2017

3D CFD Investigation of Low Pressure Turbine Aerodynamics

Jacob Andrew Sharpe

Wright State University

Follow this and additional works at: https://corescholar.libraries.wright.edu/etd_all

Part of the Mechanical Engineering Commons

Repository Citation
Sharpe, Jacob Andrew, "3D CFD Investigation of Low Pressure Turbine Aerodynamics" (2017). Browse all Theses and Dissertations. 1754.
https://corescholar.libraries.wright.edu/etd_all/1754

This Thesis is brought to you for free and open access by the Theses and Dissertations at CORE Scholar. It has been accepted for inclusion in Browse all Theses and Dissertations by an authorized administrator of CORE Scholar. For more information, please contact library-corescholar@wright.edu.
3D CFD INVESTIGATION OF LOW PRESSURE TURBINE AERODYNAMICS

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering

By

JACOB ANDREW SHARPE

B.S., Saginaw Valley State University, 2015

2017
Wright State University
May 2, 2017

I HEREBY RECOMMEND THAT THE THESIS PREPARED UNDER MY SUPERVISION BY Jacob Andrew Sharpe ENTITLED 3D CFD Investigation of Low Pressure Turbine Aerodynamics BE ACCEPTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF Master of Science in Mechanical Engineering

____________________________
Dr. Mitch Wolff
Thesis Director

____________________________
Dr. Joseph Slater
Chair, Department of Mechanical and Materials Engineering

Committee on Final Examination

____________________________
Dr. Mitch Wolff

____________________________
Dr. Rolf Sondergaard

____________________________
Dr. Rory Roberts

____________________________
Dr. Robert E.W. Fyffe
Vice President for Research and Dean of the Graduate School
ABSTRACT

Sharpe, Jacob Andrew. M.S.M.E. Department of Mechanical and Materials Engineering, Wright State University, 2017. 3D CFD Investigation of Low Pressure Turbine Aerodynamics.

A 3-D Reynolds-Averaged Navier Stokes (RANS) model of a highly-loaded blade profile has been developed using a commercial CFD code with an unstructured/structured grid and several different turbulence models. The ability of each model to predict total pressure loss performance is examined in terms of the spanwise loss distribution and the integrated total pressure loss coefficient. The flowfield predicted by each model is investigated through comparisons of isosurfaces of Q criterion to previous Implicit Large Eddy Simulation (ILES) results. The 3-equation k-kl-ω model was shown to provide the most accurate performance predictions for a baseline 3-D LPT geometry, and was then used to analyze the effect a new 3D contoured geometry. The model accurately predicted the qualitative improvement made by the contour by weakening the various vortex structures.
# TABLE OF CONTENTS

1 INTRODUCTION
   1.1 Low Pressure Turbines                                  1
   1.2 Motivation of Research                                  2
   1.3 Highly-Loaded Blades                                    3
   1.4 Flow Physics                                            6
   1.5 Transition Effects                                      8
   1.6 Current Work                                            11

2 COMPUTATIONAL METHODS
   2.1 Computational Fluid Dynamics                            12
   2.2 SC/Tetra                                               13
   2.3 Turbulence Models                                      14
   2.4 Flow Domain and Boundary Conditions                    16
   2.5 Mesh Details and Grid Independence Study               18

3 BOUNDARY LAYER THICKNESS SENSITIVITY
   3.1 k-ε                                                  22
   3.2 SST k-ω                                              25
   3.3 K-kl-ω                                              30
   3.4 Discussion                                           34

4 L2F LOSS ANALYSIS                                         35
   4.1 K-ε                                                  36
LIST OF FIGURES

Figure 1. Midspan static pressure coefficient for various blade profiles……………………3
Figure 2. L2F geometry ........................................................................................................4
Figure 3. L2F-EF geometry ..................................................................................................5
Figure 4. Endwall flow features .........................................................................................6
Figure 5. Midspan total pressure loss coefficient .............................................................9
Figure 6. Reynolds lapse plot of midspan total pressure loss coefficient ......................10
Figure 7. L2F computational domain ..............................................................................17
Figure 8. Boundary layer profiles .....................................................................................18
Figure 9. 2D grid independence study ............................................................................19
Figure 10. 3D grid independence study ..........................................................................20
Figure 11. Average total pressure loss coefficient at 150% axial chord ......................21
Figure 12. Velocity-averaged total pressure loss at 150% axial chord .......................21
Figure 13. k-ε midspan flowfield ....................................................................................22
Figure 14. k-ε convergence plot ......................................................................................22
Figure 15. k-ε secondary axial vorticity comparison .....................................................24
Figure 16. k-ε total pressure loss comparison ................................................................25
Figure 17. SST k-ω midspan flowfield ............................................................................25
Figure 18. SST k-ω convergence plot ............................................................................26
Figure 19. SST k-ω secondary vorticity comparison .....................................................28
Figure 20. SST k-ω total pressure loss comparison .......................................................29
Figure 21. k-kl-ω midspan flowfield ..............................................................................30
Figure 22. k-kl-ω convergence plot ..............................................................................30
Figure 23. k-kl-ω secondary vorticity comparison ......................................................31
Figure 24. k-kl-ω total pressure loss comparison .........................................................33
LIST OF TABLES

Table 1. Loss coefficient comparison.................................................................36
Table 2. L2F and L2F-EF coefficient comparison..............................................47
Table 3. Point locations......................................................................................63
ACKNOWLEDGEMENTS

First, I would like to thank my advisor, Dr. Mitch Wolff, for providing this opportunity and for his guidance and encouragement during my time completing this work, as well as Dr. Chris Marks and Philip Bear for their assistance in understanding the experimental data. I would also like to thank Dr. Rolf Sondergaard and Dr. Rory Roberts for their valuable input. In addition, Dr. Mark McQuilling has been a tremendous help with setting up simulations and analyzing CFD data.

In computational fluid dynamics, technical expertise and IT support is of prime importance. With this in mind, I would like to thank Dr. Mike List of the Air Force Research Laboratory, Sunil Vytla and David Welsh of Cradle Software for their CFD knowledge. I would also like to thank Mike Van Horn and Matt Kijowski of Wright State University for their information technology assistance.

Last but not least, I would like to thank Dr. Andreas Gross of New Mexico State University for his ILES data, which has been proven to be a useful benchmark for comparisons in areas where experimental data was either insufficient or absent. I would also like to thank the Dayton Area Graduate Studies Institute for supporting this research, and my family and friends, without whose support I would not have made it this far.
**NOMENCLATURE**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>LPT</td>
<td>Low Pressure Turbine</td>
</tr>
<tr>
<td>$C_x$</td>
<td>Axial Chord (m)</td>
</tr>
<tr>
<td>$S$</td>
<td>Pitch (m)</td>
</tr>
<tr>
<td>$H$</td>
<td>Span (m)</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds Number</td>
</tr>
<tr>
<td>$Y_t$</td>
<td>Total Pressure Loss</td>
</tr>
<tr>
<td>BL</td>
<td>Boundary Layer</td>
</tr>
<tr>
<td>FSTI</td>
<td>Freestream Turbulence Intensity</td>
</tr>
<tr>
<td>PV</td>
<td>Passage Vortex</td>
</tr>
<tr>
<td>SSSCV</td>
<td>Suction Side Corner Separation Vortex</td>
</tr>
<tr>
<td>SV</td>
<td>Shed Vortex</td>
</tr>
<tr>
<td>CV</td>
<td>Corner Vortex</td>
</tr>
<tr>
<td>$Z_w$</td>
<td>Zweifel Loading Coefficient</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>URANS</td>
<td>Unsteady Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>ILES</td>
<td>Implicit Large Eddy Simulation</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Fredrichs-Lewy Number</td>
</tr>
<tr>
<td>$Y^*$</td>
<td>Nondimensional Wall Distance</td>
</tr>
<tr>
<td>$U_{in}$</td>
<td>Inlet Velocity (m/s)</td>
</tr>
<tr>
<td>$U,V,W$</td>
<td>Mean Flow Velocity (m/s)</td>
</tr>
<tr>
<td>$u,v,w$</td>
<td>Velocity Fluctuations (m/s)</td>
</tr>
</tbody>
</table>
\( < > \) = Time Average

\( \rho \) = Density

\( P \) = Pressure

\( P_{\text{lin}} \) = Total Upstream Pressure

\( \lambda \) = Stagger angle

\( C_{w,s} \) = Secondary Vorticity Coefficient

\( Q \) = Q criterion

\( \Psi \) = Deformation Work

\( \dot{P}_t \) = Total Pressure Transport along a Streamline

\( k \) = Turbulent Kinetic Energy

\( \varepsilon \) = Turbulent Dissipation

\( \omega \) = Eddy Dissipation Rate

\( k_L \) = Laminar Kinetic Energy
1 INTRODUCTION

1.1 Low Pressure Turbines

Gas turbine engines are the primary means by which military and commercial aircraft are propelled, and therefore technologies that improve them are extremely important areas of research. Gas turbine engines operate in three primary stages: compression, combustion, and expansion. Air is sucked into the engine, usually via a fan, where it is compressed by the compressor. In a typical two-spool engine, the compressor (as well as the turbine) is divided into a low-pressure and high-pressure component. The air, which has undergone some compression before entering the low pressure compressor (LPC) is further compressed by the LPC, and then again by the high pressure compressor (HPC). The highly compressed air then enters the combustor where it is mixed with fuel and burned. This mixture enters the high pressure turbine (HPT), which expands the flow and extracts work to drive the high pressure compressor section. The partially expanded flow continues on to the low pressure turbine (LPT) where it is expanded further and work is extracted to drive the LPC and usually a fan. Finally, the air is exhausted through the nozzle along with any air that passes through the fan and bypasses the core, to provide thrust. In high bypass engines, the LPT can be responsible for up to 1/3 of the weight and 80% of the thrust (via driving the fan) of an engine. Gas turbine engines are widely used for both aerospace and power generation applications. They are extremely complicated technology, therefore research into improving/understanding their performance has been
undertaken for decades since their initial development. Research regarding the high and low pressure turbine components has focused on improving the efficiency and performance, increasing the amount of useful work that can be extracted from the flow.

