Abstract

PETRILLI, JUSTIN LAWRENCE. Reynolds-Averaged Navier-Stokes Computational Study of Various Wings and Airfoils Through Post-Stall Angles of Attack. (Under the direction of Dr. Ashok Gopalarathnam.)

This thesis presents results from an ongoing effort to develop an aerodynamic database from Reynolds-Averaged Navier-Stokes (RANS) computational analysis of airfoils and wings at stall and post-stall angles of attack. The data obtained from this effort will be used for validation and refinement of a low-order post-stall prediction method developed at NCSU, and to fill existing gaps in high angle of attack data in the literature. Such data could have potential applications in post-stall flight dynamics, helicopter aerodynamics and wind turbine aerodynamics.

An overview of the NASA TetrUSS CFD package used for the RANS computational approach is presented. Detailed results for three airfoils are presented to compare their stall and post-stall behavior. The results for finite wings at stall and post-stall conditions focus on the effects of taper-ratio and sweep angle, with particular attention to whether the sectional flows can be approximated using two-dimensional flow over a stalled airfoil. While this approximation seems reasonable for unswept wings even at post-stall conditions, significant spanwise flow on stalled swept wings preclude the use of two-dimensional data to model sectional flows on swept wings. Thus, further effort is needed in low-order aerodynamic modeling of swept wings at stalled conditions. Comparisons between finite wing CFD solutions and the NCSU low-order post-stall method are also presented. In general, the comparisons between the low-order method and the CFD results were found to be good. With the rectangular wing, in deep stall the method tends to over predict the wing lift coefficients as compared to CFD. In addition to this, the assumption of chordwise flow through stall into post-stall is not a valid assumption with swept wings. It was also found that the maximum sectional lift coefficients can exceed the 2-D airfoil's maximum near inboard sections of the semispan of swept wings. These issues warrant additional study and research with the use of CFD to calibrate the low-order method to better handle such geometries.
Reynolds-Averaged Navier-Stokes Computational Study of Various Wings and Airfoils Through Post-Stall Angles of Attack

by

Justin Lawrence Petrilli

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

Raleigh, North Carolina 2013

APPROVED BY:

Dr. Fred DeJarnette
Dr. Neal Frink

Dr. Hong Luo
Dr. Ashok Gopalarathnam
Chair of Advisory Committee
Dedication

To my parents and fiancé for their unwavering support over the past six years while pursuing my education.
Biography

Justin L. Petrilli was born to Henry and Lisa Petrilli January 24th, 1989 in Pittsburgh, Pennsylvania. In 1995, Justin and his family moved to Ocean City, Maryland where he attended elementary, middle and high school. After graduating from Stephen Decatur High School in 2007, Justin began his undergraduate career at the University of Maryland Department of Aerospace Engineering. After one semester, Justin decided to transfer to North Carolina State University in Raleigh to pursue his Bachelors of Science in Aerospace Engineering. After a vigorous seven semesters, Justin graduated Magna-Cum-Laude with his B.S. in Aerospace Engineering in May 2011. During the summer of 2011, Justin interned at Honda Aircraft Company in Greensboro, North Carolina before deciding to continue his education at North Carolina State. In August of 2011, he began his pursuit of a Masters of Science in Aerospace Engineering. Justin joined the NCSU Applied Aerodynamics Group under the supervision of Dr. Ashok Gopalarathnam shortly after beginning graduate school. Funding for his research was received from NASA Langley Research in January 2012. Justin spent the summer of 2012 working at NASA Langley Research Center in Hampton, VA in the Configuration Aerodynamics Branch under the supervision of Dr. Neal Frink. After the completion of his Masters Degree, Justin will begin his career at NASA’s Langley Research Center in Hampton, Virginia as an Aerospace Engineer in the Flight Dynamics Branch.
Acknowledgements

I would like to acknowledge some of the people and organizations which assisted me during my graduate schooling. First, I would like to thank my advisor Dr. Ashok Gopalarathnam for assisting me during the past two years at NC State. I would like to thank Gautam Shah for providing the funding for my graduate research the VSST project along with the National Institute of Aerospace for facilitating the funding. A special thank you to Dr. Neal Frink for the help and mentoring he provided to me during my summer working at NASA Langley. Without his help, this thesis as it stands would not have been possible.
# Table of Contents

List of Figures .......................................................... vii  
Nomenclature ........................................................... x  

**Chapter 1 Introduction** .............................................. 1  
1.1 Post Stall Aerodynamics ........................................... 1  
1.2 Why Is Studying Post-Stall Aerodynamics of Importance? ........... 2  
1.3 Previous Post-Stall Work ......................................... 7  
1.4 Layout of this Thesis ............................................... 7  

**Chapter 2 Predicting Aerodynamics In Post-Stall Regime** ............. 9  
2.1 Existing Low-order Methods ....................................... 9  
2.2 Higher Order Methods ............................................. 10  
2.3 NCSU Method for Post Stall Aerodynamics Prediction ............... 11  
2.4 Implementation in Flight Dynamics ................................ 18  
2.5 Chapter 2 Summary ................................................ 22  

**Chapter 3 Approach for Generating Post-Stall Database** ............... 23  
3.1 High Fidelity Computational Tool .................................. 25  
3.2 Chapter 3 Summary ................................................ 28  

**Chapter 4 Methodology for Post-Stall Reynolds-Averaged Navier-Stokes CFD Simulations** .................................................. 29  
4.1 Geometry Generation ............................................... 29  
4.2 Grid Generation .................................................... 30  
4.3 Flow Solution Generation ......................................... 32  
4.4 Solution Convergence ............................................. 33  
4.5 Post-Processing .................................................... 33  
4.5.1 Total Forces and moments acting on the entire wing/configuration ........................................... 34  
4.5.2 Spanwise distribution of forces on a surface .................... 34  
4.6 Chapter 4 Summary ................................................ 35  

**Chapter 5 Results** .................................................. 37  
5.1 2D Airfoil Results .................................................. 37  
5.1.1 Reynolds Number Effects Through Post-Stall ................... 38  
5.1.2 Sharp vs. Blunt Trailing Edge Geometries ....................... 40  
5.1.3 Comparison of Post-Stall Characteristics of Three Airfoil Geometries ........................................... 41  
5.2 3D Wing Results ................................................... 46  
5.2.1 Rectangular Wing ............................................... 48  
5.2.2 Tapered Wing .................................................. 53  
5.2.3 Swept Wing Cases ............................................. 57  
5.3 Comparisons Between the Low-Order Post-Stall Method and CFD Simulations .................................................. 69
List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Representative airfoil lift curve showing potential flow predictions compared to viscous flows.</td>
</tr>
<tr>
<td>1.2</td>
<td>NACA0012 airfoil in deep stall AoA (36 deg.)</td>
</tr>
<tr>
<td>1.3</td>
<td>Coverage of A330 simulators compared with AF447 data [17].</td>
</tr>
<tr>
<td>1.4</td>
<td>Additional L.O.C. accident data compared to known aero envelope [18].</td>
</tr>
<tr>
<td>2.1</td>
<td>Visualization of CFD solution on a semispan rectangular wing AR = 12 at stall conditions. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. Black line indicates plane of symmetry.</td>
</tr>
<tr>
<td>2.2</td>
<td>Decambering functions utilized by the low-order method.</td>
</tr>
<tr>
<td>2.3</td>
<td>Plots showing that a potential flow solution for an equivalent decambering can match viscous $C_l$ and $C_m$ curves (solutions for a NACA0012 Airfoil).</td>
</tr>
<tr>
<td>2.4</td>
<td>Trajectory lines for a given geometry with effects of changing number of spanwise sections.</td>
</tr>
<tr>
<td>2.5</td>
<td>Post-Stall method implementation with 6-DOF model.</td>
</tr>
<tr>
<td>2.6</td>
<td>Typical roll damping vs. Angle of Attack for a rectangular wing.</td>
</tr>
<tr>
<td>2.7</td>
<td>Comparison of pre-stall and post-stall $C_l$ distributions for a rectangular wing with a non dimensional roll rate of 0.02.</td>
</tr>
<tr>
<td>2.8</td>
<td>Stall-Spin trajectories For a rectangular and tapered Wing.</td>
</tr>
<tr>
<td>3.1</td>
<td>Approach for using CFD for initial validation and further refinement of the low-order method. Blue emphasizes that a consistent grid spacing is used with the 2D and 3D CFD discretizations.</td>
</tr>
<tr>
<td>3.2</td>
<td>Screen capture of geometry imported into GridTool.</td>
</tr>
<tr>
<td>3.3</td>
<td>Screen capture of viscous layers and inviscid grid near leading edge of an airfoil.</td>
</tr>
<tr>
<td>4.1</td>
<td>Process of geometry generation to grid generation.</td>
</tr>
<tr>
<td>4.2</td>
<td>Screen capture of GridTool source placement on a tapered wing.</td>
</tr>
<tr>
<td>4.3</td>
<td>Screen capture of airfoil grid.</td>
</tr>
<tr>
<td>4.4</td>
<td>Typical USM3D convergence of lift coefficient for an airfoil. NACA 4415, USM3D/SA, RE = 3 million, $M_\infty = 0.2$.</td>
</tr>
<tr>
<td>4.5</td>
<td>Converged vs. AOA for 2D airfoil and 3D rectangular wing.</td>
</tr>
<tr>
<td>4.6</td>
<td>Screen shot of PREDISC code (left) and a plot of the extracted spanwise $C_l$ distribution (right) on a rectangular wing AR = 12.</td>
</tr>
<tr>
<td>4.7</td>
<td>Positive sign convention for $C_f$ and $C_p$.</td>
</tr>
<tr>
<td>5.1</td>
<td>Comparison of airfoil geometry for three cases studied.</td>
</tr>
<tr>
<td>5.2</td>
<td>Illustration of approach used for averaging oscillatory airfoil CFD convergence history.</td>
</tr>
<tr>
<td>5.3</td>
<td>Effect of Reynolds number on lift coefficient for a NACA 4415 airfoil. USM3D/SA, Re= 3 million, $M_\infty= 0.2$.</td>
</tr>
</tbody>
</table>
Figure 5.4 Effects of trailing-edge sharpness for a NACA 4415 airfoil. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 41

Figure 5.5 Comparison of $C_l$ vs. $\alpha$ for three airfoil types. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 42

Figure 5.6 Comparison of $C_l$ vs. $\alpha$ from CFD and experiment [2] for the NACA 63006 thin airfoil. .......................................................... 43

Figure 5.7 Instantaneous Mach contours and $C_p$ distribution for an NACA 4415 airfoil. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. .................................. 44

Figure 5.8 Instantaneous Mach contours and $C_p$ distribution on an NACA 63006 airfoil. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. .................................. 45

Figure 5.9 Effect of taper on unswept wing $C_L$ vs. $\alpha$. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. .......................................................... 47

Figure 5.10 Effect of sweep on constant-chord wing $C_L$ vs. $\alpha$. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 47

Figure 5.11 Comparison of local $C_l$ distribution at various angles of attack for a rectangular wing AR = 12. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ........... 48

Figure 5.12 $C_p$ contours and surface streamlines on a semi-span rectangular wing, AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 50

Figure 5.13 Comparison of local $C_l$ distribution at varying angles of attack for a tapered wing, $\lambda = 0.5$ and AR = 12. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 53

Figure 5.14 $C_p$ contours and surface streamlines on a tapered wing, $\lambda = 0.5$, AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 54

Figure 5.15 Comparison of local $C_l$ distribution at various angles of attack on a swept wing, $\Lambda = 10$ deg., AR = 12. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 57

Figure 5.16 Comparison of local $C_l$ distribution at various angles of attack on a swept wing, $\Lambda = 20$ deg., AR = 12. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 58

Figure 5.17 Comparison of local $C_l$ distribution at various angles of attack on a swept wing, $\Lambda = 30$ deg., AR = 12. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 58

Figure 5.18 $C_p$ contours and surface streamlines on a swept wing, $\Lambda = 10$ deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 60

Figure 5.19 $C_p$ contours and surface streamlines on a swept wing, $\Lambda = 20$ deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 63

Figure 5.20 $C_p$ contours and surface streamlines on a swept wing, $\Lambda = 30$ deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 66

Figure 5.21 Comparison between CFD and Low-Order Method of wing $C_L$ v.s. $\alpha$ for a rectangular wing, AR = 12. Low-Order Method -20 panels, CFD - USM3D/SA, Re = 3 million, $M_\infty = 0.2$. ......................................................... 70
Figure 5.22 Local $C_l$ distribution on a rectangular wing, AR = 12. Low-Order Method - 20 panels, CFD - USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 71

Figure 5.23 Comparison of local $C_l$ distributions on a rectangular wing, AR = 12 at 24 degrees angle of attack. Low-Order Method - 20 panels, CFD - USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 72

