Design Optimization of a Non-Axisymmetric Endwall Contour for a High-Lift Low Pressure Turbine Blade

Jacob Allen Dickel

Wright State University

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

Part of the Mechanical Engineering Commons

Repository Citation
https://corescholar.libraries.wright.edu/etd_all/2016

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.
DESIGN OPTIMIZATION OF A NON-AXISYMMETRIC ENDWALL CONTOUR
FOR A HIGH-LIFT LOW PRESSURE TURBINE BLADE

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

By

JACOB ALLEN DICKEL
B.S.M.E., Wright State University, 2017

2018
Wright State University
The views expressed in this article are those of the author and do not reflect the official policy or position of the United States Air Force, Department of Defense, or the U.S. Government.
I HEREBY RECOMMEND THAT THE THESIS PREPARED UNDER MY SUPERVISION BY Jacob Allen Dickel ENTITLED Design Optimization of a Non-Axisymmetric Endwall Contour for a High-Lift Low Pressure Turbine Blade BE ACCEPTED IN PARTIAL FULLFILLMENT OF THE REQUIREMENT FOR DEGREE OF Master of Science in Mechanical Engineering.

Mitch Wolff, Ph.D.
Thesis Director

Joseph C. Slater, Ph.D., P.E.
Chair, Department of Mechanical and Materials Engineering

Committee on final examination:

Mitch Wolff, Ph.D.

Christopher Marks, Ph.D.

Rolf Sondergaard, Ph.D., P.E.

Barry Milligan, Ph.D.
Interim Dean of the Graduate School
ABSTRACT


Various approaches have been used to shape the geometry at the junction of the endwall and the blade profile in high-lift low-pressure turbine passages in order to reduce the endwall losses. This thesis will detail the workflow to produce an optimized non-axisymmetric endwall contour design for a front-loaded high-lift research turbine profile. Validation of the workflow was performed and included a baseline planar and test contour case for a future optimization study. Endwall contours were defined using a series of Bezier curves across the passage to create a smooth surface. A parametric based approach was used to develop the test contour shape with a goal of directing incoming endwall flow at the leading edge towards the suction side of the blade. A commercial RANS flow solver was used to model the flow through the passage. The test contour performance was measured in a low-speed linear cascade wind tunnel to verify that the numerical tools adequately captured the necessary endwall flow physics. The numerical model showed excellent agreement of total pressure loss and endwall flow structure compared with experimental measurements. Utilizing the validated workflow, the grid size, mesh deformation method, and commercial RANS flow solver, previously determined to be adequate, were used to optimize the endwall and gave confidence that the optimized
contour would perform well experimentally. A genetic algorithm was used to optimize the endwall and to improve the total pressure loss characteristics. Experimental measurements for the final optimized endwall were obtained in the low-speed wind tunnel. Comparisons between the planar endwall, test case endwall, and optimized endwall shapes were made to show how different shapes affect the flowfield. The test case endwall was found to reduce the losses associated with the passage vortex, while the optimized endwall reduced losses associated with the suction side corner separation vortex.
# Table of Contents

1  Introduction ........................................................................................................... 1

1.1 Motivation ............................................................................................................ 1

1.2 Basic Flow Physics ............................................................................................... 2

1.3 Flow Control Methods ......................................................................................... 6

1.4 Optimization Methods ......................................................................................... 10

1.5 Experimental Facility ......................................................................................... 12

2  Planar L2F Flow Modeling ..................................................................................... 16

2.1 Flow Solver and Grid Independence .................................................................... 17

2.2 Experimental Comparison .................................................................................. 20

3  Design Methodology .............................................................................................. 23

3.1 Design Methodology .......................................................................................... 23

3.2 Endwall Design Tool ........................................................................................ 24

3.3 Optimization Method ......................................................................................... 27

4  L2F-EWC Test ....................................................................................................... 33

4.1 150% Cx Analysis .............................................................................................. 34

5  Optimization of Non-axisymmetric Endwall Contour ........................................... 42

5.1 Optimization .................................................................................................... 42

5.2 Computational Results ......................................................................................... 47

5.3 Experimental Verification .................................................................................. 52
5.4 Design Variation Experiments ................................................................. 55

6 Conclusions ........................................................................................................ 57

6.1 Recommendations for Future Work ............................................................. 59

References ............................................................................................................. 61
List of Figures

FIGURE 1-1. BASIC LPT ENDWALL FLOW STRUCTURES FROM LITERATURE (SHARMA, 1987).................................................3

FIGURE 1-2. LPT BLADE LOADING FOR THE L2F GEOMETRY COMPARED WITH OTHER RESEARCH PROFILES (LYALL 2012) ........4

FIGURE 1-3. TRAILING EDGE VIEW OF Q-CRITERION ISOSURFACES (Q=10) COLORED BY AXIAL VORTICITY FROM ILES OF FLOW THROUGH THE PASSAGE AT RE = 100,000. (BARE, 2017).................................................................5

FIGURE 1-4 SUCTION SIDE BLOWING HOLES USED BY BENTON AT AL. (2014)..............................................................7

FIGURE 1-5 L2F-EF PROFILE CONTOUR USED TO REDUCE THE STAGGER ANGLE CLOSE TO THE ENDWALL (LYALL, 2012).........8

FIGURE 1-6. LSWT TEST SECTION..........................................................................................................................12

FIGURE 1-7. ENDWALL SPLITTER PLATE DESIGN FOR PLANAR AND CONTOURED ENDWALLS.................................13

FIGURE 2-1. WAND GENERATED PLANAR GRID WITH a.) BOUNDARY CONDITIONS B.) MESH TYPE ......................................17

