2013

Turbulence Modeling for Subsonic Separated Flows Over 2-D Airfoils and 3-D Wings

Aaron Michael Rosen
Purdue University

Follow this and additional works at: https://docs.lib.purdue.edu/open_access_theses
Part of the Aerospace Engineering Commons

Recommended Citation
https://docs.lib.purdue.edu/open_access_theses/121

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
This is to certify that the thesis/dissertation prepared

By Aaron Michael Rosen

Entitled
Turbulence Modeling for Subsonic Separated Flows Over 2-D Airfoils and 3-D Wings

For the degree of Master of Science in Aeronautics and Astronautics

Is approved by the final examining committee:

Gregory A. Blaisdell
Chair
Alina A. Alexeenko

Tom I-P. Shih

To the best of my knowledge and as understood by the student in the Research Integrity and Copyright Disclaimer (Graduate School Form 20), this thesis/dissertation adheres to the provisions of Purdue University’s “Policy on Integrity in Research” and the use of copyrighted material.

Approved by Major Professor(s): Gregory A. Blaisdell

Approved by: Weinong Chen 12/02/2013
Head of the Graduate Program Date
TURBULENCE MODELING FOR SUBSONIC SEPARATED FLOWS
OVER 2-D AIRFOILS AND 3-D WINGS

A Thesis
Submitted to the Faculty
of
Purdue University
by
Aaron M. Rosen

In Partial Fulfillment of the
Requirements for the Degree
of
Master of Science in Aeronautics and Astronautics

December 2013
Purdue University
West Lafayette, Indiana
ACKNOWLEDGMENTS

Many people deserve thanks for making this research possible. First and foremost, I would like to thank my advisor Dr. Gregory Blaisdell for taking me on as a graduate student and for his guidance throughout my graduate career. He has taken time each week to meet with me and discuss my research, offering advice on how to proceed and guiding me towards helpful resources. He has also balanced this by allowing me to work through issues on my own and to gain my own understanding of the topics at hand. I am also grateful to Dr. Tom Shih and Dr. Alina Alexeenko who served on my research committee.

I would also like to acknowledge Thomas Shurtz who introduced me to the OVERFLOW code as well as the technique of overset gridding and spent many hours working through problems with me. Matt Churchfield was also of great help in learning the principles of overset gridding. I would also like to thank Brandon Oliver who provided Purdue with his OVERFLOW postprocessing code, as well as Randy Lillard who provided me with the modified lagRST version of OVERFLOW. Kurt Aikens was also of great value throughout this process and I very much appreciate his willingness to talk through various CFD and turbulence modeling topics which pertained to this research.

Lastly, I would like to thank my parents, Barry and Debby Rosen, for their endless support throughout my graduate studies.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>LIST OF TABLES</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>v</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LIST OF FIGURES</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>vi</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SYMBOLS</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ABBREVIATIONS</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>xii</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ABSTRACT</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>xiii</td>
</tr>
</tbody>
</table>

1  Introduction
   1.1 Motivation
   1.2 Background
      1.2.1 Previous Computational Work with Separated Flows
      1.2.2 Test Cases Available for Comparison
   1.3 Objectives

2  Turbulence Modeling
   2.1 Equilibrium Turbulence Modeling
   2.2 Nonequilibrium Turbulence Modeling Overview
      2.2.1 Lag Model
      2.2.2 lagRST Model

3  Computational Setup
   3.1 OVERFLOW Code Description
   3.2 Wadcock and Coles Two-Dimensional NACA 4412 Airfoil
      3.2.1 Experimental Setup
      3.2.2 Computational Domain
      3.2.3 Grid Convergence Study
      3.2.4 Timestep Convergence Study and Run Length
      3.2.5 Validation Case
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.3</td>
<td>NACA 0021 Wing</td>
<td>46</td>
</tr>
<tr>
<td>3.3.1</td>
<td>Experimental Setup</td>
<td>46</td>
</tr>
<tr>
<td>3.3.2</td>
<td>Computational Domain</td>
<td>49</td>
</tr>
<tr>
<td>3.3.3</td>
<td>Grid Convergence Study</td>
<td>52</td>
</tr>
<tr>
<td>3.3.4</td>
<td>Unsteadiness in Steady-State Computations</td>
<td>56</td>
</tr>
<tr>
<td>4</td>
<td>Test Cases</td>
<td>60</td>
</tr>
<tr>
<td>4.1</td>
<td>NACA 4412 Airfoil Comparisons</td>
<td>61</td>
</tr>
<tr>
<td>4.1.1</td>
<td>Separation Point and Velocity Profiles</td>
<td>63</td>
</tr>
<tr>
<td>4.1.2</td>
<td>Pressure Coefficient Comparisons</td>
<td>72</td>
</tr>
<tr>
<td>4.2</td>
<td>NACA 0021 Wing Comparisons</td>
<td>77</td>
</tr>
<tr>
<td>4.2.1</td>
<td>Surface Streamlines and Separation Patterns</td>
<td>81</td>
</tr>
<tr>
<td>4.2.2</td>
<td>Pressure Coefficients and Comparisons to Experiment</td>
<td>90</td>
</tr>
<tr>
<td>5</td>
<td>Conclusions and Recommendations</td>
<td>95</td>
</tr>
<tr>
<td></td>
<td>LIST OF REFERENCES</td>
<td>99</td>
</tr>
<tr>
<td></td>
<td>VITA</td>
<td>103</td>
</tr>
</tbody>
</table>
## LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>A summary of the parameters for the NACA 4412 grid convergence study.</td>
</tr>
<tr>
<td>3.2</td>
<td>Richardson extrapolation results for lift, drag, and pressure coefficient for the 2-D NACA 4412 case.</td>
</tr>
<tr>
<td>3.3</td>
<td>Summary of the timestep convergence study parameters.</td>
</tr>
<tr>
<td>3.4</td>
<td>Summary of the timestep convergence study results.</td>
</tr>
<tr>
<td>3.5</td>
<td>Summary of the averaging study results.</td>
</tr>
<tr>
<td>3.6</td>
<td>A summary of the parameters for the NACA 0021 grid convergence study.</td>
</tr>
<tr>
<td>3.7</td>
<td>Richardson extrapolation results for lift, drag, and pressure coefficient for the 3-D NACA 0021 case.</td>
</tr>
<tr>
<td>4.1</td>
<td>Turbulence models used and their associated reference number in the modified OVERFLOW 2.2e.lagRST Code.</td>
</tr>
<tr>
<td>4.2</td>
<td>Reference conditions for the testing of a NACA 4412 airfoil in the GALCIT wind tunnel at Caltech.</td>
</tr>
<tr>
<td>4.3</td>
<td>Separation points resulting from each model for a NACA 4412 airfoil at 13.87°.</td>
</tr>
<tr>
<td>4.4</td>
<td>Estimated run conditions for the testing of a NACA 0021 wing in the Boeing wind tunnel at Purdue.</td>
</tr>
<tr>
<td>4.5</td>
<td>Estimated freestream conditions for the testing of a NACA 0021 wing in the Boeing wind tunnel at Purdue.</td>
</tr>
<tr>
<td>4.6</td>
<td>Separation points on a 3-D NACA 0021 wing at wing root for angles of attack of 10 and 16 degrees.</td>
</tr>
<tr>
<td>4.7</td>
<td>Lift coefficient calculated on the 3-D NACA 0021 wing for angles of attack of 10 and 16 degrees.</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>1.1</td>
<td>Inviscid, incompressible solution of flow over an airfoil.</td>
</tr>
<tr>
<td>1.2</td>
<td>Streamwise velocity contours plotted for a NACA 4412 airfoil at 13.87 degrees. Note the reverse flow region close to the suction surface at X = 70.</td>
</tr>
<tr>
<td>1.3</td>
<td>Corresponding pressure coefficient plot. The low pressure at the leading edge causes a steep adverse pressure gradient downstream on the upper surface.</td>
</tr>
<tr>
<td>3.1</td>
<td>Depiction of the GALCIT Wind Tunnel test section setup with a NACA 4412 airfoil mounted inside.</td>
</tr>
<tr>
<td>3.2</td>
<td>Wind tunnel boundary conditions.</td>
</tr>
<tr>
<td>3.3</td>
<td>C-grid trailing edge abutting grid topology. The green grid defines the trailing edge of the wing and also the wake, and the pink grid defines the wing.</td>
</tr>
<tr>
<td>3.4</td>
<td>C-grid trailing edge band-aid grid topology.</td>
</tr>
<tr>
<td>3.5</td>
<td>NACA 4412 O-grid topology.</td>
</tr>
<tr>
<td>3.6</td>
<td>NACA 4412 leading edge spacing.</td>
</tr>
<tr>
<td>3.7</td>
<td>NACA 4412 trailing edge spacing and wake region.</td>
</tr>
<tr>
<td>3.8</td>
<td>Linear plot of lift coefficient as a function of 1/(number of grid points)^2.</td>
</tr>
<tr>
<td>3.9</td>
<td>Velocity profile plots of the NACA 4412 case showing the effects of grid density.</td>
</tr>
<tr>
<td>3.10</td>
<td>Pressure coefficient plots of the NACA 4412 case showing the effects of grid density.</td>
</tr>
<tr>
<td>3.11</td>
<td>Minimum pressure region of the pressure coefficient plots.</td>
</tr>
<tr>
<td>3.12</td>
<td>Velocity profiles taken off the upper surface at x/c = 0.7, showing the effects of timestep length.</td>
</tr>
<tr>
<td>3.13</td>
<td>Velocity profiles taken off the upper surface at x/c = 0.9, showing the effects of timestep length.</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
</tr>
<tr>
<td>---------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>3.14 Velocity profile taken in the wake at $x/c = 1.1$, showing the effects of timestep length.</td>
<td>41</td>
</tr>
<tr>
<td>3.15 Unsteady lift coefficient plotted as a function of physical time, showing the effects of timestep length.</td>
<td>42</td>
</tr>
<tr>
<td>3.16 Unsteady lift plotted as a function of timestep for the 5,000 timestep run.</td>
<td>44</td>
</tr>
<tr>
<td>3.17 Unsteady lift plotted as a function of timestep for the 10,000 timestep run.</td>
<td>44</td>
</tr>
<tr>
<td>3.18 Unsteady lift plotted as a function of timestep for the 20,000 timestep run.</td>
<td>45</td>
</tr>
<tr>
<td>3.19 Velocity profile plots of the NACA 4412 case showing the effects of grid density.</td>
<td>47</td>
</tr>
<tr>
<td>3.20 Chordwise bands of wind tunnel pressure measurement, at 8 and 22 inches from the wing root on either side of the root indicated by the dark black lines.</td>
<td>49</td>
</tr>
<tr>
<td>3.21 Center line of the wingcap surface to be marched inboard along the surface of the wing.</td>
<td>50</td>
</tr>
<tr>
<td>3.22 Wingtip cap for the NACA 0021 wing.</td>
<td>50</td>
</tr>
<tr>
<td>3.23 NACA 0021 leading edge spacing.</td>
<td>51</td>
</tr>
<tr>
<td>3.24 NACA 0021 trailing edge spacing and wake region.</td>
<td>51</td>
</tr>
<tr>
<td>3.25 Pressure coefficient plots of the NACA 4412 case showing the effects of grid density.</td>
<td>55</td>
</tr>
<tr>
<td>3.26 Minimum pressure region of the pressure coefficient plots.</td>
<td>55</td>
</tr>
<tr>
<td>3.27 Lift coefficient as a function of timestep for the lagRSTSST model with $\sigma_k = 0.8$, run with an upwind and central difference spatial discretization. For the central difference method, timesteps 6,000-17,000 were run with a CFL number of 5.0 and timesteps 17,000-23,000 were run with a CFL number of 1.0.</td>
<td>56</td>
</tr>
<tr>
<td>3.28 Lift coefficient as a function of timestep, run with the central difference method for the lagRSTSST model with $\sigma_k = 0.8$, denoting the peak and valley of the quasi-steady state solution.</td>
<td>57</td>
</tr>
<tr>
<td>3.29 Skin friction distribution on the upper surface at the wing root showing the differences between the Peak and Valley lift values.</td>
<td>58</td>
</tr>
<tr>
<td>3.30 Pressure coefficient distribution at the wing root showing the differences between the Peak and Valley lift values.</td>
<td>58</td>
</tr>
</tbody>
</table>
Figure 4.1 Velocity profiles comparisons of the Lag model taken off the upper surface at $x/c = 0.7$. .......................................................... 64

4.2 Velocity profile comparisons of the lagRST model taken off the upper surface at $x/c = 0.7$. .......................................................... 65

4.3 Velocity profile comparisons of the lagRSTSST model taken off the upper surface at $x/c = 0.7$. .......................................................... 65

4.4 Velocity profiles comparisons of the Lag model taken off the upper surface at $x/c = 0.9$. .......................................................... 66

4.5 Velocity profile comparisons of the lagRST model taken off the upper surface at $x/c = 0.9$. .......................................................... 67

4.6 Velocity profile comparisons of the lagRSTSST model taken off the upper surface at $x/c = 0.9$. .......................................................... 67

4.7 Velocity profiles comparisons of the Lag model taken in the wake at $x/c = 1.1$. .......................................................... 68

4.8 Velocity profile comparisons of the lagRST model taken in the wake at $x/c = 1.1$. .......................................................... 69

4.9 Velocity profile comparisons of the lagRSTSST model taken in the wake at $x/c = 1.1$. .......................................................... 69

4.10 Velocity contours of the separation region for the lagRST models with a) $\sigma_k = 0.5$, b) $\sigma_k = 0.8$, c) $\sigma_k = 1.5$, in comparison to d) Wadcock and Coles’ data (1979). .......................................................... 71

4.11 Velocity contours of the separation region for the lagRSTSST models with a) $\sigma_k = 0.5$, b) $\sigma_k = 0.8$, c) $\sigma_k = 1.5$, in comparison to d) Wadcock and Coles’ data (1979). .......................................................... 73

4.12 Pressure coefficient comparison between each of the turbulence models and the Wadcock and Coles data. .......................................................... 74

4.13 Pressure coefficient comparison between the Lag models at the minimum pressure peak. .......................................................... 74

4.14 Pressure coefficient comparison between the lagRST and lagRSTSST models at the minimum pressure peak. .......................................................... 75

4.15 Pressure coefficient comparisons in the separation region. .......................................................... 76

4.16 Wind tunnel setup depicting the pressure ports hanging below the wing. .......................................................... 80

4.17 Skin friction plots at the wing root for the 10° cases showing the differences between turbulence models. .......................................................... 85
Figure | Page
--- | ---
4.18 Plot showing the relationship between the artificial transition point and the subsequent separation location. | 85
4.19 Plot showing the effect of moving back the transition point on the skin friction plot at the wing root for the lagRSTSST model with $\sigma_k = 0.8$. | 86
4.20 Surface streamline plots at $\alpha = 10^\circ$ of the Lag model with a) $\sigma_k = 0.5$, b) $\sigma_k = 0.8$, and c) $\sigma_k = 1.5$, showing the effect of changing $\sigma_k$ on the separation locations. | 87
4.21 Surface streamline plots at $\alpha = 10^\circ$ of the lagRST model with a) $\sigma_k = 0.5$, b) $\sigma_k = 0.8$, and c) $\sigma_k = 1.5$, showing the effect of changing $\sigma_k$ on the separation locations. | 88
4.22 Surface streamline plots at $\alpha = 10^\circ$ of the lagRSTSST model with a) $\sigma_k = 0.5$, b) $\sigma_k = 0.8$, and c) $\sigma_k = 1.5$, showing the effect of changing $\sigma_k$ on the separation locations. | 89
4.23 Surface pressure coefficients at the wing root for the $\alpha = 10^\circ$ case. | 90
4.24 Minimum pressure peak at the wing root at 10 degrees, comparing the Lag models with the standard models and the lagRSTSST models. | 91
4.25 Minimum pressure peak at the wing root at 10 degrees, comparing the lagRST models with the standard models and the lagRSTSST models. | 91
4.26 Pressure coefficient comparison between CFD and experimental at the spanwise location 8 inches from the wing root for the $\alpha = 10^\circ$ case. | 91
4.27 Diagrams showing the a) upper and b) lower surface free transition locations for a 10 degree angle of attack. The purple section represents the turbulent area, all other colors represent a laminar region. | 93
4.28 Pressure coefficient plots of the fully turbulent and simulated free transition CFD data using the lagRSTSST turbulence model with $\sigma_k = 0.8$, in comparison to the experimental data. Data was taken at the spanwise location 8 inches from the wing root for the $\alpha = 10^\circ$ case. | 93
SYMBOLS

Roman Symbols:

\( a \)  Lag turbulence modeling constant
\( c \)  Chord length
\( c_f \)  Skin friction coefficient
\( c_p \)  Pressure coefficient
\( f \)  Frequency
\( L \)  Total lift on a three-dimensional body
\( l \)  Section (two-dimensional) lift
\( k \)  Turbulent kinetic energy
\( M \)  Mach number
\( N \)  Number of data samples in a time-accurate simulation
\( p \)  Static pressure
\( \mathcal{P} \)  Turbulent production term
\( Q \)  Dynamic pressure
\( R \)  Specific gas constant
\( R_{ij} \)  Reynolds stress tensor
\( Re \)  Reynolds number
\( S \)  Strain rate tensor
\( St \)  Strouhal number
\( T \)  Temperature
\( t \)  Time
\( \overline{u'v'} \)  Turbulent shear stress
\( U \)  Velocity magnitude
\( u, v, w \) \hspace{1cm} \text{x, y, z components of velocity}

\( x, y, z \) \hspace{1cm} \text{Cartesian axis components}

\( Y \) \hspace{1cm} \text{General quantity used in Richardson extrapolation}

Greek Symbols:

\( \alpha \) \hspace{1cm} \text{Angle of Attack}

\( \gamma \) \hspace{1cm} \text{Ratio of specific heats}

\( \epsilon \) \hspace{1cm} \text{Turbulent dissipation}

\( \mu \) \hspace{1cm} \text{Dynamic viscosity}

\( \nu \) \hspace{1cm} \text{Kinematic viscosity}

\( \nu_T \) \hspace{1cm} \text{Eddy viscosity}

\( \rho \) \hspace{1cm} \text{Density}

\( \sigma_k \) \hspace{1cm} \text{Diffusion coefficient in the kinetic energy equation}

\( \tau_{ij} \) \hspace{1cm} \text{Reynolds stress tensor}

\( \tau_w \) \hspace{1cm} \text{Wall Shear Stress}

\( \omega \) \hspace{1cm} \text{Specific turbulence dissipation rate}
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ASL</td>
<td>Aerospace Sciences Laboratory</td>
</tr>
<tr>
<td>BB</td>
<td>Baldwin-Barth Turbulence Model</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>DTPHYS</td>
<td>OVERFLOW’s Nondimensional Physical Timestep Input Variable</td>
</tr>
<tr>
<td>GALCIT</td>
<td>Graduate Aeronautical Laboratories, California Institute of Technology</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>R.E.</td>
<td>Richardson Extrapolation</td>
</tr>
<tr>
<td>RST</td>
<td>Reynolds Stress Tensor</td>
</tr>
<tr>
<td>RMS</td>
<td>Root Mean Square</td>
</tr>
<tr>
<td>SA</td>
<td>Spalart-Allmaras Turbulence Model</td>
</tr>
<tr>
<td>SARC</td>
<td>Spalart-Allmaras Turbulence Model with Rotation/Curvature Correction</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport Turbulence Model</td>
</tr>
<tr>
<td>SWTBLI</td>
<td>Shockwave/Turbulent Boundary Layer Interaction</td>
</tr>
<tr>
<td>URANS</td>
<td>Unsteady Reynolds-Averaged Navier-Stokes</td>
</tr>
</tbody>
</table>
ABSTRACT

Rosen, Aaron M. M.S.A.A., Purdue University, December 2013. Turbulence Modeling for Subsonic Separated Flows Over 2-D Airfoils and 3-D Wings. Major Professor: Gregory A. Blaisdell.