Figure 5.24 Possible method to address swept wing input data. 73

Figure A.1 Non-dimensional y direction vorticity contours on a NACA 4415 airfoil. USM3D/SA, Re = 3 million, $M_\infty = 0.2$ 84

Figure A.2 Non-dimensional y direction vorticity contours on a NACA 0012 airfoil. USM3D/SA, Re = 2 million, $M_\infty = 0.2$ 96

Figure A.3 Non-dimensional y direction vorticity contours on a NACA 63006 airfoil. USM3D/SA, Re = 3 million, $M_\infty = 0.2$ 107

Figure B.1 Surface streamlines highlighting separation patterns on a rectangular wing, AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 119

Figure B.2 Surface streamlines highlighting separation patterns on a tapered wing, $\lambda = 0.5$, AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 125

Figure B.3 Surface streamlines highlighting separation patterns on a swept wing, $\Lambda = 10$ deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 131

Figure B.4 Surface streamlines highlighting separation patterns on a swept wing, $\Lambda = 20$ deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 136

Figure B.5 Surface streamlines highlighting separation patterns on a swept wing, $\Lambda = 30$ deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 141
Nomenclature

Symbols

AoA  angle of attack, deg
a    VGRID stretching/growth factor
b    VGRID stretching/growth factor
BL   boundary layer
CFD  computational fluid dynamics
C\(_d\) airfoil or section drag coefficient
C\(_f\) skin friction coefficient
C\(_L\) wing lift coefficient
C\(_l\) airfoil or section lift coefficient
C\(_m\) airfoil or section pitching moment coefficient about the quarter chord
C\(_p\) pressure coefficient
c\(_{ref}\) reference chord length
LLT  lifting line theory
L.O.C loss of control
Re   Reynolds number based on airfoil chord length
SA   Spalart-Allmaras turbulence model
TE   trailing edge
t    physical time, sec
Greek

$\alpha_{stall}$  airfoil stall angle of attack, deg

$\Delta C_l$  difference between viscous and potential flow lift coefficient

$\Delta C_m$  difference between viscous and potential flow pitching moment coefficient

$\Delta t$  physical time step, sec

$\delta_x$  correction vector for all $\delta$ variables

$\delta_1$  decambering function for lift coefficient

$\delta_2$  decambering function for pitching moment coefficient

$\Gamma$  circulation strength of a vortex ring

$U_\infty$  freestream reference velocity

$y+$  non-dimensional wall distance

$\Delta z_i$  height of the $i^{th}$ viscous layer

$\alpha$  angle of attack, deg

$\Lambda$  angle of sweepback, deg

$\lambda$  taper ratio
Chapter 1

Introduction

1.1 Post Stall Aerodynamics

Aircraft typically operate in what is known as the linear region of aerodynamics. That is, for every increase in angle of attack there is a linear increase in the amount of lift generated. This linear region is well understood and has been shown with both experimental, numerical and theoretical analysis of both finite wings and two dimensional airfoils. This linear lift curve is the result of the attached boundary layer and potential flow approximations are valid. Potential flow theory can be used for low angle of attack analysis with decent accuracy, however, if the angle of attack is continuously increased it is found to be no longer valid. The reasoning is that potential flow theory does not include the effects of viscosity. Viscosity plays an intricate role in the aerodynamics of a wing or airfoil and is manifested through the boundary layer (BL). At lower angles of attack, the boundary layer is thin and attached to the surface. However, as angle of attack is increased, the boundary layer will increase in thickness on the upper surface. The formation of an increasing adverse pressure gradient encountered by the flow eventually causes the B.L. to separate from the upper surface Fig. 1.1. This flow separation (typically seen starting at the trailing edge) causes a deviation of the lift curve from what potential flow would predict [1]. If the angle of attack is further increased beyond the maximum lift coefficient, the
regime of aerodynamics known as Post Stall has been encountered.

![Diagram of airfoil lift curve showing potential flow predictions compared to viscous flows.]

In deep post stall angles of attack, the flow is massively separated (Fig. 1.2) and may even have unsteady characteristics associated with vortices shedding from the upper surface. As a result of the extreme nonlinearity of the aerodynamics encountered in this region, it becomes increasingly difficult to predict with accuracy or confidence what forces and moments would be acting on an aircraft. This large amount of uncertainty significantly lowers the fidelity of flight dynamic simulations at stall and post-stall conditions.

### 1.2 Why Is Studying Post-Stall Aerodynamics of Importance?

Airfoil lift and moment data in the low angle of attack regime is readily available from a multitude of sources— from experimental data such as those presented in Abbott and von Doenhoff [2]...
and those from the University of Stuttgart [3] and University of Illinois at Urbana-Champaign [4, 5, 6], to modern computational approaches designed to predict sectional aerodynamic characteristics based on arbitrary input geometry, such as XFOIL [7]. For many applications, data in this linear regime is sufficient. However, fields such as wind turbine aerodynamics, helicopter aerodynamics and post-stall flight dynamics of fixed-wing aircraft require data to extend beyond aerodynamic stall. Efforts have been made in the wind turbine community and the helicopter aerodynamics community to extend airfoil data into the post-stall regime. Models for airfoil force and moment coefficients at high angle of attack conditions have been developed experimentally [8, 9], which has lead to researchers proposing empirical models based on flat plate theory [10]. The empirical models developed from experiment require that both the maximum $C_l$ and the corresponding $\alpha_{stall}$ at which this lift coefficient occurs be known reliably before theoretical flat plate data may be fitted to extend the data well into post-stall. Very little data exists in the literature covering the stall behavior of finite wings, especially that which extends
deep into post-stall. Published work in post-stall wing aerodynamics often covers general stall behavior dependent on factors such as planform, without providing much in the way of detailed force and moment data beyond initial stall [11]. Some studies propose empirical methods based on theory to extend force and moment coefficient data-sets deep into post-stall, taking into account 3D effects with some correction for aspect ratio [9]. An interesting, purely experimental study, that covers the stall of both a 2D airfoil section and 3D wings of various aspect ratios was performed by Ostowari and Naik in 1985 [12]. The study presented consistent lift coefficient versus angle of attack data for various NACA 44XX series airfoils and 3D rectangular wings with a range of aspect ratios having the same airfoils as cross sections.

From the view of aircraft dynamics, post-stall aerodynamics can lead to what are called “Loss of Control (LOC) Scenarios. The Joint Safety Analysis Team (JSAT) defined loss of control as “... significant, unintended departure of the aircraft from controlled flight, the operational envelope, or usual flight attitudes, include ground events” [13]. Given this definition, it is not entirely surprising that the majority of aviation accidents and fatalities are attributed to loss of control situations [14]. According to a report published by The Boeing Company on worldwide commercial aircraft accidents [15], there were 1,926 fatalities due to in flight loss of control between 1999 and 2008.

One of the more recent commercial aviation accidents that falls into the LOC category was the tragic accident of Air France Flight 447 in June 2009 [16]. On May 31st 2009, AF 447 departed from Galeo International Airport in Brazil in route to Charles de Gaulle International Airport in France. During the early morning hours of June 1st (2:10 am) 2009, abnormal airspeed indications caused the autopilot and auto-thrust to disengage. The abnormal speed indications were later attributed to the build up of ice on the air data probes [16]. The two co-pilots in the cockpit of the aircraft took control and through a serious of nose up control inputs, caused the aircraft to climb from 35,000 feet to 38,000 feet (its maximum altitude). The Airbus A330-203 entered a high altitude stall shortly there after. Not recognizing this fact, the pilots in control continued to add more aft stick control inputs. The descent rate for the
following minutes was in excess of 10,000 ft/minute and the angle of attack was consistently between 30 and 40 degrees [16]. The aircraft impacted the Atlantic Ocean at 2:14 am killing all 228 individuals on board.

This is but one example where post stall aerodynamics has contributed to a tragic aviation accident. There is a dire need and new requirements for pilots to train in the stall and post-stall regimes. Such training is increasingly done in flight simulators rather than in a physical aircraft. A significant issue is encountered here because of the lack of data at high angles of attack for use in flight simulations. As the use of flight simulators has become a large part of a pilots training, the ability to have high fidelity aerodynamics models is of utmost importance. In Figures 1.3 and 1.4 [17, 18], the plot on the left shows the coverage of the A330-203 flight simulation aerodynamics model and on the right another plot showing known areas of aerodynamic data. In both of these plots, it is very apparent that the flight conditions encountered in a loss of control accident goes well beyond the acceptable ranges of the aerodynamic data used in simulation models. It seems plausible that the pilots who were in control of AF447 did not have adequate training due to the model’s deficiencies at high angles of attack and side-slip. Without increasing the capabilities of the aerodynamic models used in simulations such as these, improving the training of commercial transport pilots in this regime of flight will not be possible.
Figure 1.3: Coverage of A330 simulators compared with AF447 data [17].

Figure 1.4: Additional L.O.C. accident data compared to known aero envelope [18].
1.3 Previous Post-Stall Work

In 2004 work began at NCSU on a low-order methodology with the goal of predicting stall and post-stall aerodynamics on wings and configurations of lifting surfaces [1]. The method accomplishes this via use of existing linear low-order methods (VLM, Weissinger or LLT), corrected for nonlinear sectional airfoil behavior. The term low-order here indicates that the method is a rapid, albeit approximate, way to predict post-stall aerodynamics. The method uses fully viscous two dimensional sectional airfoil data as an input and outputs the total aerodynamic forces and moments. The crux of the methodology is based on a decambering analogy, that is, in stall and post stall, the separated flow effectively decambers each airfoil section along a lifting surface’s span. Each two dimensional panel can be decambered in such a way that the control points satisfy the zero normal flow requirement of potential flow. Thus the resulting lift distribution is consistent with the distribution of induced flow angles, and the operating point for each airfoil section falls somewhere on the viscous input data supplied to the model. This solution procedure turns out to be iterative in order to simultaneously satisfy these two requirements. The iterative procedure may be incorporated into aerodynamic prediction methods such as Lifting Line Theory (LLT), Weissinger’s method or vortex-lattice methods all of which are commonly used for wing and configuration aerodynamic analysis.

1.4 Layout of this Thesis

In Chapter 2, a high level overview of methods for predicting the aerodynamic forces and moments on wings will be discussed. The advantages and disadvantages of these methods will be presented as well as a more detailed look at the the NCSU post-stall methodology. The approach for generating a post-stall aerodynamics database and the computational tool that was used will be discussed in Chapter 3. The methodology that was developed to apply the computational tool to post-stall simulations is then presented in Chapter 4. Chapter 5 presents results of the computational study and comparisons to the NCSU post-stall method. Conclusions based on
the results of the thesis work along with recommendations for future work along this line of research are discussed in Chapter 6. Additional results are also presented in the appendices of this thesis.
Chapter 2

Predicting Aerodynamics In Post-Stall Regime

2.1 Existing Low-order Methods

The aerodynamic prediction methods mentioned in Section 1.3 are well known to successfully predict the lift and induced drag behavior of medium to high aspect ratio wings at low angles of attack. These methods are also quick in their calculations, only taking a few seconds to compute a solution. However, these methods are not suitable for high angle of attack aerodynamic prediction. The reason is that they do not model actual flow physics, instead they utilize potential flow theory, where a linear lift curve is assumed for each two dimensional airfoil section on a wing or lifting surface. Viscosity is ignored, thus the effects of flow separation on the wing are lost in these analyses. As was shown in Figure 1.1, when utilizing potential flow theory for an airfoil, an increase in the angle of attack will yield a corresponding increase in lift. This is the predicted result from a Vortex Lattice method or Lifting Line Theory method. As the goal is to predict aerodynamic quantities in the nonlinear regions, these simple methods are not suitable as they stand.
2.2 Higher Order Methods

Higher order prediction methods, namely Computation Fluid Dynamics (CFD), exist which can successfully predict the effects of flow separation on aerodynamic surfaces within a degree of uncertainty. Unlike the lower order counterparts, CFD numerically solves the equations of fluid flow and models actual flow physics. The advantage here is that the effects of viscosity can be incorporated and the desired trends it causes near stall and into post-stall can be predicted. In Figure 2.1 below, an example of a flow visualization of a CFD solution on a semispan rectangular wing is shown. The surface streamlines assist in understanding the stall progression along the span of the wing. It can be seen that some very complex flow features have developed in the form of stall cells. This sort of phenomenon is obviously lost in lower order methods discussed.

![Figure 2.1: Visualization of CFD solution on a semispan rectangular wing AR = 12 at stall conditions. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. Black line indicates plane of symmetry.](image)

Although computational methods such as CFD can offer a greater level of accuracy and fidelity to an aerodynamics model, they require massive amounts of time and computational resources. Just as an example, a fully viscous simulation of a three dimensional wing can take more than 12 hours to complete at just a single angle of attack. It is therefore not feasible to attempt to use a method such as CFD for rapid post stall aerodynamic predictions for an in the
loop calculation for a flight simulation. A method that is on the order of what was discussed in Section 2.1 would be required.