1.2 Motivation of Research

By decreasing the weight and complexity of the LPT while maintaining the same performance, the total engine cost can be reduced and the thrust-to-weight ratio increased. The weight can be reduced by reducing the number of blades used in the LPT section. In order to do this, the same work extraction must be spread across fewer blades, leading to a higher loading on each blade. This allows either the extraction of the same amount of work for a LPT with fewer blades, or the extraction of additional work for an LPT with a similar number of blades. Multiple aerospace industry trends are resulting in the increase in electrical power required by aircraft, including increases in the complexity of electronic aircraft subsystems and the development of aircraft weapons systems requiring large amounts of electrical power. This trend adds to the challenge of producing more power with fewer LPT blades as typically power extraction occurs on the LPT spool.

Low pressure turbines must be able to operate effectively across a range of Reynolds numbers (Re), leading to the use of Reynolds lapse as an important performance indicator. The Reynolds lapse curve is the total pressure loss coefficient plotted as a function of Re (which is inversely related to altitude for a given operating Mach number), and presents the effectiveness of a particular blade across the operating range. Total pressure loss represents irreversible losses in the flow that negatively affects performance. Due to separation effects at lower Re, there are sharp increases in the total pressure loss experienced at the lower end of the operating Re region (high altitudes). In addition, there
are a variety of complicated flow structures near the endwall region that interact with each other to cause increased losses. These flow phenomenon are challenging to accurately model using computational fluid dynamics models because they are sensitive to details of flow separations that occur in multiple places in the passages. Various sets of equations are used in different CFD models to capture the fluid dynamics involved. Each of these models has a different level of complexity which is directly related to the cost of computation in terms of both computing resources (memory/processors) and time. Commercial CFD models solving the Reynolds-Averaged Navier Stokes (RANS) equations are prevalent in industry and widely used for component design. RANS solvers have been developed to a reasonably mature state and require moderate computational costs. An important feature of RANS solvers is the turbulence closure model used with there being many different models available. It is important to understand the capability of RANS-based CFD models to predict these LPT flow features and their performance. Proper understanding of this capability will lead to increased confidence in the use of RANS models for LPT blade design.

1.3 Highly-Loaded Blades

The Zweifel coefficient was developed by Zweifel\textsuperscript{2} to provide a parameter that can be used to compare the loading levels of different turbine blade designs with the same gas angles. This coefficient is a function of both

![Figure 1. Midspan static pressure coefficient for various blade profiles\textsuperscript{1}](image)
the geometry (chord, pitch, turning angles) and the velocity change produced by the blade. McQuilling defined a highly-loaded LPT blade profile under the current state-of-the-art technology level is one having a $Z_w > 1.15$, which is the Zweifel coefficient of the Pack B design developed by Pratt & Whitney. This means that the Pack B is the current benchmark to which highly-loaded LPT blades are compared. The Turbomachinery Branch of the Air Force Research Laboratory at Wright Patterson Air Force Base has developed a series of research airfoils similar to the Pack B, but with significantly higher loading, which are being used for the study of the flow physics of highly-loaded blades. These airfoils are designated the L1M, L1A, L2A, and L2F. “L” stands for an LPT blade design. The numeral represents the increase in lift over the Pack B, with the 1 standing for a Level 1 increase in lift of 17%, while the 2 stands for a Level 2 increase in lift of 38%. The letter indicates the location of the peak of the pressure distribution, with A for aft-loaded, M for mid-loaded, and F for front-loaded.

Highly-loaded blades are prone to separation at lower Reynolds numbers due to the presence of a strong adverse pressure gradient on the suction side that opposes the flow,
decreasing the boundary layer momentum and causing separation. An aft-loaded profile (L1A Figure 1) has its pressure peak towards the trailing edge. By modifying the shape of the camber, the peak of the pressure distribution can be moved towards the leading edge (L2F Figure 1). This causes the magnitude of the pressure gradient from the peak location to the trailing edge to be lower, spreading the same change in pressure over a greater distance. This has been shown to reduce profile loss, particularly at low Re. Therefore, the L2F is the focus of this research into highly-loaded LPT blade profiles.

The L2F (Figure 2) was developed by McQuilling³ with Zw=1.59 and the same gas angles as the Pack B. It was found to provide 38% more lift compared to the Pack B profile. The axial chord of the L2F used in the LSWT facility is 6 inches, with a pitch of 7.326 inches and a span of 25 inches. The inlet angle of the flow is 35 degrees, while the exit angle is 58.4 degrees. The blade is designed to promote earlier boundary layer transition that helps overcome the adverse pressure gradient and leads to a thinner boundary layer, reducing losses. However, it was found by Lyall¹ that the high stagger angle inherent in the L2F design promotes the separation of the inlet endwall boundary layer, which contributes to higher endwall losses than an aft loaded blade like the L2A. Lyall¹ designed a variant of the L2F that decreased the stagger angle of the profile to create the L2F-LS, then blended this shape into the L2F away from the wall. They found that by using the low-stagger profile (L2F-LS) near the endwall and the high-stagger profile (L2F) at higher spanwise locations, they reduced the overall passage loss. This 3D contoured blade profile is referred to as the L2F-EF, shown in Figure 2. The L2F-
EF was found by Marks et al\textsuperscript{4} to reduce passage total pressure loss across a range of Reynolds numbers, with an increase in Reynolds number corresponding to an increase in loss reduction relative to the L2F.

1.4 Flow Physics

The endwall region of the L2F blade profile is the location of several important flow structures that contribute to passage loss, as shown in Figure 4. These are mainly vortices that form due to the cross-passage pressure driven separation of the inlet boundary layer and its interaction with the blade surface and the stream-wise and cross-stream pressure gradients. The inlet boundary layer forms two main structures when it encounters the leading edge of the blade: the suction side horseshoe vortex (SSHV) and the pressure side horseshoe or passage vortex (PV). For the L2F, the SSHV moves away from the suction side and then dissipates near mid-passage. Meanwhile, the PV moves from the pressure side of one blade towards the suction side of the next due to the cross-passage pressure gradient and then lifts off the endwall. Another vortex, the suction side corner separation vortex (SSCSV) is also present, forming in the same region in which the SSHV dissipates and where the SSHV interacts with the suction side blade boundary layer.

In addition to the vortices, for the L2F a laminar separation bubble (examined by Marks et al\textsuperscript{4}) forms in a 2D manner across the span on the suction surface of the blade in

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig4.png}
\caption{Endwall flow features\textsuperscript{5}}
\end{figure}
the vicinity of minimum pressure. This bubble reattaches at roughly 57% axial chord away from the endwall. This bubble interacts with and is affected by the endwall vortex structures, with the strength of the vortices affecting the penetration height of the separated region. The appearance of this on the L2F is believed by the authors to be related to the interaction of the passage vortex and the separation of the flow. They also examined the effect of Reynolds number on the passage loss of the L2F at a plane of 150% axial chord, finding that decreasing the Reynolds number increased the area of the high loss region. This also results in a movement of the endwall lift-off line of the passage vortex towards the middle of the passage and away from the suction surface as the Reynolds number is decreased. The suction surface lift-off line of the passage also moves towards the endwall.

Bear et al.\(^5\) examined the sources of loss production and deformation work in the endwall flow field of the L2F. They found that locations of peak deformation work match up with locations of high total pressure loss. The total pressure gradient also matches up well with the convective total pressure transport. In addition, it was determined that the component of deformation work that is oriented normal to the flow direction is the best predictor of loss development. The authors compared the flowfields of the L2F and the L2F-EF, finding that the turbulence intensity and deformation work were reduced by the addition of the 3D endwall contour. They theorize that the higher level of turbulence intensity of the L2F is due to higher levels of turbulent and unsteady flow features and that adding the contour has a dampening effect, reducing the total pressure loss.

Gross et al.\(^6\) performed a 3-D implicit large-eddy simulation (ILES) of the L2F profile at Re=100k, and showed that the flow features are heavily dependent on the state of the incoming boundary layer. The authors used a research code with a finite volume
method. The study found that the code agreed reasonably with previously obtained PIV data for an L2F profile at a Reynolds number of 100,000 with an incoming turbulent boundary layer. However, for a laminar endwall boundary layer, the passage vortex (the dominant endwall flow feature) is absent and the total pressure loss coefficient is reduced compared to the turbulent boundary layer. The reduction in passage vortex strength for the L2F-EF found in the experimental results is also shown in the computational results.

