y^+ = 1$, is desired. Near-wall treatments are further explained in Section 4.2.6.

ICEM CFD provides many options to help generate the highest quality prism mesh. The quality of the mesh is defined by aspect ratio and skewness. Prisms with large aspect ratios must always be avoided as they degrade the accuracy and stability of the solution. Although skewness will always be present as a characteristic of the prism elements, limiting the amount of prism elements below a certain skewness quality is important. To ensure a good transition from the surface mesh to the boundary layer mesh, each prism layer was generated with a 40 % volume growth rate from the previous layer. Also, to ensure a good transition from the boundary

layer mesh to the volume mesh, the last prism element was generated to have a 1:1 volume ratio to the adjacent tetrahedral element. A visual check of the boundary layer mesh is necessary to ensure that there is a good transition throughout the rest of the geometry. Figure 3.10 shows a typical boundary layer mesh transitioning to the tetrahedral volume mesh.

Figure 3.10 Close up of the boundary layer mesh on the engine nacelle.

3.3 Grid Refinement

As stated previously throughout this thesis, there are many factors that can lead to unreliable and misleading CFD results. One of those factors that can greatly affect the CFD solution is mesh refinement. Findings from the NASA Circulation Control workshops even suggest that one of the problems in the solutions were attributed to meshing issues¹²³. It's important to generate a sufficiently refined mesh in order to capture all the underlying flow features that will be experienced. This can be done by performing a grid independence study.

3.3.1 Richardson's Extrapolation Method

Celik²⁰ recommends using Richardson's Extrapolation (RE) method for estimating the discretization error of CFD simulations. It's known to be the most reliable method available for predicting numerical uncertainty in the solution. Celik then recommends the Grid Convergence Index (GCI) method when performing a grid independence study. The GCI method, which is based on the RE method, is the accepted and recommended method as it has been used on hundreds of CFD cases. In short, the GCI method predicts a physical result by reproducing an "exact" solution when a sufficiently fine grid is used²⁰. The following is the procedure in calculating the discretization error of the CFD model.

First, the representative mesh size, h , was defined. For 3D meshes, h is defined as

$$h = \left(\frac{1}{N} \sum_{i=1}^N \Delta V_i \right)^{1/2} \quad (3.1)$$

where ΔV is the volume of the i^{th} cell and N is the total number of cells. As a measure of refinement from one grid to the next, the grid refinement factor, r , was calculated using

$$r = \frac{h_{coarse}}{h_{fine}} \quad (3.2)$$

The apparent order, n , of the method was then calculated using the expressions

$$n = \frac{1}{\ln r_{21}} |\ln |\epsilon_{32}/\epsilon_{21}| + q(n)| \quad (3.3)$$

$$q(n) = \ln \left(\frac{r_{21}^n - s}{e_{32}^n - s} \right) \quad (3.4)$$

$$s = 1 \cdot \text{sign}(\epsilon_{32}/\epsilon_{21}) \quad (3.5)$$

where $\epsilon_{32} = \sigma_3 - \sigma_2$, $\epsilon_{21} = \sigma_2 - \sigma_1$ and σ is the variable important to the object of the simulation of the study. For this validation, σ was chosen to be C_L of the CCW geometry. Richardson's extrapolation was then used to extrapolate the solution for an infinitely large mesh.

The extrapolated solution was found from the expression

$$\sigma_\infty = \frac{r_{23}^n \sigma_3 - \sigma_2}{r_{23}^n - 1} \quad (3.6)$$

and the approximate and extrapolated relative errors were then found using

$$e_{app} = \left| \frac{\sigma_3 - \sigma_2}{\sigma_3} \right| \quad (3.7)$$

$$e_\infty = \left| \frac{\sigma_3 - \sigma_2}{r_{23}^n \sigma_3 - \sigma_2} \right| \quad (3.8)$$

Finally, the grid convergence index was calculated using

$$GCI = \frac{1.25 e_{app}}{r_{23}^n - 1} \quad (3.9)$$

Chapter 4

SIMULATION SETUP

ANSYS FLUENT 6.3 was used to solve the Reynolds-averaged Navier-Stokes (RANS) equations. For turbulent flows, the system of governing equations was solved using turbulence equations. FLUENT is a finite volume numerical solver that can be used for both structured and unstructured meshes. Pressure-based segregated and density-based coupled solvers are both available in FLUENT.

4.1 Governing Equations

FLUENT's governing equations are briefly covered in this section as described in the FLUENT User Manual¹⁰. Full derivations of the governing equations are described by Tannehill et. al²¹.

4.1.1 Continuity Equation

The conservation of mass, or continuity, is solved using the continuity equation

$$\frac{\partial \rho}{\partial t} + \rho \nabla \cdot \vec{U} = 0 \quad (4.1)$$

where $\vec{U}$ is the the velocity vector. This most general form of the continuity equation can be used for both incompressible and compressible flows. When a mass is added

to or taken away from the system, the source S_m is added to the right hand side of the continuity equation, as shown in the following equation

$$\frac{\partial \rho}{\partial t} + \rho \nabla \cdot \vec{U} = S_m \quad (4.2)$$

where S_m is the additional source term. This source term accounts for any user-defined sources. For example, a source term accounting for the vaporization of liquid droplets in the system.

4.1.2 Momentum Equation

The conservation of momentum is solved using the momentum equation. For an inertial reference frame, the momentum equation in it's general form is written as

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \quad (4.3)$$

where p is the static pressure, $\bar{\bar{\tau}}$ is the stress tensor, $\rho \vec{g}$ is the gravitational force on the body, and $\vec{F}$ is any external body forces. The stress tensor, $\rho \vec{g}$, is defined as

$$\bar{\bar{\tau}} = \mu \left[(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I \right] \quad (4.4)$$

where μ is the molecular viscosity and I is the unit tensor.

4.1.3 Energy Equation

The energy equation is expressed as

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot [\vec{v}(\rho E + p)] = \nabla \cdot \left[k_{eff} \nabla T - \sum h_j \vec{J}_j + (\bar{\tau}_{eff} \cdot \vec{v}) \right] + S_h \quad (4.5)$$

- The term $k_{eff} \nabla T$ is the energy transfer due to conduction.
- $\sum h_j \vec{J}_j$ is known as the species diffusion.
- $\bar{\tau}_{eff} \cdot \vec{v}$ is known as the viscous dissipation.
- S_h is the heat source term which contains contributions from radiation, chemical reactions, as well as any other volumetric heat sources.

- $\vec{J}_j$ is the diffusion flux
- k_{eff} is the effective thermal conductivity which equals to

$$k_{eff} = k + k_t \quad (4.6)$$

and k_t is the thermal conductivity from the turbulence model.

- The energy, E , can be written as

$$E = h - \frac{p}{\rho} + \frac{v^2}{2} \quad (4.7)$$

where h is the sensible ethalpy defined as

$$h = \sum Y_j h_j \quad (4.8)$$

where Y_j is the mass fraction of species j and h_j is defined as

$$h_j = \int_{T_{ref}}^T c_{p,j} dT \quad (4.9)$$

where T_{ref} is 288.15K.