2.3 NCSU Method for Post Stall Aerodynamics Prediction

As was previously stated, work began in 2004 at North Carolina State University Applied Aerodynamics Group to address the need for rapid stall and post-stall aerodynamic prediction capabilities [1]. The method’s central concept (described easiest in two dimensions) is that the flow separation encountered at stall and into post stall can be modeled as an effective chordwise decambering of the airfoil section on a wing. Mainly, the boundary layer displacement thickness and separation is the root cause in the change in effective “shape” of the airfoil. This decambering causes the airfoil section to follow nonlinear $C_l$ and $C_m$ curves rather than the linear potential flow curves. If the effective decambering of the airfoil can be determined, then a potential flow prediction on the newly decambered airfoil should match closely with the fully viscous airfoil with no decambering. This effective decambering of the airfoil was defined in [1] using two decambering functions $\delta_1$ and $\delta_2$.

![Decambering functions utilized by the low-order method.](image-url)

Figure 2.2: Decambering functions utilized by the low-order method.
Figure 2.2 shows a representative explanation of the two decambering functions. Two functions are required in order to match both the viscous $C_l$ and $C_m$ curves. It is important to note that the combination of these two functions will not necessarily match the physical decambering, which could be determined using computational methods such as CFD, but the goal was to find an “equivalent” decambering. Still looking at only the two dimensional problem, for a given $\alpha$ the $\Delta C_l$ and $\Delta C_m$ are defined by equations 2.1 and 2.2

$$\Delta C_l = (C_l)_{viscous} - (C_l)_{potential} \quad (2.1)$$

$$\Delta C_m = (C_m)_{viscous} - (C_m)_{potential} \quad (2.2)$$

From these two equations, the values of $\delta_1$ and $\delta_2$ in radians can be determined using thin airfoil theory and a three-term Fourier series approximation for a flat plate with an added flap deflection [19]. The equations are written as follows

$$\delta_2 = \frac{\Delta C_m}{\frac{1}{4}sin2\theta_2 - \frac{1}{2}sin\theta_2} \quad (2.3)$$

$$\delta_1 = \frac{\Delta C_l - [2(\pi - \theta_2) + 2sin\theta_2]\delta_2}{2\pi} \quad (2.4)$$

where $\theta_2$ is the angular location in radians for the starting point of the $\delta_2$ decambering function as shown in Figure 2.2. It can be viewed as essentially the “hinge point” for the $\delta_2$. It is important to note that in equation 2.3 only the $\Delta C_m$ and $\theta_2$ need be specified while in equation 2.4, the knowledge of $\delta_2$ is required in addition. This can be attributed to the fact that the addition of the $\delta_2$ function will not only perturb $C_m$ but it will also affect $C_l$, thus it must be accounted for.

Example results for a case analyzing a NACA 0012 airfoil can be seen in Figure 2.3. The two plots show that a potential flow analysis of an equivalent decambering of the NACA 0012
can match the fully viscous data.

Figure 2.3: Plots showing that a potential flow solution for an equivalent decambering can match viscous $C_l$ and $C_m$ curves (solutions for a NACA0012 Airfoil).

The method is very simple when only considering a two dimensional airfoil, it becomes less trivial when considering a finite wing. If the assumption is made that each section of a three dimensional wing can be assumed to act as a two dimensional airfoil, then this decambering method can be used at each section. However, because this is no longer just a two dimensional problem, a change in the $\delta$ variables at one section can have an effect on neighboring and down stream sections [1]. In order to account for this, a 2N-dimensional Newton Iteration is utilized to determine the values of $\delta_1$ and $\delta_2$ at each wing section. The goal essentially is still to drive the values of $\Delta C_l$ and $\Delta C_m$ towards zero at each section as the iterations progress. A $2N \times 2N$ matrix equation, Eq. 2.5, is solved at each step of the Newton Iteration

$$J \cdot \delta x = -F \tag{2.5}$$

where, $F$ is a 2N-dimensional vector containing the residuals to be zeroed, $\delta x$ is a 2N-dimensional vector containing the corrections to the 2N $\delta$ decambering variables and $J$ is the 2N $\times$ 2N
Jacobian of the system which contains all the gradient information. Eq. 2.7 - 2.10 below show each of the four sub-matrix elements in the Jacobian.

\[ J = \begin{pmatrix} J_{l1} & J_{l2} \\ J_{m1} & J_{m2} \end{pmatrix} \]  \hfill (2.6)

\[
(J_{l1})_{i,j} = \frac{\partial \Delta C_{li}}{\partial \delta_{1,j}} = \frac{(C_{lp})_i - (C_{ls})_i}{[(\delta_{1s})_j + p] - (\delta_{1s})_j} \]  \hfill (2.7)

\[
(J_{m1})_{i,j} = \frac{\partial \Delta C_{mi}}{\partial \delta_{1,j}} = \frac{(C_{mp})_i - (C_{ms})_i}{[(\delta_{1s})_j + p] - (\delta_{1s})_j} \]  \hfill (2.8)

\[
(J_{l2})_{i,j} = \frac{\partial \Delta C_{li}}{\partial \delta_{2,j}} = \frac{(C_{lp})_i - (C_{ls})_i}{[(\delta_{2s})_j + p] - (\delta_{2s})_j} \]  \hfill (2.9)

\[
(J_{m2})_{i,j} = \frac{\partial \Delta C_{mi}}{\partial \delta_{2,j}} = \frac{(C_{mp})_i - (C_{ms})_i}{[(\delta_{2s})_j + p] - (\delta_{2s})_j} \]  \hfill (2.10)

Where \( C_{lp} \) is the local lift coefficient due to a small perturbation \( p \) and \( C_{ls} \) and \( \delta_{1,2s} \) are the starting values of local lift coefficient and decambering respectively. The terms for the \( \delta_2 \) variables and \( C_m \) follow the same logic. At each step of the iteration, this equation is used to determine \( \delta x \) which is added to \( \delta_1 \) and \( \delta_2 \). A detailed step by step breakdown of each iteration of the method can be review in Ref. [1, 20, 21]. One downside to this process is that it requires the use of a VLM, Weissinger or LLT code in the loop of each iteration. That is, once the \( \delta_1 \) and \( \delta_2 \) of each wing section has been updated, the VLM, LLT etc. must be executed to determine new values of \( C_l \) and \( C_m \) at each section. Although these are low order methods, the time it takes to run each iteration is significant. To converge a solution for a single post stall angle of attack would take approximately a minute of computational time.

As previously stated, the goal is to have the method run in the loop during a flight simulation to calculate the aerodynamic forces and moments. If the method requires a minute of computation time for a single angle of attack, then it can not be used in real time. In order to
speed up the time to convergence, the principles of lift superposition were implemented into the method. Within the assumptions of linear aerodynamics, that is, a linear relation between $C_l$ and $\alpha$, the spanwise distribution of bound circulation can be written as a sum of two different loadings: i) a basic spanwise loading, $\Gamma_b(y)$, and ii) an additional spanwise loading, $\Gamma_a(y)$.[22, 23]

$$\Gamma(y) = \Gamma_b(y) + \Gamma_a(y) \quad (2.11)$$

The basic spanwise loading is the $\Gamma$ distribution at a wing $C_L = 0$, which is a result of spanwise variations in geometric and aerodynamic twist and flap deflections. As the decambering functions, $\delta_1$ and $\delta_2$ are essentially similar to flap deflections, they can be implemented as another basic loading. The additional loading is the $\Gamma$ distribution is result of only changes to $\alpha$ for the wing with no twist and no flap deflections. This additional loading conveniently scales with the wing $C_L$. Therefore the additional loading for a $C_L = 1$ can be precomputed and stored as, $\Gamma_{a,1}$ and used to computed the additional loading for any other wing $C_L$ as seen in Eq. 2.12.

$$\Gamma_a(y) = C_L \Gamma_{a,1}(y) \quad (2.12)$$

From Ref. [20], the net $C_l$ distribution for a given wing $C_L$ can be shown in terms of the unknown decambering $\delta$ variables. Eq. 2.13 below shows that the $C_l$ distribution can be calculated by adding Eq. 2.12 with $C_{lb,0}$ (the basic loading distribution) and the incremental changes in the basic loading ($C_{lb,i}$) due to a unit $\delta_1$ for each of the total N sections on the wing ($\delta_2$ excluded for clarity).

$$C_l = C_L C_{la,1} + C_{lb,0} + C_{lb,1}\delta_1,1 + C_{lb,2}\delta_1,2 + \cdots + C_{lb,N}\delta_1,N \quad (2.13)$$

Equation 2.13 is expressed in terms of the total wing lift coefficient, $C_L$, however for post-stall computations at a given wing angle of attack, $\alpha_w$, it is required to write $C_L$ in terms of $\alpha_w$:  

15
\[ C_L = a(\alpha_w - \alpha_{b,0} - (\alpha_{b,1}\delta_{1,1} + \alpha_{b,2}\delta_{1,2} + \cdots + \alpha_{b,N}\delta_{1,N})) \] (2.14)

In this equation, \( a \) is the wing lift curve slope \( (a = \frac{1}{\alpha_{a,1}}) \), \( \alpha_{b,0} \) is the wing angle of attack corresponding to the zero-decambering basic \( C_l \) distribution and \( \alpha_{b,i} \) is the wing angle of attack corresponding to the increment in basic \( C_l \) distribution due a unit \( \delta_1 \) for section \( i \). The elements of the Jacobian can be easily precomputed as shown in Eq. 2.7-2.10. Additional details can be seen in Ref. [20]. Implementing superposition yielded faster calculation times for the method, reducing the time to convergence for a single angle of attack to less than 0.1 seconds.

\[ (J_{\|})_{i,j} = \frac{\partial \Delta C_{li}}{\partial \delta_{1,j}} = -a\alpha_{b,j}C_{la,1}(i) + C_{lb,j}(i) \] (2.15)

Improvements to this method are still ongoing, with the most recent published updates to the methodology shown in Ref. [24]. One issue encountered with the method as described above and in the associated references [1, 20, 21] is the calculation of what is known as the “trajectory lines”. The trajectory line is defined as the line of a given section along the wing in the airfoil’s \( C_l \) vs. \( \alpha \) plane, which describes how incremental changes in the \( \delta \) variables affect the movement of that section’s operating point [24]. Figure 2.4 shows a representation of the trajectory lines on the airfoil’s lift curve. The slope of these trajectory lines is calculated using the following equation using the precomputed basic and additional loadings:

\[ \left( \frac{dC_l}{d\alpha} \right) = \frac{-\alpha_{b,i}C_{la,1}(i)/\alpha_a + C_{lb,i}(i)}{C_{lb,i}(i)/\alpha_0 - \alpha_{b,i}C_{la,1}(i)/\alpha_a\alpha_0 - 1} \] (2.16)

Two separate problems were noticed with this approach: 1) The slope of the trajectory line changed significantly as the number of sections along the span was changed and 2) in certain cases where the viscous airfoil input curve had sections with large negative slope, these trajectory lines may not have any intersection with the curve [24]. The effects of panel density on the trajectory lines of each wing section can be seen in Figure 2.4. As the goal is to move the operating point so it is at some point along the input curve, these problems needed to be
A modified method had to be developed in order to alleviate both of these problems. In the new method, the trajectory lines are calculated using the original method for only the first iteration. After the completion of the second iteration, a new trajectory line is calculated using the first iteration operating point and the second iteration operating point. The convenient thing about this method is that it automatically incorporates the effects of the changing decambering variables in the other wing sections. Also, the total number of iterations required for the post-stall method to converge is reduced [24].

Figure 2.4: Trajectory lines for a given geometry with effects of changing number of spanwise sections.
2.4 Implementation in Flight Dynamics

The method described in the preceding sections was conceived to eventually be used as an aerodynamics model for use in flight dynamics simulations. As a part of this, the effects of angular rates were incorporated into the model. Angular rates essentially cause local velocity changes based on the location of a wing section’s location relative to the axes of rotation. This effectively changes the local angles of attack and hence the operating points of all wing sections. Details of implementation of angular rate effects into the low-order model formulation can be seen in Refs. [24, 25]. The low-order method has been integrated with a 6-DOF flight dynamics model using standard aircraft equations of motion with quaternion parameters. Figure 2.5 shows a flow chart of how the low-order method is used to determine the body axis forces and moments that are then used in the 6-DOF simulation.

Figure 2.5: Post-Stall method implementation with 6-DOF model.