1.5 Transition Effects

The effect of the transition of the incoming flow from laminar to turbulent is an important factor in designing and analyzing low pressure turbine blades. This is particularly true near the endwall, which is dominated by separation driven flow features. Since the end goal is a blade that has improved performance across a wide range of Reynolds numbers, the mechanisms of transition and the capability of computational tools to accurately predict it is an essential part of this research. There has been a large amount of work done in determining the effectiveness of various models in predicting transition. Gross and Fasel\(^7\) performed a study of a wide variety of turbulence models, finding that none of them managed to accurately predict the flowfield. Keadle and McQuilling\(^8\) tested a three equation model developed by Walters and Cokljat\(^9\) in an unsteady simulation, finding that it provided improved performance for transition and separation but failed to accurately predict total pressure loss in the wake due to over-predictions of the coherency of vortices being shed from the suction side of the blade. This was compared to experimental data as well as results from the AKN model developed by Abe et al\(^{10,11}\) and the MPAKN model developed by Kato and Launder\(^{12}\). In addition, a region of total pressure gain was predicted for the three equation model and the MPAKN model (Figure 5). Their study was performed
at a Reynolds number of 25k, and they theorized that the pressure loss predictions would be better at a higher Re. They also predicted that a three-dimensional simulation would produce flow features that would help to break up some of these vortices and lessen their effect.

![Image](image_url)

**Figure 5.** Midspan total pressure loss coefficient\(^8\).

Empirical transition models developed from experimental data are also used in transition prediction, such as the model developed by Praisner and Clark\(^13\). This model was applied to the Wildcat flow solver and provided much improved Reynolds lapse prediction when compared to the commercial RANS codes LEO and Fluent. The physics-based k-kl-\(\omega\) model developed by Walters and Cokljat\(^9\) and based on the model created by Walters and Leylek\(^14\) was implemented in the commercial code Fluent and accurately predicted total pressure loss at Re=50K and above, but failed at lower Reynolds numbers. Figure 4 shows this comparison, clearly displaying the issues that such models have as well as the accuracy of the empirical model implemented in Wildcat over the same domain.
Important in the prediction of transition and turbulence is the effect of the freestream turbulence on the pre-transitional boundary layer. One of the primary mechanisms by which energy is transferred from the freestream to the boundary layer is referred to as the “splat” mechanism. This mechanism, described by Bradshaw\textsuperscript{15}, as occurring when vortices in the freestream with length scales similar to that of the boundary layer thickness collide with the wall, causing their energy to be redirected along the wall. This contributes to the production of laminar kinetic energy in the boundary layer, which was shown by Jacobs and Durbin\textsuperscript{16} to be related to fluctuations in freestream turbulence. As mentioned previously, the models of Walters and Leylek\textsuperscript{14}, and Walters and Cokljat\textsuperscript{9}, as well as that of Sveningsson\textsuperscript{17} include transport equations to account for the laminar kinetic energy.

In the case of simulating a low pressure turbine blade, a laminar velocity profile is specified at the inlet to the computational domain, and it is this boundary layer that
transitions and rolls over while encountering the leading edge of the blade, forming the two legs of the horseshoe vortex. Therefore, the importance of transition prediction is essential in predicting the performance of low pressure turbine blades. In addition, the model of Walters and Cokljat\textsuperscript{9} was shown by Keadle and McQuilling\textsuperscript{8} to improve separation prediction. As separation can be a large source of pressure loss, accurate prediction of separation and reattachment is also essential to performance prediction.

### 1.6 Current Work

As mentioned previously, it is essential that the performance of RANS models be understood within the scope of the LPT design space. This includes understanding the capability of these models to predict total pressure loss, both at midspan and throughout the passage. In addition, this requires examining the accuracy of the predicted flow features, including the separation bubble and the various vortex structures. In order to achieve this, several turbulence models and several inlet boundary layer thicknesses were applied to a computational flow domain imitating that of the LSWT. The sensitivity of the flow features and the models to inlet boundary layer thickness is explored. The accuracy of downstream total pressure loss prediction for each model is examined, as well as the ability of each model to accurately predict the size and structure of relevant flow features. The L2F-EF profile is also analyzed to determine the effectiveness of RANS in predicting a performance improvement from 3D profile contouring. Finally, unsteady RANS is investigated to better understand the difference between steady and unsteady RANS CFD predictions of LPT flow physics.
2 COMPUTATIONAL METHODS

2.1 Computational Fluid Dynamics

The field of CFD has transformed engineering, harnessing computing power to enable the analysis of fluid dynamics problems with a level of detail and efficiency unachievable with experiment alone. While wind tunnel testing is required for the validation of computational results, CFD allows for very detailed examination of flow physics for countless different cases at a fraction of the cost. Oftentimes, an initial prototype is constructed and tested in the tunnel to obtain global flow parameters. These global parameters allow the validation of the CFD results. CFD is then used to analyze the prototype in a wide variety of flow conditions and variations of local flow parameters. This combination of experimental and computational work has greatly improved the efficiency and effectiveness of the design process, delivering superior results at a lower cost.

Computational fluid dynamics is divided into three general categories of fidelity: Reynolds-Averaged Navier-Stokes (RANS), large-eddy simulation (LES), and direct numerical simulation (DNS). DNS entails solving the equations of fluid flow at all turbulent length scales, all the way down to the Kolmogorov scale. This is extremely expensive, requiring an extraordinarily high level of mesh refinement and computational power. This prohibits the use of DNS for all but the simplest cases. LES solves only the large turbulent scales, ignoring the smaller scales that are also solved by DNS. While still being computationally costly, LES has become more common and useful as computing
power has improved. RANS is the least computationally expensive method since it only computes the coarsest scales, but requires the use of turbulence models to provide turbulence closure of the Navier-Stokes equations, leading to a lower level of accuracy than DNS and LES.

Traditionally, the most widely used equation set for CFD modeling is the RANS equations. A Reynolds decomposition is performed on the instantaneous Navier-Stokes equations, which separates them into a time-averaged component and a fluctuating component. These equations are discretized using a finite-element, finite-volume, or finite-difference algorithm, with a network of nodes representing the geometry. In the finite volume method, fluxes are calculated across the faces of a volume surrounding the node. In the finite difference method, differences between neighboring nodes are used to calculate flux terms. In the finite element method, the problem is broken down into smaller pieces and the equations are applied to these individually, then reassembled to analyze the entire system. The finite-element method is usually used for structural analysis, while the finite-volume and finite-difference methods are much more common in CFD.

2.2 SC/Tetra

Tetra is a CFD code developed by Cradle Software\textsuperscript{18}. It is a full CFD model, containing a preprocessor, flow solver, and postprocessor as well as a variety of utilities. It is capable of utilizing steady and unsteady RANS equations as well as LES equations for both compressible and incompressible flows. The Reynolds-averaged Navier-Stokes equations are solved using the SIMPLEC\textsuperscript{19} pressure correction and MUSCL\textsuperscript{20} flux scheme for incompressible flows. An algebraic multigrid solver is used to improve computation convergence along with an unstructured mesh including the option of using prism layers.
2.3 Turbulence Models

Reynolds-Averaged Navier-Stokes methods require the use of turbulence models to provide closure of the equations. A wide variety of turbulence models have been developed for various purposes, ranging from models specialized for low Reynolds number flows focusing on transition and separation to ones developed for high speed flows. When it comes to modeling transition, there is an ongoing discussion as to whether or not physics-based or empirical models are the proper route for the research community to pursue. Empirical models such as that of Praisner and Clark\textsuperscript{13} have so far proven to be the most accurate and reliable means of prediction. However, they require having experimental data for a particular configuration and therefore are not universally applicable, as they are not derived from the flow physics involved. Physics-based models have struggled to reach the same level of accuracy due to the incomplete understanding of transition physics. Several more recent models have been developed to improve prediction of transition and separation. Some of these models will be utilized. Classical shear stress transport models (k-ω and k-ε) were used in an attempt to accurately predict the performance and flow features of a LPT blade, in order to understand the capabilities of these. These models are discussed in the following sections.

Two-Equation k-ε Turbulence Model

The standard k-ε model is a very popular model that solves for turbulent kinetic energy $k$ and turbulent dissipation $\varepsilon$. The model equations are provided below.

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \rho \varepsilon - Y_M + S_k \quad (1)$$

$$\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) \right] + C_1 \varepsilon \frac{\varepsilon}{k} (P_k + C_3 \varepsilon P_b) - C_2 \rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (2)$$
Two-Equation SST k-ω Turbulence Model

The shear stress transport formulation of the SST k-ω model was developed by Menter\textsuperscript{21,22} and is often used as a low-Reynolds number turbulence model. This model is a combination of the k-ω and k-ε models, using the k-ω equations in the boundary layer and the k-ε model in the freestream flow. The model is meant to provide improved performance in cases dealing with flow separation, due to the improved modeling of the boundary layer. It solves for the turbulent kinetic energy k and the eddy dissipation rate ω. The primary equations are provided below, see the references to Menter’s development of the model for a complete formulation.

\[
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ (v + \sigma_T \nu_T) \frac{\partial k}{\partial x_j} \right]
\]

(3)

\[
\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ (v + \sigma_T \nu_T) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \sigma \omega \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}
\]

(4)