4.2 Turbulence Modelling

A 2D CCW airfoil study was performed by Storm and Marshall²² to determine the turbulence model that was best suited for circulation control. The study was to determine which of FLUENT's existing turbulence models, Spalart-Allmaras, $k - \epsilon$, or $k - \omega$, performed best for this type of simulation. It was found that the $k - \epsilon$ turbulence model was the most accurate in capturing turbulence features of CCW airfoils. However, the general trend of two-equation turbulence models when applied to 2D circulation control applications is to over-predict C_L as C_μ increases. Storm²³ also developed a modified $v2 - f$ turbulence model for use with FLUENT. The $v2 - f$ turbulence model²⁴ is known to capture the non-linear eddy viscosity effects and streamline curvature effects of circulation control flows. Results indicate that the $v2 - f$ turbulence model generated lift coefficients within approximately 20% of experimental values. Results also show that FLUENT's Spalart-Allmaras, $k - \epsilon$, and $k - \omega$ produce errors as high as 85%, 36%, and 39%, respectively. However, this turbulence model was not investigated for this work. The reader can refer to Storm²³ for further details of the study.

A turbulence model study was performed to determine which of FLUENT's built in turbulence models was best suited for 3D circulation control applications. The turbulence models investigated for this work were the standard $k - \epsilon$ model, the realizable $k - \epsilon$, the RNG $k - \epsilon$, the standard $k - \omega$ model and the shear stress transport $k - \omega$ model. Full derivations for each of these models are not presented

for this work; however, their equations are briefly covered in this section as described in the FLUENT User Manual¹⁰

4.2.1 Standard $k - \epsilon$ Model

The standard $k - \epsilon$ model²⁵ is the most widely used model and is well known for its robustness and stability in the solution. It is known to be accurate and stable for a large range of flow fields. It's good for wall bounded, internal flows as long as there are no severe pressure gradients in the flow. The assumptions of fully turbulent flow and negligible molecular viscosity effects were made in deriving the $k - \epsilon$ model. For this reason, the $k - \epsilon$ model is valid only for turbulent flows. Its accuracy goes down for more complex flows that involve large adverse pressure gradients, separation, and strong streamline curvatures^{26,27}. Despite these limitations, it still continues to be the most commonly used turbulence model.

For the standard $k - \epsilon$ model, the turbulence kinetic energy, k , and its dissipation rate, ϵ , are derived from the following transport equations.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial t}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (4.10)$$

and

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (4.11)$$

where G_k is the turbulence kinetic energy from the mean velocity gradients, G_b is the turbulence kinetic energy due to buoyancy, and Y_M is the contribution of the

fluctuating dilatation in compressible turbulence to the overall dissipation rate. σ_k and σ_ϵ are the turbulent Prandtl numbers for k and ϵ respectively.

The turbulent viscosity, μ_t , is defined as

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (4.12)$$

where C_μ is a constant.

In FLUENT, the default values for the constants $C_{1\epsilon}$, $C_{2\epsilon}$, C_μ , σ_k , and σ_ϵ are given as the following²⁵

$$C_{1\epsilon} = 1.44, C_{2\epsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\epsilon = 1.3$$

These constant values have been determined from experimental data and have been found to work very well with wall-bounded and free shear flows.

4.2.2 Realizable $k - \epsilon$ Model

The realizable $k - \epsilon$ model²⁸ differs from the standard model such that it uses a different formulations for the turbulent viscosity and for the dissipation rate, ϵ . The realizable $k - \epsilon$ model is more accurate at predicting the spreading rate of jets and its more superior with handling flows involving rotation, boundary layers under strong adverse pressure gradients, separation and recirculation compared to the standard model. Although the this model is fairly new in development compared to the standard model, it's been known to provide the best performance of all the $k - \epsilon$ models for validating separated flow and flows with complex secondary flow features.

The transport equations for the realizable $k - \epsilon$ are

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (4.13)$$

and

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S_\epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\nu \epsilon}} + C_{1\epsilon} \frac{\epsilon}{k} C_{3\epsilon} G_b + S_\epsilon \quad (4.14)$$

where

$$C_1 = \max \left[0.43 \frac{\eta}{\eta + 5} \right] \quad (4.15)$$

$$\eta = S \frac{k}{\epsilon} \quad (4.16)$$

$$S = \sqrt{2 S_{ij} S_{ij}} \quad (4.17)$$

G_k is the generation of turbulence kinetic energy due to the mean velocity gradients and G_b is the generation of turbulence kinetic energy due to buoyancy. σ_k and σ_ϵ are the turbulent Prandtl numbers for k and ϵ , respectively. S_k and S_ϵ are user defined source terms and C_2 and $C_{1\epsilon}$ are both constants.

In FLUENT, the default values for the constants $C_{1\epsilon}$, $C_{2\epsilon}$, σ_k , and σ_ϵ are given as the following

$$C_{1\epsilon} = 1.44, C_{2\epsilon} = 1.9, \sigma_k = 1.0, \sigma_\epsilon = 1.2$$

Just like the standard $k - \epsilon$ model, the eddy viscosity is defined as

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (4.18)$$

However, the main difference between the realizable $k - \epsilon$ model and the standard model is that C_μ is no longer constant. C_μ is defined as

$$C_\mu = \frac{1}{A_0 A_S \frac{k U^x}{\epsilon}} \quad (4.19)$$

The realizable $k - \epsilon$ model has been validated for a wide range of flows, such as jet flows, boundary layer flows, and separated flows. and all of these validation cases have shown that the $k - \epsilon$ performs better than the standard $k - \epsilon$ model.

4.2.3 RNG $k - \epsilon$ Model

Using the renormalization group (RNG) methods, the standard $k - \epsilon$ turbulence model was refined to develop the RNG $k - \epsilon$ turbulence model²⁹. It is very similar to the standard model but it includes an additional R_ϵ term in the dissipation rate, ϵ , equation, which is later defined in this section. The RNG method also uses analytical formula for the turbulent Prandtl numbers, while the standard model uses user-defined, constant values. Finally, the RNG model provides an effective turbulent viscosity derived from analytical formulas that accounts for low Reynolds number effects. These refined features make the RNG $k - \epsilon$ turbulence model more suitable for wider range of flows than the standard model.

The RNG transport equations are

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (4.20)$$

and

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j} \left[\alpha_k \mu_{eff} \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} - R_\epsilon + S_\epsilon \quad (4.21)$$

G_k is the generation of turbulence kinetic energy due to the mean velocity gradients and G_b is the generation of turbulence kinetic energy due to buoyancy. σ_k and σ_ϵ are the inverse effective Prandtl numbers for k and ϵ , respectively. S_k and S_ϵ are user defined source terms.

Just like the standard $k - \epsilon$ model, the turbulent viscosity is defined as

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (4.22)$$

with a value of $C_\mu = 0.0845$ which is derived from the RNG theory, compared to a $C_\mu = 0.09$ for the standard k_ϵ model.