Simulations have shown promising results thus far when using the low-order method in the
loop with the 6-DOF model. Detailed results can be seen in [24, 25]. The loss in roll damping seen near stall has been simulated using the low-order method. Figure 2.6 shows the change in roll damping for a rectangular wing as the angle of attack is increased. As the wing approaches stall, the rolling moment coefficient due to roll rate \( C_{lp} \) trends towards zero. Typically, \( C_{lp} \) is negative meaning a roll rate would be opposed by the load distribution over the wing span, the opposite would be true if it were a positive value.

![Figure 2.6: Typical roll damping vs. Angle of Attack for a rectangular wing.](image)

Figure 2.7 shows the lift distribution for the same rectangular wing at a pre-stall and post-stall condition with a non dimensional roll rate of 0.02. This roll rate causes an increase in the local angle of attack on the right side of the wing and decrease in the angle on the left.
Figure 2.7: Comparison of pre-stall and post-stall $C_l$ distributions for a rectangular wing with a non dimensional roll rate of 0.02.
The pre-stall condition can be seen in Fig. 2.7a where higher local lift coefficients are seen on the right side of the span as opposed to the left. This induces a rolling moment in the opposite direction of the simulated roll rate. At the post-stall condition seen in Fig. 2.7b, the local lift coefficients show partial wing stall occurring, which yields a distribution that would add to the roll rate already prescribed (i.e. loss in roll damping).

The trajectories of two different aircraft, one with a rectangular wing and the other with a tapered wing, are shown in Figure 2.8.
The initial conditions for the simulation had the two aircraft operating near stall with a large elevator and rudder input. Asymmetric stall occurs in the simulation causing a rapid spiral decent of the aircraft. The trajectories highlight a well known phenomenon known as a stall-spiral descent, which can be dangerous to aircraft at low altitude (take off/departure and approach for landing).

2.5 Chapter 2 Summary

Background information was presented for post-stall aerodynamic prediction methods. A discussion was presented of existing linear methods which fail to predict stall and post-stall as well as high order CFD methods that are computationally expensive and ill suited for real time simulations. The crux behind the the NCSU low-order post-stall method was shown as well as the implementation of the model into aircraft flight dynamics. Early results from a 6-DOF model using the post-stall method has shown expected trajectories and behaviors for high angle of attack conditions.
Chapter 3

Approach for Generating Post-Stall Database

A database of post-stall airfoil and wing data is needed partly to address the lack of data in the literature, but primarily to support the ongoing effort of developing the low-order model of post-stall aerodynamics for finite wings mentioned previously. Several approaches could be used to generate the necessary data, one being through experimentation in a wind tunnel and another through computational approaches. The experimental route provides an opportunity to obtain a very large amount of data for any given geometry of interest, making it ideal for an in-depth analysis of a single wing or airfoil. However, for the wide range of geometries that will be of interest to study with respect to validation of the low-order method, the experimental approach is less practical. There are also issues associated with obtaining data at the desired flow conditions which makes having adequate experimental facilities that can provide such conditions a top priority. A computational approach through the use of CFD was chosen for initial studies of various wings and airfoils through post-stall. Using a CFD tool provides an avenue for studying numerous geometries with control of the flow conditions of interest.

Figure 3.1 describes the relationship between the low-order method and the CFD effort, and how the results from the latter are to be used for initial validation and further refinement.
of the former.

Figure 3.1: Approach for using CFD for initial validation and further refinement of the low-order method. Blue emphasizes that a consistent grid spacing is used with the 2D and 3D CFD discretizations.

Two dimensional CFD is performed on airfoil sections at varying Reynolds, Mach numbers and a full angle of attack sweep from 0 to 90 degrees. The outputs are fully viscous, nonlinear $C_l$, $C_d$ and $C_m$ curves which are then used as inputs to the low-order method. As previously described, the low-order method essential utilizes these nonlinear aerodynamic curves to "calibrate" an already existing method (VLM, Weissinger etc.) to predict stall and post-stall aerodynamic forces and moments. In order to begin validating this method that uses the 2D data to determine post-stall characteristics, CFD solutions are generated for 3D geometries.
utilizing the same 2D airfoil cross section. The 3D CFD solutions provide a comparison upon which initial validation of the low-order post-stall method can be based. The results of the 3D CFD and low-order method are compared on the basis of total force/moments and spanwise force/moment distributions. These comparisons as well as flow visualizations provided by CFD guide further development and refinement of the low-order method. As noted in Figure 3.1 it is important that the 2D and 3D CFD have similar discretization per unit span since the 2D solutions are used in the low-order method and the 3D CFD is used as a comparison. This was an effort to eliminate differing discretization from affecting the comparisons.

3.1 High Fidelity Computational Tool

The NASA Tetrahedral Unstructured Software System (TetrUSS) [26] is a package of loosely integrated software, developed to allow rapid aerodynamic analysis from simple to complex problems. The system has its origins in 1990 at the NASA Langley Research Center and has won the NASA Software of the Year Award twice. TetrUSS has been used on high priority NASA programs such as Constellation and the new Space Launch System for aerodynamic database generation along with work in the Aviation Safety Program.

The component software packages are assembled such that a user follows a systematic process during each application of TetrUSS. There are software packages for preparing geometries for grid generation (GridTool), generating unstructured surface and volume grids (VGRID) and calculating flow solutions (USM3D). Post-processing the solutions with TetrUSS can be done using the included SimpleView software or by easily converting for use with other commercial packages (eg Tecplot, EnSight etc.).

For preparing geometries for grid generation, GridTool is used to generate the necessary VGRID [27] input files. GridTool can read Non-uniform Rational B-Spline (NURBS) curves and surfaces through an Initial Graphics Exchange Specification (IGES) file, as well as PLOT3D point cloud surface definitions. The geometric surfaces are then defined by way of surface patch construction. Each patch has a specified boundary condition such as a viscous or inviscid surface
and a family definition for users to group related patches together. Grid spacing parameters are also defined and controlled within GridTool. Sources are placed in three dimensional space by a user in order to control the size and growth rates of the tetrahedral cells. Numerous classes of sources are available to control the grid topography. Nodal sources and line sources are typically used in most cases, while volume sources are available for use in specific cases requiring control over a large volume of the domain. Other parameters controlled in GridTool are the viscous layer spacing parameters and the maximum and minimum tetrahedral sizes.

VGRID is the unstructured grid generation tool used in the TetrUSS Package. Viscous grid generation is accomplished via the Advancing Layers Method (ALM) [28]. Tetrahedral cells are generated in an orderly manner, ”marching” nodes away from the surface. The size and growth of these cells is controlled by Equation 3.1.

\[
\Delta z_{i+1} = \Delta z_1 \left[ 1 + a(1 + b)^i \right]^i
\]  

(3.1)

In this equation, the height of the \(i^{th}\) layer is determined by an initial spacing parameter, \(\Delta z_1\),
and two stretching/growth factors $a$ and $b$. Once the height of the $i^{th}$ layer reaches the size of the background sources specified by the user in GridTool, no more cells are formed and viscous layer generation is complete. After the viscous layers are generated, VGRID then utilizes the Advancing Front Method (AFM) [29] for the generation of the inviscid portion of the volume grid. An example of the generated viscous grid generation and transition to the inviscid grid can be seen in Figure 3.3. VGRID can not always completely close the grid. When this occurs, an auxiliary code called POSTGRID is used to complete the formation of the remaining tetrahedral cells.

![Figure 3.3: Screen capture of viscous layers and inviscid grid near leading edge of an airfoil.](image)

The flow solver at the core of the TetrUSS package is USM3D [30]. USM3D is a parallelized, tetrahedral cell-centered, finite volume Reynolds Averaged Navier-Stokes (RANS) flow solver. It computes the finite volume solution at the centroid of each tetrahedral cell and provides several upwind schemes to compute inviscid flux quantities across tetrahedral faces. USM3D has several turbulence models implemented for use. The Spalart-Allmaras (SA) one-equation model, the Menter Shear Stress Transport (SST) two equation model and the k-epsilon two equation
model are some of the more common models. Some additional capabilities that USM3D has implemented are dynamic grid motion and overset grids which are currently used for stability and control calculations.

3.2 Chapter 3 Summary

The method for using CFD tools to assist in the development and validation of the low-order post-stall method was presented. The correlation between the 2D CFD solutions with the 3D CFD solutions in terms of grid density requirements was discussed. Also the basis upon which results would be utilized for comparisons between CFD and the low-order method was shown. Additionally, background on the NASA TetrUSS CFD package and all of it’s component parts was described.
Chapter 4

Methodology for Post-Stall
Reynolds-Averaged Navier-Stokes
CFD Simulations

The main aim of the current research is to develop an aerodynamic database of high fidelity flow solutions for a wide variety of 2D airfoils and 3D geometries. The flow solutions would include a large range of angles of attack to encompass pre-stall, stall and post-stall flow regimes. The data gained from these simulations are intended for use in assisting the further development and validation of the low-order method mentioned in the introduction as well as to fill gaps in the currently available high angle of attack aerodynamic data for arbitrary geometries. An efficient process to go from a geometry to a converged flow solution was developed and is discussed in the following sub-sections.

4.1 Geometry Generation

Traditional Computer Aided Design (CAD) software would be more than adequate for the creation of the desired geometries, however these tools are not geared specifically towards the
modeling of wings and airfoils. Understanding this, the recently released parametric modeling tool, Open Vehicle Sketch Pad (OpenVSP) [31], was chosen as the geometry generation tool. A flow chart showing the process of geometry generation can be seen in Figure 4.1. OpenVSP is a modeling package developed and released by NASA Langley Research Center in Hampton, Virginia. The unique concept that OpenVSP provides is that it allows a user to drag and drop generic aircraft components (such as a wing) into the modeling area, and directly manipulate familiar geometric parameters. Consequently, it is simple to insert a wing, change its root chord, tip chord, span, etc. and view the resulting geometry in real time. Aerodynamic reference quantities can also be automatically calculated for the user. Airfoil cross-section generation is also simplified. A user can select any 4 or 5 digit series NACA airfoil or load in a formatted airfoil coordinate file for use on any lifting surface.

The 3D wing and corresponding airfoil for each case to be analyzed were generated in OpenVSP. In order to read the geometry into GridTool, the file must be in the IGES format. Vehicle Sketch Pad does not output IGES files, thus each geometry must be exported as a Rhino3D formatted file. The Rhinoceros NURBS modeling package was used to convert the geometry into the necessary IGES file and to make small modifications to the geometry. Some grid generation failures were encountered due to how OpenVSP closes the trailing edge of the wing/airfoil geometries (it always forces a sharp trailing edge). In some cases the sharpness had to be removed to ensure successful grid generation.

4.2 Grid Generation

Grid generation parameters were generalized such that, between different geometries, parameters such as source placement and viscous spacing had minimal required changes. This meant that from initial geometry generation to a completed grid would require only a matter of hours. Establishing this commonality and routine for grid generation enabled the generation of adequate grids for many configurations in a short time span. An example placement of sources for a simple tapered wing is shown in Figure 4.2. A series of line sources are utilized, in the
spanwise direction of each wing at differing chord-wise locations. Anisotropic stretching [32] as high as 10:1 was used near the root of the wing, transitioning to isotropic cells near the wing tips. For viscous tetrahedral layer generation, the height of the first layer ($\Delta z_1$) is Reynolds number dependent. A viscous layers spacing tool called USGUTIL, was used to determine the height of the initial viscous layer. In order to have adequate number of cells in the viscous layers, the $y^+$ of the first node was set to be 3, this would ensure that the $y^+$ of the first cell center would be less than 1 (approx. 0.75) as is required for a fully viscous Navier-Stokes solution. The values used for the grid growth parameters ($a$ and $b$) in Eq. 3.1 were 0.15 and 0.02 respectively [32]. A grid sensitivity study was performed on a rectangular wing with aspect ratio (AR) of 12 to determine adequate grid sizing. It was found that a grid sizing of 5–9 million tetrahedral cells showed changes in $C_{L,\text{max}}$ of approximately 0.02 between the grids. This method of grid generation was applied to all 3D wing geometries with the typical grids averaging between 9–12 million tetrahedral cells.

For airfoil calculations, a quasi 2D grid was generated on a constant-chord, short-span wing, between two reflection plane boundary condition patches. Figure 4.3 shows a completed grid for an NACA 0012 airfoil. As noted in Chapter 3, a general goal was set to maintain very
similar grid density between the 3D wing grids and the airfoil grids per unit span. This is necessary because the airfoil results were being used as input data into the low-order method discussed in the introduction while the 3D wing results from USM3D were being used to assess the accuracy of the low-order method (Figure 3.1). Therefore a separate grid sensitivity study was not performed specifically for airfoils. Typical airfoil grids were on the order of 300,000 tetrahedral cells and were generated using a nearly identical source placement as the 3D wing.