Three-Equation k-kl-ω Turbulence Model

Walters and Cokljat\textsuperscript{9} developed a three-equation turbulence model that adapts the recommendation of Mayle and Schultz\textsuperscript{23}, adding an equation accounting for the laminar kinetic energy in the boundary layer to the existing k-ω framework. This model is similar to the one developed by Walters and Leylek\textsuperscript{14}. The additional equation is meant to improve the models prediction of the magnitude of low-frequency fluctuations in the incoming boundary layer. As discussed previously, this is an important factor in boundary layer transition. The model displays transition as a redistribution of energy from laminar kinetic energy (k_L) to turbulent kinetic energy (k_T) via the increase in magnitude of the pressure strain terms in the Reynolds stress equation. This differs from interpretations of transition as turbulent production or dissipation. The incompressible single phase form of the model equations are defined as:
\[
\frac{\partial k_T}{\partial t} = P_{kT} + R_{BP} + R_{NAT} - \omega k_T - D_t + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\alpha_T}{\sigma_k} \right) \frac{\partial k_T}{\partial x_j} \right]
\] (5)

\[
\frac{\partial k_L}{\partial t} = P_{kL} - R_{BP} - R_{NAT} - D_L + \frac{\partial}{\partial x_j} \left[ \nu \frac{\partial k_L}{\partial x_j} \right]
\] (6)

\[
\frac{\partial \omega}{\partial t} = C_{\omega 1} \frac{\omega}{k_T} P_{kT} + \left( \frac{C_{\omega R}}{f_W} - 1 \right) \frac{\omega}{k_T} (R_{BP} + R_{NAT}) - C_{\omega 2} \omega^2 + C_{\omega 3} f_\omega \alpha_T f_W^2 \frac{k_T}{d_s} +
\]

\[
\quad \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\alpha_T}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right]
\] (7)

Equation 5 and 7 are the k-\(\omega\) equations, while Equation 6 is the laminar kinetic energy equation. As discussed by the authors of the model, the second term and the fourth on the right-hand side of the \(\omega\) equation function to decrease the turbulent length scale during the breakdown of transition and in the outer region of the turbulent boundary layer, respectively. Equations for the wall effects, and others required to close the model are provided in the reference.

### 2.4 Flow Domain and Boundary Conditions

For the L2F computation, the same 2-D airfoil shape is maintained from hub to tip. The computational flow domain is shown in Figure 7. The boundary conditions used required a velocity inlet to set a Reynolds number of 100,000, as well as a static pressure outlet boundary condition. Three boundary layer profiles are specified at the inlet. In the first no profile is specified, allowing the boundary layer to develop from the interaction of the freestream inlet velocity and the endwall. The second is a thin boundary layer profile measured from experiments in the LSWT using the short splitter plate length. The third is a thick profile measured during experiments with the long splitter plate length, and is the profile present in all the experimental data presented. Special attention is given to the comparisons between the experimental data and the thick boundary layer profile results. The thin and thick profiles are shown in Figure 8. The CFD freestream turbulence intensity
(FSTI) matches the experimental inlet turbulence of 3.1%, as well as a viscosity ratio of 6.1447 calculated from the experimental length scale of 0.01m. The inlet plane is located 1.2 chords upstream of the leading edge and the outlet plane is located 3.5 chords downstream of the trailing edge. The inlet and exit grid planes were specified from the airfoil leading and trailing edges to ensure minimal effect on the solution. The computational domain is one full pitch in the pitchwise direction with periodic boundary conditions utilized. Stationary wall boundary conditions are specified on the blade surface and endwall. Wall contoured prism layers are used on both the blade surface and endwall to capture the boundary layer velocity profile. A symmetry plane was created at midspan as the blade profile is constant in the spanwise direction with a wall boundary at both the blade hub and tip in the LWST linear cascade.

Figure 7. L2F computational domain
2.5 Mesh Details and Grid Independence Study

In order to generate the mesh for the L2F, both two-dimensional and three-dimensional grid independence (GI) studies were carried out. The turbulence model used for the GI studies was the $k$-$\varepsilon$-$\omega$ model. The two-dimensional study compared the pressure coefficient distribution and total pressure loss coefficient at 150% axial chord downstream of the leading edge for each mesh to determine which 2D mesh to use for the generation of the 3D grid. 2D grids were generated with 40,000, 140,000, and 200,000 mesh points. The 140k mesh achieves the same level of accuracy as the 200k mesh for the $C_p$ distribution. The difference between the 140k and 200k mesh for the $P_{loss}$ distribution is only visible in the wake region and is very small. Therefore, the 140k mesh is considered sufficient and is used as the basis for 3D grid generation. The results of the 2D GI study are shown in Figure 9.
After completion of the 2D GI study, 3D meshes were generated to determine the amount and location of the spanwise points required for grid independence. Since one of the goals is to determine the effectiveness of the models in predicting total pressure loss, the total pressure loss distribution is examined at 50% (midspan), 25%, 7.5%, 5%, and 2.5% span. These results are shown in Figure 10. Examining the midspan plots, the 16M mesh is not independent on the left side of the passage. Small changes become apparent at 7.5% and 5% span, showing the independence of the 21M mesh. More difference is evident at 2.5% span, where 21M mesh is again independent. As a further measure of convergence, the total pressure loss coefficient was both area-averaged and velocity-averaged over a vertical plane located at 150% axial chord downstream of the leading edge. Figures 11 and 12 contain plots of these results. Clearly the 21M grid results are the same as the 32M grid, so the 21M mesh will be used for the general 3D study. This mesh was used as a guide to generate a mesh for the L2F-EF, resulting in a 30 million element mesh for the L2F-EF results. The higher element count is due to the necessity of using tetrahedral elements in the region where the contour exists, since it is not symmetric along the vertical axis.
Figure 10. 3D grid independence study
Figure 11. Area-averaged total pressure loss coefficient at 150% axial chord

Figure 12. Velocity-averaged total pressure loss at 150% axial chord
3 BOUNDARY LAYER THICKNESS SENSITIVITY

3.1 Two-Equation \( k-\varepsilon \) Turbulence Model

The midspan velocity flowfield for the \( k-\varepsilon \) turbulence model is shown in Figure 13. There is a region of very low velocity/separation beginning at 60% axial chord and then extending downstream past the trailing edge where it forms a wake. The convergence plot is shown in Figure 14.

**Figure 13.** \( k-\varepsilon \) midspan flowfield

**Figure 14.** \( k-\varepsilon \) convergence plot
Planes of secondary axial vorticity coefficient were computed to allow more detailed examination of the vortex structures in the flowfield and how they change as the flow moves through the passage. The secondary direction is taken as the exit flow angle of 58.4 degrees. This means that the plots shown are normal to the exit flow direction. The secondary vorticity coefficient is computed by the equation

$$C_\omega = \frac{dW}{dy_s} - \frac{dV_s}{dz}$$  \hspace{1cm} (8)

where $V_s$ is the pitchwise velocity in the secondary direction, and $y_s$ is the y-coordinate transformed by the exit angle, which is the secondary y-coordinate.

The secondary vorticity results for the $k$-$\epsilon$ turbulence model are shown in Figure 15. Examining the no boundary layer case shows that the movement of the region of negative vorticity up the suction surface of the blade is present at 50% axial chord, and results in the formation of a distinct region at 70% and 85% axial chord. This behavior is repeated for all boundary layer thicknesses. The spanwise location of the distinct suction side region is shown to be dependent on the boundary layer height. This is most easily observed with the thick boundary layer results. While the no boundary layer and thin boundary layer profiles match the experimental profile more closely at 50%, 60%, and 70% axial chord, the thick boundary layer case is the most accurate at 85% axial chord.
Figure 15. k-ε secondary axial vorticity comparison

The results of the total pressure loss distribution for the k-ε turbulence model are shown in Figure 16. A large region of total pressure loss is shown to be present for all boundary layer thicknesses throughout the axial locations examined which over-predicts the loss compared to the experimental results. The effect of boundary layer thickness on the loss is to increase the spanwise extent as well as a slight increase in magnitude. The thick boundary layer cases predicts the most accurate spanwise location, around 7% span for the 80% axial chord location (close to the experimental location of peak loss) and around 9% span for the 100% axial chord location. However, the overprediction of loss is
significant and negatively affects the ability of the k-ε model to be used for performance prediction.

![Figure 16. k-ε total pressure loss comparison](image)

3.2 Two-Equation SST k-ω Turbulence Model

The midspan velocity flowfield for the thick boundary layer case of the SST k-ω turbulence model is shown in Figure 17. The region of low velocity starts around 60% axial chord but does not reattach, instead forming a wake that continues past the trailing edge. This results in high two-dimensional total pressure losses. The

![Figure 17. SST k-ω midspan flowfield](image)
convergence history is shown in Figure 18.