The additional term R_ϵ in the ϵ equation is given by

$$R_\epsilon = \frac{C_\mu \rho \eta^3 (1 - \eta/\eta_0) \epsilon^2}{1 + \beta \eta^3} \frac{1}{k} \quad (4.23)$$

where

$$\eta \equiv Sk/\epsilon, \eta_0 = 4.38, \beta = 0.012$$

In FLUENT, the default values for the constants $C_{1\epsilon}$ and $C_{2\epsilon}$, as derived analytically using the RNG method, are given as the following

$$C_{1\epsilon} = 1.42, C_{2\epsilon} = 1.68$$

4.2.4 Standard $k - \omega$ Model

The standard $k - \omega$ model³⁰ is another most commonly used turbulence models that's available today. It is a modified version of Wilcox's $k - \omega$ model³¹. Wilcox has modified the old $k - \omega$ model by revising the set of model coefficients, improving the free-shear-flow spreading rates of the model and incorporating low-Reynolds number corrections that allow the standard $k - \omega$ model to be more applicable for low-Reynolds number and wall-bounded flows³². Compared to the $k - \epsilon$ models, the standard $k - \omega$ allows for more accuracy with near wall treatments, provided a sufficient resolution of the mesh near the wall, and it's known to show significant improvements with flows under adverse pressure gradient conditions.

The turbulent kinetic energy, k , and the specific dissipation rate, ω , are derived from the following transport equations

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \quad (4.24)$$

and

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + S_\omega \quad (4.25)$$

G_k is the generation of turbulence kinetic energy due to the mean velocity gradients and G_ω represents the generation of ω . Γ_k and Γ_ω are the effective diffusivities for k and ω , respectively. Y_k and Y_ω are the dissipation of k and ω due to turbulence. Finally, S_k and S_ω are user defined source terms.

The effective diffusivities, Γ_k and Γ_ω , for k and ω are defined as

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \quad (4.26)$$

and

$$\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \quad (4.27)$$

where σ_k and σ_ω are the turbulent Prandtl numbers for k and ω , respectively. By combining these two equations, the turbulent viscosity, μ_t , can be defined as

$$\mu_t = \alpha^* \frac{\rho k}{\omega} \quad (4.28)$$

where α^* is the damping coefficient that damps the turbulent viscosity, resulting in the correction for low-Reynolds number flows.

Despite its significant improvements, the standard $k-\omega$ model still poses numerous disadvantages. One of these disadvantages is the model's instability and divergence in the the flow outside of the boundary layer. For this reason, the SST- $k-\omega$ model was developed.

4.2.5 Shear Stress Transport $k-\omega$ Model

The shear stress transport $k-\omega$ was developed by Menter^{26,27} in order to combine the accuracy of the $k-\omega$ model in the near-wall region with the freestream independence of the standard $k-\epsilon$ model away from walls. This is done by using a blended function that blends the standard $k-\omega$ model with the $k-\epsilon$ model that's been

transformed to have a similar formulation as the $k - \omega$'s formulation. The blending function is designed to activate the $k - \omega$ model near the wall while at the same time activating the $k - \epsilon$ away from the surface. For these reasons, the SST $k - \omega$ model has become a very popular turbulence model in the aerospace industry where it's necessary to resolve the viscous flow and apply turbulence models throughout the boundary layer. This makes the SST $k - \omega$ model very attractive to the circulation control application of this thesis.

The transport equations for the SST $k - \omega$ are

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + \tilde{G}_k - Y_k + S_k \quad (4.29)$$

and

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (4.30)$$

$\tilde{G}_k$ is the generation of turbulence kinetic energy due to the mean velocity gradients and G_ω represents the generation of ω . Γ_k and Γ_ω are the effective diffusivities for k and ω , respectively. Y_k and Y_ω are the dissipation of k and ω due to turbulence. S_k and S_ω are user defined source terms.

Γ_k and Γ_ω are defined just like in the standard $k - \omega$ model as

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \quad (4.31)$$

and

$$\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \quad (4.32)$$

where σ_k and σ_ω are the turbulent Prandtl numbers for k and ω , respectively. They are defined as

$$\sigma_k = \frac{1}{F_1/\sigma_{k,1} + (1 - F_1)/\sigma_{k,2}} \quad (4.33)$$

$$\sigma_\omega = \frac{1}{F_1/\sigma_{\omega,1} + (1 - F_1)/\sigma_{\omega,2}} \quad (4.34)$$

where F_1 and F_2 are the blending functions.

4.2.6 Near-Wall Treatment

Selecting the best turbulence model to successfully predict turbulent flows can greatly impact the accuracy of the solution. Typically, the presence of walls provide the main source of mean vorticity and turbulence in wall-bounded flows. Therefore, accurate representation of the near-wall flow features is very important in turbulence modeling. The presence of walls significantly affect turbulent flows because of the no-slip condition at the wall that the flow has to satisfy. The no-slip boundary condition requires the flow at the surface of a wall to be zero and the temperature of the flow to be equal to the temperature of the surface. The velocity of the flow then increases farther away from the wall until it reaches freestream velocity. It is at the walls that the flow experiences the largest gradients in pressure and momentum.

The near-wall region can be divided into three layers: the viscous sublayer, the

buffer layer, and the fully turbulent region. The region closest to the wall is the viscous sublayer, where the flow is almost laminar and viscous forces are dominant in the flow. As the flow moves farther away from the wall and into the buffer layer, viscous forces and turbulence are both dominant. Finally, flow in the fully turbulent region is dominated by turbulence. Figure 4.1 shows the division of the near-wall regions in a semi-log plot of the non-dimensional velocity of the flow on the vertical axis and the non-dimensional wall distance, y^+ , on the horizontal axis.

Figure 4.1 Subdivisions of the near-wall region¹⁰.

The non dimensional wall distance, y^+ , is defined as

$$y^+ = \frac{\rho u_t y}{\mu} \quad (4.35)$$

where u_t is the friction velocity which is defined as

$$u_t = \sqrt{\frac{\tau}{\rho}} \quad (4.36)$$

Two common practices are used in modeling the near-wall region and both can be applied in FLUENT. These two approaches are depicted in Figure 4.2.

Figure 4.2 Near-wall treatments used in FLUENT¹⁰.

4.2.7 Wall Functions

The first method, also known as the “wall function” approach, does not require a fully resolved mesh in the viscous and buffer layer. Semi-empirical formulas called “wall functions” are used to interpolate the flow variables in these regions. In essence, the wall functions approach is used to connect the viscosity dominant regions to the fully turbulent region. The “wall functions” approach can save computational resources because the mesh does not need to be resolved near the wall, decreasing time it takes to generate the mesh, which in turn speeds up the solution. Wall functions also perform well in steady, non-separated flow. The robustness and reasonable accuracy of this approach make it a practical alternative to the second method, also known as the “near-wall treatment” approach.

4.2.8 Near-Wall Treatment

The “near-wall treatment” approach uses a mesh that is refined all the way to the wall, including the viscous and buffer layers. Resolving the mesh in these viscosity dominant regions eliminates the use of wall functions. A near wall mesh is considered sufficiently fine if the first node of the boundary layer prism cell is placed at $y^+ \approx 1$. However, generating a near-wall mesh that is sufficiently fine everywhere can be computationally expensive. The total number of cells increases considerably which in turn increases the time it takes to generate solutions. Therefore, it’s ideal to have a turbulence model that uses near-wall formulations that can be applied with both fine and coarse meshes. “Enhanced wall treatment” allows for wall functions to be used on coarser cells, which typically have y^+ values greater than 5, while at the same time using wall treatment on the finer cells, which typically have y^+ values less than 5.

In FLUENT, the $k - \epsilon$ turbulence model can be used with either the wall functions approach or with enhanced wall treatment. The model’s ability to utilize both near wall approaches make it a very attractive option for turbulence modeling. The $k - \omega$ turbulence model, however, applies the near-wall treatment approach to all meshes, whether the near-wall mesh is sufficiently fine or not. For this reason, it’s important to have a sufficiently fine near-wall mesh when using the $k - \omega$ model.