4.3 Flow Solution Generation

All solutions with the USM3D solver were computed for steady state conditions using time-accurate Reynolds-Averaged Navier-Stokes (RANS). The limitations of RANS for modeling massively separated flows are well known. The more preferred Detached Eddy Simulation (DES) modeling will provide better physical representation of 3D separated flow, but with an order-of-magnitude more expense. Since this investigation requires generation of many flow solutions, the initial focus is to determine if time-accurate RANS can provide sufficient engineering accuracy for capturing the salient aerodynamic characteristics of wings at stall and post-stall conditions. Furthermore, a consistent modeling is desired between the 2D airfoils and 3D wings.

All computations were advanced at a characteristic time step of $\Delta t^* = \Delta t \cdot U_{\infty} / c_{ref} = 0.02$
using a second order time-accurate scheme with three-point backward differencing and physical
time stepping. The number of sub-iterations for each time step was set to between 10 and
15 to ensure adequate sub-iteration convergence. The Spalart-Allmaras (SA) [33] one equation
turbulence model was used almost exclusively, however some simulations were performed with
the two equation SST turbulence model to understand the difference in final solution quantities.
The solver was run on both a NASA Langley computer cluster and the North Carolina State
University High Performance Computing (NCSU HPC) cluster. Making use of USM3D’s parallel
computation capabilities, each grid was partitioned into 28-64 equal zones which could be loaded
onto 28–64 individual processors, reducing calculation times proportionally. To further increase
productivity, a series of Unix scripts were developed to generate the required input files for job
submission and minor post-processing of completed jobs.

4.4 Solution Convergence

Convergence of the solutions was monitored by generating convergence plots such as that seen
in the Figure 4.4. Unix scripts were used to compile all of the convergence information contained
in the USM3D output files into a Tecplot format.

Each plot shows the logarithm of the residual over each iteration and the changes in the
aerodynamic coefficients. The criteria for a fully converged solution was for each plot to show a
leveling off of the quantities under consideration. These plots allowed for rapid determination
of whether any given solution had reached a converged state (Figure 4.4a) or if the solution
had attained an unsteady solution shown by oscillatory convergence (Figure 4.4b).

4.5 Post-Processing

After solution convergence was verified, the data is processed so that total forces and moments,
the spanwise distribution of forces and moments, and flow visualization may be studied. As was
previously mentioned, in some cases an oscillatory solution develops rather than single steady-
state values for the forces and moments. This has only been observed for 2D airfoil solutions at very high angles of attack.

4.5.1 Total Forces and moments acting on the entire wing/configuration

The first outputs of interest are the total force and moment coefficients acting for a wing/configuration. A script was used to extract the body-axis force and moment coefficients $[C_X C_Y C_Z C_{Mx} C_{My} C_{Mz}]$, defined parallel and perpendicular to the body coordinate system, and stability-axis coefficients $[C_L C_D]$, defined parallel and perpendicular to the free-stream velocity. Figure 4.5 shows an example of lift coefficient results for a 2D airfoil and a 3D wing which used the same airfoil cross section.

4.5.2 Spanwise distribution of forces on a surface

The other output of interest is the spanwise distribution of force coefficients, particularly the spanwise lift coefficient. The PREDISC utility [34], was used for extracting this information. PREDISC simultaneously loads the grid files containing the surface grid and a converted
Figure 4.5: Converged vs. AOA for 2D airfoil and 3D rectangular wing.

TetrUSS solution file containing only surface data. Data extraction planes can be arbitrarily defined (see Figure 4.6) and PREDISC will output surface pressure and skin friction coefficients, $C_p$ and $C_f$ respectively, along the surface discretized according to a fine mesh of $x/c$, $y/c$ locations. The surface pressure and skin friction force coefficients are integrated to approximate body-axis force coefficients. An example lift coefficient distribution obtained by integrating the $C_p$ and $C_f$ values at each extraction plane is shown in Figure 4.6. Figure 4.7 below shows positive sign conventions for both $C_p$ and $C_f$ along an airfoil profile.

4.6 Chapter 4 Summary

The method for using the NASA TetrUSS CFD package was presented in this chapter. Methods for geometry generation with OpenVSP and manipulation with Rhino 3D was discussed along with grid generation techniques. The post-processing of results was also discussed in terms of ensuring adequate solution convergence as well as using PREDISC for sectional data extraction.
Figure 4.6: Screen shot of PREDISC code (left) and a plot of the extracted spanwise $C_l$ distribution (right) on a rectangular wing AR = 12.

Figure 4.7: Positive sign convention for $C_f$ and $C_p$. 
Chapter 5

Results

The results shown in this section represent the current level of development of the CFD database. The results from CFD analysis of cambered, symmetric and thin airfoils will be presented along with 3D wing solutions with rectangular, tapered and swept planforms.

5.1 2D Airfoil Results

Three different airfoils have been studied and added to the CFD aerodynamic database to date. Additional airfoils will be added as needed in future efforts for further development of the database and as required for validation of the low-order post-stall method discussed previously. The airfoils chosen exhibit different stall and post-stall behavior and offer insight into how certain airfoil geometries will tend to behave at high angles of attack. Results for a symmetric airfoil (NACA 0012), a cambered airfoil (NACA 4415) and a very thin airfoil (NACA 63006) are shown in this section. The geometries of the airfoils are shown in Figure 5.1.

An interesting phenomenon that was encountered while developing the airfoil database was the tendency for the airfoil solutions to exhibit oscillatory behavior in the force and moment convergence histories at angles of attack of approximately 40 degrees and above. The cause of these oscillations was determined using flow visualization which revealed periodic vortex shedding from the upper surface of the airfoil. A process to average the oscillatory behavior
and determine peak to peak amplitudes was established. A post-processing MATLAB script was developed to read all of the force and moment history files for an airfoil and detect the oscillatory behavior. For any angle of attack that displayed this behavior, the script identified two complete cycles at the end of the convergence history and determined the mean value along with the peak-to-peak amplitude. The plots in Figure 5.2 illustrate the approach used for the processing of the raw CFD data with this code.

5.1.1 Reynolds Number Effects Through Post-Stall

The general effects of Reynolds number on the aerodynamic quantities, specifically $C_{l,max}$, are known and have been observed with the current work as well. However, it is important to extend this to the post-stall region.
Figure 5.2: Illustration of approach used for averaging oscillatory airfoil CFD convergence history.

Figure 5.3: Effect of Reynolds number on lift coefficient for a NACA 4415 airfoil. USM3D/SA, Re= 3 million, $M_{\infty} = 0.2$. 
Figure 5.3 shows a comparison of the lift curves for the NACA 4415 airfoil at three different Reynolds numbers (3, 6 and 10 million). The increased maximum lift coefficient with increasing Reynolds number is expected. In the post stall region between angle of attack of 40 and 70 degrees, an additional effect of the Reynolds number is seen. The region falls directly where the oscillatory solutions develop; the values shown in Figure 5.3 are the averaged values from any oscillating data. After this recovery region, the solutions tend to follow a similar path out to 90 degrees.

5.1.2 Sharp vs. Blunt Trailing Edge Geometries

It has been noted by Hoerner [35], that the trailing edge shape of an airfoil has a distinguishable effect on the $C_l$ vs. $\alpha$ curve. When comparing an airfoil with a sharp trailing edge with the same airfoil but with a blunt trailing edge, it is found the the maximum lift coefficient is seen to be higher for the blunt trailing edge airfoil. The plot in Figure 5.4 displays results for an NACA 4415 airfoil with both a sharp and blunt trailing edge at a Reynolds number of 3 million. The blunt TE geometry was generated by removing the final one percent of the chord of the sharp trailing edge geometry, thus no other alterations to the geometry are present. The expected trend is seen with the blunt trailing edge case having a higher maximum lift coefficient. It is interesting to see that this effect seems to continue all the way past stall and through to approximately $\alpha = 70$ degrees, after which the lift curves coincide. The flow mechanism that allows for this all the way through post stall is not readily apparent from the CFD at this time, but warrants further investigation.
Figure 5.4: Effects of trailing-edge sharpness for a NACA 4415 airfoil. USM3D/SA, Re= 3 million, $M_\infty = 0.2$.

5.1.3 Comparison of Post-Stall Characteristics of Three Airfoil Geometries

Airfoils that exhibit different characteristics in terms of maximum lift coefficient and stall behavior were analyzed for addition to the post-stall database. In Figure 5.5, lift curves for three airfoils are shown from 0 to 90 degrees angle of attack. A cambered airfoil (NACA 4415), a symmetric airfoil (NACA 0012) and a very thin airfoil (NACA 63006) are compared in the figure. The NACA 4415 and NACA 0012 both exhibit trailing edge stall behavior. That is, flow separation begins at the trailing edge and progresses towards the leading edge as the angle of attack increases. This can be seen in the lift curves as both the airfoils have a more “typical” stall behavior, with a clear maximum $C_l$ followed by a fairly steep negative slope. It is interesting to note that although the NACA 4415 airfoil has a higher $C_{l,max}$ as compared to the NACA 0012, the two airfoils have very similar maximum recovery lift coefficients at approximately 50 degrees angle of attack. The NACA 63006 produced a much more unusual lift curve. This
airfoil is categorized as having “thin airfoil” stall characteristics. Due to the presence of a severe adverse pressure gradient around the leading edge at even low angles of attack, the flow over the upper surface completely separates. This sudden onset of flow separation can been seen by the sudden leveling of the lift coefficient at around 10 degrees. After this initial “stall”, the airfoil recovers past the initial stall lift coefficient and reaches nearly the same maximum recovery $C_l$ at $\alpha = 50$ degrees as the NACA 4415 and NACA 0012. The differences seen between these three airfoils in the 40 to 60 degree range can mainly be attributed to thickness and leading-edge radius effects.

Figure 5.5: Comparison of $C_l$ vs. $\alpha$ for three airfoil types. USM3D/SA, Re= 3 million, $M_\infty=0.2$.

An enlarged plot of the NACA 63006 in the region of incipient stall shown in Figure 5.6 with comparison to experimental data obtained from Abbott and von Doenhoff [2] and also
noted by Hoerner [35]. It should be noted that since the CFD solutions were fully turbulent (no transition model was used) no effects of a laminar separation bubble near the leading edge were modeled, a phenomenon noted by Leishman [36] and in other studies of similar airfoils [37].

![Figure 5.6: Comparison of $C_l$ vs. $\alpha$ from CFD and experiment [2] for the NACA 63006 thin airfoil.](image)

Figures 5.7 and 5.8 show Mach number contour plots along with surface $C_p$ distributions for the NACA 4415 and NACA 63006 airfoils at different stages along their respective lift curves.
The NACA 4415 flow visualization in Fig. 5.7 at $\alpha = 17$ deg. ($C_{l,\text{max}}$) shows the trailing edge flow separation propagating forward along the chord. This is also evident in the $C_p$ distribution.
As the angle of attack is extended above stall and into post-stall, a large wake region is seen to develop accompanied by a loss of much of the suction peak. At $\alpha = 50$ deg, the Mach contours and $C_p$ distribution show a snapshot of the unsteady behavior that exists. At $\alpha = 88$ degrees the solution shows massively separated flow typical of bluff bodies. A small suction peak is still generated around the leading edge resulting in a small amount of lift generation.
The flow visualization of the NACA 63006 CFD solutions in Fig. 5.8 provides an interesting view of thin-airfoil stall behavior. It can be seen from Figure 5.8 that, at $\alpha = 6$ degrees, there is a significant suction peak followed by a steep adverse gradient. The leading edge at this angle shows a small area of separated flow. Looking back at Figure 5.6, a slight change in the slope of the lift curve can be identified in this region. At 10 degrees the flow separation has propagated completely from the leading edge to trailing edge; this corresponds with the maximum lift coefficient prior to the airfoil entering into the recovery region in post-stall. The $C_p$ distribution at $\alpha = 24$ degrees shows a larger internal area signaling an increase in the normal force generated, the majority of which still falls along the lift direction. The airfoil reaches another “$C_{l,max}$” at approximately 50 degrees and also shows unsteady characteristics.

### 5.2 3D Wing Results

Results of five different 3D wing geometries are presented in the following subsection. The plots in Figures 5.9 and 5.10 display the lift curves for wing planform geometries with varying taper and varying degrees of aft sweep form 0 to 90 degrees angle of attack.
Figure 5.9: Effect of taper on unswept wing $C_L$ vs. $\alpha$. USM3D/SA, $Re= 3$ million, $M_\infty= 0.2$.

Figure 5.10: Effect of sweep on constant-chord wing $C_L$ vs. $\alpha$. USM3D/SA, $Re= 3$ million, $M_\infty=0.2$. 
The database will eventually include wing geometries that have a combination of both sweep and taper, similar to civilian transport aircraft. This data will be of interest to continuing efforts aimed at improving safety of transport aircraft. Detailed spanwise load distributions and surface streamline flow visualizations are presented in this thesis for five wings of AR = 12 with rectangular, tapered and swept planforms.