FIGURE 2-2 GRID INDEPENDENCE STUDY, GRID INDEPENDENCE DECLARED AT 770,000 POINTS ...............................18

FIGURE 2-3 SIMULATION RMS RESIDUALS SHOWING NUMERICAL CONVERGENCE FOR GRID AT MESH SIZE OF 770,000 POINTS. ........................................................................................................................................19

FIGURE 2-4 EXPERIMENTAL VS RANS TOTAL PRESSURE LOSS COMPARISON.................................................................20

FIGURE 2-5 RANS Q=5 ISOSURFACES FLOODED BY AXIAL VORTICITY ........................................................................21

FIGURE 3-2. BEZIER CURVES WITH CONTROL POINTS ...................................................................................................25

FIGURE 3-1. DESIGN GRID FOR ENDWALL CONTAINING SIX CONTROL CURVES...........................................................25

FIGURE 3-3. TRANSFORMED MESH GRID WITH CONTOUR APPLIED (DESIGN SPACE) ..................................................26

FIGURE 3-4. ORIGINAL MESH GRID WITH CONTOUR APPLIED (PHYSICAL SPACE) ..........................................................26

FIGURE 3-5. 5-BIT AND 6-BIT BINARY STRING EXAMPLE ...................................................................................................28

FIGURE 3-6 SAMPLE BINARY STRING DEFINING A NON-AXISYMMETRIC ENDWALL CONTOUR.........................................28

FIGURE 3-7 UNIFORM, SINGLE-POINT, AND TWO-POINT CROSSOVER EXAMPLES............................................................29

FIGURE 3-8 WEIGHTED ROULETTE WHEEL USED IN GENETIC ALGORITHM’S SELECTION FUNCTION ..........................30

FIGURE 3-9. EXAMPLE OF A NON-PHYSICAL BEZIER CURVE FOR ENDWALL CONTOURING...........................................31

FIGURE 4-1 TEST NON-AXISYMMETRIC ENDWALL CONTOUR WITH INLET FLOW DIRECTION. A HILL IS ON THE PRESSURE SIDE OF THE ENDWALL WITH A PEAK NEAR THE LEADING EDGE AND A VALLEY NEAR THE SUCTION SIDE.............................................33
FIGURE 5-10. EXPERIMENTAL MEASUREMENTS OF TOTAL PRESSURE LOSS COEFFICIENT THROUGH THE EXIT AND DOWNSTREAM OF THE PASSAGE FOR THE A.) PLANAR AND B.) CONTOURED ENDWALLS. ................................................................. 52

FIGURE 5-11. PITCHWISE AVERAGED PRESSURE LOSS DISTRIBUTION ................................................................. 53

FIGURE 5-12. 150% C. EXPERIMENTAL PRESSURE LOSS COMPARISON A.) PLANAR B.) OPTIMIZED CONTOUR ............. 54

FIGURE 5-13. EWC1 TOTAL PRESSURE LOSS COEFFICIENT THROUGH THE EXIT AND DOWNSTREAM OF THE PASSAGE FOR A.) RANS AND B.) EXPERIMENTAL. ................................................................. 55
List of Tables

TABLE 1-1. LINEAR CASCADE PROPERTIES ........................................................................................................13
Acknowledgments

This work could not have been completed without the support and guidance of my colleagues, friends and family. I would like to thank those at the Air Force Research Laboratory and Wright State University for their technical expertise and knowledgeable advice. I would also like to acknowledge my friends and family for accepting my sporadic work hours as well as continuously promoting success in all areas of my life.
# Nomenclature

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AFRL</td>
<td>Air Force Research Laboratory</td>
</tr>
<tr>
<td>AR</td>
<td>Aspect Ratio</td>
</tr>
<tr>
<td>B</td>
<td>Bezier control curve</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Pressure Coefficient, $C_p = \frac{p_s - p_{s, in}}{0.5 \rho U^2}$</td>
</tr>
<tr>
<td>CV</td>
<td>Corner Vortex</td>
</tr>
<tr>
<td>$C_x$</td>
<td>Axial Chord, [in]</td>
</tr>
<tr>
<td>FSTI</td>
<td>Free Stream Turbulence Intensity, $FSTI = \frac{u'}{U}$</td>
</tr>
<tr>
<td>GTE</td>
<td>Gas Turbine Engine</td>
</tr>
<tr>
<td>$H$</td>
<td>Span [in]</td>
</tr>
<tr>
<td>HPT</td>
<td>High Pressure Turbine</td>
</tr>
<tr>
<td>ILES</td>
<td>Implicit Large Eddy Simulation</td>
</tr>
<tr>
<td>LHS</td>
<td>Latin Hypercube Sampling</td>
</tr>
<tr>
<td>LPT</td>
<td>Low Pressure Turbine</td>
</tr>
<tr>
<td>LSWT</td>
<td>Low Speed Wind Tunnel</td>
</tr>
<tr>
<td>$P_t$</td>
<td>Total Pressure [Pa]</td>
</tr>
<tr>
<td>P</td>
<td>Bezier Control Point</td>
</tr>
<tr>
<td>PV</td>
<td>Passage Vortex</td>
</tr>
<tr>
<td>Q</td>
<td>Q-Criterion</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier Stokes</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynold’s Number, $Re = \frac{u_{\infty} c}{\nu}$</td>
</tr>
</tbody>
</table>
S  Pitch [in]
SKE  Secondary Kinetic Energy
SPIV  Stereographic Particle Image Velocimetry
SSCSV  Suction Side Corner Separation Vortex
SV  Shed Vortex
T  Temperature [K]
t  Bezier independent variable
TDAAS  Turbine Design and Analysis System
u  velocity vector [m/s]
U  Velocity [m/s]
\bar{U}  Mean Velocity [m/s]
V  Velocity [m/s]
x  Axial Direction [in]
y  Pitchwise Direction
z  Spanwise Direction [in]
Z_w  Zweifel Loading Coefficient