4.3 Operating Conditions

The experimental investigations were performed with a dynamic pressure of $q = 5.5\text{psf}^6$. The velocity was determined using the definition for dynamic pressure as

$$q = \frac{1}{2}\rho V^2 \quad (4.37)$$

where ρ is the freestream air density and V is the velocity. With the freestream air density assumed to be at sea level condition, velocity was determined to be $V = 20.73\text{m/s}$. For standard day, this is equivalent to Mach=0.061.

Turbulent intensity is defined as

$$I \equiv \frac{u'}{u_{avg}} \quad (4.38)$$

where u' root-mean-square of the velocity fluctuations and u_{avg} is the mean flow velocity. Typically, a turbulence intensity of 1% or lower is considered low while intensities greater than 10% is considered high. Turbulence intensities of different wind tunnels are usually available as part of the tunnel characteristics. Freestream turbulence intensity of wind tunnels can be as low as 0.05%¹⁰. For the work presented in this thesis, the turbulence intensity was assumed to be 1%. The turbulent viscosity ratio was left as the FLUENT default of 10.

The operating conditions were calculated to match the experimental conditions as close as possible. For the operating conditions that could not be determined, such as temperature and pressure, they were assumed to be standard day conditions. The freestream velocity was calculated using the dynamic pressure measured during the

experimental analysis, while the mass flow rates for the CCW slot and for the engine were determined using slot blowing momentum coefficient, and thrust coefficient, respectively. This is discussed in detail in the following subsections. Table 4.1 shows the CFD operating conditions for all cases that were investigated.

Table 4.1 CFD Operating Conditions.

Altitude, m	0
Mach	0.061
Velocity, m/s	20.3
Pressure, Pa	101325
Temperature, K	297
Density, kg/m ³	1.225
Viscosity, kg/m-s	1.7894×10^{-5}
Turbulent Intensity, %	1
Turbulent Viscosity Ratio	10

4.3.1 Boundary Conditions

The inlet to the wind tunnel was modeled using a “velocity inlet” boundary conditions while the outlet was set as “pressure outlet”. To achieve a certain mass flow rate, a “mass-flow inlet” boundary condition was set for the engine and CCW outlet faces.

4.3.2 Wind Tunnel Inlet and Outlet

The wind tunnel inlet was prescribed as a “velocity inlet” boundary condition in order to define the inflow velocity. With this boundary condition, the velocity magnitude and its direction can be prescribed. To ensure continuity in the flow, the wind tunnel outlet can be prescribed as a “velocity outlet” boundary condition. However, the flow from the engine simulator and the circulation control count as

additional mass to the overall system and it's assumed that the flow is dumped out into the atmosphere. For this reason, the outlet was modeled as a “pressure outlet” boundary condition with a zero gage pressure. Figure 4.3 shows the “velocity inlet” and “pressure outlet” boundary conditions that were set in FLUENT.

Figure 4.3 Velocity inlet and pressure outlet boundary conditions.

4.3.3 Engine Boundary Condition

The engine nozzle exit is controlled using a “mass-flow inlet” boundary condition. With this boundary condition, a mass flow can be prescribed at the engine outlet face. The mass flow out of the engine can be calculated using the definition of thrust

coefficient, C_T

$$C_T = \frac{\dot{m}_{eng} U_{eng}}{q_\infty S} \quad (4.39)$$

where U_{eng} is the outlet flow velocity of the engine, q_∞ is freestream dynamic pressure, and S is the wing reference area. $\dot{m}_{eng}$ is defined as

$$\dot{m}_{eng} = \rho_{eng} A_{eng} U_{eng} \quad (4.40)$$

which can be re-written as

$$U_{eng} = \frac{\dot{m}_{eng}}{\rho_{eng} A_{eng}} \quad (4.41)$$

where A_{eng} is the engine outlet area. Multiplying Eqs. 4.40 and 4.41 results in the thrust equation out of the engine outlet, $\dot{m}_{eng} U_{eng}$

$$\dot{m}_{eng} U_{eng} = \frac{\dot{m}_{eng}^2}{\rho_{eng} A_{eng}} = \frac{\dot{m}_{eng}^2 R T_{eng}}{P_{eng} A_{eng}} \quad (4.42)$$

Substituting this definition of thrust in Eq. 4.39 results in

$$C_T = \frac{\dot{m}_{eng}^2 R T_{eng}}{P_{eng} A_{eng} q_\infty S} \quad (4.43)$$

Assuming temperature, T_{eng} , and pressure, P_{eng} , to be equal with freestream conditions, the mass flow out of the engine outlet face can be calculated by using

$$\dot{m}_{eng} = \sqrt{\frac{C_T P_\infty A_{eng} q_\infty S}{RT_\infty}} \quad (4.44)$$

where R is the universal gas constant. The mass flow out of the engine outlet can now be calculated for any C_T specified by Englar [2009]⁶.

Table 4.2 Calculated engine mass flow rates.

Thrust Coefficient, C_T	Mass Flow Rate, kg/s
0	0
2.1	0.3830

4.3.4 Slot Boundary Condition

There are two approaches to set the boundary condition for the circulation control slot, both are explained in detail by Baker [2004]³³. Figure 4.4 shows a schematic of both computational boundary conditions. Both approaches are used to achieve a certain momentum blowing coefficient, C_μ , at the slot exit, which is defined as

$$C_\mu = \frac{\dot{m}_{slot} U_{slot}}{q_\infty S} \quad (4.45)$$

The first approach is to model the computational boundary at the slot exit and apply a “mass-flow inlet” boundary condition. The mass flow can be calculated using C_μ and applying the same method used in calculating the mass flow for the engine outlet, resulting in

$$\dot{m}_{slot} = \sqrt{\frac{C_\mu P_{slot} A_{slot} q_\infty S}{RT_{slot}}} \quad (4.46)$$

Figure 4.4 Two approaches to model the circulation control slot boundary.

The second approach requires modeling the inside of the plenum and applying a “pressure inlet” boundary condition. A “mass-flow inlet” boundary condition can also be applied on the plenum face, upstream of the slot. The first approach proves to be the simpler alternative of the two because it doesn’t require a mesh inside the plenum. Meshing upstream of the slot flow into the plenum requires a sufficiently fine mesh to model the flow, increasing the total cell count dramatically. The problem with the first approach, however, is that knowing the slot conditions at the slot exit face, which in this case is 0.02 inches in height, is nearly impossible. Calculating the mass flow through the slot face requires a few assumptions, as done in the calculation of mass flow out of the engine outlet. The temperature and pressure at the outlet face are unknown and therefore assumed to be freestream conditions. Applying a “mass-flow inlet” boundary condition at the slot exit also eliminates the ability of the flow to build up the appropriate velocity profile at the slot exit. In order to specify

the correct velocity profile at the slot face, FLUENT's User Defined Functions must be used to implement a velocity profile similar to 2D circulation control studies. Even with its disadvantages, this approach was still chosen for majority of the cases because of the many different mass flows that need to be prescribed at the slot exit. It results in a smaller mesh, as the plenum does not need to be meshed, and proves to be the simplest way to prescribe a certain mass flow at the slot exit.