5.2.1 Rectangular Wing

The results presented are for a rectangular wing with an aspect ratio of 12 and a NACA 4415 airfoil cross section. The solutions for this case were produced for a Reynolds number of 3 million based on the chord and a Mach number of 0.2.

The plots in Figure 5.11 show spanwise $C_l$ distributions calculated from the CFD solutions for angles of attack near stall and into post-stall. It is seen that at an angle of attack of 18 degrees, a sawtooth pattern in the $C_l$ distribution is present. The spanwise extent of this sawtooth pattern is seen to increase as angle of attack is increased to 22 degrees and 28 degrees. Correlating these
load distributions with flow visualization at the same angles of attack enlightened the reason for the sawtooth patterns. Through the use of surface streamlines it can be seen in Figure 5.12 that as the angle of attack increases, reversed flow is seen aft of the separation line and shows the presences of multiple stall cells forming along the semi-span of the wing. This causes certain sections along the wing to have more attached flow than others, generating the oscillations in the local lift coefficients going from the root to the tip. This stall cell formation eventually dissipates as the flow over the upper surface becomes fully separated and the region of reversed flow reaches the leading edge of the wing as can be seen at 28 degrees angle of attack. The streamlines in Figure 5.12 seem to suggest that even in high angle of attack situations the surface flow is still relatively in the chord-wise direction for the majority of the semi-span. There are some variations near the borders of stall cells and near the wing tip as would be expected due to the influence of the tip vortex.
Figure 5.12: $C_p$ contours and surface streamlines on a semi-span rectangular wing, AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 
(a) $\alpha = 14$ deg.

(b) $\alpha = 18$ deg.
(c) $\alpha = 22 \text{ deg.}$

(d) $\alpha = 28 \text{ deg.}$
5.2.2 Tapered Wing

Results for a tapered wing with a taper ratio of 0.5 and aspect ratio of 12 are presented in Figures 5.13 and 5.14. This wing also has the same NACA 4415 airfoil cross section and the solutions were generated for a Mach number of 0.2 and Reynolds number of 3 million based on the mean aerodynamic chord. The $C_l$ distributions in Figure 5.13 show that the highest $C_l$ is seen near the middle of the semi-span at 14 degrees angle of attack. Such increases in local span $C_l$ can be an indicator of impending stall in that region. As anticipated, a stall cell develops near this high $C_l$ area of the semi-span at 18 degrees, which is denoted by the severe drop in the local lift coefficients. Looking at the surface streamlines in Figure 5.14 shows the existence of the stall cell as predicted by CFD. Similar to the rectangular case, these stall cells disappear once the angle of attack is high enough that the flow on the upper surface is entirely separated and nearly all flow is reversed. This reversed flow is still seen to be mainly in the chordwise direction along the majority of the semispan at these highly separated conditions.

(a) Local $C_l$ distribution at various angles of attack (b) Corresponding points on $C_L$ vs. $\alpha$ curve

Figure 5.13: Comparison of local $C_l$ distribution at varying angles of attack for a tapered wing, $\lambda = 0.5$ and AR = 12. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 

53
Figure 5.14: $C_p$ contours and surface streamlines on a tapered wing, $\lambda = 0.5$, $AR = 12$, symmetry plane indicated by black line. USM3D/SA, $Re = 3$ million, $M_{\infty} = 0.2$. 
(c) $\alpha = 14$ deg.

(d) $\alpha = 18$ deg.
(e) $\alpha = 30$ deg.

(f) $\alpha = 42$ deg.
5.2.3 Swept Wing Cases

Results are presented for constant chord swept wing cases with 10, 20 and 30 degrees of sweep, aspect ratio of 12 and an NACA 4415 cross section parallel to the plane of symmetry. The solutions were generated at a Mach number of 0.2 and Reynolds number of 3 million based on the chord parallel to the free stream. Spanwise $C_l$ distributions are shown in Figures 5.15 - 5.17.

Stall progression from near the tip of the wing towards the inboard sections of the wing is evident and expected in all cases, as is the decrease in the maximum lift coefficient as sweep is increased. Looking back to the unswept rectangular wing, stall cells dominate the regions of separated flow just before, during and just after stall. It should be noted that as sweep angle increases, the presence of the stall cells diminishes. Comparing the local $C_l$ distributions for 20 degrees angle of attack for the 10 and 20 degree swept wings (Figures 5.15 and 5.16) shows a less pronounced sawtooth like oscillation between the two geometries with no sawtooth pattern emerging in the 30 degree swept case in Figure 5.17.

![Graph](attachment:image.png)

(a) Local $C_l$ distribution at various angles of attack  
(b) Corresponding points on $C_L$ vs. $\alpha$ curve

Figure 5.15: Comparison of local $C_l$ distribution at various angles of attack on a swept wing, $\Lambda = 10$ deg., AR = 12. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 

57
Figure 5.16: Comparison of local $C_l$ distribution at various angles of attack on a swept wing, $\Lambda = 20$ deg., $AR = 12$. USM3D/SA, $Re = 3$ million, $M_\infty = 0.2$.

Looking at the 30 degree swept wing in Figure 5.17, as the angle of attack increases to 30 degrees the inboard local lift coefficients (calculated here based on a streamwise section, not
chordwise) are operating at well above observed maximum lift coefficient seen for the NACA 4415 2D airfoil CFD solutions (approx 1.75). This phenomenon has also been observed in past studies of swept wings. Hunton and James [38] as well as Harper and Maki [11] note this same behavior, which is most pronounced at inboard sections near the root, but can occur along most of the semi-span. They describe this as an effect of a "natural boundary-layer control" that delays stall on the inboard section of a swept wing. It should also be noted that the percent increase in the local sectional $C_{l,max}$ goes up with the sweep angle.

Flow visualizations of the three swept wing geometries are presented in Figures 5.18 - 5.20. Through surface streamlines, the separation patterns can be observed for the three cases. The 10 degree swept case in Figure 5.18, flow separation starts at the trailing edge and progresses towards the leading edge. The inboard section separation progression is seen to be slower than the outboard portion. The presence of stall cells is evident on the outboard portion of the wing at 20 degrees angle of attack (corresponding to the $C_l$ distribution). The flow is essentially fully separated from the upper surface at 30 degrees angle of attack. The presence of stall cells is almost nonexistent in the 20 degree swept wing in Figure 5.19. A small cell is seen at 20 degrees angle of attack, but the extent is very small compared with the 10 degree swept case and the non-swept wing. Evidence of the delayed stall near the root of the 30 degree swept wing can be seen in Figure 5.20. The inboard section does not show significant flow separation even at 30 degrees angle of attack, while the rest of the wing is fully separated. No evidence of stall cell formation can be seen at any of the angles of attack presented.
Figure 5.18: $C_p$ contours and surface streamlines on a swept wing, $\Lambda = 10$ deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 
(c) $\alpha = 10$ deg.

(d) $\alpha = 16$ deg.
(e) $\alpha = 20$ deg.

(f) $\alpha = 30$ deg.
Figure 5.19: $C_p$ contours and surface streamlines on a swept wing, $\Lambda = 20$ deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 
(a) $\alpha = 10$ deg.

(b) $\alpha = 16$ deg.
(c) $\alpha = 20$ deg.

(d) $\alpha = 30$ deg.
Figure 5.20: $C_p$ contours and surface streamlines on a swept wing, $\Lambda = 30$ deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 
(a) $\alpha = 10 \text{ deg.}$

(b) $\alpha = 16 \text{ deg.}$
(c) $\alpha = 20$ deg.

(d) $\alpha = 30$ deg.
5.3 Comparisons Between the Low-Order Post-Stall Method and CFD Simulations

The following subsections show preliminary comparisons between 3D CFD simulations and the corresponding low-order method simulations for the rectangular wing and tapered wing geometries. A short discussion is also made on the issues that low-order modeling of swept wings at stall and post-stall will present. Through comparisons such as this and with other geometries, improvements to the model’s ability to effectively predict forces and moments through post-stall can be made.

5.3.1 Rectangular Wing

A comparison between the low-order method and CFD for the same semispan rectangular wing with an AR = 12 is presented below in Figures 5.21 and 5.22. It can be seen that in general, the low-order has very good behavioral agreement with the CFD wing $C_L$ and $C_l$ distributions. The low-order method predicts nearly the same lift coefficients up to stall and even slightly into post-stall. After approximately 30 degrees angle of attack, the low-order method tends to diverge from the CFD predictions. Looking at the At pre-stall angles of attack such as the case at 16 degrees, there is almost no difference between the CFD and the low-order method $C_l$ distributions. Moving past stall and into post-stall, there are some differences in magnitudes, especially when stall cells are present in the CFD solutions. However, the low-order method has shown some encouraging results for this region. First, the extent of the region of stall along the semispan is well predicted in all of the cases presented. Second, the low-order method has generated sawtooth pattern solutions just like what was observed in the CFD solutions. This can be easily seen in the load distributions at 22 degrees angle of attack. An additional $C_l$ distribution at 24 degrees angle of attack is presented in Figure 5.23. In this figure, it can be seen the the sawtooth patterns predicted by both the low-order method and CFD are in very good agreement with regards to extent and magnitudes of the oscillations. In early efforts with
the low-order method, these oscillations were seen as "non physical" and were thought to be a problem with the numerical algorithm. However through this comparison, it seems that what the low-order method is predicting may actually be a physical phenomenon. More investigation is warranted as to how and why the low-order method produces these solutions.

Figure 5.21: Comparison between CFD and Low-Order Method of wing $C_L$ v.s. $\alpha$ for a rectangular wing, AR = 12. Low-Order Method -20 panels, CFD - USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 

Figure 5.22: Local $C_l$ distribution on a rectangular wing, AR = 12. Low-Order Method - 20 panels, CFD - USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 

(a) $\alpha = 16$ deg. 

(b) $\alpha = 20$ deg. 

(c) $\alpha = 22$ deg. 

(d) $\alpha = 28$ deg.
Figure 5.23: Comparison of local $C_l$ distributions on a rectangular wing, AR = 12 at 24 degrees angle of attack. Low-Order Method -20 panels, CFD - USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 

![Graph showing comparison of local $C_l$ distributions](image)
5.3.2 Swept Wing

No comparisons between CFD and the LO method for the swept wing geometries have been run as of the writing of this thesis. The CFD analysis of the swept wing geometries presented in this chapter as well as in Appendix B bring to light two things that must be modified to predict swept-wing aerodynamics in the low order model: (1) The assumption of chord-wise flow is not valid for the swept-wing cases especially in the approach to stall and in immediate post-stall. The low-order model must take this into account by defining 2-D input data along the expected flow direction. A more thorough CFD analysis of various swept wing geometries will likely be required to implement this in the low order method.

![Possible method to address swept wing input data.](image)

(2) The sectional lift coefficient distributions along the span computed from the CFD show that sweep causes a rather significant $\Delta C_{l_{\text{max}}}$ compared to the 2-D airfoil input data that appears
to be a function of the span. A comprehensive CFD study is needed to develop a lookup
table quantifying the $\Delta C_{l_{\text{max}}}$ as a function of spanwise distance and sweep angle. One possible
method, shown in Figure 5.24, is to determine the lift curve at multiple spanwise locations.
This can be done by extracting the $C_p$ and $C_f$ values and integrate them to determine the local
$C_l$.

5.4 Chapter 5 Summary

Results were presented from CFD solutions for three airfoil geometries and five different finite
wing planform geometries. General discussion was made based on the trends seen in the solutions
through post-stall as well as analyzing flow visualizations of the solutions. Comparisons between
the CFD solutions and the low-order method were then presented. It was seen that in general,
the low-order method is showing good behavior agreement with the CFD solutions. There is still
more research to be conducted to understand why the CFD is over predicting in the stall and
post-stall regimes, specifically looking at tapered and swept wing geometries. It is recommended
that more tapered wing geometries with various taper ratios and airfoil cross-sections be run in
CFD for comparison. Additional effort needs to be also placed on additional studies of swept
wings in order to correctly modify how the low-order method uses 2D input data and how that
data needs to be modified.
Chapter 6

Conclusions

This thesis presents research related to the development of an aerodynamic database of airfoils and wings through post-stall angles of attack. Such data has potential for use in modeling post-stall flight dynamics of fixed-wing and rotary-wing aircraft, and for prediction of wind turbine performance. Except for a few sources, there is a dearth of post-stall aerodynamic data from experimental or computational studies. One objective of the work is to fill this gap in knowledge. Further motivation behind developing this database is its potential for use in the validation and refinement of a low-order post-stall aerodynamics prediction method being developed for wings and aircraft configurations. The current approach to the development of this database is to use the NASA TetrUSS CFD package to analyze the geometries using Reynolds-Averaged Navier-Stokes equations. The CFD analyses enables flow solutions to be generated for various geometries (both 2D and 3D) in a more rapid fashion than would be possible with experimental work, besides providing detailed spanwise lift coefficients, and separation patterns that are not easily available from experimental studies.