Accurate predictions of turbulent boundary layers and flow separation through computational fluid dynamics (CFD) are becoming more and more essential for the prediction of loads in the design of aerodynamic flight components. Standard eddy viscosity models used in many commercial codes today do not capture the nonequilibrium effects seen in a separated flow and thus do not generally make accurate separation predictions. Part of the reason for this is that under nonequilibrium conditions such as a strong adverse pressure gradient, the history effects of the flow play an important role in the growth and decay of turbulence. More recent turbulence models such as Olsen and Coakley’s Lag model and Lillard’s lagRST model seek to simulate these effects by lagging the turbulent variables when nonequilibrium effects become important. The purpose of the current research is to assess how these nonequilibrium turbulence models capture the separated regions on various 2-D airfoils and 3-D wings. Nonequilibrium models including the Lag model and the lagRST model are evaluated in comparison with three baseline models (Spalart-Allmaras, Wilcox’s $k-\omega$, and Menter’s SST) using a modified version of the OVERFLOW code. Tuning the model coefficients of the Lag and lagRST models is also explored. Results show that the various lagRST formulations display an improvement in velocity profile predictions over the standard RANS models, but have trouble capturing the edge of the boundary layer. Experimental separation location measurements were not available, but several trends are noted which may be useful to tuning the model coefficients in the future.
1. INTRODUCTION

As computers grow in size and high-performance computing becomes more and more viable, computational predictions have become an essential part of the design process for many fields of study. In the case of fluid mechanics problems, Computational Fluid Dynamics (CFD) can be used to predict complex fluid flows around complicated geometries. The vast majority of applicable flows are turbulent, which makes turbulence modeling one of the most important topics in the field of aerodynamics. There is a hierarchy to these models and the main problem faced is capturing all of the relevant flow physics while still reaching a solution in a reasonable amount of time.

Turbulent simulations such as Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) directly simulate the flowfield and allow for a more accurate representation of the physics, but are still largely impractical for complicated geometries because of the grid spacing requirements. Thus, for most practical flows, a time-averaged form of the governing equations known as the Reynolds-Averaged Navier-Stokes (RANS) equations is used. During the averaging process, however, some information is lost leaving no explicit relationship between the mean flowfield quantities and the terms introduced by the averaging process. Turbulence models are used to model these extra terms which cannot be computed directly; namely the Reynolds Stress Tensor (RST) and the turbulent heat flux vector.

Eddy viscosity models typically use the Boussinesq approximation,

\[ R_{ij} = \frac{2}{3} k \delta_{ij} - 2 \nu_t \left( s_{ij} - \frac{1}{3} s_{kk} \delta_{ij} \right) \tag{1.1} \]

which relates the RST to the mean flow variables using the assumption that the mean strain rate tensor \( (s_{ij}) \) and the RST \( (R_{ij}) \) are aligned and are linearly related.
These can take the form of algebraic, one-equation, or two-equation models. The most successful eddy viscosity models used today such as the Spalart-Allmaras (SA) one-equation model [1], and Shear Stress Transport (SST) model [2] have been designed for equilibrium flows where the turbulent time scales are much smaller than the mean flow time scales and react almost instantaneously to changes in the mean strain rate. This is a decent assumption to make for flows with no rapid changes in the flowfield. In adverse pressure gradient regions or other nonequilibrium flows where the mean conditions change abruptly, however, eddy viscosity models are limited by the Boussinesq approximation and do not perform well. More complex RANS models not limited by this assumption are based on the concept of more advanced modeling of the physics associated with the RST. These models have been investigated rigorously, yielding mixed results. Reynolds stress models do inherently account for more non-equilibrium effects by removing the explicit dependency of the RST on the mean flow variables, but they are notorious for causing numerical instabilities, are more computationally expensive, and do not generally yield significantly better results than eddy viscosity models. Supporting this is a study done by Viti et al. [3] which did not find any significant advantages of the more complex Reynolds stress models over Menter’s SST model for various flows with moderate amounts of anisotropy. Although Reynolds stress models inherently capture anisotropy effects more effectively than eddy viscosity models, they are also more computationally intensive [4], more difficult to implement, and can be numerically stiff. Because of these issues with Reynolds stress transport models and the computational expense of LES and DNS, eddy viscosity models are still widely used in industry and research today.

The major drawback of one- and two- equation eddy viscosity models is that they do not account for history effects of the Reynolds stresses. Essentially, the Reynolds stresses at a given time are related not only to the mean flow variables at the current time, but also to the mean flow variables and Reynolds stresses at past times. In non-equilibrium flows, history effects play an important role in determining the behavior of the Reynolds stresses. Turbulence is produced by the mean strain
rate in the flow. Under a nonequilibrium condition, the components of the strain rate tensor are constantly changing. The Reynolds stresses change also, but take time to react to changes in the mean strain rate. Alternatively, the Boussinesq approximation states explicitly that the Reynolds stresses are related linearly to the mean strain rate tensor at the current time. For non-equilibrium flows this is not a valid assumption. Several corrections to eddy viscosity models have been proposed to deal with this, such as the Spalart-Allmaras model with rotation and curvature correction (SARC) which suppresses turbulent production when the Reynolds stress tensor lags the mean strain rate tensor but does not actually model the physics of the lagging process [5,6]. Another model, proposed by Olsen and Coakley [7] is the Lag model which adds an additional equation to an existing eddy viscosity model designed to relax the eddy viscosity towards an equilibrium value. More recently, models have been tested which lag the RST instead of the eddy viscosity. This model is called the lagRST model developed by Lillard [8], and was introduced by Olsen and Coakley [7].

While experimental testing remains a crucial part of aerodynamic design, CFD is becoming more important as a cheaper preliminary design test, and thus it is desirable to be able to accurately model turbulent flows without having to expend the resources on computationally intensive techniques. Specifically for the aerodynamics of airplane wings, it is important to be able to predict certain mean flow quantities such as pressure and skin friction values on the surface of the wing. Contemporary one-equation and two-equation eddy viscosity models have been calibrated for attached boundary layer cases, and generally perform to an acceptable degree of accuracy for equilibrium flows where changes in the mean flow variables are gradual. Problems arise, however, when the boundary layer begins to separate due to large adverse pressure gradients. To calculate the mean flow properties in these cases, a turbulence model must be able to perform in regions of high anisotropy. The focus of this work is to assess how the various lagged Reynolds stress models perform in identifying and capturing the separated region on airfoils at high angles of attack.
1.1 Motivation

The main motivation in pursuing this research is to assess the nonequilibrium turbulence models using data available from experiments. When calculating aerodynamic forces on a body, the characterization of the flow in the separation zone has an enormous impact on the lift and drag. At small angles of attack, the viscous effects generate only minor modifications to the inviscid flow and the region close to the body will be characterized by a general turbulent log-law boundary layer. When the angle of attack is increased, however, the boundary layer will separate downstream on the wing and create a region of anisotropy. Standard one- and two-equation eddy viscosity models typically cannot be trusted to capture where the flow will separate (to a certain degree of precision) and at which angles of attack separation will occur. Additionally, the surface values of pressure and skin friction determine the lift and drag on a body, and thus being able to accurately predict nonequilibrium regions of the flow is crucial to calculating the forces and moments. The nondimensional pressure coefficient is defined as

\[ c_p = \frac{p - p_\infty}{\frac{1}{2} \rho_\infty U_\infty^2} \]  

(1.2)

where \( c_p \) is the pressure coefficient, \( p \) is the pressure, \( \rho_\infty \) is the freestream density, and \( U_\infty \) is the freestream velocity magnitude. The skin friction coefficient is a nondimensionalized measure of the shear stress between the fluid and the wall, and is defined as

\[ c_f = \frac{\tau_w}{\frac{1}{2} \rho_\infty U_\infty^2} \]  

(1.3)

where \( c_f \) is the skin friction coefficient and \( \tau_w \) is the shear stress of the fluid at the wall.

Flow separation is a direct result of the fluid viscosity. When fluid motion exists relative to a solid wall, a boundary layer forms with the fluid velocity going to zero at the wall and increasing in magnitude moving away from the wall. Under an adverse
pressure gradient or no pressure gradient, the fluid constantly loses momentum and the velocities of the fluid particles are decreased. For inviscid, incompressible flow over an airfoil, one stagnation point exists at the leading edge and one at the trailing edge (Figure 1.1).

Fig. 1.1.: Inviscid, incompressible solution of flow over an airfoil.

Since the velocity magnitude goes to zero at these locations, the pressure coefficient is 1. For incompressible flow, Equation 1.2 can be modified using Bernoulli’s equation to clarify this assertion:

\[ c_p = 1 - \frac{U^2}{U_\infty^2}. \tag{1.4} \]

When viscosity is added to the analysis, a stagnation point will still form at the leading edge. However, as the flow moves downstream along the wing, the boundary layer grows under an adverse pressure gradient and the flow outside the boundary layer does not come together at a sharp corner, meaning there is no stagnation point
at the trailing edge. Therefore, the pressure coefficient does not necessarily increase all the way up to 1, although it will increase to a certain extent as long as the flow is attached.

This is applicable to airplane wings at high angles of attack. The fluid reaches a stagnation point at the leading edge and then speeds up around the upstream region of the upper surface which has a small radius of curvature, creating a low pressure region. When the value of the minimum pressure coefficient becomes low, the adverse pressure gradient must become very strong so that the pressure coefficient will have increased to a value close to 1 by the time it reaches the trailing edge. Under large adverse pressure gradients, the velocity of the flow in the boundary layer will decrease until the flow actually reverses (Figure 1.2, Figure 1.3).

Fig. 1.2.: Streamwise velocity contours plotted for a NACA 4412 airfoil at 13.87 degrees. Note the reverse flow region close to the suction surface at X = 70.

Fig. 1.3.: Corresponding pressure coefficient plot. The low pressure at the leading edge causes a steep adverse pressure gradient downstream on the upper surface.

The separation point is associated with an increase in pressure. Downstream, the fluid is no longer attached and the surface pressure no longer reacts to changes in
the geometry of the surface, leading to very little pressure recovery at the trailing edge (the pressure coefficient will be significantly less than 1 at the trailing edge). With regard to drag, a lower pressure at the downstream end of a wing than at the upstream end results in an increase in drag. With regard to lift, the pressure on the majority of the separated region on the upper surface is higher than it would be for an attached flow, causing a decrease in lift. Thus, a separated flow can significantly decrease lift and increase drag, and the magnitude of this effect has an important dependence on how large the separated region is and where it is located.

As mentioned in the previous section, traditional one- and two-equation eddy viscosity models rely on the assumption that changes in the mean flow conditions affect the Reynolds stresses immediately. In this case, under large adverse pressure gradients the flow will decelerate quite rapidly along the airfoil. Following this assumption, traditional eddy viscosity models may predict the Reynolds stresses to change just as rapidly in response to the changes in pressure and velocity. In reality, the Reynolds stresses do change, but take time to react to the changes in the mean flow conditions. Predicting the Reynolds stresses incorrectly results in changes to the mean flow conditions - velocity and pressure, both of which directly affect the skin friction and pressure distributions on a body. Formulations like the Lag and lagRST models seek to mitigate this inherent disadvantage by “lagging” the effect of the mean flow on the Reynolds stresses to avoid a direct response to changes in the flowfield.

1.2 Background

The current study seeks to compare the computational results acquired using the lagRST model with experimental data which has been tabulated for a separated flow on a wing due to large adverse pressure gradients on high angle of attack cases. The phenomenon of separated flow has been explored over many years, both experimentally and computationally.
1.2.1 Previous Computational Work with Separated Flows

For many years, the standard RANS model for predicting separated flow regions was $k - \epsilon$ [9] which has a history of performing poorly in these conditions. Rhie and Chow [10] ran two-dimensional cases with the $k - \epsilon$ model for NACA 0012 and NACA 4412 airfoils with and without separation. They found relatively good agreement with experimental data for attached flow cases but poor agreement for separated flow cases. A more recent study by Chaouat [11] of flow over a high-lift airfoil also found that in comparison with a Reynolds Stress model, the $k - \epsilon$ model performed well for attached flows, but did much worse predicting the skin friction coefficient and the velocity profiles when the angle of attack was increased to a near-stall configuration.

David Wilcox has historically seen better success with his formulation of the $k - \omega$ model [12] under adverse pressure gradients. In a study which compared low and high Reynolds number formulations of the $k - \omega$ and $k - \epsilon$ models for various pressure gradient magnitudes in a boundary layer [13], Wilcox found that both models calculated skin friction values relatively accurately for favorable pressure gradients. However, the $k - \epsilon$ model did much worse under adverse pressure gradients, sometimes predicting skin friction values as much as 50% to 100% higher than measured. In contrast, the $k - \omega$ models were within 6% to 12% of the measured values for even the strongest adverse pressure gradients. However, other studies have concluded that both $k - \omega$ and $k - \epsilon$ do not perform well under adverse pressure gradients. Ekaterinaris and Menter [14] showed that other models, such as SA, Baldwin-Barth (BB) [15], and SST perform better in predicting the separation point and velocity profiles in the separated region on a NACA 4412 airfoil at a high angle of attack. This, combined with the known freestream dependencies of the $k - \omega$ model [16], have resulted in the SST and SA models becoming the standard RANS models in application to adverse pressure gradient flows, although both of them still have serious modeling issues. Both models are limited by the Boussinesq approximation, and in the case of the SST model this leads to increased production of turbulence levels in regions with
large normal strain which can cause early separation predictions. The SA model is also highly tuned for wall-bounded flows and is dependent on the distance from the wall, which can be ambiguous for complicated geometries and free shear flows.

Other RANS-type models have since arisen with corrections to the traditional RANS models for nonequilibrium flows. These models, including the Lag model and the lagRST model, have potential to predict separated flowfields more accurately; however, the lagged Reynolds stress models have not yet been rigorously tested for separated flows on a wing. These models are investigated in the current study.

1.2.2 Test Cases Available for Comparison

Separated flows due to adverse pressure gradients have been studied in detail for many years. However, at the present time, experimental studies of wing structures at separated or near-stall conditions in the open literature are scarce; and acquiring the tabulated data for these studies proved to be even more difficult. Thus, the main requirement for comparison is that tabulated data of the experimental case must be easily obtainable. The study also requires that the experimental setup be well-described and reproducible. It was not a requirement that the data be two-dimensional, but it turned out that the vast majority of available tabulated wind tunnel data for a wing is two-dimensional. The final requirement is that the experimental case deals with separated flow for high angle of attack cases. Four different studies were considered out of which one was selected for comparison. In addition to this case a three-dimensional NACA 0021-based wing at 10 and 16 degree angles of attack was tested as part of an unrelated study at Purdue University, although the experimental data was not yet available for comparison.

The first dataset considered was Alan Wadcock and Donald Coles’ flying hotwire study of a NACA 4412 wing at a near-stall angle of attack (13.87 degrees) and a low Mach number of 0.0785 [17, 18]. The experiment was done in a 10-foot cylindrical wind tunnel with false walls built in to make the sidewalls rectangular. The wing
was built into the false walls so that the airfoil took up the entire span of the test section. Various active and passive methods of flow control were employed in order to ensure two-dimensionality of the flowfield. Velocity and pressure data were available for comparison in the separated region.

The second dataset was Seetharam and Wentz’s experimental investigation of a 17% thick GA(W)-1 airfoil section at a nearly-incompressible Mach number of 0.13 [19]. Datasets consisting of pressure and velocity measurements were available for the separated region at angles of attack of 10, 14, and 18 degrees. The experiment was done in a 7x10-foot wind tunnel modified with a rectangular insert of 7x3-foot dimensions, and measurements were taken at the wing root. The wing once again spanned the entire test section so as to simulate a two-dimensional flowfield. The major error in this study, according to Wadcock [17] is that the paper provides no evidence that the flow is actually two-dimensional. Indeed, Seetharam and Wentz do not provide any indication of a flow control study or any data which proves two-dimensionality of the flowfield, and so the validity of the data as a two-dimensional dataset may be in question.