Greek
\delta_{99}  Boundary Layer Thickness [in]
\gamma  Total Pressure Loss Coefficient, \frac{P_{t,in}-P_{t,out}}{\frac{1}{2} \rho U_{in,st}^2}
\lambda  Transformed Spanwise Direction
\xi  Transformed Pitch Direction
\rho  Density, [kg/m^3]
\[ \chi \]
Transformed Chord Direction

\[ \omega \]
Vorticity [\text{s}] 

**Subscripts**

- \( \text{a} \)  Ambien
- \( \text{in} \)  Inlet Condition
- \( \text{i, j, k} \)  Tensor Directions
- \( \text{out} \)  Outlet Condition
- \( \text{ps} \)  area-averaged
- \( \text{t} \)  pitchwise-averaged
1 Introduction

1.1 Motivation

The Low Pressure Turbine (LPT) is a vital component in modern multi-spool aero-engines. In a high-bypass engine, the LPT accounts for a third of the overall engine weight, and through the LPT-driven fan, powers 80% of the engine thrust (Howell, 2002). Improvements to the LPT can significantly contribute to improvements in the overall engine performance and weight. Current research is aimed at improving overall aircraft efficiency by reducing the weight of the LPT while maintaining performance. This weight reduction is achieved by increasing the aerodynamic loading levels on individual LPT blades and increasing pitch spacing, allowing for fewer blades per blade row while extracting the same amount of power. The increase in aerodynamic loading is directly related to how highly curved the airfoils are, resulting in airfoils that can have stronger adverse pressure gradients on the suction surface. The stronger adverse pressure gradient causes the stall Reynolds number to increase, which negatively affects engine performance.

An additional characteristic of the LPT component related to weight is the blades significantly contribute to the weight of the LPT both through their own weight and the weight of the disk structure required to support them.

Since LPT airfoils typically have a large aspect ratio (AR), most research focus had been in the midspan region of the blade. Superior high-lift blade designs that perform well at low Reynolds numbers have been designed by using front-loaded pressure distributions,
but performance in the endwall region was neglected for midspan design purposes (McQuilling, 2007). Increasing the loading levels can lead to unacceptably high endwall losses that need to be mitigated for implementing high lift blades in an engine. In order to mitigate the endwall losses in high lift blades, an understanding is needed of the flow physics generating these losses.

The structure of this thesis will be in three parts. The first will focus on the planar case verifying that the computational tools are adequate for use by comparing the computational results to previous experimental results. The second will investigate the methodology of designing a non-axisymmetric endwall contour using a parametric design approach. This will compare the predicted computational results with experimental measurements for confidence that the design methodology will work in an optimization routine. The third will cover genetic algorithm-based optimization and the contour obtained. The focus area is predominantly on the fluid phenomena exhibited through the design process.

1.2 Basic Flow Physics

The turbine flow field had been examined by several previous researchers and investigated approaches to reduce losses in the flow field (e.g. Langston 2001). Several vortical structures are present in both conventional lift and high lift airfoils. Common to most junction flows, an incoming boundary layer (BL) approaches the blunt leading edge of an object creating an adverse pressure gradient. The BL then separates and becomes entrained in what is called a horseshoe vortex (HV). For the LPT, the HV wraps around the leading edge of the airfoil and its two legs are commonly denoted by the respective half of the passage which the vortex resides (i.e. pressure side (PS) and suction side (SS)).
passage vortex (PV), which is an extension of the PS leg of the HV which had been strengthened by the passage pressure field, is the largest vortex when judged by the amount of affected fluid flow as it migrates across the passage. In contrast, the SS leg of the HV hugs the SS of the blade as it traverses from the leading edge (LE) to trailing edge (TE) of the blade. As shown in Figure 1-1, the PS and SS legs of the HV interact with each other in the downstream half of the passage, producing loss. This loss is produced predominantly through shear and the entrainment of high-speed flow inside of the vortices. Another common vortex for turbine blades is the shed vortex at the TE. This vortex forms in the wake of the blade due to the spanwise change of the blade circulation (Sharma, 1987).

This thesis will focus on the flow field of AFRL’s LPT research profile, the L2F. The L2F is a front-loaded, high-lift, airfoil with the same design gas angles as the 1990’s state-of-the-art Pratt & Whitney Pack B research airfoil at 35 and 58.5 degrees for the inlet and exit angles, respectively (Schmitz, 2010). The L2F has a Zweifel loading level of 1.59

*Figure 1-1. Basic LPT Endwall Flow Structures from Literature (Sharma, 1987)*
and is compared with other pressure coefficient ($C_p$) distributions for other research profiles in Figure 1-2. The peak loading location is about 25% axial chord resulting in a more gradual pressure recovery along the remainder of the blade suction surface. At the midspan location, the L2F had superior performance at low Reynolds numbers according to McQuilling (2007). Substantially high endwall losses, however, make the L2F undesirable for immediate usage in the LPT.

Several recent investigations performed by Sangston et al. (2014), Marks et al. (2016), and Bear et al. (2016) of high-lift LPT endwall aerodynamics were accomplished using the L2F geometry in a low-speed linear cascade wind tunnel. An accompany Implicit Large Eddy Simulation (ILES) was developed and described in Gross et al. (2017). The experiments and simulation together provide a very detailed understanding of the endwall flow structures and loss generation through the passage. This knowledge of the L2F endwall loss development is used in this thesis to develop a non-axisymmetric endwall contour.