Results are first presented for three airfoils that exhibit different stall and post-stall behaviors. The results for these and other airfoils are used as input data for the low-order post-stall prediction method for finite wings. Another objective in obtaining these results is that studying the post-stall behavior of these airfoils may lead to the development of a rapid method of gener-
ating airfoil force and moment curves from 0 to 90 degrees angle of attack using a combination of results from XFOIL and other simplified models such as the flat plate theory for very high angles of attack. Next, results for finite wings were presented to illustrate the effects of taper ratio and sweep angles on stall and post-stall behavior. The results for the rectangular and tapered unswept wings show that the flow along wing sections is nominally two-dimensional even at post-stall conditions. The nominally two-dimensional flow provides confidence in the use of sectional data in modeling post-stall aerodynamics of finite wings. However, there is noticeable variation in the shape of the upper-surface flow separation line, resulting in saw tooth oscillations in the spanwise lift-coefficient distributions.

In contrast to the results for unswept wings, the results for swept wings are seen to be highly three dimensional, as expected. At and beyond stall, there is significant spanwise flow on the upper surface. As a result of this spanwise flow (resulting from spanwise pressure gradients) and the higher lift coefficients on the outboard portions of the wing in pre-stall conditions, the outboard portions of the swept wings stall first while the root portion maintains lift coefficients much higher than the maximum lift coefficient in two-dimensional flow. The consequences of such behavior for aircraft stall characteristics, namely tip stall with associated rolling moment and pitch-up moment at stall, are well known. In the context of the low-order modeling, the significant three-dimensional flow on swept wings at and beyond stall poses serious stumbling blocks. It remains to be seen how these effects can be captured correctly and efficiently in a low-order aerodynamic model that can be used in real-time flight dynamics simulation. The benefit of such CFD studies, however, is that the detailed results do provide the very type of sought-after information for developing phenomenological augmentation of low-order approaches even when the flow is not entirely two-dimensional. In follow-on work, continued expansion of the database will progress with the addition of more airfoil and wing geometries. Studies of wing-tail geometries and effects of angular velocities are also planned.

Comparisons of the finite wing results from the CFD simulations with results from the low-order method show promise with the current method. Comparisons with the unswept rect-
angular wing showed excellent prediction of both the maximum wing lift coefficient and the corresponding stall angle of attack. The results differ slightly when entering deeper post-stall, but given that the flow is massively separated in this region, the relative magnitudes and behavior are encouraging for the low-order method. Local lift coefficient distributions showed very good agreement both in pre-stall and post-stall. It has been noticed in some low-order method solutions, that a sawtooth like pattern develops in the load distributions. It is not clear if these sawtooth oscillations have any correspondence with similar oscillations seen in the results predicted by high fidelity CFD solutions.

Tapered wing comparisons showed encouraging results. In prestall conditions, the lift curves are nearly identical and the load distributions are in good agreement with a small offset. This offset seems to grow slightly when approaching stall and post-stall. However, there is generally excellent comparison to the spanwise extent of stall. The prediction of \( \alpha_{stall} \) by the low-order method compares well with CFD but seems to over predict \( C_{L,max} \) as well as post-stall lift coefficients. This was also noticed in the rectangular wing case. One possible cause of this is the fact that the airfoil input data has a recovery region while the finite wing CFD results show no indication of this recovery. It is recommended that a study be performed that would determine whether a sectional airfoil on a finite wing has a recovery region. This could be done by extracting sectional \( C_p \) and \( C_f \) data from 3-D wing results from 0 to 90 degrees angle of attack. Integrating the sectional data would yield a "3-D" airfoil \( C_l \) vs. \( \alpha \) curve and enlighten as to whether an airfoil section on a 3-D wing truly exhibits the same behavior as it does in 2-D flow.

Swept wing comparisons were not made in this thesis. This is mainly due to the known fact that the current approach of using airfoil input data in the chordwise direction will not be sufficient at stall and post-stall conditions. It was shown with flow visualization of CFD results that the assumption of chordwise flow is severely violated once trailing edge separation begins along the semispan. Additionally, it was shown that there is a \( \Delta C_{L,max} \) at sectional locations along the semispan of a swept wing. This change in the local maximum lift coefficient is seen to
be a function of both sweep angle and the non-dimensional semispan location. Another study of 
the swept wing results and additional CFD work is need to discern what input data alterations 
would be required for the low-order method to predict stall and post-stall comparisons.
References


Appendices
Appendix A

Airfoil Flow Visualization

A.1 NACA 4415

The following Figures present the instantaneous non-dimensional Y-direction vorticity contours for the NACA 4415 Airfoil as predicted by USM3D using the SA turbulence models at a Mach number of 0.2 and Reynolds number of 3 million. The Y-direction was plotted as it is the dominate term in 2-D flows.
Figure A.1: Non-dimensional y direction vorticity contours on a NACA 4415 airfoil. USM3D/SA, Re = 3 million, $M_{\infty} = 0.2$
A.1. 1: $\alpha = 00.0$ deg.

A.1. 2: $\alpha = 02.0$ deg.

A.1. 3: $\alpha = 04.0$ deg.

A.1. 4: $\alpha = 06.0$ deg.

A.1. 5: $\alpha = 08.0$ deg.

A.1. 6: $\alpha = 09.0$ deg.
A.1. 7: $\alpha = 10.0$ deg.
A.1. 8: $\alpha = 11.0$ deg.
A.1. 9: $\alpha = 12.0$ deg.
A.1. 10: $\alpha = 13.0$ deg.
A.1. 11: $\alpha = 14.0$ deg.
A.1. 12: $\alpha = 15.0$ deg.
A.1. 13: $\alpha = 16.0$ deg.

A.1. 14: $\alpha = 17.0$ deg.

A.1. 15: $\alpha = 18.0$ deg.

A.1. 16: $\alpha = 19.0$ deg.

A.1. 17: $\alpha = 20.0$ deg.

A.1. 18: $\alpha = 21.0$ deg.
A.1. 19: $\alpha = 22.0 \text{ deg.}$

A.1. 20: $\alpha = 23.0 \text{ deg.}$

A.1. 21: $\alpha = 24.0 \text{ deg.}$

A.1. 22: $\alpha = 25.0 \text{ deg.}$

A.1. 23: $\alpha = 26.0 \text{ deg.}$

A.1. 24: $\alpha = 27.0 \text{ deg.}$
A.1. 25: $\alpha = 28.0$ deg.

A.1. 26: $\alpha = 29.0$ deg.

A.1. 27: $\alpha = 30.0$ deg.

A.1. 28: $\alpha = 31.0$ deg.

A.1. 29: $\alpha = 33.0$ deg.

A.1. 30: $\alpha = 35.0$ deg.
A.1. 31: \( \alpha = 37.0 \) deg.

A.1. 32: \( \alpha = 39.0 \) deg.

A.1. 33: \( \alpha = 41.0 \) deg.

A.1. 34: \( \alpha = 43.0 \) deg.

A.1. 35: \( \alpha = 45.0 \) deg.

A.1. 36: \( \alpha = 46.0 \) deg.
A.1. 37: $\alpha = 48.0$ deg.

A.1. 38: $\alpha = 50.0$ deg.

A.1. 39: $\alpha = 52.0$ deg.

A.1. 40: $\alpha = 54.0$ deg.

A.1. 41: $\alpha = 56.0$ deg.

A.1. 42: $\alpha = 58.0$ deg.
A.1. 43: $\alpha = 60.0$ deg.

A.1. 44: $\alpha = 62.0$ deg.

A.1. 45: $\alpha = 64.0$ deg.

A.1. 46: $\alpha = 66.0$ deg.

A.1. 47: $\alpha = 68.0$ deg.

A.1. 48: $\alpha = 70.0$ deg.
A.1. 49: $\alpha = 72.0$ deg.

A.1. 50: $\alpha = 74.0$ deg.

A.1. 51: $\alpha = 76.0$ deg.

A.1. 52: $\alpha = 78.0$ deg.

A.1. 53: $\alpha = 80.0$ deg.

A.1. 54: $\alpha = 82.0$ deg.
A.1. 55: $\alpha = 84.0$ deg.

A.1. 56: $\alpha = 86.0$ deg.

A.1. 57: $\alpha = 88.0$ deg.
A.2 NACA 0012

The following Figures present the instantaneous non-dimensional Y-direction vorticity contours for the NACA 0012 Airfoil as predicted by USM3D using the SA turbulence models at a Mach number of 0.2 and Reynolds number of 2 million. The Y-direction was plotted as it is the dominate term in 2-D flows.
Figure A.2: Non-dimensional y direction vorticity contours on a NACA 0012 airfoil. USM3D/SA, Re = 2 million, $M_\infty = 0.2$
A.2.1: $\alpha = 00.0$ deg. 

A.2.2: $\alpha = 02.0$ deg. 

A.2.3: $\alpha = 04.0$ deg. 

A.2.4: $\alpha = 06.0$ deg. 

A.2.5: $\alpha = 08.0$ deg. 

A.2.6: $\alpha = 09.0$ deg.
A.2. 7: $\alpha = 10.0$ deg.

A.2. 8: $\alpha = 11.0$ deg.

A.2. 9: $\alpha = 12.0$ deg.

A.2. 10: $\alpha = 13.0$ deg.

A.2. 11: $\alpha = 14.0$ deg.

A.2. 12: $\alpha = 15.0$ deg.
A.2. 13: $\alpha = 16.0$ deg.

A.2. 14: $\alpha = 17.0$ deg.

A.2. 15: $\alpha = 18.0$ deg.

A.2. 16: $\alpha = 19.0$ deg.

A.2. 17: $\alpha = 20.0$ deg.

A.2. 18: $\alpha = 22.0$ deg.
A.2. 19: $\alpha = 24.0$ deg.

A.2. 20: $\alpha = 26.0$ deg.

A.2. 21: $\alpha = 28.0$ deg.

A.2. 22: $\alpha = 30.0$ deg.

A.2. 23: $\alpha = 32.0$ deg.

A.2. 24: $\alpha = 34.0$ deg.
A.2. 25: $\alpha = 36.0$ deg.
A.2. 26: $\alpha = 38.0$ deg.
A.2. 27: $\alpha = 40.0$ deg.
A.2. 28: $\alpha = 42.0$ deg.
A.2. 29: $\alpha = 44.0$ deg.
A.2. 30: $\alpha = 46.0$ deg.
A.2. 31: $\alpha = 48.0$ deg.

A.2. 32: $\alpha = 50.0$ deg.

A.2. 33: $\alpha = 52.0$ deg.

A.2. 34: $\alpha = 54.0$ deg.

A.2. 35: $\alpha = 56.0$ deg.

A.2. 36: $\alpha = 58.0$ deg.
A.2. 37: $\alpha = 60.0$ deg.
A.2. 38: $\alpha = 62.0$ deg.
A.2. 39: $\alpha = 64.0$ deg.
A.2. 40: $\alpha = 66.0$ deg.
A.2. 41: $\alpha = 68.0$ deg.
A.2. 42: $\alpha = 70.0$ deg.
A.2. 43: $\alpha = 72.0$ deg.

A.2. 44: $\alpha = 74.0$ deg.

A.2. 45: $\alpha = 76.0$ deg.

A.2. 46: $\alpha = 78.0$ deg.

A.2. 47: $\alpha = 80.0$ deg.

A.2. 48: $\alpha = 82.0$ deg.
A.2. 49: $\alpha = 84.0$ deg.
A.2. 50: $\alpha = 86.0$ deg.
A.2. 51: $\alpha = 88.0$ deg.
A.2. 52: $\alpha = 90.0$ deg.
A.3 NACA 63006

The following Figures present the instantaneous non-dimensional Y-direction vorticity contours for the NACA 63006 Airfoil as predicted by USM3D using the SA turbulence models at a Mach number of 0.2 and Reynolds number of 3 million. The Y-direction was plotted as it is the dominate term in 2-D flows.
Figure A.3: Non-dimensional y direction vorticity contours on a NACA 63006 airfoil. USM3D/SA, Re = 3 million, $M_\infty = 0.2$
A.3.1:  \( \alpha = 00.0 \text{ deg.} \)
A.3.2:  \( \alpha = 02.0 \text{ deg.} \)
A.3.3:  \( \alpha = 04.0 \text{ deg.} \)
A.3.4:  \( \alpha = 06.0 \text{ deg.} \)
A.3.5:  \( \alpha = 08.0 \text{ deg.} \)
A.3.6:  \( \alpha = 09.0 \text{ deg.} \)
A.3. 7: $\alpha = 10.0$ deg.
A.3. 8: $\alpha = 11.0$ deg.
A.3. 9: $\alpha = 12.0$ deg.
A.3. 10: $\alpha = 13.0$ deg.
A.3. 11: $\alpha = 14.0$ deg.
A.3. 12: $\alpha = 15.0$ deg.
A.3. 13: $\alpha = 16.0$ deg.