![Figure 18. SST k-ω convergence plot](image)

Figure 19 shows contours of secondary vorticity coefficient at 50%, 60%, 70%, and 85% axial chord, with the SST k-ω turbulence model. Examining the contours of secondary vorticity for the no boundary layer case, it is shown that the magnitude and shape of the vorticity profile is similar to experiment, especially at 50%. At 60%, the profile is located lower than that of the experimental results and a vortex forms on the blade surface further up the span, not seen until 85% experimentally. Several other vortices form around this second vortex at 85%. In addition, a region of high positive vorticity is shown to be present at midpitch at 50% axial chord, moving towards the suction surface. The behavior of the thin and thick boundary layer cases generally mirrors that of the no layer case. The region of negative vorticity is larger for the thin layer than for the no layer, and so is the positive region of vorticity. This near-wall region also shows up slightly closer to the blade surface. The second vortex forms further up the span, and the additional vortices that form around 85% axial chord are of greater magnitude. However, the large region of negative vorticity
weakens more quickly. For the thick boundary layer case, this pattern of behavior continues. The negative vorticity region is initially larger than the other two cases and the experimental data, with a larger positive region also present closer to the suction surface of the blade. At 60%, the vortex that forms near the blade surface is further up than that of the thin or no layer case, which suggests that its formation height directly correlates with the height of the incoming boundary layer. While the magnitude of vorticity is the strongest at 50% and 60%, at 70% and 85% much more dissipation has occurred for the thick layer case than for the other two. In addition, the vortices present at 85% axial chord are much closer to the wall than for the others, with only a small region of near-wall negative vorticity remaining. These results show that the boundary layer specified at the inlet has a large effect on the magnitude of the flow features, but that the general flow structure stays constant with the exception of the spanwise location of the system of vortices at the various axial planes.
Examining the results of the total pressure loss for the no boundary layer case in Figure 20, it becomes obvious that the region of high loss located along the span of the blade is much larger and higher in magnitude than the experimental result. This is due to the model’s lack of any reattachment to the blade surface. This is also true for the thin and thick layer cases, which differ from the no boundary layer case primarily by the spanwise location of the high loss region. It is clear that the effect of increasing boundary layer thickness is to increase the spanwise location at which the high loss region begins. This is evident by the fact that the lower edge of the region at 60% axial chord is located at 5%
span for the no layer case, 7.5% span for the thin layer case, and 10% span for the thick layer case. This is true for all axial locations, again confirming the effect of boundary layer thickness on the spanwise location of flow features and loss structures. While the wake region close to the endwall is predicted reasonably well in magnitude, it is much smaller in the computational results than in the experimental data.

Figure 20. SST k-ω total pressure loss comparison
3.3 Three-Equation $k$-$k_l$-$\omega$ turbulence Model

The midspan velocity flowfield for the thick boundary layer case $k$-$k_l$-$\omega$ turbulence model is shown in Figure 21. The convergence plot is shown in Figure 22. The flow separates at 60% axial chord and reattaches at 80% axial chord. This turbulence model does an excellent job of modeling the separation/reattachment region for $Re=100,000$. This is more accurate than the complete separation shown in the other two models. Figure 23 shows contours of secondary vorticity coefficient at 50%, 60%, 70%, and 85% axial chord. The y-axis of each plot is the suction surface of the blade. On the left is the experimental results obtained by Bear et al\textsuperscript{22}, on the right are Tetra’s flow field predictions.
At 50% axial chord, the large region of negative vorticity (the passage vortex) is wide and flat dominating across the passage. Tetra predicts a small region of high positive vorticity close to the endwall, which is faint in the experimental data but is present to some degree in all plots. This is also present along the suction surface of the blade. In the CFD results for the no inlet boundary layer thickness, the negative vorticity region is shown to creep up the suction surface, an occurrence not present in the experimental data until 60% axial chord. As the flow moves to 60%, the region of negative vorticity continues higher up the
blade surface, with the experimental data showing the formation of a distinct core at the far end of the passage. The CFD model does not show this, with the flow field at 60% being similar to that at 50%. However, at 70% changes start to appear. The vorticity continues up the blade as before, but a core separates from the endwall region and decrease in magnitude as it moves through the passage to 85% chord. The vertical jet is found to form another core of higher negative vorticity at 70%, but this dissipates. This general flow field is present for all three boundary layer settings. This differs from the experimental data, where the core formed at 60% continues to move toward the suction surface of the blade, with another region of high negative vorticity forming at about 6.5% span, which is the suction side corner separation vortex. The computational region of negative vorticity moves closer to the suction surface at 70% and 85%, but not with the strong core still present at 85%. Examining the sensitivity of the inlet boundary layer thickness to these results provides several interesting details. First, it is obvious that a larger inlet boundary layer leads to the code predicting a larger region of negative vorticity, and that the general structure of this region is independent of the boundary layer thickness. For 50% and 60%, the thin boundary layer results most accurately match the experimental data, but at 70% and 85% the thick boundary layer results are best. At this location the large negative vorticity region stays nears the endwall for the thin boundary layer case, instead of moving up the span like the experimental data and thick boundary layer results. This movement is due to the movement of the passage vortex.

Figure 24 shows the comparison between total pressure loss at 60%, 80%, and 100% axial chord downstream of the leading edge for the three boundary layers. The overall shape of the loss distributions for the thick and thin layers are shown to resemble
that of experiment, but the magnitude is significantly greater. The thin boundary layer is slightly closer to the experiment than the thick boundary layer. At 60% chord, the loss structure shown in the computational results is significantly larger and stronger than the experimental result, especially for the thick layer. At 80% and 100%, the difference between the experimental and computational results is smaller but still prevalent. The results for the case without a specified inlet boundary layer profile under predicts the amount of losses through the passage. Comparing between the no boundary layer case and the other two cases, the losses predicted by the CFD model are clearly influenced by the inlet boundary layer profile.

**Figure 24.** k-kl-ω total pressure loss comparison
3.4 Discussion

By examining and comparing the contours of secondary vorticity and total pressure loss, the effect of the turbulence models can be determined and isolated from behavior that is universally predicted. The fact that the vortex is initially located in the right side of the passage and moves up and towards the suction surface as the flow moves towards the trailing edge of the blade is predicted by all turbulence models. This is due to the fact that the passage vortex, which begins as the pressure side of the horseshoe vortex, moves towards the suction surface due to the pressure gradient and then lifts off of the endwall. The effect of the different turbulence models on this is to change the predicted initial location of the vortex, with the k-kl-ω and SST k-ω predicting a vortex that is initially closer to the suction surface. In addition, the turbulence model can also affect the amount of movement experienced by the vortex, although the effect is not as pronounced. The effect of the boundary layer thickness on the initial vortex location is larger than that of the turbulence model, with a thicker boundary layer leading to a vortex that is higher and closer to the suction surface. This may be caused by a thicker boundary layer rolling over to create the passage vortex earlier, leading to the movement of the flow features upstream. An interesting result is that for many of the turbulence models, the thick boundary layer is the most accurate at 85% axial chord despite overpredicting the size of the negative vorticity region earlier in the passage. The thickness of the boundary layer also affects the spanwise location of all features, particular those close to the suction side of the blade. The importance of separation is also noted, with the k-ε and SST k-ω predicting separation but not reattachment, leading to much greater losses. The k-kl-ω model predicts reattachment, and is therefore closer to experiment.
4 L2F LOSS ANALYSIS

In order to understand the ability of RANS to accurately predict endwall flow features and their effect on performance, an analysis of the ability of each turbulence model to predict spanwise total pressure loss distribution and endwall flow structures was carried out. The models developed by Abe et al\textsuperscript{10,11} and Kato and Launder\textsuperscript{12} were also analyzed, but massive flow separation without reattachment was predicted. This is a nonphysical result and led to highly erroneous total pressure loss results. Therefore, it is not included in this work. Figure 25 shows the spanwise distribution of the total pressure loss coefficient for each of the turbulence models examined for the thick boundary layer conditions. It is immediately clear that the k-kl-\omega model is superior, due to the aforementioned effect of separation/reattachment. The loss distribution can be broken down into four regions: the 2D flow region, the suction side corner separation/shed vortex region (SSCSV and SV), the passage vortex region (PV), and the endwall/corner vortex (CV) region. By examining the flows in each region, the effect of the flow feature predictions made by the turbulence models can be linked to the total pressure loss. Table 1 shows a comparison of integrated total pressure loss coefficients for each turbulence
model compared to experiment. Regarding the overall total pressure coefficient, k-ε performs the worst while k-kl-ω performs the best.

<table>
<thead>
<tr>
<th></th>
<th>Yps</th>
<th>%Diff</th>
</tr>
</thead>
<tbody>
<tr>
<td>k-ε</td>
<td>0.197618</td>
<td>61.7172</td>
</tr>
<tr>
<td>SST k-ω</td>
<td>0.165673</td>
<td>35.57534</td>
</tr>
<tr>
<td>k-kl-ω</td>
<td>0.113441</td>
<td>7.167628</td>
</tr>
<tr>
<td>Exp</td>
<td>0.1222</td>
<td></td>
</tr>
</tbody>
</table>

Table 1. Loss coefficient comparison

4.1 Two-Equation k-ε Turbulence Model

Figure 26 shows the comparison between the k-ε results and the experimental data. The k-ε model overpredicts the amount of total pressure loss associated with the 2D flow region and especially the suction side corner separation vortex region. In addition, the losses associated with the PV are also overpredicted. A 3D diagram of the flowfield compared to that predicted by Gross’ ILES results is presented in Figure 27. These are isosurfaces of Q criterion (Q=10) and flooded by secondary axial vorticity. Axial vorticity contains both shear and rotation, while Q criterion isolates the rotational component of the flow. Vorticity is used to provide the direction of rotation, while Q is used to analyze the size of the region. Q criterion is calculated using Equation 9, where
\[ Q = \frac{1}{2} [W_{ij} W_{ij} - S_{ij} S_{ij}] \] (9)

\[ W_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \] (10)

\[ S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \] (11)

Figure 27. k-\( \varepsilon \) flowfield comparison

Comparing the RANS CFD flow features to the ILES shows that the k-\( \varepsilon \) turbulence model significantly overpredicts the size of all four of the major vortex structures. The SSCSV and SV extend much further from the trailing edge of the blade than they do in the ILES results, as do the PV and the CV. This correlates well with Figure 26, indicating that the model is overpredicting losses associated with these flow features. Figure 28 shows plots of Q criterion with total pressure loss coefficient contour lines compared with experimental data taken by Bear et al\textsuperscript{24} and ILES results obtained by Gross et al\textsuperscript{25}. This allows analysis of the losses associated with the various endwall flow features. Examining the computational results at 70\% axial chord shows that the losses associated with the passage vortex are already overpredicted by the k-\( \varepsilon \) model, with a center loss contour of 0.6 compared to 0.4 for the experiment. At 95\% axial chord, the loss core for the passage
vortex is at 0.8, while the experimental result has only increased to 0.6. The suction side corner separation vortex is in both the computational and experimental results, with a peak loss coefficient of 1.2 for the computational data. At 150% axial chord, this value has stayed constant for the k-ε model’s SSCSV prediction and shows 1.2 for the passage vortex and 1.4 for the SSCSV, while the experimental data shows a loss coefficient of 0.6 for both the PV and the SSCSV. The ILES results are closer to the experimental data at all three locations, and do not overpredict losses. Along with the isosurfaces and spanwise distribution plots, these plots clearly show that the model is overpredicting the total pressure loss produced by the PV and SSCSV.