![Figure 1-2. LPT blade loading for the L2F geometry compared with other research profiles (Lyall 2012)](image-url)
The Q-criterion is a method of vortex identification that compares vorticity magnitude and shear strain rate (Holmén, 2012). The equation for Q-criterion is

\[ Q = \frac{1}{8} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)^2 - \frac{1}{4} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)^2. \]  

(1.1)

Positive values of Q-criterion occur in regions where rotation is dominant, and vortices are present. Figure 1-3 shows isosurfaces of Q-criterion from the ILES of the flow through an L2F blade passage. This is a view of the TE from downstream looking upstream towards the suction side trailing edge. The isosurfaces show the dominant vortical structures within the passage of the L2F. The isosurfaces in Figure 1-3 have been colored by axial vorticity to indicate the rotational direction. Equation 1.2 defines the vorticity vector,

\[ \omega = \nabla \times V. \]  

(1.2)

The color blue reflects a clockwise (relative to the view) or negative rotation in the axial direction. The red isosurfaces represent counterclockwise or positive rotation flow in the
axial direction. Arrows have been placed over the isosurfaces to represent the general rotational direction.

The ILES results show several vortical endwall structures through the passage. Some of these structures are the same found in LPT literature, and others are not common in literature and may be unique to highly loaded LPTs. The PV starts as one leg of the Horseshoe Vortex, and then migrates across the passage under the influence of the passage pressure field and interacts with a strong corner separation originating on the suction side as a rotational region developing near the pressure minimum and extending toward the trailing edge. The vortical structure has the same direction of rotation as the PV and is referred to as the suction side corner separation vortex (SSCSV). Because of their common rotation, the interaction between the PV and the SCSV generates significant mixing losses. The flow in the interaction zone between the structures is in opposite directions, slowing the flow down. This shear causes the structures to grow larger and climb up the SS blade surface, thereby affecting more flow within the passage, thus generating more losses. The shed vortex (SV) is formed in the wake of the blade and is common in turbine flows. The small corner vortex (CV) is formed by the lift off of the PV and SCSV from the endwall. Prior research discusses how each of these flow structures generate losses in more detail (i.e Bear, 2017).

1.3 Flow Control Methods

Two broad methods are commonly used to affect endwall losses inside a LPT: passive and active flow control. Passive techniques are modifications to the blade and/or endwall geometries to reduce passage loss values, and are fixed. Active flow control techniques include momentum addition or subtraction along the endwall and blade surfaces
to manipulate the flow and reduce losses. Each have their respective pros and cons when implemented in an engine.

Adding blowing in a passage requires the air to come from somewhere in the system, normally from the high-pressure compressor before it enters the combustor. This results in improved efficiency in the turbine, but potentially reduces overall engine efficiency as work was done on that bleed air to compress it. An advantage of active flow control is that it can adapt to different operating conditions to minimize the negative system impact. The amount of momentum can be increased or decreased based on the need of the current operating condition.

Benton et al. (2014) used suction surface blowing on a front loaded blade to reduce the interaction of the PV with the SS (Figure 1-4). This resulted in a very large reduction in passage total pressure loss (~40%). Using unsteady jets and fewer holes reduced the amount of mass flow required to manipulate the flow. The resulting change in the passage is that the PV is pushed away from the SS.

![Figure 1-4 Suction side blowing holes used by Benton et al. (2014).](image)

Passive flow control is the preferred method as the benefits are directly added to the overall engine performance and don’t require any form of momentum or energy input. The negative characteristic of passive flow control techniques is that they are designed for
a single operating condition and are very hard to adapt to off-design conditions as they are normally built into the geometry of the blade row. Passive flow control for the endwall region can take the form of an endwall fence, a profile contour, or a non-axisymmetric endwall contour. Chung et al. (1991) implemented an endwall fence in a turbine passage to improve film cooling effectiveness. In their baseline configuration, the secondary flows would pull the cool air away from the suction surface of the blade and reduce any effects of film cooling. The addition of the endwall fence increased the film cooling effectiveness and decreased the amount of total pressure loss for the turbine row (Chung, 1991). Lyall et al. (2014) showed that the endwall losses of a front-loaded high-lift blade can be reduced by contouring the blade shape near the endwall with a low stagger angle profile (Figure 1-5). Reducing the stagger angle near the endwall reduced the strength of the PV.

Profile contouring is limited to changing the blade profile, while a powerful method at reducing losses, does not utilize a big portion of the available design space in a turbine passage. Another approach is to shape the endwall between the blades rather than just near the blades themselves. Adding strategic hills and valleys inside the passage to direct the flow is referred to as non-axisymmetric endwall contouring. Non-axisymmetric endwall contouring has been shown to reduce losses inside of a turbine passage (Praisner, 2013; Knezevici, 2009).

![Figure 1-5 L2F-EF profile contour used to reduce the stagger angle close to the endwall (Lyall, 2012)](image)
Praisner et al. (2013) performed a study on high-lift ($Z_w=1.4$) and conventional-lift blades with and without non-axisymmetric endwall contouring to compare the benefits of using endwall contouring. Ultimately, the study was an attempt to reduce the losses of the high-lift blades to the same level of the conventional lift blade (Praisner, 2013). The endwall contours were produced in an optimization routine using sequential quadratic programming with the design goal of reducing row total pressure loss. They found non-axisymmetric contouring had less benefit on an aft-loaded blade compared to a front-loaded blade. The front-loaded blade had a reduction in row losses of 12% predicted and 13.3% experimentally for an AR of 2.7, Reynolds number of $1.26 \times 10^5$, BL thickness of 16% of the span, and a FSTI of 4% (Knezevici, 2009). The conventional lift blade had amplitudes for the peak hill and valleys at 7.1% and 4.7% axial chord, respectively (Knezevici, 2008). The front-loaded, high-lift airfoil had a more extreme contoured shape compared to the aft-loaded airfoil.