The third dataset was of a similar setup investigated by Goradia, Mehta, and Shrewsbury [20, 21] of a 17% thick GA(W)-1 airfoil section at a low Mach number of 0.18. Wadcock [17] displayed the same reservations about this dataset as he did with the Seetharam and Wentz study, calling into question its legitimacy as two-dimensional since no flow control ideas were mentioned and no evidence was put forth to support the two-dimensionality claim. The paper describing the experimental setup was not easily available which makes it difficult to confirm these concerns, but it was determined that the dataset should not be used based on Wadcock’s claims.

The final dataset considered was a two-dimensional investigation of the wake region of a NACA 0012 wing at angles of attack of 0 and 6 degrees [22, 23]. The data provided included two components of velocity and the Reynolds stress at various streamwise locations in the wake measured by hot-wire anemometers. The report mentions that the flowfield along the measuring plane was two-dimensional so pre-
sumably two-dimensionality validation tests were performed; however none of that data is present in the report. Additionally, one can assume that angles of attack of 0 and 6 degrees are not large enough to see flow separation upstream of the trailing edge point on a NACA 0012 airfoil, so this data describes only the wake region downstream of the trailing edge, which is not as relevant to this study as any of the previous three datasets described.

All four of these studies have strengths and weaknesses, but the only case with available data, assured two-dimensionality, and an experimental setup applicable to separated flow under an adverse pressure gradient on a wing was the Wadcock and Coles NACA 4412 data. As mentioned before, the two-dimensionality of the flowfield was not a requirement but it was important to be able to reproduce the experimental case correctly. Thus, the two cases to be studied are a two-dimensional NACA 4412 airfoil at 13.87 degrees angle of attack and a three-dimensional NACA 0021-based wing at 10 and 16 degree angles of attack.

1.3 Objectives

The lagRST model is a relatively new model which has not been thoroughly tested for nonequilibrium flows. In general terms, the model solves an additional transport equation which relaxes the current value of the Reynolds stresses towards the equilibrium value calculated by a baseline two-equation model. Lillard [8] implemented various formulations of the lagRST model into a version of OVERFLOW [24]. To the best knowledge of the author, this implementation in OVERFLOW has been tested in a very limited context: a flat plate boundary layer, the modeling of subsonic and supersonic shock wave/turbulent boundary layer interaction (SWTBLI) separated flows, Orion capsule wake flows [8, 25], and the modeling of $q$ and wingtip vortices [26, 27]. The main objective of this research is to extend the analysis of this model to other types of nonequilibrium flows; specifically separation on a wing under a strong adverse pressure gradient.
The expectation from this study is that the models which account for nonequilibrium effects will perform better than the models which rely solely on the Boussinesq approximation to calculate the eddy viscosity and RST. Thus, we hypothesize that the Lag and lagRST models will predict the separation region more accurately than the standard eddy viscosity models. The results are compared to Wadcock and Coles’ NACA 4412 experiments to determine the accuracy of the models. Additionally, data from the three-dimensional NACA 0021 cases are analyzed in comparison to the experimental results.
2. TURBULENCE MODELING

The analysis of any fluids problem begins with conservation of mass and momentum. By starting with the incompressible Navier Stokes equations and Reynolds averaging them, the incompressible RANS equations are derived:

\[
\frac{\partial \pi_i}{\partial t} + u_j \frac{\partial \pi_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial \pi_i}{\partial x_j} - \overline{u_i u_j} \right).
\] (2.1)

Each of these variables except for the final term represents a flowfield quantity which is calculated directly by a CFD solver. This final term, \(-\frac{\partial \overline{u_i u_j}}{\partial x_j}\), is the \(x_j\) derivative of the RST, which functions as a “fluid stress due to turbulence” term. The quantity \(\overline{u_i u_j}\) can be denoted by \(R_{ij}\), or \(-\tau_{ij}/\rho\). From Equation 2.1 along with the conservation of mass equation, there are now four equations (conservation of mass along with three components of momentum) with ten unknowns (three components of velocity, pressure, and six components of the RST). This is what is commonly referred to as the turbulent closure problem.

Eddy viscosity models use the Boussinesq approximation, which is formulated as an explicit and linear relationship between the Reynolds stresses and the mean flow variables:

\[-\overline{u_i u_j} + \frac{1}{3} \overline{u_k u_k} \delta_{ij} = \nu_T \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right).\] (2.2)

In this equation, \(\nu_T\) is called the “eddy viscosity” which is a value related to turbulent stress in the same way that \(\nu\) is related to the viscous stress. Solving for the RST and substituting back into Equation 2.1, the following equation is derived:

\[
\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} = -\frac{\partial}{\partial x_i} \left( \frac{p}{\rho} + \frac{1}{3} \overline{u_k u_k} \right) + \frac{\partial}{\partial x_j} \left( (\nu + \nu_T) \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) \right)
\] (2.3)
Using the Boussinesq approximation, there is only one quantity left which needs to be modeled: the eddy viscosity. Eddy viscosity models differ by how the eddy viscosity is calculated. The current study looks at traditional one- and two- equation eddy viscosity models as well as the SARC model, Lag model, and lagRST model.

2.1 Equilibrium Turbulence Modeling

Equilibrium turbulence is characterized by flow with slowly-varying properties. In equilibrium boundary layers, the production and dissipation of kinetic energy is relatively balanced and external conditions which affect the large-scale structures remain nearly unchanged, or change slowly. Traditional eddy viscosity models are generally designed and calibrated for equilibrium turbulence where the mean flow variables change slowly and the Boussinesq approximation is fairly accurate.

Eddy viscosity models are built strictly on the Boussinesq approximation. For the current study, these are run for comparison purposes and are expected to perform poorly in comparison to the other models. The models tested are SA, the 1998 formulation of $k - \omega$, and SST. These models were chosen because they are the base formulations for the nonequilibrium turbulence models tested and (in the cases of the SA and SST model) they are standard RANS models. The SA model is the base formulation for the SARC model. The $k - \omega$ model is no longer considered a standard model because of its known freestream dependencies, but it is useful for comparison because it is the base model for the Lag model and some lagRST formulations. The SST model is the baseline model for other lagRST formulations.

The SA model [1] is one of the most widely used one-equation turbulence models for aerodynamic applications because of its simplicity, low cost, and robustness. It solves one partial differential equation directly for the eddy viscosity. The model is well-tuned for equilibrium flows but the formulation breaks down in other circumstances as it does not account for the production and diffusion of kinetic energy.
The SARC model [5] is based on the idea that turbulence production is increased if the principal axes of the RST lead the principal axes of the strain rate tensor, and turbulence production is decreased if the opposite is true. The Boussinesq approximation is unable to account for these effects as it assumes the RST is aligned with the strain rate tensor. In rotating flow, the principal axes of the RST lag the principal axes of the strain rate tensor and thus turbulence is suppressed. In other words, relying on the Boussinesq approximation for vortical flows will result in an over-prediction of turbulence production and eddy viscosity. The SARC model calculates the magnitude of the “lag” or “lead” of the RST in relation to the strain rate tensor, and from this calculates a “rotation function”. This is then multiplied by the production term of the Spalart-Allmaras model to suppress turbulence production under weak rotation. The Boussinesq approximation is still used to determine the Reynolds stresses, but the rotation correction will ideally correct the turbulent production term.

The $k-\omega$ model [12] solves partial differential equations for kinetic energy and the specific dissipation rate. The eddy viscosity is then calculated as the ratio of kinetic energy to the specific dissipation rate. The $k-\omega$ model has been shown to do well for adverse pressure gradients but has suffered from its dependency on freestream conditions.

The SST model [2] is a two-equation model which is a hybrid of the $k-\epsilon$ and $k-\omega$ models. Again, equations are solved for the kinetic energy and specific dissipation rate. The $k-\omega$ model is used closer to the wall and the $k-\epsilon$ model is used further from the wall. This allows for the accuracy of the $k-\omega$ near-wall behavior while reducing its dependency on the freestream conditions far from the wall and in the wake.
2.2 Nonequilibrium Turbulence Modeling Overview

When the production and dissipation of kinetic energy are balanced, the mean flow time scales and turbulent time scales are proportional (as in the Boussinesq approximation). However, in the presence of sudden changes to the flowfield such as a pressure gradient, curvature, or rotation, an equilibrium condition for kinetic energy may not exist and the calculation of the eddy viscosity can no longer rely on this assumption.

Many models have been proposed to account for nonequilibrium effects. The simplest model mathematically is the Johnson-King $1/2$ Equation Model which is based on the idea that the Reynolds shear stress differs by some unknown non-equilibrium factor from the equilibrium value predicted by an algebraic model $[28,29]$. The model uses an algebraic-type formulation for the equilibrium value of the eddy viscosity inside the boundary layer. It then accounts for nonequilibrium effects by solving an ordinary differential equation for the development of the maximum Reynolds shear stress. Since the non-equilibrium factor is unknown the solution process must be iterative. This original formulation has been shown to perform poorly in regions where the flow is in equilibrium. Ahmed and Tannehill $[30]$ modified the formulation to eliminate the iterative process and improved the results for equilibrium regions. Abid and Ridha $[31]$ also extended the formulation to three-dimensional flows and found an improvement in comparison to traditional algebraic models. Despite these modifications, the Johnson-King model has not been widely accepted in the aerospace community because of its limited robustness. A major concern is that it relies on calculating values inside and outside the boundary layer, meaning the distance from the wall must be known. This is not a robust methodology for grids with complex geometries where the location of the relevant wall may be ambiguous.

More robust models have been introduced, such as Knight and Saffmans’s “Reynolds Stress Relaxation Model” $[32]$ which relaxes the RST through a model which includes molecular and turbulent transport terms. The equilibrium value of the RST is not
based on the Boussinseq approximation but rather a nonlinear relation involving the mean strain rate and rotation rate tensors. It also includes a “gyroscopic stability term” to account for rotation effects. More recently, Hamlington and Dahm [33] proposed a model in which the relaxation equation for the RST is solved along a streamline, reducing it to an ordinary differential equation.

The current study investigates various models which account for nonequilibrium effects; namely the Lag and lagRST models. A third model which is closely related to the lagRST model is denoted as the lagRSTSST model, and uses the SST model to calculate the equilibrium turbulence values instead of the lagRST’s $k - \omega$ baseline.

2.2.1 Lag Model

The Lag model introduced in 2001 by Olsen and Coakley [7] takes advantage of the idea that the turbulent time scales are larger than the mean flow time scales, and so turbulent quantities do not react immediately to changes in the mean flow. The model takes a baseline two-equation model and couples it with a third equation which accounts for the time required for the turbulence to respond to changes in the mean flow variables. This allows the model to account for non-equilibrium effects while still using the Boussinesq approximation’s ability to accurately calculate the equilibrium flow conditions. In this case, the eddy viscosity is the quantity which is lagged in the third equation.

It should be noted that the original model was based on Wilcox’s 1998 $k - \omega$ formulation which suffers from well-known freestream dependencies. It was found that the Lag model does not suffer from the same dependencies [7]. Olsen, Lillard, and Coakley [34] modified the original Lag formulation to a simpler form in 2005,

\[
\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho u_i k - (\mu + \sigma_k \rho \nu_i) \frac{\partial k}{\partial x_i} \right) = \mathcal{P}_k - \epsilon_k \tag{2.4}
\]

\[
\frac{\partial \rho \omega}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho u_i \omega - (\mu + \sigma_\omega \rho \nu_i) \frac{\partial \omega}{\partial x_i} \right) = \mathcal{P}_\omega - \epsilon_\omega \tag{2.5}
\]
\[
\frac{\partial \rho \nu t}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i \nu t) = a_0 \rho \omega (\nu_{eq} - \nu_t), 
\]  \hspace{1cm} (2.6)

with the following auxiliary relations:

\[
\nu_{eq} = \frac{k}{\omega} \hspace{1cm} P_\omega = \alpha \rho S^2
\]

\[
P_k = \tau_{ij} s_{ij} \hspace{1cm} \epsilon_\omega = \beta \rho \omega^2
\]

\[
\epsilon_k = \beta^* \rho k \omega \hspace{1cm} S = \sqrt{2s_{ij} s_{ij}}
\]

\[
\tau_{ij} = \rho \left( \frac{2}{3} k \delta_{ij} - \nu_t \left( 2s_{ij} - \frac{2}{3} s_{kk} \delta_{ij} \right) \right) \hspace{1cm} s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

and the following parameters:

\[
a_0 = 0.35 \hspace{1cm} \beta = 0.075 \hspace{1cm} \sigma_k = 1.5
\]

\[
\alpha = \frac{5}{9} \hspace{1cm} \beta^* = 0.09 \hspace{1cm} \sigma_\omega = 0.5.
\]

The parameter \(a_0\), known as the “lag constant”, functions as a measure of the amount of lag in the system. A higher value of \(a_0\) drives the system toward equilibrium values more quickly while a lower value of \(a_0\) introduces more lag into the system. The original Lag model used a more complex lag equation,

\[
\frac{\partial \rho \nu_t}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i \nu_t) = a(R_T) \rho \omega (\nu_{eq} - \nu_t), 
\]  \hspace{1cm} (2.7)

where the equivalent of the lag constant is actually a variable function of \(R_T\), given by:

\[
a(R_T) = a_0 \left[ \frac{R_T + R_{T_0}}{R_T + R_{T_\infty}} \right] 
\]  \hspace{1cm} (2.8)

with the following parameters:

\[
R_T = \rho k/\mu \omega \hspace{1cm} R_{T_0} = 1
\]

\[
a_0 = 0.35 \hspace{1cm} R_{T_\infty} = 0.01.
\]
These parameters allow the lag variable to vary from 0.35 where dissipation largely outweights the kinetic energy in the system, to 350 where the kinetic energy is much larger than the dissipation rate. Thus, the system is lagged to a greater extent when the kinetic energy is high and energy is not being dissipated. The dependence of the lag variable on $R_T$ was deemed unnecessary and using a constant value even led to minor improvements in some cases.

The value of the diffusion constant in the kinetic energy equation, $\sigma_k$, was also changed from 0.5 in the original model to 1.5 in the 2005 version. More recently [8], Olsen again changed the constant $\sigma_k$ to 0.8 based on jet predictions. These values now define the “Standard Lag Model”. Olsen et al. [34] tested the Lag model for various high-speed flows such as a Mach 5 flat plate, an axisymmetric cylinder, and an overexpanded nozzle. The latter two cases involve shock-induced separation. For these cases, the Lag model calculated relatively good separation predictions although the heat transfer calculations did not agree with experiment. Olsen et al. [35] later tested the model using a URANS (Unsteady Reynolds Averaged Navier-Stokes) solver for various compressible and incompressible flowfields involving massive separation and found very good agreement for predictions of the mean flowfield.

More recently, Loganathan [36] used the Lag model for stall prediction verification for wind turbine blades and found better agreement with experimental data by changing the lag constant ($a_0$) to 0.2, and the diffusion constant for the kinetic energy equation ($\sigma_k$) to 0.5. It can be noted that Loganathan’s implementation of the Lag model into the ANSYS Fluent code [37] required the use of a different parameter named $\sigma_{kA}$, which is the inverse of $\sigma_k$. He sets this parameter to 2.0, meaning the value of $\sigma_k$ is 0.5.

### 2.2.2 lagRST Model

The lagRST Model, introduced by Olsen and Coakley [7], follows the same principle as the Lag model in that it assumes the turbulent variables do not react to
changes in the mean flowfield immediately. In this case, the third transport equation lags the RST instead of the eddy viscosity:

\[
\frac{DR_{ij}}{Dt} = a_0 \omega (R_{ij_{eq}} - R_{ij}),
\]

(2.9)

where the quantity \( R_{ij_{eq}} \) is the equilibrium Reynolds stresses tensor which would be calculated by the Boussinesq approximation. This is given below in a slightly modified form of Equation 2.2 which does not assume incompressibility:

\[
R_{ij_{eq}} = 2 \frac{k_{eq}}{3} \delta_{ij} - 2 \nu_{t_{eq}} \left( s_{ij} - \frac{1}{3} s_{kk} \delta_{ij} \right).
\]

(2.10)

\( R_{ij} \) is the “true” Reynolds stress, defined as:

\[
R_{ij} = -\tau_{ij} / \rho = \overline{u_i'u_j'}. \tag{2.11}
\]

This process is very similar to a full Reynolds Stress Model in that it models the RST separately from the Boussinesq approximation. However, instead of using the full Reynolds stress transport equation, it models the RST using the equilibrium value calculated by the two-equation model.

Olsen and Coakley originally introduced the lag equation in the form:

\[
\frac{D(\rho R_{ij})}{Dt} = a(R_T) \omega \rho (2 \nu_t s_{ij} - R_{ij}),
\]

(2.12)

where it is assumed the quantity \( a(R_T) \) would be defined as something similar to Equation 2.8. Churchfield and Blaisdell [26] identified a few problems with this formulation. One problem is that the left side of the equation is the material derivative of \( \rho R_{ij} \) while the right side is the material derivative of only \( R_{ij} \). Thus, a better formulation of the lag equation is Equation 2.9. The second problem is that when comparing Equation 2.12 to Equation 2.9, it is clear that Olsen and Coakley define the equilibrium value of the Reynolds stresses to be \( R_{ij_{eq}} = 2 \rho \nu_t s_{ij} \). Comparing this to Equation 2.10, it appears the \((2/3)k_{eq} \delta_{ij}\) and the \((2/3)s_{kk} \delta_{ij}\) terms are not included in the equilibrium value of the Reynolds stress. Churchfield and Blaisdell thus
suggested instead that the model be formulated in a slightly different manner. The Boussinesq approximation can be rewritten as

$$\frac{DR_{ijeq}}{Dt} - \frac{1}{3} R_{kk eq} \delta_{ij} = -2 \nu_{eq} \left( s_{ij} - \frac{1}{3} s_{kk} \delta_{ij} \right).$$ (2.13)

Churchfield and Blaisdell leave off the $\frac{1}{3} s_{kk} \delta_{ij}$ term, presumably because they assume incompressible flow where the trace of the strain rate tensor is zero. This can then be simplified to:

$$\frac{DR_{ijeq}^D}{Dt} = 2 \rho \nu_{eq} s_{ij}.$$ (2.14)

where the superscript D denotes the deviatoric part of the tensor. The deviatoric part of the RST then becomes the lagged variable. Churchfield and Blaisdell tested this formulation alongside Lillard’s formulation which lags the entire RST and found very similar results with both models. The formulation used in this study lags the entire RST and not just the deviatoric part.