Figure 28. k-ε flow features Ptloss comparison
4.2 Two-Equation SST K-Omega Turbulence Model

Figure 29 shows the spanwise total pressure loss coefficient distribution for the SST k-ω turbulence model compared to experimental data. The model overpredicts the losses caused by 2D flow effects. The model also overpredicts the losses associated with the SCSV and SV, but is better than the k-epsilon turbulence model. The most important detail is that the model vastly overpredicts the losses associated with the passage vortex, and along with the separation prediction this is what leads the model to poor overall performance prediction.

Figure 29. SST k-ω spanwise loss distribution

The flowfield predicted by the SST k-ω turbulence model is compared to the ILES flowfield in Figure 30. The model greatly overpredicts the size of the passage vortex, extending very far past the trailing edge. Since the model predicts the PV to extend much
further, it is predicting that it is initially much stronger, causing the large spike in total pressure loss coefficient in Figure 29. In addition, the PV moves further away from the endwall than in the ILES data, explaining the higher spanwise location of the loss core. The shed vortex and the corner vortex are also much larger than in the ILES data. The most interesting result is the SSCSV result. Several smaller vortexes with the opposite direction of rotation separate from the blade surface region. This does not develop from the suction side leg of the horseshoe vortex. The true SSCSV forms before this and ends up mixing with the PV. Since the model overpredicts all the other vortex structures, the actual SSCSV is being replaced which leads to a slightly overpredicted loss core for this region in Figure 29.

Figure 31 shows the Q-criterion and total pressure loss contours for the SST k-ω model compared to experimental data and the ILES results. Examining the RANS CFD results at 70% axial chord shows that the loss core associated with the passage vortex is initially extremely strong, with a contour level at the core of 1.4, compare to the experimental value of only 0.4. At 95% axial chord, the experimental value has increased to only 0.6, while the computational value at the center is 1.4. At 150% axial chord, the values are similar to 95% axial chord, except that the experimental value of the contour is 0.8 at 15% span. The computational value at 15% span is 1.2, leading to the relative accuracy of the model in predicting the upper loss core as compared to the k-ε turbulence model. The value is 1.6 for the lower core in the computational results, compared to 0.6 for the experimental data. The previous observations about the ILES data holds for this comparison. It is clear from these results that the overprediction of the losses associated
with the passage vortex is the cause of the gross overprediction of the loss coefficient between 5% and 10% span in Figure 30.

Figure 31. SST k-ω flow feature and Ptloss comparison
4.3 Three-Equation $k$-$kl$-$\omega$ Turbulence Model

Figure 32 shows the spanwise distribution of the total pressure loss coefficient for the 3-equation $k$-$kl$-$\omega$ turbulence model. The model does an excellent job predicting losses associated with 2D flow, but slightly underpredicts the upper loss core associated with the SCSV. It again does an excellent job predicting the loss between the upper and lower loss cores, but underpredicts the lower loss core and close to the endwall. However, the qualitative spanwise behavior is extremely well predicted, with only the magnitudes being off in certain places.

Figure 33. $k$-$kl$-$\omega$ flowfield comparison

Figure 33 shows the RANS $k$-$kl$-$\omega$ flowfield compared to the ILES flowfield. The passage vortex is accurately predicted in size and extent, as is the corner vortex. The shed
vortex is much smaller than in the ILES data. The suction side corner separation vortex is slightly smaller, but is not overpredicted like in the k-ε results and the behavior is accurately model, unlike in the SST k-ω results. Of the three models, this is the most accurate flowfield picture, and the accurate prediction of vortex size and behavior is an important factor in the improved performance prediction capabilities of the three equation turbulence model.

Figure 34 shows the Q criterion and total pressure loss contour plots for the k-kl-ω model compared to experimental data and ILES results. Examining the RANS CFD results at 70% axial chord shows a peak loss core for the passage vortex of 1, compare to the experimental value of 0.4. At 95% axial chord, the computational loss core peaks at 1, where the experimental data shows a PV loss peak of 0.6. While the contour levels at 70% and 95% are similar to the k-ε prediction, the contours and flow structures match the experiment more closely than the other two models. At 150% axial chord, the contours match very well, with a contour of 0.6 in the center and a contour of 0.8 further up the span. The RANS contour levels are much closer to the ILES data than for the other two turbulence models, especially at 150% axial chord. These improvements in the prediction of the flowfield result in the improved prediction of total pressure loss.
4.4 Discussion

While the magnitudes were off for the 70% and 95% axial chord planes, the shapes of the total pressure loss contours for the k-kl-ω model matched those of experiment very well at all three locations. In addition, the magnitudes and shapes matched the experiment extremely well at 150% axial chord. As shown by the isosurface plots, the streamwise extent and behavior of the flow features as predicted by the three equation turbulence model matched the ILES data much better than the other two models. These results show that 3D RANS CFD with the appropriate turbulence model can be used to analyze the endwall flow features and total pressure performance of a low pressure turbine blade accurately.
5 L2F-EF LOSS ANALYSIS

5.1 Introduction

As discussed in Section 1.3, highly loaded blade profiles such as the L2F are prone to increased endwall losses. This is due to the high stagger angle causing early separation of the inlet endwall boundary layer. The stagger angle of the L2F was lowered near the endwall to create the L2F-LS. The low-stagger profile (Figure 35) used at the endwall was tapered into the L2F by 9.5% of the span. This results in the L2F-EF 3D blade profile, shown in Figure 36. This combines the midspan performance of the L2F with the endwall performance of the L2F-LS, leading to a highly-loaded blade that mitigates high endwall losses.

An understanding of the capability of 3D RANS CFD to predict the effect of a 3D profile contour on the LPT flow physics is necessary to determine if RANS CFD models can be used as a LPT design tool. A mesh was generated for the L2F-EF based on the mesh used for the L2F, and the k-kl-ω turbulence model was used along with the thick inlet boundary layer.

Figure 35. Profile comparison

Figure 36. L2F-EF blade profile
5.2 L2F-EF Results

Figure 37 shows the spanwise distribution of the RANS CFD average total pressure loss coefficient results for both the L2F and L2F-EF compared to the experimental data. The computational results are very accurate from roughly 20% span to midspan, but the results for the L2F diverge slightly between 12% and 20% span, with the computational losses being lower than the experimental losses from 15% to 20%, and higher from 12% to 15% span. The L2F-EF results are very accurate from midspan to 12% span, including the SSCSV loss region, but the predictions for both the L2F and L2F-EF diverge from the experiment from 10% span to the endwall, although the general behavior is shown to be the same. In particular, the model underpredicts the amount of L2F-EF loss around 7% to 10% span, being closer to the L2F experimental data. The computational results from 7% span to the endwall are lower than the experiment therefore resulting in the overall coefficient being lower than the experimental coefficient. The computational results are able to show the qualitative improvement in total pressure loss due to the reduction of the passage vortex, located at roughly 5% span.

Table 2 shows the integrated total pressure loss coefficient comparison for the L2F and L2F-EF profiles. A 5.44% reduction is shown for the computational results, compared to a 4.99% reduction for the experimental results. The coefficients are underpredicted for both profiles, but overall the trend of the CFD results are consistent with the experimental
results. The ability to predict this trend is important if RANS based CFD is to be used for modeling advanced LPT blade designs.

<table>
<thead>
<tr>
<th></th>
<th>L2F</th>
<th>L2F-EF</th>
<th>% Diff</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>0.113441</td>
<td>0.10727</td>
<td>5.440228</td>
</tr>
<tr>
<td>Exp</td>
<td>0.1222</td>
<td>0.1161</td>
<td>4.991817</td>
</tr>
<tr>
<td>% Diff</td>
<td>7.167628</td>
<td>7.60577</td>
<td></td>
</tr>
</tbody>
</table>

Table 2. L2F and L2F-EF coefficient comparison

To gain a deeper understanding the LPT flow physics associated with the L2F-EF, 3D flow features are compared to experimental results using secondary vorticity plots. Figure 38 shows the CFD secondary vorticity for the L2F-EF and L2F compared to the experimental L2F-EF results. One of the major differences between the experimental and computational results for the L2F-EF is the spanwise thickness of the negative vorticity (blue region). This was also shown with the L2F results. The behavior of the flow near the blade surface is accurately predicted by the code, with the region of negative vorticity moving up the blade surface and forming a separate core. However, this distinct core forms at 70% axial chord in the computational results as opposed to 85% in the experimental results. Another difference is the shape and size of the high vorticity core near the endwall. In the experimental results, this core is wide and flat along the endwall region. Computationally, it is a horseshoe shape and focused near the blade surface. The region of negative vorticity is weaker for the L2F-EF than for the L2F for all locations shown, indicating the effect of the EF contour is to weaken the strength of the passage vortex, the main rotational structure in the endwall region.
Figure 38. L2F-EF secondary vorticity comparison

Planes of total pressure loss coefficient at different axial chord locations are shown in Figure 39 for the L2F and L2F-EF computationally and experimentally for the L2F-EF.
The RANS CFD results show large differences compared to experiment at 60% and 80% axial chord, but are much closer at 100% axial chord. The main difference at 60% axial chord is the presence of a large region of total pressure loss at the endwall, while at 80% axial chord it is the spanwise extent of the region of high total pressure loss along the blade surface (resulting from separation). The CFD L2F-EF results are similar to those of the L2F, with a reduced intensity of the total pressure loss region near the endwall at 80% and 100% axial chord.