Other studies that use non-axisymmetric endwall contouring with conventionally loaded blades show varying results. Researchers have focused on reducing the cross-passage pressure gradient and secondary kinetic energy (SKE) and have gained a significant reduction in loss (Harvey, 2000; Yan, 1999). Ingram et al. (2004) focused on reducing SKE and had drastically reduced the SKE with little to no row loss reductions.

The studies mentioned above show that both blade profiling and endwall contouring are methods capable of reducing endwall losses through high-lift blade passages. Both a better fundamental understanding of how these contouring approaches reduce loss, accurate numerical design tools are required to move forward with improved high-lift turbine designs.
1.4 Optimization Methods

Optimization routines are commonly used in the design of new products as they allow the designer to specify certain limits of the design space and desired goals, and the computer explores the design space, presenting one or more “optimal” solutions. The selection of an optimization method requires detailed knowledge about the problem being studied and the strengths and weaknesses of the different optimization methods available.

Design of experiments methods such as $2^k$ factorial design and response surface methods use a statistical analysis of the design cases to make a prediction on where to generate new designs (Arora, 2004). These methods allow a designer to gain a deep knowledge of how each design variable affects the problem at hand. Once an analysis has been completed, the data for each design case can be used to produce an equation relating the design parameters, which can be used to generate new designs. This equation is normally a first- or a second-order polynomial and the approach is not well suited for non-linear optimization problems, which are typical in fluid dynamics (Simpson, 2001).

Kriging is similar to response surfaces, except it chooses new members based on the predicted result and estimate of the amount of variance at that design point. This method is beneficial as it explores the design space in an efficient manner. It will not choose a design which has a poor predicted result, nor does it waste a design case calculation in a region already well known. Kriging uses a response surface portion of the model that creates a “global” model of the design space, and a “localized” portion which accounts for individual sample points for refining a model (Simpson, 2001). Downsides of this method are the initial setup of the routine and the computational complexity of fitting a kriging model.
Generational methods, like genetic algorithms and particle swarms, are commonly used for complex optimization problems. A particle swarm calculates a “velocity” that drives design cases from a previous generation to a newer generation with a different distribution in the design space. Genetic algorithms obtain an optimum by determining the fitness of a design case and keeping the best genetic material from each generation to populate the next. Genetic algorithms calculate the fitness value for a generation member and weights the probability that a member will pass on its genetic material. The genetic material is passed on similar to the way genes are passed in nature (Arora, 2004). There is a crossover between two parents and then there is an allowance for mutation. Genetic algorithms can be encoded in different ways, such as integer-encoded and binary-encoded. Johnson et al. (2012) successfully performed a film-cooling optimization using a binary-encoded genetic algorithm. The downside of genetic algorithms are that they are very computationally expensive since many design cases need to be evaluated for functionality. Another downside is the need to start off with a diverse initial population. Without a diverse initial population, the optimization would likely only find a local minimum or maximum and not the desired global extrema. The upsides of a genetic algorithm are that they are easily implemented, have been shown to reliably find a global optimum in n-dimensional optimization problems, and provide insight on design parameters by showing which are important and which are not.
1.5 Experimental Facility

ARFL’s Low Speed Wind Tunnel (LSWT) Facility was used for experimental verification of the numerical prediction tools. The low-speed linear cascade test section (Figure 1-6) is configured with seven L2F blades. A turbulence grid is installed upstream of the test section to increase the FSTI to 3.1%. The test section includes a splitter plate which creates an artificial endwall with a clean controllable incoming BL. The splitter plate surrounds the blades forming the measurement endwall and extends upstream and downstream of the blade row. The splitter plate configuration used in the experiments resulted in a BL thickness at 1.5 Cx upstream of 2.24% span (9.3% Cx) at the nominal Reynolds (Re) number of 1.0 x 10^5. Here Re is based on inlet velocity and axial chord. The blades have a 6-inch axial chord, pitch/axial chord spacing of 1.221, and an AR of 4.17. Table 1-1 summarizes these quantities.

The contoured endwalls were printed out of plastic using an additive manufacturing technique. They were created in several pieces so that the endwall could be reconfigured in three of the six passages as a flat or contoured endwall without removing the blades to replace the endwall inserts. The part labeled “splitter plate” in Figure 1-7 is the base endwall with the valley (concave) regions removed. The “filler piece” fits inside the concave region of the endwall and is inserted to create the flat endwall configuration. The
third “positive piece” is used to create the raised region of the contour. Both setups for a planar endwall and contoured endwall are shown in Figure 1-7.

![positive piece and filler piece diagram]

<table>
<thead>
<tr>
<th>Table 1-1. Linear Cascade Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td># of Blades</td>
</tr>
<tr>
<td>Axial Chord (Cₓ)</td>
</tr>
<tr>
<td>Pitch/Axial Chord (S/Cₓ)</td>
</tr>
<tr>
<td>Span/Axial Chord (H/Cₓ)</td>
</tr>
<tr>
<td>Reₓ</td>
</tr>
<tr>
<td>Boundary Layer Thickness at 1.5 Cₓ upstream (δ₉₉%)</td>
</tr>
<tr>
<td>Free Stream Turbulence Intensity (FSTI)</td>
</tr>
</tbody>
</table>

Since the LSWT is a linear cascade and not a true turbine disk, a check for flow periodicity is necessary as each passage and blade should experience similar flow conditions to minimize measurement errors. Similar flow conditions are needed as only one passage in the center of the test section is used for in-depth measurements. This is done to save experimental time, and since periodicity is checked for each setup, the other passages are expected to behave similarly.