The original formulation of the lagRST model uses the $k - \omega$ model as its baseline formulation to calculate the equilibrium values. In addition to this, Lillard implemented another model which he calls the lagRSTSST model. This model uses the SST model as its baseline formulation. The transport equations for $k$ and $\omega$ are the same as defined by Menter [2] in his original model:

$$\frac{D(\rho k)}{Dt} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left( (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right)$$ (2.15)

$$\frac{D(\rho \omega)}{Dt} = \frac{\gamma}{\nu_t} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left( (\mu + \sigma_\omega \mu) \frac{\partial \omega}{\partial x_j} \right) + 2 \rho (1-F_1) \sigma_{\omega} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}.$$ (2.16)

The modeling constants and blending functions are also the same as in Menter’s original reference. The equilibrium value of the eddy viscosity is calculated the standard way:

$$\nu_{eq} = \frac{a_1 k}{max(a_1 \omega; \Omega F_2)}.$$ (2.17)
The equilibrium values of the Reynolds stresses can then be calculated by substituting into Equation 2.10.

It can be noted that in deriving this formulation, there are two different equations for the turbulent kinetic energy. One comes from the baseline turbulence model (Equation 2.4 or 2.15). The other comes from the fact that by contracting the indices of the RST, the following relation is derived:

\[ R_{kk} = \overline{u'_k u'_k} = 2k. \] (2.18)

To reconcile these differences, the turbulent kinetic energy calculated by the baseline turbulence model can be thought of as the equilibrium value of turbulent kinetic energy. The turbulent kinetic energy value used in the flowfield will come from the trace of the RST (Equation 2.18).

The lagRST and lagRSTSST models have been implemented into the OVERFLOW code by Randy Lillard and have been tested for only a limited number of flows. In 2008, Churchfield and Blaisdell [26] modeled a q-vortex and found that the model was capable of predicting the lag between the principal axes of the RST and strain rate tensors, but that for a vortical flow, the amount of lag is coupled with the magnitudes of the Reynolds stresses and thus unintentionally affects the growth and decay of turbulence. Churchfield and Blaisdell later modeled a wingtip vortex [27] and found that the lagRST model performed much better than linear eddy viscosity models, and showed similar or more accurate computations of the mean flowfield as compared with the SARC model. However, the dependency of the magnitude of the RST on the amount of lag between the principal axes of the RST and strain rate tensor was again problematic, and Churchfield gave some suggestions for decoupling the two effects. Since the current study does not explicitly deal with vortical flows, this problem is not discussed.

Lillard, Olsen, Blaisdell, and Lyrintzis [25] used the lagRST model for various simple separation cases as well as more complicated SWTBLI problems. They found that the lagRST model performed well using the same coefficients as the standard
Lag model \( (a_0 = 0.35, \sigma_k = 0.8) \) for incompressible and transsonic predictions, but performed poorly for mixing layer components. The lagRSTSST model with the same coefficients performed well for the more complicated SWTBLI cases but the separation predictions for the simpler test cases were poor.
3. COMPUTATIONAL SETUP

3.1 OVERFLOW Code Description

NASA’s OVERFLOW code [24] is a three-dimensional, time-marching, implicit, compressible Navier-Stokes solver. It runs on structured, overset grids, meaning the RANS equations are solved on one or more grids which interpolate from each other at boundaries where no boundary conditions are set. The equations are discretized using a finite-difference formulation with flow quantities computed at the grid nodes. Although the code is three-dimensional, it can also solve two-dimensional cases with three parallel planes and symmetric boundary conditions set on the two end-planes.

OVERFLOW solves the RANS equations in generalized coordinates with the array $q$ containing all conserved variables. The explicit viscous and inviscid fluxes are separated, where the inviscid fluxes can be computed with a Roe Upwind or central differencing scheme. The viscous fluxes are always computed using a second-order central differencing scheme. A wide variety of implicit solution algorithms are implemented for the timestepping terms. Local timestepping, multigrid techniques, and grid sequencing can all be used to accelerate convergence to a steady state solution. Several RANS models are hard-coded including algebraic models like Baldwin-Lomax [38], one-equation models (BB [15] and SA [1]), two-equation models ($k - \omega$ [12] and SST [2]), as well as various detached-eddy RANS-LES hybrid models. Rotation and curvature corrections are available for one- and two- equation models.

All calculations were performed using a modified version of OVERFLOW v2.2e. In this version the turbulence models are spatially discretized with a second-order operator, which is notable because previous versions of OVERFLOW discretized turbulence models instead using a first-order operator. It can be noted that the first-order discretization of lag equation of the Lag and lagRST models leads to excessive grid
density requirements [7], which are relaxed when the discretization is second order. The ARC3D diagonalized Beam-Warming scalar pentadiagonal scheme [39] is used as an implicit solver for the timestepping terms.

Convergence is monitored in a few different ways. OVERFLOW outputs the $l^2$ and $l_\infty$ norm of the conserved variables, as well as the $l^2$ and $l_\infty$ norm of the turbulent variables for each grid. Mass flow rate residuals are also monitored, as well as the forces on the airfoil or wing. Theoretically these will all converge to a steady state value for the steady state runs. For the unsteady cases, the number of Newton subiterations for each physical timestep is set so that the $l^2$ norm of the RHS converges at least two orders of magnitude for each physical timestep. Once the case is run past the transient region, it is averaged over a specified number of timesteps.

For both cases studied here, the Mach number is well within the incompressible range. For the NACA 0021 3-D wing the freestream Mach number was 0.05 and for Wadcock and Coles’ NACA 4412 2-D airfoil the freestream Mach number was around 0.085. Because OVERFLOW is a compressible flow solver, the equations become numerically stiff in the incompressible regime. To alleviate this, OVERFLOW’s Smith-Weiss low-Mach number preconditioning [40] is enabled to improve stability and convergence.

The grids for this study were all created in Chimera Grid Tools, which outputs Plot3D files that can be read by OVERFLOW. Chimera Grid Tools has many utilities designed for building overset grids. Another package, called Pegasus, takes separate Plot3D grid files and connects them into a single domain. This involves first cutting out grid points which exist inside of solid bodies. Points on the edges of grids where boundary conditions are not set are then denoted as “fringe points” which means information from other grids will be interpolated onto these boundaries. The interpolation stencils are then generated between the boundaries of the grids. All of this information is passed on to the flow solver.
3.2 Wadcock and Coles Two-Dimensional NACA 4412 Airfoil

3.2.1 Experimental Setup

The Wadcock and Coles flying hotwire data was collected in the GALCIT (Graduate Aeronautical Laboratories, California Institute of Technology) 10-ft wind tunnel. The wing was mounted horizontally between vertical false walls in a cylindrical test section and pitched to an angle of attack of 13.87 degrees. Because of the flying hotwire setup, the wing was mounted facing downwards and velocity and turbulence intensity data were collected above the suction surface upstream of the trailing edge and also in the wake. The test section was 322.8 cm in the streamwise direction and 10 ft. high (304.8 cm). The wing was mounted with the leading edge on the centerline of the wind tunnel and 51.4 cm downstream of the inlet to the test section, and pitched to an angle of attack of 13.87 degrees. The chord length of the airfoil is 90.12 cm, and the boundary layer is tripped to turbulent at x/c = 0.025 on the upper surface and x/c = 0.103 on the lower surface. This setup is depicted below in Figure 3.1, flipped upside down from its original configuration so that the lift vector points upwards.

![Figure 3.1. Depiction of the GALCIT Wind Tunnel test section setup with a NACA 4412 airfoil mounted inside.](image-url)
Wadcock and Coles designed the experiment to account for tunnel blockage effects by nondimensionalizing all variables (velocity, pressure, etc.) by reference values instead of freestream values. The reference point is depicted in Figure 3.1, 98.1 cm downstream of the leading edge and 123 cm below the tunnel centerline (about 30 cm from the lower wall). This is far enough from the wall to be outside of the boundary layer, and far enough from the airfoil to not be disturbed by the separation zone.

The reference quantities measured at this point are reported as follows [18]: \( T_{ref} = 24.3^\circ C \), \( p_{0,ref} = 72.9 \text{ cm Hg} \), \( Q_{ref} = 3.13 \text{ mm Hg} \), \( \rho_{ref} = 1.14 \text{ kg/m}^3 \), \( \nu_{ref} = 0.000161 \text{ m}^2/\text{s} \), \( U_{ref} = 27.13 \text{ m/s} \), where \( Q \) denotes the dynamic pressure, given below:

\[
Q = p_0 - p_{static} = \frac{1}{2} \rho U^2.
\] (3.1)

The report does not list what any of the freestream conditions are; however by setting the outflow static pressure equal to the freestream static pressure, the ratio between the reference velocity and the freestream velocity in the CFD result was found to be \( u_{ref}/u_\infty = 0.93 \). This ratio is taken into account when comparing the two datasets. A similar correction is used for the pressure data. By assuming incompressibility, the following relation can be derived, where the subscript “ref” denotes a quantity nondimensionalized by the reference conditions, and the subscript “\( \infty \)” denotes a quantity nondimensionalized by the freestream conditions:

\[
c_{p,ref} = 1 + \frac{U^2_\infty}{U^2_{ref}} (c_{p,\infty} - 1).
\] (3.2)

To achieve a speed of \( U_{ref} = 27.13 \text{ m/s} \) at the reference point as specified above the inlet speed for the computational run was chosen to be \( U_{ref}/0.93 \), which comes out to be 29.17 m/s. With the reference conditions as specified, this results in a Reynolds number of 1.63 million.
3.2.2 Computational Domain

The effects of the tunnel blockage are important for this setup, especially at high-lift conditions, so it is important to be able to model the upper and lower walls of the test section. It becomes clear from analyzing the dimensions of Figure 3.1 that the domain must be extended upstream and downstream of the test section. However, the boundary layer growth in the test section has an important effect on tunnel blockage. Therefore, the portions of the upper and lower walls of the background domain which are within the physical test section are given a viscous wall boundary condition; thus the streamwise and perpendicular components of the velocity are both zero at the wall. The portions of these walls which are outside of the test section are given an inviscid wall boundary condition, meaning the perpendicular component of the velocity is zero but the streamwise component can be nonzero. This is shown in Figure 3.2. The boundary condition at the inlet is an inflow condition which relates the stagnation pressure and stagnation temperature at the inlet to the stagnation pressure in the freestream (which is also set by the user). The ratios $p_{0, \text{inlet}}/p_{0, \infty}$ and $T_{0, \text{inlet}}/T_{0, \infty}$ are both set to 1.0. The outflow boundary condition is a static pressure condition which sets the static pressure at the downstream end of the tunnel equal to the freestream static pressure value.

![Wind tunnel boundary conditions](image)

Fig. 3.2.: Wind tunnel boundary conditions.

Wadcock mentions that no modifications to the NACA 4412 profile were made, and namely that the trailing edge had the thickness prescribed by Jacobs, Pinkerton, and Ward [41]. The trailing edge of the grid therefore reflected this. In the author’s
previous work with blunt-trailing edge airfoils, it was determined that generating a C-grid topology is not effective. The original idea used to create a C-grid was to extend the airfoil grid out behind the airfoil perpendicular to the trailing edge face, and then use another grid to fill in the gap between the upper and lower planes of the wake (Figure 3.3). When the flow is solved on this type of topology, however, fringe points exist which are also part of other interpolation stencils. Thus, there are coincident fringes where points exist in the flowfield which are fringe points for both grids. To demonstrate this, consider point A on the green grid in Figure 3.3. Since point A is a fringe point (it is one point downstream of the boundary of the green grid), the pink grid will need to interpolate onto it. Now consider point B on the pink grid. Point B is also a fringe point (it is one point away from the boundary of the pink grid in the cross-stream direction), and thus the green grid will need to interpolate onto it. The interpolation stencil for point B will include point A. Therefore the flow is not solved at point A and convergence to an accurate solution is not possible. This problem was identified by Churchfield [42]. One possible way to remedy this is to build a “band-aid” grid which is a connector between the wake and wing grids (Figure 3.4). Attempting this, however, resulted in the solution converging to a non-physical result. The reason for this is likely that the interpolation is taking place in regions of high gradients in the flow variables. Both of these ideas were discarded and instead an orthogonal grid (O-grid) topology was pursued.

Fig. 3.3.: C-grid trailing edge abutting grid topology. The green grid defines the trailing edge of the wing and also the wake, and the pink grid defines the wing.
The advantages of an O-grid are the orthogonality of the off-body spacing near the trailing edge, as well as the fact that multiple grids in the trailing edge region are not needed. Every time information is transferred to another grid, some information is lost if the grid points do not line up precisely. The main disadvantage of an O-grid is that in the far-wake region, cells become very large in the cross-stream direction. In the near-wake region, however, the grid spacing is still very small and the O-grid maintains good resolution for some distance away. Moving downstream of the trailing edge, the flow then transfers to a finer wake grid which continues to resolve the wake and separation region. This grid is also moderately splayed so that the grid spacings
in the interpolation regions at the top and bottom of the wake grid are somewhat aligned. This is shown in Figure 3.5.

Special considerations were taken with the airfoil to capture the intricacies of the experimental setup. To simulate laminar-turbulent boundary layer transition on the wing, OVERFLOW has the option of turning off the turbulent production term in certain regions of the flow. Although this does not give a perfect representation of the tripping process, it allows for some type of boundary layer trip correction. The grid was generated so that a grid point aligns precisely with the trip location on both the upper and lower surfaces, and so the turbulent production term can be turned on at precisely the correct point on the wing.

Another consideration is that the airfoil is at a high angle of attack, meaning there will be a large leading edge suction peak. To resolve this strong favorable pressure gradient at the leading edge, grid points were clustered close to the stagnation point along the upper surface (Figure 3.6). To ensure sufficient grid resolution in the near-
wake region, grid points were also clustered on the trailing edge surface (Figure 3.7). Because the spacing on the surface of the blunt trailing edge face must match the spacing on the upper and lower surfaces near the trailing edge, the grid must be quite fine in this region also.

![Fig. 3.6.: NACA 4412 leading edge spacing. Fig. 3.7.: NACA 4412 trailing edge spacing and wake region.](image)

Since this is a two-dimensional case, the flow solver is supplied with three planes of grid data. The two outer planes are set with a 2-D symmetric boundary condition for all of the grids. On the wing grid, a viscous wall boundary condition is set as well as a periodic boundary condition in the spanwise direction. Aside from the 2-D symmetry condition, no boundary conditions are set for the wake grid since it is simply a denser grid for the wing grid to interpolate onto in the wake region.

The solver used for the inviscid fluxes for this case is a 2nd-order central differencing scheme with added matrix dissipation. Because of the massive separation on the airfoil, the flowfield was unsteady enough that the solution had to be run time-accurately with OVERFLOW’s dual time integration. The physical timestep was a
point of further study and is described in Section 3.2.4. The computational timestep was set by a constant CFL number of 1.0. The time-resolved solution was calculated by setting the number of Newton subiterations to 5 in order to assure the \( l_\infty \) norm of the inviscid fluxes dropped by at least two orders of magnitude between each physical timestep. This case was run for between 20,000 and 40,000 physical timesteps to move past the transient region, and then flow quantities were averaged over the following 10,000 timesteps.

The central difference scheme was employed for the inviscid flux calculations, using second- and fourth- order matrix dissipation. The second- and fourth- order dissipation coefficients are set to 2.0 and 0.4 respectively, the matrix dissipation limit on linear and nonlinear eigenvalues is set to 0.3, and the dissipation scheme uses Roe-averaging for half-grid point flow quantities.

### 3.2.3 Grid Convergence Study

An initial grid was generated with a geometry matching the wind tunnel setup described above. This grid (referred to as the medium grid) had 329 points in the airfoil chordwise direction including 25 points on the trailing edge, and 96 points in the orthogonal direction. The off-body spacing was set so that the minimum \( y^+ \) value would be 0.7, with 30 points in the boundary layer. The wake grid extended outwards with nearly the same orthogonal spacing as the wing grid. The background spacing was 1% of the chord in the streamwise direction in the test section and extended upstream and downstream with a stretching ratio of 1.05. The boundary layer of the tunnel walls was resolved to a \( y^+ \) of 1 at the first grid point off of the wall in the boundary layer, with at least 10 points in the boundary layer. Two other grids, a “coarse” and “fine” grid were then developed based off of these initial parameters. For the coarse grid, the spacings were increased by a factor of \( \sqrt{2} \) and grid points decreased by a factor of \( \sqrt{2} \). Since the grid is two dimensions, this effectively decreases the number of grid points by roughly a factor of 2. For the fine grid, the opposite
occurs and the grid point count increases by roughly a factor of 2. The grid parameters are summarized below:

Table 3.1: A summary of the parameters for the NACA 4412 grid convergence study.