Modeling the slot boundary condition with the second approach was also investigated in order to determine the sensitivity of the work to the type of slot boundary condition used. The advantage of this approach is that the flow is allowed to develop freely as it would in the wind tunnel experiments, resulting in a more realistic velocity profile at the slot exit. As previously mentioned, this requires a volume mesh to model the air inside the plenum, resulting in an increased total cell count.

The “pressure inlet” boundary condition in FLUENT requires a total pressure, P_0 , and total temperature, T_0 , input at the boundary face. The total temperature is assumed to be approximately equal to the freestream total temperature. The total pressure must then be calculated for a prescribed C_μ using isentropic relations. First the static temperature in the plenum must be calculated using the following equation.

$$\frac{T_{0,slot}}{T_{slot}} = 1 + \frac{\gamma - 1}{2} M_{slot}^2 \quad (4.47)$$

where $\gamma = 1.4$. The slot face is assumed to be at the throat of the nozzle, which means that the jet Mach number, M_{slot} , at the slot is equal to 1. This equation can

be rewritten to become

$$U_{slot}^2 = \frac{2\gamma R}{\gamma - 1} (T_{0,slot} - T_{slot}) \quad (4.48)$$

Substituting Eq. 4.48 into Eq. 4.45 and applying the Ideal Gas Law equation, $\rho_{slot} = P_{slot}/RT_{slot}$, the following equation can be obtained.

$$C_\mu = \frac{2\gamma}{\gamma - 1} \frac{A_{slot}}{q_\infty S} \frac{P_{slot}(T_{0,slot} - T_{slot})}{T_{slot}} \quad (4.49)$$

For a given C_μ , the static pressure at the slot can be calculated which can then be used to calculate the total pressure at the slot using the following equation

$$\frac{P_{slot}}{P_{0,slot}} = \left(1 + \frac{\gamma - 1}{2} M_{slot}^2\right)^{-\frac{\gamma}{\gamma - 1}} \quad (4.50)$$

Table 4.3 Calculated CCW mass flow rates.

Blowing Momentum Coefficient, C_μ	Mass Flow Rate, kg/s	Total Pressure, Pa
0	0	0
0.25	0.0711	38676
0.50	0.1005	77353

4.4 Solver Settings

ANSYS FLUENT 6.3 was used as the numerical solver for this thesis. Table 4.4 shows a summary of the FLUENT settings used to generate the solutions.

Table 4.4 FLUENT Solver Settings.

Solver	Pressure-Based
Numerical Scheme	2nd Order
Turbulence Model	$k - \epsilon$ and $k - \omega$ Models
Turbulence Intensity	1%
Target y^+	1
Density Calculator	Ideal Gas Law
Viscosity Calculator	Sutherland's Law

Chapter 5

RESULTS

5.1 Case Listings

In order to validate my CFD methods, experimental results from GTRI's Configuration B was used as a baseline data to be used for comparison. The wind tunnel evaluation of Configuration B consisted of analyzing a sweep of blowing momentum coefficients, C_μ , for different values of engine thrust coefficient, C_T . Results from this particular test are shown in Figure 5.1 and Figure 5.2. These plots were used to determine the values of C_μ and C_T to be evaluated in CFD. To limit the total number of cases, only values of $C_T = 0$ and $C_T = 2.1$ were chosen to be evaluated for the different values of C_μ . Choosing both extreme values of C_T would allow the sensitivity effects of increasing the thrust and introducing more turbulence to be captured in the CFD solution. Similarly, C_μ values of 0.0, 0.25, 0.50 were chosen to limit the number of cases and to investigate the sensitivity of the model to the blowing momentum coefficient.

Figure 5.1 Experimental lift coefficient as a function of C_{μ} at different values of C_T

Figure 5.2 Experimental drag coefficient as a function of C_{μ} at different values of C_T

Mass flow rates were calculated for the engine and the circulation control slot according to the methods described in Section 4.3. The corresponding mass flow rates for each cases were calculated and are shown in Tables 5.1- 5.2.

Table 5.1 Case listing: Wind tunnel included, CCW slot modeled as mass-flow inlet, $C_T = 0$

Case	C_μ	$\dot{m}_{slot}$ kg/sec	C_T	Turbulence Model
101	0.00	0.0	0	$k - \epsilon$ STD
102	0.25	0.0711	0	$k - \epsilon$ STD
103	0.50	0.1005	0	$k - \epsilon$ STD
104	0.00	0.0	0	$k - \epsilon$ RLZ
105	0.25	0.0711	0	$k - \epsilon$ RLZ
106	0.50	0.1005	0	$k - \epsilon$ RLZ
107	0.00	0.0	0	$k - \epsilon$ RNG
108	0.25	0.0711	0	$k - \epsilon$ RNG
109	0.50	0.1005	0	$k - \epsilon$ RNG
110	0.00	0.0	0	$k - \omega$ SST
111	0.25	0.0711	0	$k - \omega$ SST
112	0.50	0.1005	0	$k - \omega$ SST
113	0.00	0.0	0	$k - \omega$ STD
114	0.25	0.0711	0	$k - \omega$ STD
115	0.50	0.1005	0	$k - \omega$ STD

Table 5.2 Case listing: Wind tunnel included, CCW slot modeled as mass-flow inlet, $C_T = 2.1$ ($\dot{m}_{engine}=0.3830$ kg/sec)

Case	C_μ	$\dot{m}_{slot}$ kg/sec	C_T	Turbulence Model
201	0.00	0.0	2.1	$k - \epsilon$ STD
202	0.25	0.0711	2.1	$k - \epsilon$ STD
203	0.50	0.1005	2.1	$k - \epsilon$ STD
204	0.00	0.0	2.1	$k - \epsilon$ RLZ
205	0.25	0.0711	2.1	$k - \epsilon$ RLZ
206	0.50	0.1005	2.1	$k - \epsilon$ RLZ
207	0.00	0.0	2.1	$k - \epsilon$ RNG
208	0.25	0.0711	2.1	$k - \epsilon$ RNG
209	0.50	0.1005	2.1	$k - \epsilon$ RNG
210	0.00	0.0	2.1	$k - \omega$ SST
211	0.25	0.0711	2.1	$k - \omega$ SST
212	0.50	0.1005	2.1	$k - \omega$ SST
213	0.00	0.0	2.1	$k - \omega$ STD
214	0.25	0.0711	2.1	$k - \omega$ STD
215	0.50	0.1005	2.1	$k - \omega$ STD

Cases 301-303 investigated the free-air cases, where the wind-tunnel was not modeled.

Table 5.3 Case listing: Wind tunnel not included, CCW slot modeled as mass-flow inlet, $C_T = 2.1$ ($\dot{m}_{engine}=0.3830$ kg/sec)

Case	C_μ	$\dot{m}_{slot}$ kg/sec	C_T	Turbulence Model
301	0.00	0.0	2.1	$k - \omega$ SST
302	0.25	0.0711	2.1	$k - \omega$ SST
303	0.50	0.1005	2.1	$k - \omega$ SST

There are two approaches to set the boundary condition for the circulation control slot, as discussed in Section 4.3.1. Investigation of both approaches were performed to determine the sensitivity of the solution to both approaches. Case 403 models the plenum with a “mass -low inlet” while Case 406 models the plenum with a “pressure

inlet”.