Experimental data acquisition was performed using LabVIEW in conjunction with a NI PXI-1052 chassis with NI SCXI-1303 and NI SCXI-1305 data acquisition and filtering cards. Several 0-1 inch H₂O AllSensor transducers were used to measure the total pressure loss between the total port on an upstream pitot static probe and downstream Kiel probes in a custom five probe rake downstream of the trailing edges. A single inline Kiel probe
was used for in-passage total pressure loss measurements. All pressure transducers were calibrated with a Ruska 7200 low pressure calibrator. Measurements were taken at 2kHz for 3.5 seconds and averaged for a single pressure value. A Velmex traverse was used for positioning the downstream Kiel probes in the measurement grid. The total pressure loss coefficient is calculated at each measurement point using

\[
\gamma = \frac{P_{t,in} - P_{t,out}}{\frac{1}{2} \rho U_{in}^2}.
\]  

(2.1)

Area averaged passage total pressure loss coefficient values were calculated in a measurement plane by first integrating in the pitchwise direction, and then in the spanwise direction. The pitchwise and spanwise integrals were non-dimensionalized by the pitch and span lengths respectively.

A 0-0.4 inch H₂O Druck pressure transducer connected to a pitot static probe two axial chords upstream was used to measure the incoming dynamic pressure. The incoming dynamic pressure was used to set the tunnel velocity and used in the denominator of Equation 2.1.

Low repetition rate stereographic particle image velocimetry (SPIV) was used to provide non-intrusive velocity measurements inside a plane in the passage. Velocity data is used to provide numerical values for quantities such as Q-criterion and vorticity used in vortex detection methods described earlier. SPIV is predominately used to show how the PV is moved or dispersed inside the passage.

SPIV measurements used two 5 MegaPixel sCMOS cameras and a 200mJ, 532nm double pulsed Evergreen laser controlled by LaVision’s Davis software. A Concept Smoke ViCount smoke generator was used with a mineral oil-based fluid to create particles on the order of 1 micron in diameter. Scheimpflug adapters along with a narrowband 532nm band
pass optical filter were used to focus on the laser sheet and reduce noise. Data was averaged by using 2000 double-framed images recorded at 15 Hz in the measurement plane for statistically converged results. It was found that 2000 images provided statistical independence and were processed using a single pass 64x64 interrogation window followed by two passages of a 32x32 pixel interrogation window size. Velocity uncertainties were on the order of ±1%.
2 Planar L2F Flow Modeling

Verification of computational tools is a vital step for design. A computational model needs to be grid independent and also capture the physical phenomena being investigated. Grid independence is verified by increasing the mesh resolution until the changes in the computational solution between progressively refined solutions reduces to an acceptable level.

Depending on the type of solver used, the computational solution could be different from experimental data. The difference between a higher fidelity solver and a lower fidelity solver is based on how the physics within the problem are modeled and the assumptions used in the governing equations. Higher fidelity computational solvers model more of the flow physics potentially providing a deeper understanding of the problem and fill in gaps experimental data does not provide.

The problem with higher order computational tools is the computational cost per simulation. This is because they require more grid points due to the higher-order discretization methods resulting in more complex mathematical tasks for solution. The individual equations require more inputs and as there are more points to calculate, more time is needed for convergence. A computational solver that is “good enough” is more useful to a designer so long as enough of the physical phenomena can be captured, as they take less time and computational power when compared to a high-fidelity solver. Intimate knowledge of every design case is unnecessary in the design process if the extra information is not useful for creating a better design. This chapter explores a Reynold’s
Averaged Navier-Stokes (RANS) solver as a viable option for the design of a non-axisymmetric endwall contour.

2.1 Flow Solver and Grid Independence

The Aerodynamic Solutions, Inc. mesh generator code WAND was used for creating the original grid. Figure 2-1 is an example of the grid generated by WAND and the different boundary conditions used in the simulation. One full pitch of the linear cascade was modeled. The grid was a structured OHH grid where the O-grid wraps around the blade surface, one H-grid is used in the majority of the passage, and an additional H-grid extends upstream to the inlet and downstream to the exit.

The flow solver used is Aerodynamic Solutions, Inc. solver LEO (Ni, 1999). Code LEO is a 2nd order accurate in space, compressible, finite volume RANS solver that employs a cell-vertex discretization scheme. Time integration was focused on steady state, which uses convergence acceleration by employing local time stepping and multi-grid techniques. Wilcox’s k-ω turbulence model was used for closure. Since this study is for low Mach number flows, a preconditioner included in the code was used to reduce stiffness.
in the RANS equations and accelerate the convergence. LEO requires a binary restart file, which WAND outputs with the mesh, and a flow solver file, which contains information such as iteration number and type of solver to use (e.g., steady or unsteady).

A mesh convergence study is shown in Figure 2-2. A grid independent solution was determined by comparing area-averaged total pressure loss, $\gamma_{ps}$, for each grid size with the previous grid’s value for $\gamma_{ps}$. Once $\Delta \gamma_{ps}$ between grid sizes was less than 0.5%, grid independence was declared. Further mesh increases show diminishing benefits. Grid independence was obtained at 770,000 points and used for this study due to accuracy of results and quick simulation run times. The grid independent solution was compared to both experimental and ILES data to verify the solution convergence.

Simulations were each run for a total of 4000 iterations at which point the residual density change approached zero ($<1E^{-5}$), which was used as the convergence criteria. Figure 2-3 shows the root mean square (RMS) for the residuals used to quantify the
convergence for the grid size of 770,000 points. Figure 2-3 shows the residuals falling below 1E-4 after about 2000 iterations. Most of the residuals fell to around 1E-5 after 4000 iterations.