<table>
<thead>
<tr>
<th>Grid</th>
<th>Total Grid Points</th>
<th>$y^+$ value off-body</th>
<th>DTPHYS</th>
<th>Number of Timesteps</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>374,907</td>
<td>0.98</td>
<td>0.0424</td>
<td>7,071</td>
</tr>
<tr>
<td>Medium</td>
<td>687,594</td>
<td>0.70</td>
<td>0.0300</td>
<td>10,000</td>
</tr>
<tr>
<td>Fine</td>
<td>1,296,123</td>
<td>0.50</td>
<td>0.0212</td>
<td>14,142</td>
</tr>
</tbody>
</table>

The lagRST SST turbulence model with a $\sigma_k$ value of 0.8 was used for the grid convergence study. As will be discussed in Section 3.2.4, the physical timestep used for the medium grid was a nondimensional DTPHYS = $c/(v_{ref}t) = 0.03$ and flow quantities were averaged over 10,000 timesteps. For the coarse and fine grids the physical timestep was altered by a factor of $\sqrt{2}$ in order to keep the CFL number constant between runs. Various quantities, namely lift coefficient, drag coefficient, and minimum pressure coefficient were averaged and output from these runs to be analyzed. These values, which will be denoted as a general quantity $Y$, are all sensitive to grid resolution. To quantify the results of this study, a Richardson extrapolation (R.E.) [43] was used. Because the inviscid fluxes, viscous fluxes, and turbulence models are all discretized using second-order operators, it was assumed that the method is second order. This is confirmed by the linearity of the value of $Y$ when plotted as a function of $1/(\text{number of grid points})^2$. As an example, the lift values are plotted in this manner in Figure 3.8.

The idea of the Richardson extrapolation is that the coarse, medium, and fine solutions can be expressed in terms of the “exact” solution and higher order error terms, as follows:

$$Y_{\text{coarse}} = Y_{\text{exact}} + c_1 \Delta x^2 + c_2 \Delta x^3 + \ldots$$  (3.3)
Fig. 3.8.: Linear plot of lift coefficient as a function of $1/(\text{number of grid points})^2$. 
\[ Y_{medium} = Y_{exact} + c_1 \left( \frac{\Delta x}{\sqrt{2}} \right)^2 + c_2 \left( \frac{\Delta x}{\sqrt{2}} \right)^3 + \ldots \quad (3.4) \]

\[ Y_{fine} = Y_{exact} + c_1 \left( \frac{\Delta x}{\sqrt{2}} \right)^2 + c_2 \left( \frac{\Delta x}{\sqrt{2}} \right)^3 + \ldots \quad (3.5) \]

Plugging in the value of lift coefficient, drag coefficient, or minimum pressure coefficient, the \( c_1 \) and \( c_2 \) terms can be eliminated and one only fourth- and higher-order error terms will be left. Neglecting these higher order error terms, one can solve for the value of \( Y_{exact} \) to get an idea of what the grid is converging towards. The results are tabulated in Table 3.2.

Table 3.2: Richardson extrapolation results for lift, drag, and pressure coefficient for the 2-D NACA 4412 case.

<table>
<thead>
<tr>
<th>Grid</th>
<th>Lift Coefficient</th>
<th>Drag Coefficient</th>
<th>Minimum Pressure Coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>1.622</td>
<td>0.0376</td>
<td>-6.65</td>
</tr>
<tr>
<td>Medium</td>
<td>1.627</td>
<td>0.0354</td>
<td>-6.79</td>
</tr>
<tr>
<td>Fine</td>
<td>1.629</td>
<td>0.0349</td>
<td>-6.86</td>
</tr>
<tr>
<td>R.E.</td>
<td>1.629</td>
<td>0.0346</td>
<td>-6.93</td>
</tr>
</tbody>
</table>

The values from the medium grid are close to 1% of the lift value given by the Richardson extrapolation, 2% of the drag value, and 2% of the minimum pressure coefficient value. These are extremely small differences. By increasing the grid point count by a factor of two, the values change by between 0% and 1.5% between the medium and fine grids. The fine grid is already over one million grid points and a two-dimensional grid this dense likely cannot feasibly be extended to three dimensions if this type of case is ever extrapolated to 3-D flows. Therefore, the medium grid was chosen to run on.

For further validation of this grid, the pressure distribution and velocity profiles were analyzed. Figure 3.9 depicts the velocity profiles calculated for each grid at a
location $x/c = 0.8$ on the upper surface of the airfoil, which is in the separated region. All of the plots collapse to a certain extent far enough from the separated region; however close to the surface of the airfoil there are differences. The fine and medium grids are indistinguishable at this scale while the coarse grid predicts velocities lower than those predicted by the other two grids. This suggests that further refinement will have little to no effect on the velocity profiles.

![Velocity profile plots](image)

Fig. 3.9.: Velocity profile plots of the NACA 4412 case showing the effects of grid density.

The pressure distribution is plotted in Figures 3.10 and 3.11 for the upper and lower surfaces. The fine grid generally predicts the lowest pressures on the upper surface and the highest pressures on the lower surface. Figure 3.11 enlarges the minimum pressure coefficient region to show the effect of grid resolution on resolving this strong gradient. Again, the medium grid is much closer to the fine grid than it is to the coarse grid, suggesting further refinement will not drastically alter the pressure coefficient values.
Fig. 3.10.: Pressure coefficient plots of the NACA 4412 case showing the effects of grid density.

Fig. 3.11.: Minimum pressure region of the pressure coefficient plots.
3.2.4 Timestep Convergence Study and Run Length

The time-accurate parameters include the size of the physical timestep as well as the time period the solution is averaged over. Although these are two very different variables, the studies of the two parameters as well as the grid density requirements are all very much dependent on each other and so this is somewhat of an iterative process. All of the iterations of the study determining the timestep and averaging parameters are not discussed here; only the final parameters which were determined to be the most accurate and computationally reasonable.

In OVERFLOW, the time advancement of the solution is controlled by a physical timestep (DTPHYS) which is nondimensionalized by the freestream velocity and the reference length. The timestep was initially set based on the suggestions of Olsen et al. to be $\Delta t = 10^{-3}c/U_\infty$ [35], meaning a fluid element traveling at the freestream velocity travels one chord length in $10^3$ timesteps. In OVERFLOW notation, this corresponds to a value DTPHYS of 0.001. After experimenting with different values of DTPHYS, it was found that the vortices shed travel at a much lower frequency than those described in Olsen et al., and that increasing the timestep did not change the solution to a great extent. A nominal timestep of DTPHYS = 0.0100 was chosen to begin with, and the resulting solution was compared against solutions using various other values of DTPHYS. The timestep convergence study compared DTPHYS values of 0.00333, 0.0100, 0.0300, and 0.0600. A DTPHYS of higher than 0.0600 resulted in an unstable run, presumably because the timestep is large enough that the CFL condition is not satisfied. Each simulation was run for the same amount of dimensional physical time (9.26 seconds). All cases were run on one node of 24 processors each. The parameters of each run are summarized in Table 3.3.

The solutions were run time-accurately and averaged over the specified number of timesteps. Velocity profiles were taken on the upper surface in three locations: on the airfoil before separation ($x/c = 0.7$), inside the separation region ($x/c = 0.9$), and behind the airfoil in the wake ($x/c = 1.1$). The results are shown in Figures 3.12-3.14.
Table 3.3: Summary of the timestep convergence study parameters.

<table>
<thead>
<tr>
<th>DTPHYS</th>
<th>Dimensional Simulation Time (s)</th>
<th>Number of Timesteps</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00333</td>
<td>9.26</td>
<td>90,000</td>
</tr>
<tr>
<td>0.0100</td>
<td>9.26</td>
<td>30,000</td>
</tr>
<tr>
<td>0.0300</td>
<td>9.26</td>
<td>10,000</td>
</tr>
<tr>
<td>0.0600</td>
<td>9.26</td>
<td>5,000</td>
</tr>
</tbody>
</table>

Fig. 3.12.: Velocity profiles taken off the upper surface at $x/c = 0.7$, showing the effects of timestep length.

Fig. 3.13.: Velocity profiles taken off the upper surface at $x/c = 0.9$, showing the effects of timestep length.
Fig. 3.14.: Velocity profile taken in the wake at x/c = 1.1, showing the effects of timestep length.
Timestep resolution does not have a large impact on the velocity predictions. In the separation region as well as in the wake region, there are some noticeable differences using timesteps larger than DTPHYS = 0.03, but there is very little impact below that threshold. To take a closer look at the time-resolved solution, the force oscillations were analyzed. Scaled to physical time over the period of 9.26 seconds, the results are shown in Figure 3.15.

Fig. 3.15.: Unsteady lift coefficient plotted as a function of physical time, showing the effects of timestep length.

In general, the less resolved the timestep is, the larger the amplitude of the oscillations is. Additionally, the structure of the oscillations is lost at timesteps higher than DTPHYS = 0.03. Analyzing the signals further one can calculate the mean lift ($\bar{l}$), root mean square lift ($l_{RMS}$), and strouhal number ($St$). The root mean square (RMS) of a signal is defined here as measure of the magnitude of how much a signal varies about its mean, which is related to the amplitude of oscillation. It is given by Equation 3.6 below,
\[ l_{RMS} = \sqrt{\frac{1}{N} ((l_1 - \bar{l})^2 + (l_2 - \bar{l})^2 + \ldots + (l_N - \bar{l})^2)}, \quad (3.6) \]

where \( N \) is the number of force data samples throughout the run.

The Strouhal number is a nondimensional frequency which measures the propagation of the vortices shedding downstream, which is the cause of the oscillations in lift shown in Figure 3.15. The Strouhal number nondimensionalizes the frequency \( f \) by a reference length and a reference velocity, which in this case is the chord length and the velocity measured at the reference location, as shown below:

\[ St = \frac{fc}{U_{ref}}. \quad (3.7) \]

These quantities are summarized in Table 3.4, along with the runtime of each case on 24 processors.

<table>
<thead>
<tr>
<th>DTPHYS</th>
<th>( \bar{l} )</th>
<th>( l_{RMS} )</th>
<th>( St )</th>
<th>Runtime</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00333</td>
<td>1.645</td>
<td>0.0259</td>
<td>0.0430</td>
<td>90 hours</td>
</tr>
<tr>
<td>0.0100</td>
<td>1.637</td>
<td>0.0263</td>
<td>0.0360</td>
<td>30 hours</td>
</tr>
<tr>
<td>0.0300</td>
<td>1.641</td>
<td>0.0267</td>
<td>0.0394</td>
<td>10 hours</td>
</tr>
<tr>
<td>0.0600</td>
<td>1.629</td>
<td>0.0569</td>
<td>0.0441</td>
<td>5 hours</td>
</tr>
</tbody>
</table>

From this, one can conclude the mean lift and Strouhal number do not follow much of a trend in relation to timestep resolution. The RMS lift, however, seems to increase as the timestep becomes less resolved. Figure 3.15 confirms this, as the amplitudes of the signals become larger as the timestep increases. At timesteps above DTPHYS = 0.03, however, \( l_{RMS} \) increases dramatically. This conclusion, combined with the velocity profiles in the detached flow region (Figures 3.13 and 3.14), suggests that a physical timestep of larger than DTPHYS = 0.03 is too large, and a physical timestep of smaller than DTPHYS = 0.03 does not result in major differences in the
solution. As the RMS velocity for DTPHYS = 0.03 case is only 3% larger than the RMS velocity for the DTPHYS = 0.00333 case, it was not worthwhile to have to run each case nine times longer just to decrease the amplitude of oscillations a small amount. Therefore DTPHYS = 0.03 was chosen as the physical timestep.

After deciding on a physical timestep, the next task was to determine how long the solution needs to be run. The oscillations of the lift signal suggest the unsteady solution is somewhat periodic, and so it is necessary to average over enough oscillations to make the solution statistically meaningful. At a timestep of 0.03, one oscillation is completed in about 900 timesteps. Solutions were run for 5,000, 10,000, and 20,000 timesteps to ideally capture 5.5, 11, and 22 oscillations, respectively. The unsteady lift plots are shown in Figures 3.16-3.18.

![Fig. 3.16.: Unsteady lift plotted as a function of timestep for the 5,000 timestep run.](image1)

![Fig. 3.17.: Unsteady lift plotted as a function of timestep for the 10,000 timestep run.](image2)

The $l$, $l_{RMS}$, and Strouhal number data is tabulated in Table 3.5, along with the runtime of each case on 24 processors.

The run length does not seem to significantly affect the mean lift or RMS lift. However, running over a longer period allows for more oscillations in lift, making it
Fig. 3.18.: Unsteady lift plotted as a function of timestep for the 20,000 timestep run.

Table 3.5: Summary of the averaging study results.

<table>
<thead>
<tr>
<th>Number of Timesteps</th>
<th>Dimensional Time (s)</th>
<th>( \bar{l} )</th>
<th>( l_{RMS} )</th>
<th>( St )</th>
<th>Runtime</th>
</tr>
</thead>
<tbody>
<tr>
<td>5,000</td>
<td>4.63</td>
<td>1.637</td>
<td>0.0279</td>
<td>0.0358</td>
<td>5 hours</td>
</tr>
<tr>
<td>10,000</td>
<td>9.26</td>
<td>1.641</td>
<td>0.0267</td>
<td>0.0394</td>
<td>10 hours</td>
</tr>
<tr>
<td>20,000</td>
<td>18.52</td>
<td>1.641</td>
<td>0.0271</td>
<td>0.0394</td>
<td>20 hours</td>
</tr>
</tbody>
</table>
easier to calculate the Strouhal number. This is clear when examining the convergence of the Strouhal number as the number of iterations increases. Additionally, ten hours per case is a very reasonable time period to run a parametric study of turbulence models. Therefore, the time-accurate procedure chosen was to run each case for 10,000 timesteps at a nondimensional timestep of DT\text{PHYS} = 0.03.

### 3.2.5 Validation Case

When performing calculations of the NACA 4412 flowfield, it is not expected that any of the models tested will capture all of the necessary physics to accurately predict a separated flowfield. However, to assure that the grid and solver are sufficient for general cases, a case was run with the lagRST model using the same wind tunnel grid, but rotating the airfoil to a lower angle of attack. Wadcock and Coles present pressure data for an \( \alpha = -4^\circ \) case, which is close to the zero-lift angle of attack. One would not expect to see separation or large adverse pressure gradients in this case so it is essentially an equilibrium flow. Figure 3.19 compares the lagRST result to the experimental measurements.

The pressure data calculated by OVERFLOW using the lagRST model agrees extremely well with the experimental measurements. While pressure data is not necessarily the best way to judge a turbulence model, it is encouraging to see the grid and solver can somewhat accurately represent the Wadcock and Coles experimental setup.

### 3.3 NACA 0021 Wing

#### 3.3.1 Experimental Setup

An unrelated research project ongoing at Purdue University involves the modeling of loads on a NACA 0021-based wing of aspect ratio 6 at various wind tunnel conditions, with angle of attack (\( \alpha \)) and Mach number (M) as the independent vari-
Fig. 3.19.: Velocity profile plots of the NACA 4412 case showing the effects of grid density.
ables. The wind tunnel measurements will include only pressure data. With the task of modeling separation, two cases were chosen to study rigorously with the following parameters: $\alpha = 10^\circ, M = 0.05, Re = 228,000$ and $\alpha = 16^\circ, M = 0.05, Re = 228,000$.

During the course of the parametric study the research project required, it was discovered that for the 10 degree case, there is generally a small separation zone covering between 10 and 15 percent of the airfoil near the trailing edge region. After inspecting the force convergence plot for oscillations, it appears the unsteady contribution of the propagating vortices is negligible. Therefore, this small separation region does not qualify as massive unsteadiness due to separation and the problem can be run steady state with no reservations.

The 16 degree case was investigated because the separation region is much larger and the differences between turbulence models should be seen more clearly. However, all measures of convergence show that the larger separation region results in an unsteady problem. Ideally, this should be run time-accurate in order to capture the shedding vortices propagating downstream and the effect they have on the flowfield velocities, the lift, and the separation point. However, it was determined that time-accurate runs on a grid this large are not feasible. One reason for this is the number of grid points on which the flow needs to be solved. For example, the 3-D grid NACA 0021 grid has close to 30 times more grid points than the 2-D grid for the NACA 4412 airfoil case. Additionally, because of the high grid density of the NACA 0021 wing grid, the physical timestep must also be very low so that the CFL number is not too large. Because of this, attempts to run this case time-accurately showed that the physical timestep must be an order of magnitude lower than the 2-D NACA 4412 case. Due to both the larger grid and the lower timestep requirements, it was determined that with the computer resources available at Purdue, time-accurate runs on the 3-D NACA 0021 grid are not feasible.

The wind tunnel setup is different from the two-dimensional NACA 4412 experiments in that the wing is tested in the wind tunnel as a three-dimensional object - it does not reach the full span of the tunnel. It must therefore be modeled in a three-
dimensional domain. The Boeing Wind Tunnel in the Aerospace Sciences Laboratory (ASL) has a test section of dimensions 6 ft. in width and 4 ft. in height. It has a six component force balance to measure forces as well as locations across the test section for pressure ports. The NACA 0021 wing is 4 ft. (48 inches) in span and 2/3 ft. (8 inches) in chord length, so there is a gap of one inch between the wall and the wing on either side. Pressure measurements were collected at 41 points on the wing, in four chordwise bands specified in Figure 3.20.

![Figure 3.20: Chordwise bands of wind tunnel pressure measurement, at 8 and 22 inches from the wing root on either side of the root indicated by the dark black lines.](image)

As in the Wadcock and Coles study, at high lift conditions the blockage effects of the walls of the wind tunnel are important; however modeling these effects was not part of the research project. Instead, the domain extends out to the freestream 100 chord lengths away. Because of this and a few other issues (see Section 4.2), the validity of the comparisons between the CFD and wind tunnel data may be in question. This will be discussed later.