![Figure 39. L2F-EF total pressure loss comparison](image-url)
Figure 40 shows isosurfaces of $Q=10$ flooded by secondary axial vorticity for the passage of the L2F and the L2F-EF, while Figure 41 shows the same quantities for the trailing edge region. Examining Figure 40, the L2F-EF shows a weakened passage vortex relative to the L2F, confirming the effect found by experimentation. In addition, the suction side leg of the horseshoe vortex is clearly shown in both cases, being opposite in rotation to the passage vortex. A counter-rotating vortex accompanying the passage is shown; this too is weakened by the 3D contouring. The scattered regions of positive (red) vorticity in the L2F-EF results are caused by the CFD grid and should be neglected.

![Diagram](image)

**Figure 40.** Computational passage comparison

Several important flow features are displayed in Figure 41. The passage vortex is clearly shown, as well as the suction side corner separation vortex immediately above it. In red, near the junction of the endwall and the suction surface, the corner vortex is shown. Downstream of the SSCSV in red is the shed vortex, coming off the trailing edge of the blade. The same flowfield is shown for the L2F-EF. The passage vortex does not extend as far past the trailing edge as it does for the L2F, but the SSCSV is much larger in size. This is partially a result of the element size being somewhat larger in this area than for the L2F, which is also the cause of the comparative smoothness of the L2F isosurfaces. The SSCSV
also extends further than in the L2F. This was also shown in the vorticity plots, Figure 38. The shed vortex (SV) is much clearer and more significant in the L2F-EF results, as well as the corner vortex (CV).

![Diagram](image.png)

**Figure 41.** Computational trailing edge region comparison

Figure 42 shows a comparison of Q criterion with total pressure loss coefficient contours at different axial chord locations for the L2F-EF RANS CFD and experimental data. At 70% axial chord, the RANS model captures the flattening of the passage vortex, but the peak value of the total pressure loss contours is 1 versus 0.4 for the experimental data. At 95% axial chord, the computational results peaks at a contour of 0.8 for the PV and 1.2 for the SSCSV region. Experimentally, the value is 0.4 for the PV and 1.4 for the SSCSV. At 150% axial chord, the computational results have a lower loss core with a value of 0.8 and 1 for the upper core, whereas the experimental data has 0.6 for both the upper and lower cores. While the magnitudes of the CFD contours differ from the experiment, the shapes are similar.
Figure 42. L2F-EF flow feature and Ptloss comparison

Figure 43 shows the comparison of Q and total pressure loss contours between the RANS CFD results for the L2F and L2F-EF. At 70% axial chord, the reduction in the size of the passage vortex is immediately clear in addition to the contours being much smaller. At 95% axial chord, the loss core associated with the passage vortex is smaller in size and magnitude for the L2F-EF, further illustrating the reduction in PV intensity caused by the EF 3D contour. At 150% axial chord, the contours and levels are similar, with the L2F-EF contours being slightly larger and the upper loss core being 1 instead of 0.8. This indicates a stronger SSCSV similar to the experimental results.
5.3 Discussion

The RANS CFD results for the L2F-EF geometry predicted the reduction in magnitude of the negative secondary vorticity, confirming the experimental results that the 3D contour reduces the strength of the passage vortex. The comparison between the spanwise distributions of the total pressure loss coefficients for the L2F and L2F-EF reveals several interesting flow features. First, the general behavior of the L2F-EF distribution relative to the L2F distribution for the RANS CFD results is similar to the experimental results. Second, the underpredicted losses in the endwall region are the largest contributor to the underprediction of the overall loss coefficient, due to the limited ability of experimental velocity measurements close to the endwall, this difference may not be significant. The results for this investigation show that RANS and the k-kl-ω 3-equation model can qualitatively predict the performance improvement and changes in endwall flow structures due to a 3D profile contour and is therefore useful as a design tool.
6 UNSTEADY ANALYSIS

6.1 Introduction

Using computational fluid dynamics to analyze loss development could be very useful as a design tool, allowing the LPT designer to predict how a geometry change such as a 3D profile or endwall contour will affect the LPT performance. This would allow a variety of different geometries to be examined, with the best candidates selected for experimental study. Recently researchers at AFRL have identified two possible flow field terms which correlate accurately to overall LPT performance but don’t require a significant amount of experimental data to be acquired. These terms are the deformation work and the total pressure transport. Unfortunately, both of these terms require knowledge of the fluctuating velocity components, necessitating the use of unsteady rather than steady Reynolds-Averaged Navier Stokes equations for CFD modeling.

The ability to analyze loss production and development throughout the passage of a LPT was analyzed by Bear. It was determined that the suction side corner separation vortex is a major driver of total pressure loss, more so than the passage vortex. As the shed vortex forms at the trailing edge of the blade, it contributes to loss development aft of the trailing edge. He also found that deformation work/turbulence production and total pressure transport are useful for predicting loss as they align more closely with contours of total pressure loss coefficient than the turbulence intensity or the component of Reynolds’ shear stress normal to the flow direction.
Arko\textsuperscript{26} analyzed the 2D vortex shedding process on several blade profiles, including the L2F, L1M, and the Pack B, finding that a vortex structure sheds from the blade suction surface when the integrated pitchwise pressure force reaches a local minimum after becoming periodic. Arko\textsuperscript{26} also examined the effect of three dimensionality on the 2D vortex shedding process of the Pack B at a Reynolds number of 25K, creating a 3D mesh with a spanwise extent of 20% axial chord. This was created using the same methodology as the 2D meshes. Using the MPAKN turbulence model, Arko\textsuperscript{26} found that the 3D case resulted in higher unsteadiness and a larger separation region than predicted by the 2D case.

### 6.2 3D Unsteady RANS

Two-dimensional unsteady simulations were carried out at a Reynolds number of 25K by Keadle and McQuilling\textsuperscript{8} using the Cradle CFD model, Tetra. As part of this research, a time step sensitivity study was accomplished. The results of this study were used since the same CFD model is being used in this study. In addition, the steady 3D grid sensitivity results previously reported were used for the URANS results.

In order to provide an initial understanding of the ability of URANS to predict turbulence production and total pressure transport, an investigation was carried out for the three-dimensional L2F profile using the laminar kinetic energy model (3-equation) of Walters and Cokljat\textsuperscript{9}, as this turbulence model provided the best steady flow results. Due to concerns regarding the computational time required for an unsteady solution of a mesh with 21 million elements, a mesh containing 7 million elements was used instead. Since the resolution of the fluctuating velocity components is highly dependent on having both an adequately refined grid and a sufficiently small time step, the timestep used was 4.7619
$\mu s$, which is slightly smaller than that used by Keadle and McQuilling\textsuperscript{8}. This ensured that the CFL number remained low and any small time scale unsteadiness would be resolved.

The highest unsteadiness is expected in the endwall region. Therefore, several grid points in the endwall region were used to obtain time traces of the unsteady velocity components. These points are located at 50\%, 70\%, and 85\% axial chord. The exact point locations are provided in the Appendix. For examination, plots of the $U$ velocity at points 1 and 7 are provided in Figure 44. The unsteady $U$ velocity achieves a steady state value without any unsteady fluctuating component. This result is consistent with all of the velocities recorded and examined. The obvious conclusion is that at a Re\# of 100K that the flow field does not have any measureable unsteadiness. This is surprising and was checked by a URANS calculation at a Reynolds number of 25K for both 2D and 3D geometries, which did have unsteady content.
The lack of unsteadiness predicted has an effect on the prediction of turbulence production/deformation work and total pressure transport, as these calculations include fluctuating velocity components. Figure 45 shows comparisons between experimental data and computational predictions for deformation work with contours of total pressure loss coefficient, while Figure 46 does the same for total pressure transport. Deformation work is calculated by Equation 12, while total pressure transport is calculated by Equation 13.

\[ \Psi_{ij} = \overline{u_i u_j} \frac{\partial u_i}{\partial x_j} \]  

(12)

\[ \dot{P}_t \approx \frac{dP_t}{dt} \approx -U_i \frac{\partial \overline{u_i u_j}}{\partial x_j} \]  

(13)
The unsteady solution predicted the fluctuating components of velocity to be extremely small, leading to quantities such as deformation work and total pressure transport that rely on these components to be extremely close to zero. The deformation work is shown in Figure 45, where the entire computational flowfield is predicted to have zero...
deformation work in the secondary direction. In Figure 46 for the total pressure transport this is again the result.