Inputs for the solver were chosen to best match the experimental values inside the low speed wind tunnel. The Reynolds number was matched to the experiments (\( \text{Re}_{\text{Cx}} = 100k \)); however, the Mach number was increased slightly to 0.15 to improve convergence. This was not expected to significantly change the computational solution since it is still well into the incompressible regime. A turbulence intensity of 3.1% and a length scale of 0.01 inches were used. The FSTI matched the experiment, but the length scale was varied to produce a better agreement in total pressure losses with experiment. The length scale was found to best match experimental measurements at a sufficiently low value. Too high of a value resulted in very erroneous results but having too low of a value had negligible effects.
2.2 Experimental Comparison

Figure 2-4 shows both the 150% $C_x$ total pressure loss coefficient obtained using the RANS code compared to the experimental total pressure loss and the pitchwise averaged total pressure loss coefficient up to 50% span. Figure 2-4 shows the CFD results have a narrower wake compared to the experiment but had a higher peak within the loss region indicating somewhat reduced mixing. The CFD also had a higher 2-D loss compared to the experiment. Overall, the RANS code over predicted the passage total pressure loss coefficient by 13.8% compared to the experimental measurements on the plane shown in Figure 2-4. While a large difference, this study focused on changes to the endwall flow structures by implementing a non-axisymmetric endwall contour and not exact representations of the flow field.

Figure 2-5 shows Q-criterion isosurfaces flooded by axial vorticity in a similar view as was shown in Figure 1-3. The major vortical flow structures shown in the ILES simulation are captured by the RANS code. The rotational directions are consistent...
between the two as well as the locations of each. The PV, SSCSV, and the SV in both the RANS and the ILES have very similar rotational directions and locations.

The RANS solver was used previously for profile contouring and showed that changes to the endwall could be captured (Lyall, 2012). Finer resolution of the endwall features to match the experimental and ILES results can be achieved with the solver at the cost of more computational resources. The flow field in the passage is very complex. A RANS-based solver is only capable of capturing a certain amount of the physics for a problem. Previous studies showed the unsteady characteristics inside the passage (Veley, 2018). The steady-state RANS solver simplifies the unsteadiness resulting in different mixing within the passage and increasing the loss values. Turbulence models are mathematical equations representing complicated physics. These equations attempt to provide closure to the simplified RANS equations. Turbulence models are based on flat-plate experiments to simplify the turbulence within a flow field. A LPT passage does not

---

*Figure 2-5 RANS Q=5 Isosurfaces Flooded by axial vorticity*
have the exact flow characteristics as a flat plate and results in small errors. These errors are acceptable for design due to the computational cost savings turbulence models provide. The general trends for the different flow features are captured indicating that the RANS solver should be sufficient for further investigation with endwall contouring.
3 Design Methodology

3.1 Design Methodology

Optimizing LPT turbines requires a robust system which can create and process LPT blade rows with many design parameters. The Turbine Design and Analysis System (TDAAS) has been developed by AFRL to design turbine profiles and geometry. In addition, 3-D RANS analysis is performed using the Aerodynamic Solutions, Inc. solver LEO (Ni, 1999). TDAAS was used to generate the L2F profile as discussed in McQuilling (2007). The LEO code was used to model the endwall flow through a passage and develop a non-axisymmetric contoured design.

A module has been developed to enhance TDAAS to provide the ability to modify endwall shapes. Once a non-axisymmetric endwall contour can be produced by the module, this module can be used to explore the design space and control parameters. Manually changing some of the design parameters and running some test cases gives a working knowledge to the designer for which parameters will impact the shape generation the most.

A non-axisymmetric endwall contour was designed with the aforementioned tools to verify that the process could be used to develop an endwall contour with reduced losses. The contour was designed for the L2F profile at baseline flow conditions of Re_{Cx} = 100k and AR = 4.17. The contoured endwall design was produced using a manual gradient based approach utilizing total pressure loss in the downstream plane as a cost function. The intent of the shape was to force low-speed flow near the LE of the endwall towards the SS of the
blade before it enters the passage. The intent was to decrease the size and strength of the PS leg of the HV while strengthening the SS leg of the HV. The PS leg of the HV develops into the PV, so the interaction downstream in the passage with the SCSV and SS would be weakened by weakening the PV. The best endwall shape tried was fabricated and tested in the low-speed linear cascade wind tunnel for experimental validation. Validating a contour experimentally provides confidence that the design tools are accurately capturing the flow phenomena occurring and therefore can be used to produce an optimized non-axisymmetric endwall contour with an optimization routine.

After verifying that the design tools capture the flow phenomena adequately, a genetic algorithm method similar to the HPT vane film-cooling optimization accomplished by Johnson et al. (2012) was used to produce an optimized non-axisymmetric endwall contour. The genetic algorithm is used to reduce any bias from the designer that is imposed in the aforementioned parametric study. This optimized endwall contour was investigated both computationally and experimentally for changes in the fluid flow near the endwall region.

3.2 Endwall Design Tool

A tool was created to generate contoured endwall surfaces based on an initial (baseline) planar (or radial) endwall. The planar endwalls were generated by WAND. The contoured endwalls for this study were generated using a design grid with a small number of control curves. The steps to create the non-axisymmetric endwall contour for this study are shown in Figure 3-2 through Figure 3-4. Figure 3-2 is an example design grid for this study with six pitchwise control curves. This design grid is the foundation on which a non-axisymmetric endwall contour is built upon. A series of Bezier curves (shown in Figure
3-1) are used to define the contoured endwall shape. These Bezier curves act as the control curves in the design grid. There are a total of six planes shown in the design grid, but only the four interior planes are used to generate a contoured endwall. The two at the inlet and exit of the passage are held at a zero spanwise height. The general equation for a cubic Bezier curve is

\[ B = (1 - t)^3 P_0 + 3(1 - t)^2 t P_1 + 3(1 - t) t^2 P_2 + t^3 P_3, \quad 0 \leq t \leq 1. \quad (3.1) \]

Where \( P_i \) in Equation 3.1 are the control points for the Bezier curve, \( t \) is a parametric independent variable ranging from zero to one, and \( B \) contains the coordinates for the curve itself. This vector equation defines both the pitchwise (\( \xi \)) and spanwise (\( \lambda \)) coordinates in the design grid. The spacing between the control curves defines the axial coordinates of the curves. The main objective of the design grid is to specify the respective height (spanwise values) for the grid used in the flow solver. Cubic Bezier curves were used based on maximizing the possible contour shapes while minimalizing the amount of variables to do so.