### 3.3.2 Computational Domain

Because this is a three-dimensional case there are various complications in building a grid which are not applicable for a two-dimensional grid. The first thing to note is that the wingtips must be closed so that flow cannot travel through the surface of the wing. A wingtip cap grid was created first by revolving the wingtip to create a solid surface. The center line of points was extracted and then extruded inboard at the leading and trailing edges (Figure 3.21). Chimera Grid Tools’ SURGRD tool used
the reference wing surface to to march this line along the upper and lower surfaces. The final result was that the wingcap grid overlaps with the wing grid and wraps around the leading and trailing edges (Figure 3.22). Because the wingcap grid must define a sharp corner when wrapping around the trailing edge, a significant number of points were necessary for this grid.

![Fig. 3.21.: Center line of the wingcap surface to be marched inboard along the surface of the wing.](image1)

![Fig. 3.22.: Wingtip cap for the NACA 0021 wing.](image2)

The same complications of using a blunt trailing edge described in Section 3.2.2 exist for the 3-D wing as for a 2-D wing, so an O-grid was used instead of a C-grid. The trailing edge of the NACA 0021 profile is thicker than the NACA 4412 profile by a factor of 1.75, so 45 points are fit onto the trailing edge face (as opposed to 25 points used for the NACA 4412 case), and the spacings near the trailing edge on the upper and lower surfaces were adjusted appropriately to match the spacing on the trailing edge face (Figure 3.23). As in the previous case, since the wing is at a high angle of attack, it will have a large leading edge suction peak and the grid spacing
must be fine on the upper surface at the leading edge to capture the strong favorable pressure gradient (Figure 3.24).

In contrast to the wind tunnel case, the grid is generated with the airfoil oriented horizontally within the domain at a 0 degree angle of attack relative to the background Cartesian grid. The boundary conditions are then set so that the flow enters the domain at an angle of incidence of 10 degrees. Therefore, the lower and upstream faces of the domain are set with a freestream uniform flow boundary condition. The upper and downstream faces of the domain are set with an exit flow boundary condition, meaning no information travels backwards across these boundaries. The wing and wingcap grids both have viscous wall boundary conditions, and the wing grid is periodic in the spanwise direction.

It was found that in most cases the steady state solver was unstable using the original initial condition (all values initialized to the freestream conditions). Thus, a second-order upwind scheme was run steady-state initially to increase the quality of
the initial condition. The upwind scheme is used with multigrid and grid sequencing to speed up convergence. Three levels of grid sequencing are used for all cases: 1000 steps on the coarsest grid, 2000 steps on the next-coarsest grid, and 3000 steps on the full grid with multigrid acceleration. The resulting solution then became the initial condition for the steady state solver. The same 2nd-order central differencing scheme with added matrix dissipation is used for this case, run with a steady state solver to cut down on computational time, as discussed earlier. The central difference scheme uses second- and fourth- order matrix dissipation with the coefficients set as before (the second- and fourth- order coefficients are 2.0 and 0.4 respectively), and the matrix dissipation limit on linear and nonlinear eigenvalues is 0.3. The dissipation scheme uses Roe-averaging for half-grid point flow quantities. Local timestepping is used with the central difference scheme with a constant CFL number varying between 1.0 and 5.0.

3.3.3 Grid Convergence Study

An initial grid was generated with the parameters described above. The majority of the grid was created in a similar manner to the NACA 4412 geometry, except the background region which is a large domain instead of a wind tunnel with viscous walls. The wing consists of 307 spanwise points including 45 on the trailing edge surface, and 97 grid points in the off-body direction with an initial off-body spacing of 0.000625% chord, which results in a maximum $y^+$ value of 0.80 at 10 degrees angle of attack. The off-body grid stretches out to a distance of one chord length. The spanwise direction consists of 201 points, with spacing at the wingtips equal to 0.1% of the semi-span. For wings at high angles of attack, the spanwise spacing at the wingtip is very important because a strong wingtip vortex is generated when the pressure difference between the upper and lower surfaces is large.

As mentioned before, the wingcap grid had to be quite fine to refine the sharp trailing edge corner. The grid settled on was 227 x 113, with 227 points in the
streamwise direction. The off-body grids for the wingcaps were designed with the same parameters as the wing grid. The domain extends out to 100 chord lengths away from the wing in all directions; the near-body spacing is 1.88% of the chord, and the stretching ratio to the freestream boundaries is 1.2.

Two other grids were designed, this time with $\sqrt[3]{2}$ as the spacing factor. Since this grid is three dimensions, increasing the spacing in each dimension by a factor of $\sqrt[3]{2}$ and decreasing the number of grid points in each dimension by a factor of $\sqrt[3]{2}$ will result in a grid with half the grid points. The opposite operation results in a grid with twice the number of grid points. Table 3.6 summarizes the resulting grid parameters.

Table 3.6: A summary of the parameters for the NACA 0021 grid convergence study.

<table>
<thead>
<tr>
<th>Grid</th>
<th>Total Grid Points</th>
<th>$y^+$ value off-body</th>
<th>Near-Body Spacing (% of chord)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>8,869,752</td>
<td>1.0</td>
<td>2.37</td>
</tr>
<tr>
<td>Medium</td>
<td>17,457,568</td>
<td>0.80</td>
<td>1.88</td>
</tr>
<tr>
<td>Fine</td>
<td>34,817,581</td>
<td>0.63</td>
<td>1.49</td>
</tr>
</tbody>
</table>

The SST model was used for this grid convergence study, and grid resolution-dependent quantities such as lift, drag, and minimum pressure coefficient values were compiled for each case. This convergence study was conducted a bit differently than the 2-D case, namely because the spacing was increased and decreased by a factor of $\sqrt[3]{2}$ instead of $\sqrt{2}$. Therefore, the Richardson extrapolation equations differ slightly than Equations 3.3-3.5. The equations for the 3-D case are:

$$Y_{coarse} = Y_{exact} + c_1 \Delta x^2 + c_2 \Delta x^3 + ... \quad (3.8)$$

$$Y_{medium} = Y_{exact} + c_1 \left( \frac{\Delta x}{\sqrt[3]{2}} \right)^2 + c_2 \left( \frac{\Delta x}{\sqrt[3]{2}} \right)^3 + ... \quad (3.9)$$
\[ Y_{\text{fine}} = Y_{\text{exact}} + c_1 \left( \frac{\Delta x}{\sqrt{2}} \right)^2 + c_2 \left( \frac{\Delta x}{\sqrt{2}} \right)^3 + \ldots \] (3.10)

Once again solving for the “exact” values of the various flow variables by eliminating \(c_1\) and \(c_2\) and neglecting the higher order error terms, the following values were calculated:

Table 3.7: Richardson extrapolation results for lift, drag, and pressure coefficient for the 3-D NACA 0021 case.

<table>
<thead>
<tr>
<th>Grid</th>
<th>Lift Coefficient</th>
<th>Drag Coefficient</th>
<th>(C_{p_{\text{min}}} \text{ at Root} )</th>
<th>Run Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>0.607</td>
<td>0.0438</td>
<td>-2.13</td>
<td>12 hours</td>
</tr>
<tr>
<td>Medium</td>
<td>0.614</td>
<td>0.0442</td>
<td>-2.15</td>
<td>20 hours</td>
</tr>
<tr>
<td>Fine</td>
<td>0.617</td>
<td>0.0443</td>
<td>-2.17</td>
<td>40 hours</td>
</tr>
<tr>
<td>R.E.</td>
<td>0.618</td>
<td>0.0443</td>
<td>-2.17</td>
<td></td>
</tr>
</tbody>
</table>

The results of this study are quite close; the coarse grid is generally within 2% or 3% of the extrapolated value, the medium grid is within 1%, and the fine grid outputs results essentially equal to the extrapolated value. The fine grid is much too large and takes far too long to run for a realistic study, so this grid was abandoned. The course grid on the other hand, while in close agreement with the extrapolated value in most cases, exhibited oscillations as it converged towards a steady state value, suggesting it may not be a completely steady-state solution. Therefore the medium grid was chosen for this study.

Further supporting this choice is the plot of the pressure coefficient taken at mid-span for each of the coarse, medium, and fine grids in Figures 3.25 and 3.26. The fine grid predicts lower pressures on the upper surface and higher pressures on the lower surface. The zoomed-in image in Figure 3.26 verifies that the medium grid is closer to the fine grid than it is to the coarse grid, suggesting that further refinement to the grid will result in only minor changes to the pressure distribution.
Fig. 3.25.: Pressure coefficient plots of the NACA 4412 case showing the effects of grid density.

Fig. 3.26.: Minimum pressure region of the pressure coefficient plots.
3.3.4 Unsteadiness in Steady-State Computations

After running the Lag, lagRST, and lagRSTSST models at higher angles of attack, it became clear that using a steady state central difference solver with the lagged eddy viscosity and lagged Reynolds stress models for high angles of attack (the 16° case) will not result in a steady state solution. The problem stems from the fact that the flow separates early on the wing (in many cases, near mid-chord) and vortices begin to shed off the trailing edge, a process which cannot be adequately captured with a steady state solver. It was found that the central difference solver would not converge to a steady state, so running with a more dissipative upwind method was also attempted. The integrated lift forces, calculated with the lagRSTSST model with $\sigma_k = 0.8$, as a function of timestep are plotted in Figure 3.27.

![Lift Coefficient vs Timestep](image)

**Fig. 3.27.** Lift coefficient as a function of timestep for the lagRSTSST model with $\sigma_k = 0.8$, run with an upwind and central difference spatial discretization. For the central difference method, timesteps 6,000-17,000 were run with a CFL number of 5.0 and timesteps 17,000-23,000 were run with a CFL number of 1.0.
The central difference calculation does not converge to a steady state lift value, although it does converge to an area bounded by force oscillations. As an upwind method includes more numerical dissipation than a central difference method, it might be expected that the upwind method would dampen out these oscillations and converge to a steady state value. However, this is not that case as it does not converge to any steady state value, and the lift values calculated are much lower than would be expected. Therefore, the central difference scheme was used as the discretization method.

The problem faced at the point is how to deal with the oscillatory nature of the solution. Before going into this issue, the solution was output at both a peak and valley in the lift plot (see Figure 3.28) to determine the nature of the quasi-unsteadiness.

![Lift Coefficient as a function of timestep](image)

Fig. 3.28.: Lift coefficient as a function of timestep, run with the central difference method for the lagRSTSSST model with $\sigma_k = 0.8$, denoting the peak and valley of the quasi-steady state solution.
The solutions output at these two locations are denoted as “Peak” for the higher lift value and “Valley” for the lower lift value. The pressure coefficient and skin friction distributions at the wing root are displayed in Figures 3.29 and 3.30.

Fig. 3.29.: Skin friction distribution on the upper surface at the wing root showing the differences between the Peak and Valley lift values.

Fig. 3.30.: Pressure coefficient distribution at the wing root showing the differences between the Peak and Valley lift values.

The skin friction plots are nearly identical up until the separation region, where oscillations begin to form close to the trailing edge. The differences in pressure are much more apparent, with a more negative minimum pressure coefficient for the “Valley” solution but more lift produced in the separation region for the “Peak”. The oscillatory values are much more prevalent here, likely an indicator that the solution is not fully converged. This comparison makes it clear that this is an unsteady problem which cannot be resolved in a steady-state run. However, as mentioned in Section 3.3.1, Purdue does not have the computational resources to run this case unsteady and so a steady-state approximation is the best that can be done. Therefore, the results presented from the 16° cases should not be taken as true, converged
CFD results, but rather approximations which can show the general trends of the turbulence models for this type of flow. Although this is a steady state run, flow quantities are averaged over the final four oscillations (as shown in Figure 3.28) in order to average out the differences apparent in the force oscillations.
4. TEST CASES

The two test cases run were the Wadcock and Coles NACA 4412 experiments and the Purdue NACA 0021-based wing experiments. The goal of this was to assess how the various Reynolds stress lag models perform in capturing the flowfield in the separated region as well as the predicting the location of the separated region. As a baseline comparison, calculations were also run using standard RANS models which function as base models for the lagged models. Thus, the following turbulence models were used for each test case:

Table 4.1: Turbulence models used and their associated reference number in the modified OVERFLOW 2.2e.lagRST Code.

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>Modified OVERFLOW 2.2e.lagRST Model Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>SA</td>
<td>102 - IRC set to 0</td>
</tr>
<tr>
<td>SARC</td>
<td>102 - IRC set to 1</td>
</tr>
<tr>
<td>k-ω</td>
<td>202</td>
</tr>
<tr>
<td>SST</td>
<td>203</td>
</tr>
<tr>
<td>Lag</td>
<td>304</td>
</tr>
<tr>
<td>lagRST</td>
<td>905</td>
</tr>
<tr>
<td>lagRSTSST</td>
<td>909</td>
</tr>
</tbody>
</table>

Computational time was not allotted to perform a grid convergence study for each of these turbulence models, but it is assumed that similar results would be seen when performing a grid resolution study with any of the models. Therefore, only the grids decided upon in Sections 3.2.3 and 3.3.3 will be used to compare to the experimental results. An important component of this study is to generate further
data which could contribute to the understanding of which set of coefficients best models subsonic nonequilibrium flowfields. Therefore, various values of $\sigma_k$ are tested in this study. A value of $\sigma_k = 0.5$ represents the model coefficient of the original $k - \omega$ or SST model, which Loganathan found computed the stall angle of attack of thick wind turbine airfoils to a higher degree of precision [36]. However, for a flat plate boundary layer both Lillard [8] and Loganathan [36] noted that a nonphysical inflection point forms at the edge of the boundary layer. The current study deals only with adverse pressure gradients and will not investigate this. A value of $\sigma_k = 0.8$ represents the lowest possible value of $\sigma_k$ which does not result in an inflection point at the edge of the flat plate boundary layer [8]. A value of $\sigma_k = 1.5$ is what Olsen recommended for the Lag model [34] based on rounding the edge of the boundary layer.

4.1 NACA 4412 Airfoil Comparisons

Wadcock and Coles [18] conducted a low-speed subsonic wind tunnel tests in the GALCIT wind tunnel at Caltech. The reference flow conditions are listed in Table 4.2. The measured values included surface pressure data and streamwise velocity in the separation and wake region. The airfoil was tested at a 13.87 degree angle of attack. Other comparisons to this experimental dataset are available in various other computational studies, such as Rhie and Chow’s computations using the $k - \epsilon$ model [10] and Abid and Rumsey’s study using explicit algebraic stress models [44]. The grid geometry used in the current study models the trailing edge more accurately than these studies, leaving a nonzero thickness instead of closing it to zero thickness. Rhie and Chow do model the wind tunnel wall effects while Abid and Rumsey do not.
Table 4.2: Reference conditions for the testing of a NACA 4412 airfoil in the GALCIT wind tunnel at Caltech.

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_0$</td>
<td>72.9 cm Hg</td>
</tr>
<tr>
<td>$T_{ref}$</td>
<td>24.3 °C</td>
</tr>
<tr>
<td>$Re_c$</td>
<td>1,520,000</td>
</tr>
<tr>
<td>$U_{ref}$</td>
<td>23.17 m/s</td>
</tr>
<tr>
<td>$Q_{ref}$</td>
<td>3.13 mm Hg</td>
</tr>
<tr>
<td>$\rho$</td>
<td>0.00114 g/cm$^3$</td>
</tr>
<tr>
<td>$\nu$</td>
<td>0.161 cm$^2$/s</td>
</tr>
</tbody>
</table>
4.1.1 Separation Point and Velocity Profiles

Wadcock and Coles did not measure skin friction so it is difficult to get an exact measurement of the separation point. The flying hot-wire probe used to measure velocity also did not get close enough to the airfoil surface to be able to record where the reversed flow region begins, so this study does not attempt to estimate where the experimental location of the separation point is. However, it does look at the trends between the different turbulence models. The separation point given by each model is tabulated in Table 4.3.

Table 4.3: Separation points resulting from each model for a NACA 4412 airfoil at 13.87°.

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>Separation Point (x/c)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SA</td>
<td>0.752</td>
</tr>
<tr>
<td>$k - \omega$</td>
<td>0.809</td>
</tr>
<tr>
<td>SST</td>
<td>0.747</td>
</tr>
<tr>
<td>SARC</td>
<td>0.709</td>
</tr>
<tr>
<td>Lag, $\sigma_k = 0.5$</td>
<td>0.703</td>
</tr>
<tr>
<td>Lag, $\sigma_k = 0.8$</td>
<td>0.739</td>
</tr>
<tr>
<td>Lag, $\sigma_k = 1.5$</td>
<td>0.756</td>
</tr>
<tr>
<td>lagRST, $\sigma_k = 0.5$</td>
<td>0.749</td>
</tr>
<tr>
<td>lagRST, $\sigma_k = 0.8$</td>
<td>0.749</td>
</tr>
<tr>
<td>lagRST, $\sigma_k = 1.5$</td>
<td>0.791</td>
</tr>
<tr>
<td>lagRSTSST, $\sigma_k = 0.5$</td>
<td>0.797</td>
</tr>
<tr>
<td>lagRSTSST, $\sigma_k = 0.8$</td>
<td>0.777</td>
</tr>
<tr>
<td>lagRSTSST, $\sigma_k = 1.5$</td>
<td>0.782</td>
</tr>
</tbody>
</table>

The standard eddy viscosity models predict a wide range of separation locations. The SST model predicts separation only slightly earlier than the SA model. The $k - \omega$
model predicts separation much later, while the SARC model predicts separation much earlier than all of the standard models.

The Lag model predicts separation earlier than many of the other models. The separation point appears to move towards the trailing edge as the value of $\sigma_k$ increases. The lagRST model follows a similar trend with regard to $\sigma_k$, although the separation points predicted are further aft than those predicted by the Lag model. The lagRSTSST model predicts the latest separation points and does not seem to follow a trend with respect to the value of $\sigma_k$.