Table 5.4 Case listing: Wind tunnel included, CCW plenum modeled, $C_T = 2.1$ ($\dot{m}_{engine}=0.3830$ kg/sec)

Case	C_μ	Plenum BC	C_T	Turbulence Model
403	0.50	Mass Flow Inlet	2.1	$k - \omega$ SST
406	0.50	Pressure Inlet	2.1	$k - \omega$ SST

5.2 Grid Convergence Study

In order to determine an acceptable sized mesh, a grid convergence study was performed using Richardson’s Extrapolation method. The CFD simulation was performed on three different sized, proportional meshes. Lift and drag coefficient were looked at as the variable of interest and their error was calculated for each of the mesh. The three meshes analyzed for this study is shown on Figure 5.3. Tab 5.5 shows the calculated results from the method discussed in Section 3.3. Figures 5.4 and 5.5 shows the convergence of the difference meshes towards the extrapolated solution as given by the Richardson Extrapolation for the lift and drag coefficient. The errors associated with these solutions are also plotted with each grid points. The blue line indicates the theoretical solution for an infinitely fine mesh. The results show a good convergence on a refined mesh as the theoretical solution falls within the error associated with the finest mesh. For this reason, the finest mesh was used for the analysis.

(a) Coarse mesh: 3.97M cells

(b) Intermediate mesh: 7.11M cells

(c) Fine mesh: 8.64M cells

Figure 5.3 Mesh refinement on Configuration B.

Table 5.5 Calculation of GCI and discretization error using Richardson's Extrapolation method.

Parameters	Lift Coefficient	Drag Coefficient
N_1	$3.97x10^6$	$3.97x10^6$
N_2	$7.11x10^6$	$7.11x10^6$
N_3	$8.64x10^6$	$8.64x10^6$
r_{12}	1.21	1.21
r_{23}	1.07	1.07
σ_1	1.73	0.05
σ_2	1.70	0.04
σ_3	1.7	0.04
n	5.59	1.50
σ	1.65	0.03
e_{app}	0.01	0.02
e_∞	0.02	0.28
GCI	0.03	0.27

Figure 5.4 Convergence of lift coefficient towards the extrapolated solution as mesh increases proportionally in size.

Figure 5.5 Convergence of drag coefficient towards the extrapolated solution as mesh increases proportionally in size.

5.2.1 Mesh Description

The total mesh size varied depending on the analyzed geometry. Cases 100's and 200's used a mesh that consisted of the wind tunnel geometry and they modeled the circulation slot with a mass-flow inlet boundary condition, which meant that the plenum upstream of the slot was not included. This reduced the total mesh size considerably. Cases 100's and 200's used a mesh that resulted from the grid independence study performed using Richardson's Extrapolation method. A different mesh was used for Cases 300's as the wind tunnel walls were not included and the size of the domain was increased in order to analyze the configuration flying in free-air. Another mesh was used for Cases 400's, which modeled the plenum upstream of the

circulation control slot. This increased the total mesh size considerably as boundary layers were included inside the plenum and the element size had to be extremely small. All meshes were created with an initial prism height of 0.001 inches, corresponding to a target $y^+ = 1$. Boundary layers were generated on the entire geometry, including the wind tunnel walls and the plenum. This increased the total number of cells but it allowed for the use of $k - \omega$ and $k - \epsilon$ turbulence models without generating new meshes. Tab 5.6 shows a description of each of the mesh used for this study.

Table 5.6 Mesh Description

Geometry Description	Mesh Size	Target y^+	Prism First Height, in
Wind tunnel, CCW (Mass Flow Inlet)	8 M cells	1	0.001
Wind tunnel, CCW (Pressure Outlet)	15 M cells	1	0.001
Free Air, CCW (Mass Flow Inlet)	18 M cells	1	0.001

5.3 Turbulence Model Sensitivity

To determine the most suitable turbulence model for CCW applications, FLUENT's 2-equation models were investigated, as listed in the case listings in Tables 5.1-5.2. This was done in an attempt to determine which turbulence model can produce the smallest relative error in C_L and C_D when compared to experimental data. The turbulence models investigated were Standard $k - \epsilon$, $k - \epsilon$ Realizable, $k - \epsilon$ RNG, and $k - \omega$ SST. Each turbulence model was analyzed for a range of C_μ with $C_T = 0$ and $C_T = 2.1$ and was compared to experimental values as shown in Figure 5.1.

Figure 5.6 shows the CFD results compared to experimental results for the case with the engine off, $C_T = 0$. CFD results for this condition show that CFD over predicts values of C_L for all turbulence models, similar to the results from the 2004

NASA/ONR Circulation Control (CC) workshop^{12,3}, shown in Figure 1.5. The relative error for each turbulence model when compared to the experimental data increases as the values of C_μ increases. It's important to notice that each turbulence model follows the trend of increasing C_L with increasing C_μ , similar to the experimental results. Here the $k - \omega$ SST turbulence model produces the lowest relative error of 17% of C_L , which is calculated at the maximum value of C_μ . Figure 5.7 shows the CFD results compared to experimental results for the cases with the engine turned on, $C_T = 2.1$. Similar trends of increasing C_L with increasing C_μ can be seen. With the engine turned on, $k - \omega$ SST also produces the lowest relative error of 15% when compared to the experimental results.

Figure 5.6 Lift coefficient as a function of C_μ with $C_T = 0$

Figure 5.7 Lift coefficient as a function of C_{μ} with $C_T = 2.1$

The CFD ‘drag coefficient’ isn’t as informative as to which turbulence model is more suitable for predicting the drag on the model. This is because the thrust created by the engine and the circulation control is included in the reported values of C_D . For this reason, the C_D values presented by NASA, as shown in Figure 5.2, is actually the net force acting in the horizontal direction, C_x . Figures 5.8 and 5.9 show C_x as a function of C_{μ} for C_T values of 0.0 and 0.5, respectively. The momentum created by the circulation control and the engine are more dominant when compared to the actual drag. Since the same boundary condition is applied to the circulation control and the engine for every turbulence models, there is very little difference in the values of C_x between each turbulence model. A 20% relative error in C_x is calculated for

each turbulence models when compared to the experimental. Although there is no experimental data for the actual drag acting on the wind tunnel model to be used for comparison with CFD, the CFD predicted C_μ and C_T was further looked at.

Figure 5.8 Drag coefficient as a function of C_T with $C_\mu = 0$

Figure 5.9 Drag coefficient as a function of C_T with $C_\mu = 0.5$

A “mass-flow inlet” boundary condition was applied to the circulation control and the engine outlet face, which both required an input of mass-flow rates. Mass-flow rates for the circulation control and the engine were calculated for a given C_μ and C_T , as discussed in Chapter 4.3.1. In Chapter 4.3.1, it was assumed that the density at the outlet faces of the circulation control and engine were equal to freestream density. This assumption provided an error between the resulting C_μ and C_T in CFD as the density at their respective outlet faces were actually higher than the assumed freestream density. This resulted in a lower C_μ and C_T than what was originally targeted. Table 5.7 shows a comparison between the CFD calculated C_μ and C_T compared to their target values for Cases 210-212. This discrepancy shows up in the

C_x vs C_T plots in Figures 5.8 and 5.9, with CFD predicting lower magnitude C_x when compared to experimental.