A.3. 14: $\alpha = 17.0$ deg.

A.3. 15: $\alpha = 18.0$ deg.

A.3. 16: $\alpha = 19.0$ deg.

A.3. 17: $\alpha = 20.0$ deg.

A.3. 18: $\alpha = 21.0$ deg.
A.3. 19: $\alpha = 22.0$ deg.

A.3. 20: $\alpha = 23.0$ deg.

A.3. 21: $\alpha = 24.0$ deg.

A.3. 22: $\alpha = 25.0$ deg.

A.3. 23: $\alpha = 26.0$ deg.

A.3. 24: $\alpha = 27.0$ deg.
A.3. 25: $\alpha = 28.0$ deg.

A.3. 26: $\alpha = 29.0$ deg.

A.3. 27: $\alpha = 30.0$ deg.

A.3. 28: $\alpha = 32.0$ deg.

A.3. 29: $\alpha = 34.0$ deg.

A.3. 30: $\alpha = 36.0$ deg.
A.3. 31: $\alpha = 38.0$ deg.

A.3. 32: $\alpha = 40.0$ deg.

A.3. 33: $\alpha = 42.0$ deg.

A.3. 34: $\alpha = 44.0$ deg.

A.3. 35: $\alpha = 46.0$ deg.

A.3. 36: $\alpha = 48.0$ deg.
A.3. 37: $\alpha = 50.0$ deg.

A.3. 38: $\alpha = 52.0$ deg.

A.3. 39: $\alpha = 54.0$ deg.

A.3. 40: $\alpha = 56.0$ deg.

A.3. 41: $\alpha = 58.0$ deg.

A.3. 42: $\alpha = 60.0$ deg.
A.3. 43: $\alpha = 62.0$ deg.

A.3. 44: $\alpha = 64.0$ deg.

A.3. 45: $\alpha = 66.0$ deg.

A.3. 46: $\alpha = 68.0$ deg.

A.3. 47: $\alpha = 70.0$ deg.

A.3. 48: $\alpha = 72.0$ deg.
A.3. 49: $\alpha = 74.0$ deg.

A.3. 50: $\alpha = 76.0$ deg.

A.3. 51: $\alpha = 78.0$ deg.

A.3. 52: $\alpha = 80.0$ deg.

A.3. 53: $\alpha = 82.0$ deg.

A.3. 54: $\alpha = 84.0$ deg.
A.3. 55: $\alpha = 86.0$ deg.

A.3. 56: $\alpha = 88.0$ deg.

A.3. 57: $\alpha = 90.0$ deg.
Appendix B

Finite Wing Flow Visualization

B.1 Rectangular Unswept Wing

The following Figures present the instantaneous surface streamline patterns on a rectangular wing of AR = 12 with a NACA 4415 Airfoil cross-section as predicted by USM3D using the SA turbulence models at a Mach number of 0.2 and Reynolds number of 3 million.
Figure B.1: Surface streamlines highlighting separation patterns on a rectangular wing, AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_{\infty} = 0.2$. 
B.1. 1: \( \alpha = 00.0 \text{ deg.} \)

B.1. 2: \( \alpha = 02.0 \text{ deg.} \)

B.1. 3: \( \alpha = 04.0 \text{ deg.} \)

B.1. 4: \( \alpha = 06.0 \text{ deg.} \)

B.1. 5: \( \alpha = 08.0 \text{ deg.} \)

B.1. 6: \( \alpha = 10.0 \text{ deg.} \)

B.1. 7: \( \alpha = 12.0 \text{ deg.} \)

B.1. 8: \( \alpha = 14.0 \text{ deg.} \)
B.1. 9: $\alpha = 16.0$ deg.

B.1. 10: $\alpha = 18.0$ deg.

B.1. 11: $\alpha = 20.0$ deg.

B.1. 12: $\alpha = 22.0$ deg.

B.1. 13: $\alpha = 24.0$ deg.

B.1. 14: $\alpha = 26.0$ deg.

B.1. 15: $\alpha = 28.0$ deg.

B.1. 16: $\alpha = 30.0$ deg.
B.1. 17: $\alpha = 32.0$ deg.

B.1. 18: $\alpha = 34.0$ deg.

B.1. 19: $\alpha = 36.0$ deg.

B.1. 20: $\alpha = 38.0$ deg.

B.1. 21: $\alpha = 40.0$ deg.

B.1. 22: $\alpha = 42.0$ deg.

B.1. 23: $\alpha = 44.0$ deg.

B.1. 24: $\alpha = 50.0$ deg.
B.1. 25: $\alpha = 55.0 \text{ deg.}$

B.1. 26: $\alpha = 60.0 \text{ deg.}$

B.1. 27: $\alpha = 65.0 \text{ deg.}$

B.1. 28: $\alpha = 70.0 \text{ deg.}$

B.1. 29: $\alpha = 75.0 \text{ deg.}$

B.1. 30: $\alpha = 80.0 \text{ deg.}$

B.1. 31: $\alpha = 85.0 \text{ deg.}$

B.1. 32: $\alpha = 90.0 \text{ deg.}$
B.2 Unswept Tapered Wing

The following Figures present the instantaneous surface streamline patterns on an unswept tapered wing of $AR = 12$ with a NACA 4415 Airfoil cross-section as predicted by USM3D using the SA turbulence models at a Mach number of 0.2 and Reynolds number of 3 million.
Figure B.2: Surface streamlines highlighting separation patterns on a tapered wing, $\lambda = 0.5$, AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 
B.2. 1: $\alpha = 00.0$ deg.

B.2. 2: $\alpha = 02.0$ deg.

B.2. 3: $\alpha = 04.0$ deg.

B.2. 4: $\alpha = 06.0$ deg.

B.2. 5: $\alpha = 08.0$ deg.

B.2. 6: $\alpha = 10.0$ deg.

B.2. 7: $\alpha = 12.0$ deg.

B.2. 8: $\alpha = 14.0$ deg.
B.2. 9: $\alpha = 16.0$ deg.

B.2. 10: $\alpha = 18.0$ deg.

B.2. 11: $\alpha = 20.0$ deg.

B.2. 12: $\alpha = 22.0$ deg.

B.2. 13: $\alpha = 24.0$ deg.

B.2. 14: $\alpha = 26.0$ deg.

B.2. 15: $\alpha = 30.0$ deg.

B.2. 16: $\alpha = 34.0$ deg.
B.2. 17: $\alpha = 38.0$ deg.

B.2. 18: $\alpha = 42.0$ deg.

B.2. 19: $\alpha = 46.0$ deg.

B.2. 20: $\alpha = 50.0$ deg.

B.2. 21: $\alpha = 54.0$ deg.

B.2. 22: $\alpha = 58.0$ deg.

B.2. 23: $\alpha = 62.0$ deg.

B.2. 24: $\alpha = 66.0$ deg.
B.2. 25: $\alpha = 70.0\ \text{deg.}$

B.2. 26: $\alpha = 74.0\ \text{deg.}$

B.2. 27: $\alpha = 78.0\ \text{deg.}$

B.2. 28: $\alpha = 82.0\ \text{deg.}$

B.2. 29: $\alpha = 86.0\ \text{deg.}$

B.2. 30: $\alpha = 90.0\ \text{deg.}$
B.3 10 Degree Swept Wing

The following Figures present the instantaneous surface streamline patterns on an swept wing of \( \Lambda = 10 \) deg. and \( AR = 12 \) with a NACA 4415 Airfoil cross-section as predicted by USM3D using the SA turbulence models at a Mach number of 0.2 and Reynolds number of 3 million.
Figure B.3: Surface streamlines highlighting separation patterns on a swept wing, $\Lambda = 10$ deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 
B.3. 1: $\alpha = 00.0$ deg.
B.3. 2: $\alpha = 02.0$ deg.
B.3. 3: $\alpha = 04.0$ deg.
B.3. 4: $\alpha = 08.0$ deg.
B.3. 5: $\alpha = 10.0$ deg.
B.3. 6: $\alpha = 12.0$ deg.
B.3. 7: $\alpha = 14.0$ deg.
B.3. 8: $\alpha = 16.0$ deg.
B.3. 9: $\alpha = 18.0$ deg.

B.3. 10: $\alpha = 20.0$ deg.

B.3. 11: $\alpha = 22.0$ deg.

B.3. 12: $\alpha = 24.0$ deg.

B.3. 13: $\alpha = 26.0$ deg.

B.3. 14: $\alpha = 28.0$ deg.

B.3. 15: $\alpha = 30.0$ deg.

B.3. 16: $\alpha = 40.0$ deg.
B.3. 17: $\alpha = 45.0$ deg.

B.3. 18: $\alpha = 50.0$ deg.

B.3. 19: $\alpha = 55.0$ deg.

B.3. 20: $\alpha = 60.0$ deg.

B.3. 21: $\alpha = 65.0$ deg.

B.3. 22: $\alpha = 70.0$ deg.

B.3. 23: $\alpha = 80.0$ deg.

B.3. 24: $\alpha = 90.0$ deg.
B.4 20 Degree Swept Wing

The following Figures present the instantaneous surface streamline patterns on an swept wing of $\Lambda = 20$ deg. and AR = 12 with a NACA 4415 Airfoil cross-section as predicted by USM3D using the SA turbulence models at a Mach number of 0.2 and Reynolds number of 3 million.
Figure B.4: Surface streamlines highlighting separation patterns on a swept wing, Λ = 20 deg., AR = 12, symmetry plane indicated by black line. USM3D/SA, Re = 3 million, $M_\infty = 0.2$. 

136
B.4. 1: $\alpha = 00.0$ deg.

B.4. 2: $\alpha = 04.0$ deg.

B.4. 3: $\alpha = 06.0$ deg.

B.4. 4: $\alpha = 08.0$ deg.

B.4. 5: $\alpha = 10.0$ deg.

B.4. 6: $\alpha = 12.0$ deg.

B.4. 7: $\alpha = 14.0$ deg.

B.4. 8: $\alpha = 16.0$ deg.
B.4. 9: $\alpha = 18.0\ \text{deg.}$

B.4. 10: $\alpha = 20.0\ \text{deg.}$

B.4. 11: $\alpha = 22.0\ \text{deg.}$

B.4. 12: $\alpha = 24.0\ \text{deg.}$

B.4. 13: $\alpha = 26.0\ \text{deg.}$

B.4. 14: $\alpha = 28.0\ \text{deg.}$

B.4. 15: $\alpha = 30.0\ \text{deg.}$

B.4. 16: $\alpha = 34.0\ \text{deg.}$
B.4. 17: $\alpha = 40.0$ deg.

B.4. 18: $\alpha = 45.0$ deg.

B.4. 19: $\alpha = 50.0$ deg.

B.4. 20: $\alpha = 55.0$ deg.

B.4. 21: $\alpha = 65.0$ deg.

B.4. 22: $\alpha = 70.0$ deg.

B.4. 23: $\alpha = 80.0$ deg.

B.4. 24: $\alpha = 90.0$ deg.
B.5 30 Degree Swept Wing

The following Figures present the instantaneous surface streamline patterns on an swept wing of $\Lambda = 30 \text{ deg.}$ and $AR = 12$ with a NACA 4415 Airfoil cross-section as predicted by USM3D using the SA turbulence models at a Mach number of 0.2 and Reynolds number of 3 million.
Figure B.5: Surface streamlines highlighting separation patterns on a swept wing, $\Lambda = 30$ deg., $AR = 12$, symmetry plane indicated by black line. USM3D/SA, $Re = 3$ million, $M_\infty = 0.2$. 
B.5. 9: $\alpha = 20.0$ deg.

B.5. 10: $\alpha = 22.0$ deg.

B.5. 11: $\alpha = 24.0$ deg.

B.5. 12: $\alpha = 26.0$ deg.

B.5. 13: $\alpha = 28.0$ deg.

B.5. 14: $\alpha = 30.0$ deg.

B.5. 15: $\alpha = 34.0$ deg.

B.5. 16: $\alpha = 40.0$ deg.
B.5. 17: $\alpha = 45.0$ deg.

B.5. 18: $\alpha = 50.0$ deg.

B.5. 19: $\alpha = 55.0$ deg.

B.5. 20: $\alpha = 60.0$ deg.

B.5. 21: $\alpha = 65.0$ deg.

B.5. 22: $\alpha = 70.0$ deg.

B.5. 23: $\alpha = 80.0$ deg.

B.5. 24: $\alpha = 90.0$ deg.