Experimental streamwise velocity data is available, so velocity profiles present a more direct comparison between the experimental and computational data. The first profiles discussed are taken off of the upper surface before separation, at $x/c = 0.7$. In Figures 4.1-4.3, profiles calculated by the Lag, lagRST, and lagRSTSST models, respectively, are plotted for each value of $\sigma_k$ and are compared to each of the standard one- and two- equation models ($k - \omega$, SST, SA) as well as the SARC model.

![Fig. 4.1.: Velocity profiles comparisons of the Lag model taken off the upper surface at $x/c = 0.7$.](image)
The Lag model appears to underpredict the velocity at each $z/c$ location. This is reflected in that these models promote separation earlier than the many of the standard models. It can be noted that increasing the value of $\sigma_k$ directly leads to a higher prediction of velocity in the middle of the boundary layer. The SARC model nearly perfectly predicts the velocity profile while the SA and $k-\omega$ models seem to overpredict the velocity. From a design standpoint, an underprediction of velocity is generally more desirable than an overprediction of velocity, as it will lead to an underprediction of the separation point and an overdesign of the flight component. Figures 4.2 and 4.3 show the lagRST and lagRSTSST results.

Fig. 4.2.: Velocity profile comparisons of the lagRST model taken off the upper surface at $x/c = 0.7$.

Fig. 4.3.: Velocity profile comparisons of the lagRSTSST model taken off the upper surface at $x/c = 0.7$.

The lagRST model performs much better overall than the Lag model; however some of the same trends seen in the Lag model results also appear here. These trends include first that as $\sigma_k$ increases, the velocities in the lower part of the boundary layer increase. Secondly, lower values of $\sigma_k$ result in an inflection point close to the middle of the boundary layer which generally deviates from the measured velocity.
profile. Lastly, the highest value of $\sigma_k$ rounds the boundary layer (as it is designed to do) to too large an extent. The models have a hard time capturing the edge of the boundary layer because the boundary layer is too rounded. The lagRST model with $\sigma_k = 0.8$ seems to perform the best overall, although it deviates towards the middle of the boundary layer due to its inflection point.

The lagRSTSST models do not follow the Lag and lagRST trends. Firstly, the lowest value of $\sigma_k$ (0.5) predicts the highest velocities. Secondly, the small inflection in the velocity profiles is noticeable for all values of $\sigma_k$, not just for the lower values. Lastly, the rounding of the edge of the velocity profile is not significantly different between the three values of $\sigma_k$. Because of this, the lagRSTSST model with $\sigma_k = 1.5$ appears to have the best fit.

The second set of velocity profiles discussed is inside the separation zone, at $x/c = 0.9$. Figure 4.4 shows the Lag model velocity profiles in comparison to the standard models.

![Velocity profiles comparisons of the Lag model taken off the upper surface at $x/c = 0.9$.](image)

Fig. 4.4.: Velocity profiles comparisons of the Lag model taken off the upper surface at $x/c = 0.9$. 
Several of the models, including the SA, SARC, and $k-\omega$ models, do an extremely poor job of predicting the reversed flow region, and the Lag models all perform better at the closer to the surface. Moving towards the middle of the profile, however, the SARC model again appears to capture the velocities much better than the other models. The Lag models capture the upper portion and the edge of the boundary layer more effectively. Once again the trend of higher $\sigma_k$ values resulting in higher velocities is apparent. Figures 4.5 and 4.6 display the lagRST and lagRSTSST results, respectively.

![Fig. 4.5.](image1.png) Fig. 4.5.: Velocity profile comparisons of the lagRST model taken off the upper surface at x/c = 0.9.

![Fig. 4.6.](image2.png) Fig. 4.6.: Velocity profile comparisons of the lagRSTSST model taken off the upper surface at x/c = 0.9.

The reversed flow region again is better captured by the lagRST models than by the standard models. The lagRST model with $\sigma_k = 1.5$ captures the lower part and middle part of the velocity profile most effectively, although the comparisons become worse towards the edge of profile. The trends seen in the x/c = 0.7 comparisons are not as pronounced here. The deviation towards the middle of the profile is not as
apparent for lower values of $\sigma_k$, and the rounding of the edge of the velocity profile for higher values of $\sigma_k$ is not the problem it was for the attached boundary layer profiles.

The lagRSTSST models again do not show the correlation between increased $\sigma_k$ and increased velocities seen in the Lag and lagRST models. The $\sigma_k = 0.5$ model results in an overprediction of velocity, but the $\sigma_k = 0.8$ and 1.5 models perform very well up until the edge of the boundary layer. The edge of the boundary layer is not captured by any of the models, although SST comes closest.

The final set of velocity profiles discussed are captured in the wake, 0.1 chord lengths downstream of the trailing edge. Figures 4.7-4.9 display the Lag, lagRST, and lagRSTSST model results, respectively, in comparison to the standard models.

Fig. 4.7.: Velocity profiles comparisons of the Lag model taken in the wake at $x/c = 1.1$.

The Lag and lagRST formulations all do a poor job of capturing the velocity deficit in the wake in comparison to the standard models (SA specifically). The lagRSTSST models (Figure 4.9) appear to do better when compared to the Lag and lagRST models (Figures 4.7 and 4.8 respectively), although none of them do an acceptable
Fig. 4.8.: Velocity profile comparisons of the lagRST model taken in the wake at $x/c = 1.1$.

Fig. 4.9.: Velocity profile comparisons of the lagRSTSST model taken in the wake at $x/c = 1.1$. 
job predicting the velocity deficit. Many of the standard models do a better job in this respect; the $k - \omega$ model in particular.

A few conclusions can be taken away from this section. In the region before separation, the slope of the boundary layer for the Lag and lagRST models is very much dependent on the value of $\sigma_k$. When $\sigma_k$ becomes too high, the boundary layer is rounded off too steeply and the boundary layer edge is not captured correctly. Alternatively, when $\sigma_k$ is too low, the velocity profile develops a small inflection point deviating from the experimental values in the middle of the boundary layer. The lagRST model with a value of $\sigma_k = 0.8$ appears to be the value which minimizes both of these undesirable effects. The lagRSTSST model does not suffer these same concerns, and a value of $\sigma_k = 0.8$ appears to match the slope of the experimental values the best. In the separation region off the upper surface, the mid-profile inflection point and the rounding of the edge are no longer concerns, and it is more difficult to distinguish between the models. All of the values of $\sigma_k$ give reasonable velocity profile results except for the lagRSTSST model with $\sigma_k = 0.5$, which overpredicts the velocity by a wider margin. Additionally, none of the models capture the edge of the boundary layer. Lastly, in the wake, the lagRSTSST models do better than the Lag or lagRST models, but the predictions are still not adequate in comparison to many of the standard models.

For the lagRST model, as $\sigma_k$ is increased, separation is delayed and the separation point moves towards the trailing edge. This leads to a larger separation zone with stronger vortices shed off the trailing edge. Figure 4.10 displays this qualitatively. For simplicity’s sake, the “reversed flow region” in the section refers to the region in the separation zone where the streamwise component of the velocity is negative. As $\sigma_k$ increases, the reversed flow region becomes larger and the negative velocities become stronger. Figure 4.10(d) shows that the reversed flow region should extend to around $x/c = 1.07$ while the lagRST models all predict it to extend further than $x/c = 1.1$. However, it is clear that as $\sigma_k$ increases, the region becomes smaller because of the delayed separation point.
Fig. 4.10.: Velocity contours of the separation region for the lagRST models with a) \( \sigma_k = 0.5 \), b) \( \sigma_k = 0.8 \), c) \( \sigma_k = 1.5 \), in comparison to d) Wadcock and Coles’ data (1979).
In contrast to the lagRST models, the lagRSTSST models predict a reversed flow region slightly smaller than the experimental results. While the experimental results show a reversed flow region extending to around \( x/c = 1.07 \), the lagRSTSST results show a reversed flow region extending to around \( x/c = 1.05 (\sigma_k = 0.5) \) and \( x/c = 1.07 (\sigma_k = 0.8 \text{ and } 1.5) \). Qualitatively, the \( \sigma_k = 0.8 \) case appears to match the experimental data the best although there are not many differences between the \( \sigma_k = 0.8 \) and \( \sigma_k = 1.5 \) cases. Figure 4.11 depicts the lagRSTSST velocity contour plots in comparison to the experimental data.

4.1.2 Pressure Coefficient Comparisons

Wadcock and Coles also made available pressure coefficient data at the same angle of attack. Figure 4.12 presents this data in comparison to the results from each of the turbulence models tested.

As one might expect, the models all perform well on the pressure side of the wing where nonequilibrium effects are less prevalent. On the upper surface, however, all of the models predict the surface pressures to be too negative. The minimum pressure peak predicted for most of the turbulence models is nearly 2 nondimensional units lower than the minimum pressure tabulated in the experiment. It is not clear whether this is because the experimental dataset is too sparse to capture the exact point of the minimum pressure or if the models are severely overpredicting the magnitude of the peak. Moving along the upper surface towards the trailing edge, the experimental and predicted pressures move closer together until the separation zone where the agreement looks much better.

The experimental data and the simulation data do not agree at the minimum pressure peak. However, it is interesting to look at the trends in the computational data at the leading edge. Figure 4.13 shows the Lag model results in comparison to the standard models.
Fig. 4.11.: Velocity contours of the separation region for the lagRSTSST models with a) $\sigma_k = 0.5$, b) $\sigma_k = 0.8$, c) $\sigma_k = 1.5$, in comparison to d) Wadcock and Coles’ data (1979).
Fig. 4.12.: Pressure coefficient comparison between each of the turbulence models and the Wadcock and Coles data.

Fig. 4.13.: Pressure coefficient comparison between the Lag models at the minimum pressure peak.
The standard models (including the SARC model) all predict higher pressure peaks than the Lag models. Interestingly, the resulting stronger adverse pressure gradient does not result in an earlier separation point. The Lag models separate earlier than the most of the standard models, as shown in Table 4.3. The lagRST and lagRSTSST model results are displayed in Figure 4.14.

![Figure 4.14](image.png)

Fig. 4.14.: Pressure coefficient comparison between the lagRST and lagRSTSST models at the minimum pressure peak.

The pressure peaks for the lagRST and lagRSTSST models are generally less negative than the standard models. For both models the pressure peak becomes more negative as the value of $\sigma_k$ increases. Additionally, for large values of $\sigma_k$, the differences between the $k - \omega$- and SST-based versions of the lagRST model are minimal. For the $\sigma_k = 0.8$ and 1.5 cases, the differences are almost indistinguishable between the two models.

Moving down towards the separation region, Figure 4.15 shows the turbulence model surface pressure predictions in comparison the Wadcock and Coles’ measurements.
Fig. 4.15.: Pressure coefficient comparisons in the separation region.
The experimental measurements show that the surface pressure changes very little once the flow has separated. In contrast, although the slope of the pressure coefficient does change, the turbulence models predict a steeper pressure increase in the separation region. Most of the models collapse on each other inside the separation region; however the SARC, SST, and lagRST models with $\sigma_k = 0.8$ agree with the experiment marginally better. The lagRSTSST model with $\sigma_k = 0.5$ and the $k - \omega$ model perform marginally worse.

4.2 NACA 0021 Wing Comparisons

A wind tunnel test of a NACA 0021-based wing was conducted in the Boeing Wind Tunnel at the Aerospace Sciences Lab at Purdue. The CFD runs had to be completed before the experimental measurements were made, so there was some uncertainty in the input parameters. To estimate these parameters, the atmospheric values of pressure and temperature were used as the stagnation conditions $p_0$ and $T_0$, respectively. The reference flow conditions used in the CFD are listed in Table 4.4, along with the actual experimental conditions which were measured after the CFD runs. The study consisted of surface pressure measurements at various angles of attack, although only the 10° and 16° cases are analyzed here.

Table 4.4: Estimated run conditions for the testing of a NACA 0021 wing in the Boeing wind tunnel at Purdue.

<table>
<thead>
<tr>
<th>Condition</th>
<th>CFD Value</th>
<th>Experimental Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$p_0$</td>
<td>14.76 psi</td>
<td>14.96 psi</td>
</tr>
<tr>
<td>$T_0$</td>
<td>75.0°F</td>
<td>81.14°F</td>
</tr>
<tr>
<td>$Re_c$</td>
<td>228,000</td>
<td>220,000</td>
</tr>
<tr>
<td>$U_\infty$</td>
<td>17.3 m/s</td>
<td>17.0 m/s</td>
</tr>
</tbody>
</table>
The Boeing wind tunnel is a closed system so the static freestream state variables can be estimated by the stagnation conditions and the wind tunnel speed using isentropic relations. Although the wind tunnel is not a perfectly isentropic system, the change in entropy effects are minimal and isentropic relations allow for an estimation of how the static freestream temperatures differ from the stagnation quantities. The isentropic relations for temperature and pressure [45] are given as

\[ \frac{T_0}{T} = 1 + \frac{\gamma - 1}{2} M^2 \]  

\[ \frac{p_0}{p} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{\gamma}{\gamma - 1}} \]  

where \( \gamma \) denotes the ratio of specific heats, which is 1.4 in air. The density can then be calculated by the equation of state,

\[ \rho = \frac{p}{RT}, \]  

where \( R \) is the specific gas constant for air. The dynamic viscosity is then calculated by Sutherland’s law,

\[ \mu = \mu_{ref} \left(\frac{T}{T_{ref}}\right)^{3/2} \frac{T_{ref} + S}{T + S}. \]  

In this equation, the reference states \( T_{ref} \) and \( \mu_{ref} \), as well as \( S \), are all experimentally-determined constants. The kinematic viscosity can be determined from the dynamic viscosity. As the tests were designed to run at a Mach number of 0.05, the freestream conditions were calculated and are tabulated in Table 4.5.

There is some question as to the validity of the comparisons between CFD and wind tunnel for this setup. Consider the setup shown in Figure 4.16. Although only shown at one spanwise location here, there are four different spanwise locations on the wing where pressure tubes hang down below the wing. This obstruction on the lower surface very likely affects the circulation around the wing and creates form drag which affects the pressures on the lower surface. The wind tunnel results recorded
Table 4.5: Estimated freestream conditions for the testing of a NACA 0021 wing in the Boeing wind tunnel at Purdue.

<table>
<thead>
<tr>
<th>Condition</th>
<th>CFD Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$p_\infty$</td>
<td>14.74 psi</td>
</tr>
<tr>
<td>$T_\infty$</td>
<td>74.7°F</td>
</tr>
<tr>
<td>$\rho$</td>
<td>0.00231 slug/ft$^3$</td>
</tr>
<tr>
<td>$\nu$</td>
<td>0.153 cm$^2$/s</td>
</tr>
</tbody>
</table>
positive lift at a zero degree angle of attack even though the wing is theoretically perfectly symmetric, and the zero lift angle of attack was found to be -3.9°.

Fig. 4.16.: Wind tunnel setup depicting the pressure ports hanging below the wing.

Another concern about the comparisons between CFD and wind tunnel data is the setup of the CFD runs. The wind tunnel experiments are performed inside a wind tunnel where there are blockage effects from the side walls as well as from the top and bottom walls of the tunnel. This almost certainly has an effect on the flowfield at higher angles of attack, where the wingtip vortices are obstructed by the side walls and the lift-inducing circulation is obstructed by the upper and lower walls. Because of the requirements of the project these cases were run for, the effects of the wind tunnel walls were not simulated.

Still other concerns about comparing the datasets center around the transition locations. The wind tunnel runs do not attempt to trip turbulent flow at the leading edge, and so the flow transitions at some unspecified location. The CFD does not
account for this and assumes a fully turbulent boundary layer across the entire wing. Therefore, the comparisons to experiment are not to be taken as entirely valid, and this section will focus more on the trends and differences seen between the different turbulence models. The few comparisons between CFD and experimental data did not come out well, and while some of this is may be due to the shortcomings of the CFD and turbulence modeling, the majority can be attributed to the discrepancies between the physical setup of the wind tunnel model and the CFD runs.

4.2.1 Surface Streamlines and Separation Patterns

The results from the 2-D NACA 4412 cases showed that the separation point moves towards the trailing edge as $\sigma_k$ increases for the Lag and lagRST models. Table 4.6 tabulates the separation point at midchord for the 3-D NACA 0021-based wing cases for 10 and 16 degrees. It is clear that these trends are also seen for the finite NACA 0021-based wing case.

As seen previously in the NACA 4412 results, the standard eddy viscosity models predict a range of separation points for the 10 and 16 degree cases, with the $k-\omega$ model predicting separation the furthest downstream for the 10 degree case and the SA model predicting separation furthest downstream for the 16 degree case. In both cases, the SARC model predicts separation earlier than the SA model. The Lag models predict the earliest separation points, and separation delays as the value of $\sigma_k$ increases. The lagRST models predict slightly later separation. The lagRSTSST models again do not respond strongly to changes in $\sigma_k$, and the lift values produced are comparable to the lagRST model with higher values of $\sigma_k$. These separation locations directly correlate to the amount of lift produced, as shown in Table 4.7.

When the flow separates closer to the trailing edge, the lift increases. This is especially apparent in the Lag and lagRST models. As $\sigma_k$ increases, separation delays, and the lift produced is higher. For the lagRSTSST models, the value of $\sigma_k$ has little effect on the separation point.
Table 4.6: Separation points on a 3-D NACA 0021 wing at wing root for angles of attack of 10 and 16 degrees.