Table 5.7 Comparison between calculated and targeted values of C_μ and C_T .

Case	C_μ (target)	C_μ (CFD)	ρ_{slot} kg/m ³	C_T (target)	C_T (CFD)	ρ_{eng} kg/m ³
210	0	0	1.225	2.1	1.6699	1.324
211	0.25	0.2331	1.267	2.1	1.6727	1.321
212	0.50	0.4502	1.310	2.1	1.6739	1.321

5.4 Free Air Analysis

The wind tunnel geometry was included in all of the CFD cases analyzed in order to recreate the experimental conditions as close as possible. The interaction of the tunnel with the CCW model greatly impacted the resulting C_L and C_D of the CCW configuration. Additional cases without the wind tunnel were looked at to determine exactly how the wind tunnel affects the solution.

The domain was increased in size in all directions except towards the sides of the model. The sides of the new domain were prescribed as “symmetry” boundary conditions in order to keep the 2 dimensional nature of the wing. FLUENT assumes zero flux for all flow variables across a symmetry boundary, which results in a zero normal velocity and zero normal gradients. The other domain dimensions were increased by roughly 10 mean geometric chord (MGC) lengths upstream, 5 MGC lengths upward, 7 MGC lengths downward and 25 MGC lengths downstream³⁴. The increased in size ensured that the domain followed the characteristics of a far-field boundary.

Figure 5.10 shows the C_L vs C_μ plot for with and without the wind tunnel walls. This investigation was performed on the case with $C_T = 2.1$ and the $k - \omega$ turbulence

model was used. Not including the wind tunnel in the CFD analysis produces a maximum relative error of 20% when compared to the experimental results, while the case with the wind tunnel produces a maximum relative error of 15%. By removing the wind tunnel wall, the CFD model is less representative of the experimental results. The flow interaction between the side walls and the wing of the CCW configuration is no longer present as seen on Figure 5.11. Without the wind tunnel walls, the flow coming of the trailing edge of the CCW configuration is not restricted by the bottom walls, causing the flow to entrain much lower than it would if the wind tunnel was present. This could also cause the additional lift seen in Figure 5.10.

Figure 5.10 Lift coefficient as a function of C_{μ} , with and without wind tunnel, with $C_T = 2.1$, and with $k - \omega$ turbulence model

Figure 5.11 Mach streamlines plotted on the leading edge. Top: air flow is more uniform on the wing. Bottom: the interaction of the wing with the side walls causes less uniform flow near the wall

5.5 Modeling the Plenum

There were two ways to model the circulation control in CFD. One way was to apply the circulation control boundary conditions at the exit face, the other is to apply the boundary conditions in the plenum, upstream of the exit face. With the plenum modeled, a ‘pressure inlet’ and a ‘mass-flow inlet’ was applied upstream of the exit face. Table 5.8 shows a comparison of results between the different ways the circulation control was modeled in CFD.

Table 5.8 Case listing: Wind tunnel included, CCW plenum modeled, $C_T = 2.1$ ($\dot{m}_{engine}=0.3830$ kg/sec)

Case	Target C_μ	CFD C_μ	V_{slot} m/s	$\dot{m}_{engine}$ kg/s	ρ_{slot} kg/m ³	C_L	C_D
212	0.5	0.4502	197.57	0.1005	1.310	3.10	0.11
403	0.5	0.4794	219.56	0.1004	1.276	4.42	4.25
406	0.5	0.7176	312.79	0.1012	1.302	7.49	10.42

By modeling the plenum upstream of the circulation control slot, the velocity profile at the CCW slot exit face becomes more fully develop. Figure 5.12 shows the velocity profiles at the slot exit. The velocity profile for Case 212 shows a more constant velocity throughout the slot face. The high velocity gradient near the top and bottom walls of the CCW slot is due to the no-slip condition at the wall. Velocity gradient is more apparent in the velocity profiles for Cases 403 and 406 as the velocity is more fully defined at the exit slot face. Although the velocity profiles are more defined, the average velocity at the CCW slot increases when the model is plenum. Cases 403, where a mass-flow inlet is applied at the plenum, produces an average velocity at the slot exit face of 219.56 m/s, while Case 406 produces an average velocity of 312.79 m/s. This increase in velocity causes an increase in the total lift and drag, which is not ideal for the purpose of this thesis

Figure 5.12 Velocity profile at CCW slot face

5.6 Wall y^+

Capturing the flow physics within the boundary layer is important to ensure that the velocity profile near the wall surfaces are represented accurately in the solution. A mesh with a $y^+ = 1$ was used, which works for both the wall-functions of the $k - \epsilon$ model and the near-wall treatment of the $k - \omega$ model. Figure 5.13 shows contours of y^+ on the wind tunnel walls. The $y^+ = 1$ is captured throughout the entire wind tunnel walls. $y^+ < 5$ is also captured on the wind tunnel configuration for both engine on and off cases, as shown on Figure 5.14. Similar contours are seen for all cases of different turbulence models, different CCW momentum coefficient, and different engine thrust coefficient.

Figure 5.13 Contours of wall y^+ on the wind tunnel walls.

Figure 5.14 Contours of wall y^+ for Case 112 (engine off) and Case 212 (engine on).

5.7 Turbulence Intensity

All CFD cases were analyzed with a turbulence intensity of 1% at the inlet and outlet of the wind tunnel. However, turbulence intensity of 5% was also investigated, resulting in nearly identical solutions. This was done because the turbulence intensity of GTRI's MTF wind tunnel was not provided. This sensitivity study was performed on the engine on case with $k - \omega$ turbulence model.

Chapter 6

FINAL REMARKS

6.1 Conclusions

This paper focused on the ability of computational fluid dynamics to match experimental results of a 3-dimensional circulation control wind tunnel model with an over the wing engine. Similar analysis has shown that CFD over predicted values of blowing momentum coefficient, C_{μ} . However, these analyses were 2-dimensional and did not include an over the wing engine geometry. It has been theorized that the over prediction was due to gridding issues and turbulence model selections. This paper attempted to address both of those issues by performing a grid independence study and looking into FLUENT's 2-equations turbulence models. Lift and drag coefficients for the wind tunnel configuration were reported and compared to experimental results.

The grid convergence study was performed to determine an acceptable size mesh. Out of the three proportional meshes created, the finest mesh was chosen which contained a 2%error in C_L when compared to the approximated solution found by Richardson's Extrapolation method. The turbulence model study was then performed on this mesh. The analysis showed that the $k - \omega$ turbulence model produced the smallest relative in error in C_L when compared to the experimental results. A maxi-

mum error of 17% in C_L was seen when the engine was turned “off” (no mass flow) and 15% when the engine was turned “on”.

Although this paper confirmed the 2004 NASA/ONR Circulation Control (CC) workshop^{12,3} results that CFD tend to over predict C_L values, it has also shown that CFD does a good job in capturing the trends seen in the experimental results. The analysis was able to capture the trend of increasing C_L as C_μ , as well as the trend of increasing horizontal force (C_x) as C_T and C_μ were both increased. Although absolute values may not be accurate, the CFD results can be calibrated and used to predict results for future circulation control analysis.

6.2 Future Work

For future analysis, it would be very interesting to look into a few additional topics that were not included in this study.