**Figure 46.** Total pressure transport comparison
7 CONCLUSIONS

The ability of a 3D commercial Reynolds-Averaged Navier-Stokes flow solver to resolve flow features and predict performance of a high lift low pressure turbine cascade was examined. Three different turbulence models (k-\(\varepsilon\), SST k-\(\omega\), and k-\(k_l\)-\(\omega\)) were used. The effect of boundary layer thickness on the flowfield was investigated. The flowfield and total pressure loss predicted by the solver was compared to experimental data for a baseline high lift LPT blade profile and for a new 3D high efficiency blade profile. An initial investigation into the use of unsteady Reynolds-Averaged Navier-Stokes for prediction of turbulence production and total pressure transport was accomplished.

Using three different inlet boundary layer thicknesses and three different turbulence models, the sensitivity of the flowfield to boundary layer thickness was examined. It was determined that the primary effect of increasing boundary layer thickness was to increase the spanwise location of the vertical flow features. This is important since it can affect the location of total pressure loss cores, therefore affecting performance prediction. With changes in inlet boundary layer thickness, the general vertical flow structure did not significantly change. It only resulted in a shifting away from the endwall region for the vortical structures.

The ability of RANS CFD to predict total pressure loss performance downstream of the blade was investigated. The k-\(\varepsilon\) and SST k-\(\omega\) models overpredicted separation losses as well as the two main loss cores, while the k-\(k_l\)-\(\omega\) model accurately predicted separation losses and performed much better with regards to the main vertical loss cores. The
differences in total pressure loss performance of the models was found to be consistent with the differences predicted in the flow features, particularly the passage vortex and the suction side corner separation vortex, where the k-kl-ω model provide the best agreement with the available experimental data.

The ability of the k-kl-ω model to predict a performance improvement provided by the 3D EF profile contour was examined. The model accurately predicted the qualitative improvement provided by the profile design change. The absolute value of the blade performance was not predicted as accurately as the trend. The performance improvement was shown to be a result of a weakened passage vortex, matching available experimental research.

An initial analysis of the ability of URANS to predict turbulence production and total pressure transport was carried out. For a 100k Reynolds number, the solver did not predict any unsteadiness in the flow, leading to nearly zero predictions in turbulence production and total pressure transport. This could be due to a need for a more refined mesh to capture unsteadiness associated with small length scales. It is recommended that this finding be investigated in greater depth as time did not permit this to be accomplished for this thesis.

The results show that RANS when coupled with the k-kl-ω turbulence model can be very useful as a design tool for low pressure turbine blades. The model does an excellent job predicting total pressure loss performance and endwall flow features compared to the k-ε and SST k-ω turbulence models, and can predict the qualitative improvement in performance and the change in flow structures provided by a 3D profile contour such as
the EF. It was shown that RANS CFD with the k-kl-ω turbulence model is capable of resolving the endwall flow features.
**8 APPENDIX**

A.1 Time Trace Point Locations

<table>
<thead>
<tr>
<th>X/AxChord</th>
<th>Y/Pitch</th>
<th>Z/Span</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.5</td>
<td>0.916808</td>
</tr>
<tr>
<td>2</td>
<td>0.5</td>
<td>-0.2789117</td>
</tr>
<tr>
<td>3</td>
<td>0.5</td>
<td>0.8222252</td>
</tr>
<tr>
<td>4</td>
<td>0.7</td>
<td>0.7034594</td>
</tr>
<tr>
<td>5</td>
<td>0.7</td>
<td>0.7978272</td>
</tr>
<tr>
<td>6</td>
<td>0.7</td>
<td>0.6051148</td>
</tr>
<tr>
<td>7</td>
<td>0.7</td>
<td>0.7034594</td>
</tr>
<tr>
<td>8</td>
<td>0.7</td>
<td>0.7978272</td>
</tr>
<tr>
<td>9</td>
<td>0.7</td>
<td>0.6051148</td>
</tr>
<tr>
<td>10</td>
<td>0.7</td>
<td>0.7034594</td>
</tr>
<tr>
<td>11</td>
<td>0.7</td>
<td>0.7978272</td>
</tr>
<tr>
<td>12</td>
<td>0.7</td>
<td>0.6051148</td>
</tr>
<tr>
<td>13</td>
<td>0.5</td>
<td>0.916808</td>
</tr>
<tr>
<td>14</td>
<td>0.5</td>
<td>-0.2789117</td>
</tr>
<tr>
<td>15</td>
<td>0.5</td>
<td>0.8222252</td>
</tr>
<tr>
<td>16</td>
<td>0.85</td>
<td>0.7034594</td>
</tr>
<tr>
<td>17</td>
<td>0.85</td>
<td>0.7978272</td>
</tr>
<tr>
<td>18</td>
<td>0.85</td>
<td>0.6051148</td>
</tr>
<tr>
<td>19</td>
<td>0.85</td>
<td>0.7034594</td>
</tr>
<tr>
<td>20</td>
<td>0.85</td>
<td>0.7978272</td>
</tr>
<tr>
<td>21</td>
<td>0.85</td>
<td>0.6051148</td>
</tr>
<tr>
<td>22</td>
<td>0.85</td>
<td>0.7034594</td>
</tr>
<tr>
<td>23</td>
<td>0.85</td>
<td>0.7978272</td>
</tr>
<tr>
<td>24</td>
<td>0.85</td>
<td>0.6051148</td>
</tr>
</tbody>
</table>

**Table 3.** Point locations for URANS time traces
Figure 47. Point locations
### A.2 Model Coefficients

These are the coefficients of the turbulence models used in this work. These are the default coefficients implemented in Cradle Software’s SC/Tetra program.

**k-ε**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_{\mu}$</td>
<td>0.09</td>
</tr>
<tr>
<td>$\sigma_k$</td>
<td>1</td>
</tr>
<tr>
<td>$C_1$</td>
<td>1.44</td>
</tr>
<tr>
<td>$\sigma_\varepsilon$</td>
<td>1.3</td>
</tr>
<tr>
<td>$C_2$</td>
<td>1.92</td>
</tr>
</tbody>
</table>

**SST k-ω**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$CMYU$</td>
<td>0.09</td>
</tr>
<tr>
<td>$\sigma_{k1}$</td>
<td>1.18</td>
</tr>
<tr>
<td>$\sigma_{\omega2}$</td>
<td>1.17</td>
</tr>
<tr>
<td>$\beta_1$</td>
<td>0.075</td>
</tr>
<tr>
<td>$\sigma_{k2}$</td>
<td>1</td>
</tr>
<tr>
<td>$\sigma_{\omega1}$</td>
<td>2</td>
</tr>
<tr>
<td>$\beta_2$</td>
<td>0.0828</td>
</tr>
</tbody>
</table>

**k-kl-ω**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_0$</td>
<td>4.04</td>
</tr>
<tr>
<td>$C_{INT}$</td>
<td>0.75</td>
</tr>
<tr>
<td>$C_{\omega1}$</td>
<td>0.44</td>
</tr>
<tr>
<td>$A_s$</td>
<td>2.12</td>
</tr>
<tr>
<td>$C_{TS,crit}$</td>
<td>1000</td>
</tr>
<tr>
<td>$C_{\omega2}$</td>
<td>0.92</td>
</tr>
<tr>
<td>$A_{\nu}$</td>
<td>6.75</td>
</tr>
<tr>
<td>$C_{R,NAT}$</td>
<td>0.02</td>
</tr>
<tr>
<td>$C_{\omega3}$</td>
<td>0.3</td>
</tr>
<tr>
<td>$A_{BP}$</td>
<td>0.6</td>
</tr>
<tr>
<td>$C_{11}$</td>
<td>$3.4 \times 10^{-6}$</td>
</tr>
<tr>
<td>$C_{\omega R}$</td>
<td>1.5</td>
</tr>
<tr>
<td>$A_{NAT}$</td>
<td>200</td>
</tr>
<tr>
<td>$C_{12}$</td>
<td>$1.0 \times 10^{-10}$</td>
</tr>
<tr>
<td>$C_A$</td>
<td>2.495</td>
</tr>
<tr>
<td>$A_{TS}$</td>
<td>200</td>
</tr>
<tr>
<td>$C_R$</td>
<td>0.12</td>
</tr>
<tr>
<td>$C_{\mu,\text{std}}$</td>
<td>0.09</td>
</tr>
<tr>
<td>$C_{BP,crit}$</td>
<td>1.2</td>
</tr>
<tr>
<td>$C_{SS}$</td>
<td>1.5</td>
</tr>
<tr>
<td>$\sigma_k$</td>
<td>1</td>
</tr>
<tr>
<td>$C_{NC}$</td>
<td>0.1</td>
</tr>
<tr>
<td>$C_{\lambda1}$</td>
<td>4360</td>
</tr>
<tr>
<td>$\sigma_\varepsilon$</td>
<td>1.17</td>
</tr>
<tr>
<td>$C_{NAT,crit}$</td>
<td>1250</td>
</tr>
</tbody>
</table>
A.3 Zero FSTI Examination

The effect of setting the freestream turbulence intensity (FSTI) to zero on the flowfield predictions of the k-kl-ω model was examined. The turbulent viscosity ratio was kept at the value used for previous cases. It is obvious from Figures 48 and 49 that specifying the experimental FSTI is essential to convergence and accurate predictions.

![Zero FSTI Convergence](image)

**Figure 48.** Convergence history for zero FSTI case

![FSTI flowfield comparison](image)

**Figure 49.** FSTI flowfield comparison
REFERENCES