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>Separation Point ($x/c$), $\alpha=10^\circ$</th>
<th>Separation Point ($x/c$), $\alpha=16^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>SA</td>
<td>0.861</td>
<td>0.593</td>
</tr>
<tr>
<td>$k - \omega$</td>
<td>0.867</td>
<td>0.562</td>
</tr>
<tr>
<td>SST</td>
<td>0.799</td>
<td>0.372</td>
</tr>
<tr>
<td>SARC</td>
<td>0.849</td>
<td>0.529</td>
</tr>
<tr>
<td>Lag, $\sigma_k = 0.5$</td>
<td>0.735</td>
<td>0.256</td>
</tr>
<tr>
<td>Lag, $\sigma_k = 0.8$</td>
<td>0.773</td>
<td>0.249</td>
</tr>
<tr>
<td>Lag, $\sigma_k = 1.5$</td>
<td>0.806</td>
<td>0.213</td>
</tr>
<tr>
<td>lagRST, $\sigma_k = 0.5$</td>
<td>0.812</td>
<td>0.237</td>
</tr>
<tr>
<td>lagRST, $\sigma_k = 0.8$</td>
<td>0.844</td>
<td>0.496</td>
</tr>
<tr>
<td>lagRST, $\sigma_k = 1.5$</td>
<td>0.871</td>
<td>0.566</td>
</tr>
<tr>
<td>lagRSTSST, $\sigma_k = 0.5$</td>
<td>0.832</td>
<td>0.514</td>
</tr>
<tr>
<td>lagRSTSST, $\sigma_k = 0.8$</td>
<td>0.832</td>
<td>0.520</td>
</tr>
<tr>
<td>lagRSTSST, $\sigma_k = 1.5$</td>
<td>0.831</td>
<td>0.530</td>
</tr>
</tbody>
</table>
Table 4.7: Lift coefficient calculated on the 3-D NACA 0021 wing for angles of attack of 10 and 16 degrees.

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>Lift Coefficient, $\alpha=10^\circ$</th>
<th>Lift Coefficient, $\alpha=16^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>SA</td>
<td>0.675</td>
<td>1.01</td>
</tr>
<tr>
<td>$k - \omega$</td>
<td>0.653</td>
<td>0.969</td>
</tr>
<tr>
<td>SST</td>
<td>0.616</td>
<td>0.857</td>
</tr>
<tr>
<td>SARC</td>
<td>0.677</td>
<td>0.988</td>
</tr>
<tr>
<td>Lag, $\sigma_k = 0.5$</td>
<td>0.581</td>
<td>0.697</td>
</tr>
<tr>
<td>Lag, $\sigma_k = 0.8$</td>
<td>0.597</td>
<td>0.708</td>
</tr>
<tr>
<td>Lag, $\sigma_k = 1.5$</td>
<td>0.608</td>
<td>0.740</td>
</tr>
<tr>
<td>lagRST, $\sigma_k = 0.5$</td>
<td>0.625</td>
<td>0.827</td>
</tr>
<tr>
<td>lagRST, $\sigma_k = 0.8$</td>
<td>0.637</td>
<td>0.894</td>
</tr>
<tr>
<td>lagRST, $\sigma_k = 1.5$</td>
<td>0.644</td>
<td>0.945</td>
</tr>
<tr>
<td>lagRSTSST, $\sigma_k = 0.5$</td>
<td>0.630</td>
<td>0.932</td>
</tr>
<tr>
<td>lagRSTSST, $\sigma_k = 0.8$</td>
<td>0.630</td>
<td>0.932</td>
</tr>
<tr>
<td>lagRSTSST, $\sigma_k = 1.5$</td>
<td>0.629</td>
<td>0.933</td>
</tr>
</tbody>
</table>
Assessing the skin friction at the wing root for the 10° cases leads to some interesting results. Figure 4.17 shows the skin friction plots on the upper surface. Although all of the models were run with fully turbulent boundary layers, each turbulence model appears to show transition to turbulent flow at some point between 0.05c and 0.20c. This behavior was noted for the flat plate boundary layer cases run by Lillard [8] and it is not clear why this happens. One possible explanation is that the eddy viscosity takes some amount of time to build up as the boundary layer forms on the wing and there is not enough turbulence intensity to form a turbulent boundary layer until slightly further downstream. The Lag models “transition” earlier, while the lagRST models transition later and the lagRSTSST models transition between the two. The SA and SARC models transition much later and the k-ω and SST models transition earlier. An immediate concern is whether the effect of the “transition” location affects the separation point. To quantify this effect, the lowest skin friction value before the “transition” region was taken as the transition point. This was then plotted against the corresponding separation point in Figure 4.18. If the “transition” point does not affect where the flow separates, one would expect no trend with respect to the data plotted. However, using linear regression a positive trend between later “transition” and later separation is apparent. The linear fit has a slope of $\frac{x_{sep}}{x_{tr}} = 0.483$, which is significant. However the sample size is not large enough to draw a definite conclusion that the separation point is impacted by the location of the false “transition” location.

To briefly test this theory, the SARC model and the lagRSTSST model with $\sigma_k = 0.8$ were both run using the artificial “transition” point calculated with the SARC model. Figure 4.19 shows the differences in skin friction between the fully turbulent models and the models which force transition later. From these results it is shown that specifying the transition point does affect the skin friction plot. For the lagRSTSST model, the result is that the separation point moves downstream from $x/c = 0.832$ to $x/c = 0.854$. The reason for this is that a turbulent boundary layer begins to lose momentum as soon as it forms. If the turbulent boundary layer forms later, it will
Fig. 4.17.: Skin friction plots at the wing root for the 10° cases showing the differences between turbulence models.

Fig. 4.18.: Plot showing the relationship between the artificial transition point and the subsequent separation location.
begin to lose momentum later and subsequently separate later. In contrast, the SARC separation point moves forwards from \( x/c = 0.849 \) to \( x/c = 0.835 \) as the transition region is smaller and is associated with a much steeper increase in skin friction.

![Plot showing the effect of moving back the transition point on the skin friction plot at the wing root for the lagRSTSST model with \( \sigma_k = 0.8 \).](image)

**Fig. 4.19.** Plot showing the effect of moving back the transition point on the skin friction plot at the wing root for the lagRSTSST model with \( \sigma_k = 0.8 \).

One inherent difference between each of these cases is that the lengths and shapes of the transition regions are very different. It is true that specifying a region of laminar flow and turbulent flow allows for a slightly better (although imperfect) match of the transition point. However, the length of the transition regions do not match. Lillard [8] discovered similar issues when modeling flow on a flat plate boundary layer and suggested that to avoid OVERFLOW’s inability to control the transition zone length, all calculations be run fully turbulent. It appears there are many different variables at
Fig. 4.20.: Surface streamline plots at $\alpha = 10^\circ$ of the Lag model with a) $\sigma_k = 0.5$, b) $\sigma_k = 0.8$, and c) $\sigma_k = 1.5$, showing the effect of changing $\sigma_k$ on the separation locations.

play when calculating transition at the leading edge which all vary between turbulence models - the transition point, the transition zone length, and the skin friction value after transition. Which parameter is the best to match is beyond the scope of this study but is something which could be investigated in the future.

To convey a more thorough image of the separation predictions, surface streamline plots show the separation pattern on the wing. In Figures 4.20 and 4.21, it is clear that increasing the value of $\sigma_k$ for the Lag and lagRST models moves separation back towards the trailing edge. Likewise, the lack of sensitivity to the $\sigma_k$ parameter is shown in the surface streamlines of the lagRSTSST models in Figure 4.22 as the three plots appear nearly identical.
Fig. 4.21.: Surface streamline plots at $\alpha = 10^o$ of the lagRST model with a) $\sigma_k = 0.5$, b) $\sigma_k = 0.8$, and c) $\sigma_k = 1.5$, showing the effect of changing $\sigma_k$ on the separation locations.
Fig. 4.22.: Surface streamline plots at $\alpha = 10^\circ$ of the lagRSTSSST model with a) $\sigma_k = 0.5$, b) $\sigma_k = 0.8$, and c) $\sigma_k = 1.5$, showing the effect of changing $\sigma_k$ on the separation locations.
4.2.2 Pressure Coefficients and Comparisons to Experiment

Figure 4.23 shows the pressure coefficient curves at the wing root for each of the turbulence models for the 10 degree cases. Zooming in to the minimum pressure peak in Figures 4.24 and 4.25, it can be seen that the minimum pressure peak becomes more negative as the value of $\sigma_k$ increases for the Lag and lagRST models. This is consistent with the results of the 2-D NACA 4412 cases. Additionally, the lagRSTSST models collapse on each other, again reinforcing the idea that the lagRSTSST solution is not sensitive to the value of $\sigma_k$ for this flow.

![Figure 4.23: Surface pressure coefficients at the wing root for the $\alpha = 10^\circ$ case.](image)

To compare to the experimental data generated at Purdue, pressure coefficient values from the wind tunnel and CFD datasets were taken at a spanwise location of 8 inches from the wing root on the port side of the wing. These comparisons are shown in Figure 4.26.

The experimental data shown produces lift coefficients on the lower surface which are much lower than those predicted by the CFD models. While modeling error may
Fig. 4.24.: Minimum pressure peak at the wing root at 10 degrees, comparing the Lag models with the standard models and the lagRSTSST models.

Fig. 4.25.: Minimum pressure peak at the wing root at 10 degrees, comparing the lagRST models with the standard models and the lagRSTSST models.

Fig. 4.26.: Pressure coefficient comparison between CFD and experimental at the spanwise location 8 inches from the wing root for the $\alpha = 10^\circ$ case.
account for some of the discrepancy, it is more likely that the blockage and transition
effects described in Section 4.2 account for the majority of the differences. Accounting
for free transition may affect the separation point. As all of the CFD runs were done
with a fully turbulent boundary layer, the boundary layer grows fairly quickly and
the skin friction on the upper surface decreases rapidly, causing earlier separation. If
the flow is allowed to transition freely (as in the wind tunnel runs), the formation
of the turbulent boundary layer delays, which subsequently delays separation on the
upper surface and possibly leads to lower pressures closer to the trailing edge on the
upper surface. The flow blockage may affect the pressures at the leading edge. When
the flow is blocked underneath the wing, the pressure rises and forces more of the
flow to go over the upper surface which in turn leads to lower pressures on the upper
surface.

At this stage in the project, it has not been determined yet how to account for
flow blockage. However, the free transition effect can be accounted for. The XFLR5
code [46] is a 3-D extension of the 2-D XFOIL code [47], which is a panel code that
uses a boundary layer integral method to account for the growth in thickness of the
boundary layer. It also includes its own stability analysis which can be used to predict
where the flow will transition freely. XFLR5 extends this code to 3-D problems,
using Prandtl’s lifting line theory [45] to calculate the effective angle of attack at
each spanwise location due to the downwash from wingtip vortices. Thus, the code
is capable of outputting a transition location prediction as a function of spanwise
location on the wing. This data was used to trip turbulent flow and simulate free
transition on the wing and a 10° angle of attack. The resulting laminar and turbulent
sections are shown in Figure 4.27.

This free transition model was run with the lagRSTSST model with $\sigma_k = 0.8$ and
compared with the fully turbulent case run with the same turbulence model. Fig-
ure 4.28 shows the differences between the fully turbulent CFD model, free transition
CFD model, and experimental data at a span location of $y = -8$ in.
Fig. 4.27.: Diagrams showing the a) upper and b) lower surface free transition locations for a 10 degree angle of attack. The purple section represents the turbulent area, all other colors represent a laminar region.

Fig. 4.28.: Pressure coefficient plots of the fully turbulent and simulated free transition CFD data using the lagRSTSSST turbulence model with $\sigma_k = 0.8$, in comparison to the experimental data. Data was taken at the spanwise location 8 inches from the wing root for the $\alpha = 10^\circ$ case.
From this plot, it is clear that the free transition results are not converged and that OVERFLOW has trouble solving a flowfield like this where there are many different regions of laminar and turbulent flow. However, one can conclude from the unconverged data that simply accounting for free transition does not capture the minimum pressure peak seen in the experimental data. The differences between the CFD results and the experimental results are more likely due to flow blockage underneath the wing. To obtain better agreement between the data, one would have to take this flow blockage into account.
5. CONCLUSIONS AND RECOMMENDATIONS

The goal of the current research was to assess the ability of the various lagged turbulence models in predicting the separated flow regions over a 2-D airfoil and 3-D wing at high angles of attack, under subsonic flow conditions. By lagging the turbulent flow variables and relaxing them toward equilibrium values, the turbulent physics are theoretically more accurately modeled and it was expected to see an improvement in flowfield predictions. Studies were conducted on two different geometric models. One case attempted to simulate the experiments of Wadcock and Coles [18] on a NACA 4412 airfoil at high lift condition. The other case looked at a NACA 0021-based wing at a high angle of attack, run under similar conditions as ongoing experimental studies being run at the Boeing Wing Tunnel at Purdue University.

The majority of the quantitative analysis was done on the 2-D NACA 4412 cases. The Lag model performs particularly poorly in most of the comparisons. The lagRST and lagRSTSST do show some promise and are an improvement from the baseline models. For the lagRST models, a low value of $\sigma_k$ consistently results in a non-physical inflection point in the middle of the boundary layer profile under a strong adverse pressure gradient. A high value of $\sigma_k$ removes this inflection point but then rounds off the boundary layer too quickly. A value of $\sigma_k = 0.8$ may minimize both of these effects but still does not capture the boundary layer profile particularly well in many of the comparisons. In the separated region, the lagRST and lagRSTSST models perform well up until the edge of the velocity profile, where they do not round off fast enough. The lagRSTSST models with $\sigma_k = 0.8$ and 1.5 perform particularly well in the separated region close to the airfoil surface. In the wake region, none of the lagged models are an improvement over the SA, SARC, or $k-\omega$ models.

When looking at the overall velocity flowfield, it becomes clear that the size of the reversed flow region is not adequately captured by the lagRST models. In fact, the
lagRSTSST models qualitatively predict a more accurate representation especially with higher values of $\sigma_k$. Combined with the analysis from the velocity profiles in the separated region, this would seem to imply that the lagRSTSST models with higher kinetic energy equation diffusion coefficients are a better predictor of the separated region. Without an exact measurement of the separation point it is difficult to assess which model captures separation the best, but it would appear that the lagRSTSST models with $\sigma_k$ values of 0.8 and 1.5 are the best predictors.

While the 3-D NACA 0021 cases did not yield good agreement with experiment for various reasons, it is still useful to look at the separation trends. The Lag models consistently predict early separation points while the lagRST and lagRSTSST models predict separation later. It is worth noting that the lagRSTSST models are nearly unaffected by changes in $\sigma_k$ for all of the 3-D cases, in comparison to the Lag and lagRST models. In comparison to the standard turbulence models, the majority of the lagged models predict lower lift and earlier separation. Loganathan [36] also cited this trend in his studies with 2-D wind turbine airfoils, noting that the earlier separation predictions aligned better with experiment. Whether that is the case in this study is not clear with no experimental data to compare to. In future comparisons to this experimental dataset, the CFD must account for the flow blockage from the pressure tubing and pylons under the wing or else the minimum pressure peak at the leading edge will not be captured.

The lagRSTSST model’s lack of response to changes in the kinetic energy equation diffusion coefficient for this case is interesting. To the author’s knowledge, the lagRSTSST model has not been formally tested in any study other than in Lillard’s 2011 paper [8]. Lillard does not attempt to tune the parameters of the lagRSTSST model and instead uses a value of $\sigma_k = 0.8$ for all computations. It is clear from the 2-D NACA 4412 runs that the value of $\sigma_k$ does have an effect in other flowfields, but it may not have as great of an impact.

Tuning the lag constant is something which should be looked at for all of the models. In the current study, the lag constant used in all cases was $a_0 = 0.35$, which
is what Lillard proposed. However, Lillard [25], Churchfield [48], and Loganathan [36] have all looked at different values of the lag constant for various formulations of lagged models. Loganathan found that for the Lag model, a lag constant of 0.20 provided better separation predictions on 2-D wind turbine airfoils. Lillard looked at values of 0.20, 0.35, and 0.65 for the lagRST model and determined 0.35 to provide the best overall agreement for subsonic and supersonic separation predictions. Churchfield looked at values ranging from 0.35 to 1.00 for the lagRST model in modeling a wingtip vortex and found that higher values (0.53, 0.86, and 1.00) ultimately resulted in better agreement with experiment. He also proposed that for vortical flows, the lag parameter be tuned to the distance from the center of the vortex. Thus, there is a fair amount of disagreement to be settled about the lag constant.

As noted in Sections 4.2.1 and 4.2.2, running with a fully turbulent boundary layer may not be ideal. One issue is that the computational transition point is not aligned with the experimental transition point. The second issue is that there is an artificial laminar region at the leading edge of the wing as the eddy viscosity builds up, the length of which varies between turbulence models. This artificial transition point affects the separation prediction of the model.Specifying a region of laminar flow and a region of turbulent flow for all models may eliminate this artificial nonuniform transition, but the models may transition very differently and the best way to account for this is unclear.

Because this study did not have a sufficient amount of experimental data to compare to, it is difficult to draw definitive conclusions about which models perform the best. From the velocity profile data and qualitative comparisons of the velocity flowfield, the lagRSTSST models with $\sigma_k = 0.8$ and 1.5 appear to capture the separation process most effectively. This is not in agreement with Lillard’s conclusions, which state that the lagRSTSST models work well for complicated SWTBLI problems but perform poorly in predicting subsonic separation, and that the lagRST models should perform better under subsonic conditions [25]. Thus, more research needs to go into looking at the differences between separation predictions for the lagRST
and lagRSTSST models. It can be concluded, however, that for the Lag and lagRST models, a higher diffusion coefficient results in later separation and a higher lift value. This information could be used to tune the model in future studies where separation information is available.
LIST OF REFERENCES
LIST OF REFERENCES


100


VITA

Aaron Michael Rosen was born in Farmington Hills, Michigan in 1988. In May of 2007 he received his B.S. in Aerospace Engineering from the University of Michigan with a minor in Mathematics. In December of 2013 he finished his Masters in Aeronautical and Astronautical Engineering at Purdue University, with focuses in Aerodynamics and Computational Engineering. During this time, he completed internships with the Pennsylvania State University Applied Research Lab as well as with Sikorsky Aircraft. As part of his Masters research he was able to work with The Boeing Company on a CFD and wind tunnel data merging project. After completing his Master’s degree he will be working for Boeing in Huntington Beach, California.