This study looked into the effects of each of FLUENT’s 2-equation turbulence models. Expanding the turbulence model selection to the rest of the FLUENT’s available models can provide a good turbulence model sensitivity. Also, it was initially intended that Storm’s²³ modified $v2 - f$ turbulence model be part of the turbulence model study, however time and money did not permit this effort. Analyzing the modified $v2 - f$ and other higher order turbulence models could lead to decrease errors in the solution.

The decision to analyze GTRI’s configuration B was based solely on the simplic-

ity of the configuration. Configuration B was chosen from a list of low drag take-off configurations. This group of configurations provided the simplest geometry to analyze, ie no leading-edge devices and no CCW flap deflection. It would be interesting to expand this analysis to a more complicated geometry and determine how well or worse CFD can predict the experimental results.

Finally, this study only used one meshing technique in which a boundary layer mesh was generated on the surfaces and tetrahedral volume mesh was used for the rest of the domain space. Looking into different meshing techniques like structured meshing done by Pham³⁵ can provide a mesh sensitivity. Using a structured volume mesh near the model can greatly reduce the errors associated with meshing, but generating such mesh will surely increase the meshing time, time that was not available at the time of this study.

BIBLIOGRAPHY

- 1 Dunford, D., "CCW Diagram," Wikipedia, November 2007, Accessed: 6/28/2012.
- 2 NASA, "Rotor Systems Research Aircraft," 2002, Accessed: 9/09/2011.
- 3 Gregory S. Jones, R. D. J., editor, *Proceedings of the 2004 NASA/ONR Circulation Control Workshop Part 2*. NASA, 2005.
- 4 Englar, R. J., "Circulation Control Pneumatic Aerodynamics: Blown Force and Moment Augmentation and Modification; Past, Present, and Future," Tech. rep., Georgia Tech Research Institute, 2000.
- 5 Naudi, J.-L., "Coanda Effect," Website, 1997-2012, <http://jnaudin.free.fr/html/coanda.htm>.
- 6 Robert J. Englar, Richard J. Gaeta, W. J. L. V. L., "Development of Pneumatic Over-the-Wing Powered-Lift Technology; Part I: Aerodynamic/Propulsive," *27th AIAA Applied Aerodynamics Conference*, 2009.
- 7 Robert J. Englar, Graham M. Blaylock, R. J. G. G. S. J. and Wilholen, W., "Recent Experimental Development of Circulation Control Airfoils and Pneumatic Powered-Lift Systems," *48th AIAA Aerospace Sciences Meeting*, 2010.

- 8 George, P. L. and Seveno, E., “The Advancing-Front Mesh Generation Method Revisited,” *International Journal for Numerical Methods in Engineering*, Vol. 37, 1994, pp. 3605–3619.
- 9 Ansys Inc., *Documentation for ANSYS ICEM CFD/AI*Environment 11.0 SP1*.
- 10 Ansys Inc., *ANSYS FLUENT 12.0 Theory Guide*, January 2009.
- 11 G. S. Jones, J. C. Lin, B. G. A. W. E. M. C. L. R. R. and Swanson, C., “Overview of CFD Validation Experiments for Circulation Control Applications at NASA,” Tech. rep., NASA Langley Research Center, 2008.
- 12 Gregory S. Jones, R. D. J., editor, *Proceedings of the 2004 NASA/ONR Circulation Control Workshop Part 1*. NASA, 2005.
- 13 Troia, T. and Waters, M., “A Propulsion Cocept for Circulation Control Wing Technology,” Tech. rep., Georgia Tech Research Institute, 2005.
- 14 Englar, Robert J., J. H. N. J. M. J. H., “Circulation Control Technology Applied to Propulsive High Lift Systems,” *SAE Paper 841497*, Vol. -, October 1984, pp. -.
- 15 Lohner, R., “Generation of Three-Dimensional Unstructured Grids by the Advancing-Front Method,” *International Journal for Numerical Methods in Fluids*, Vol. 8, 1988, pp. 1135–1149.

- 16 Shewchuk, J. R., "Tetrahedral Mesh Generation by Delaunay Refinement," Tech. rep., School of Computer Science, Carnegie Mellon University.
- 17 Chew, L. P., "Constrained Delaunay Triangulations," *Algorithmica*, Vol. 4, 1989, pp. 97–108.
- 18 Yerry, M. A. and Shephard, M. S., "Three-Dimensional Mesh Generation by Modified Octree Techniques," *International Journal for Numerical Methods in Engineering*, Vol. 20, 1984, pp. 1965–1990.
- 19 Shephard, M. and Georges, M. K., "Three-Dimensional Mesh Generation by Finite Octree Technique," *International Journal for Numerical Methods in Engineering*, Vol. 32, 1991, pp. 709–749.
- 20 Celik, I. B., G. U. R. P. J. and Freitas, C. J., "Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications," *Journal of Fluids Engineering*.
- 21 Tennehil, J. C., A. D. A. and Pletcher, R. H., *Computational Fluid Mechanics and Heat Transfer Second Edition*, Taylor and Francis, 1997.
- 22 Travis M. Storm, D. D. M., "Assessing the v2-f Turbulence Models for Circulation Control Applications," *48th AIAA Aerospace Sciences Meeting*, 2010.

- 23 Storm, T. M., *Assessing the v_2 - f Turbulence Models for Circulation Control Applications*, Master's thesis, California Polytechnic State University, San Luis Obispo, 2010.
- 24 Durbin, P. A., "Near-Wall Turbulence Closure without 'Damping Functions'," *Theoretical and Computational Fluid Dynamics*, Vol. 3, 1991, pp. 1–13.
- 25 Launder, B. E. and Spalding, D. B., *Lectures in Mathematical Models of Turbulence*, Academic Press, London, England, 1972.
- 26 Menter, F. R., "Zonal Two Equation $k - \omega$ Turbulence Models for Aerodynamic Flows," *24th Fluid Dynamics Conference*, 1993.
- 27 Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA Journal*, Vol. 32-8, 1994, pp. 1598–1605.
- 28 T. H. Shih, W. W. Liou, A. S. Z. Y. and Zhu., J., "A New $k - \epsilon$ Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation," *Computer Fluids*, Vol. 24, 1995, pp. 227–238.
- 29 Yakhot, V. and Orszag, S. A., "Renormalization Group Analysis of Turbulence: I. Basic Theory," *Journal of Scientific Computing*, Vol. 1, 1986, pp. 1–51.
- 30 Wilcox, D. C., "Reassessment of the Scale-Determining Equation of Advanced Turbulence Models," *AIAA Journal*, Vol. 26, 1988, pp. 1299–1310.

- 31 Wilcox, D., *Turbulence Modeling for CFD*, DCW Industries, Inc., La Canada, 1998.
- 32 CD-Adapco, *CD-Adapco STAR-CCM User's Guide*, 2012.
- 33 Baker, W. J. and Paterson, E. G., "Simulation of Steady Circulation Control for A General Aviation Circulation Control (GACC) Wing," Tech. rep., Pennsylvania State University, 2004.
- 34 Blessing, B. H., *A Computational Study of Engine Deflection Using a Circulation Control Wing*, Master's thesis, California Polytechnic State University, San Luis Obispo, 2011.
- 35 Pham, J., *Comparison of Unstructured & Structured Meshing for a Circulation Control Wing & Engine Configuration Using RANS CFD*, Master's thesis, California Polytechnic State University, San Luis Obispo, 2012.