For added flexibility to the design space, two Bezier curves are used per axial plane. This allowed localized influence on the endwall shape at more points across the pitch. Each Bezier curve smoothly transitioned into another by holding the slope between two curves.

![Figure 3-2. Design Grid for Endwall containing six control curves.](image1)

![Figure 3-1. Bezier Curves with Control Points](image2)
the same. This slope was dependent on the two respective control points on either side of the connection point.

Once the height is specified on the design grid, the mesh grid is transformed to the coordinate system of the design grid. This conversion from real coordinates to the design space is straightforward since one needs only to transform the real coordinates into a rectangular grid. Once both the design grid and mesh grid are in the same coordinate system, a cubic interpolation is performed to apply the contour from the design grid to the mesh grid points (Figure 3-3). Finally, the mesh grid is converted back into the real coordinates and the non-axisymmetric endwall contour is obtained (Figure 3-4).

Figure 3-4 shows a design contour applied to the original mesh coordinates. This contour is now ready to be imposed on the baseline planar mesh. An AFRL developed mesh morphing tool (MORPH) is used to deform the grid independent planar endwall of the baseline 3-D mesh generated by WAND into the contoured surface (Kaszynski, 2015). The MORPH code uses a spring analogy to deform the mesh resulting in a high-quality deformed mesh with minimal skewing of cells (Kaszynski, 2015). MORPH calculates new nodal positions using a weighted Laplacian and then calculates a scaled Jacobian of the cells for cell quality. The scaled Jacobian ranges from -1.0 to 1.0, where zero is considered
a poor-quality cell and one is of the best quality. Negative values correspond to invalid cells, such as those with negative volume. The code iterates until the average mesh quality is above a certain threshold or until a maximum number of iterations is reached. The scaled Jacobian calculation is then used to test for valid and invalid cells. The new endwall deformation module for TDAAS allows contoured endwall surfaces to be applied to any blade and enhances the capability to accurately study the flow field within.

The next step in the workflow is to test the contour experimentally. Once the contour is tested and it is verified that the design tools are functional, an optimization can begin on the non-axisymmetric endwall contour.

3.3 Optimization Method

As described in Section 1.4, optimization methods are used to speed the search of a design space and reduce designer bias inside of a design system. The optimization method must be robust enough to both capture the best design and find it relatively quickly. The optimization method used for this thesis is a binary-encoded genetic algorithm. Binary-encoded genetic algorithms have been shown to be reliable when finding a global optimum and are able to find them within a small number of iterations (Johnson, 2014; Kekus, 2018). Since binary digits can only be either a “0” or a “1”, every additional digit used to represent a variable increases the total number of design cases by a factor of two. If only one binary digit is used, there are two design cases. If three binary digits are used, there are eight designs. The number of binary digits used in this study was 224, which results in 2.70E67 different possible design cases for an optimization.
A total of 40 variables were used in the optimization to control the shape of the endwall contour. Sixteen of the variables correspond to the pitchwise location of the different control points. In Figure 3-1, these variables correspond to the $\xi$-direction, represented by five bits each in the binary string. The twenty-four other variables correspond to the spanwise height in the $\lambda$-direction, in Figure 3-1. The spanwise control points were represented by six bits each in the binary string. Figure 3-5 shows examples of 5 bit and 6 bit binary strings that are used to define the controlled variables. A 5 bit and 6 bit binary string representation of a variable allows for 32 and 64 discrete possibilities available for optimization, respectively. The values could vary between zero and one for the pitchwise locations and from -1.5 to 1.5 for the spanwise locations, which corresponds to one pitch and ±6% H (±25% $C_x$), respectively. Higher bit counts could be used for additional discrete possible design cases, but the current bit size for each variable was declared sufficient. Increasing the number of bits per design point would also increase the complexity of the design problem. There would be diminishing returns by increasing the bit count from their current levels.

Figure 3-6 shows a portion of a binary string defining a whole contour shape. This portion shows the boundary in the string where the pitchwise and spanwise control points...
meet. The five-digit and six-digit segments represent the pitchwise and spanwise control points, respectively.

The genetic algorithm used in this study uses uniform, two point, and single point crossover. The percentages of each crossover type are 70%, 20%, and 10%, respectively. The percentages represent the probability of that crossover type being used after two parents are selected for the reproduction of the next generation. An example of each crossover method is shown in Figure 3-7. The figure shows how the same parents can produce different offspring based on the type of crossover used. Single-point crossover randomly selects one point in the genetic string. All the genetic material from each parent on either side of the point is then combined with the inverse side from the other parent to create two new offspring. Two-point crossover performs the same except there is an additional location dividing where genetic material is selected from each parent. Uniform crossover randomly selects genetic material sites from each parent and creates new offspring, allowing for the greatest diversity in reproduction.

Figure 3-7 Uniform, Single-Point, and Two-Point Crossover examples
The selection criterion is a weighted roulette wheel style and uses a mutation rate of 0.1%. An example for how a weighted roulette wheel works is shown in Figure 3-8. The percentages shown are the probabilities of choosing certain designs to become the parents of the next generation during the selection process. All the designs available for the selection process have a slice on the roulette wheel and the size of that slice is dependent on the design’s fitness value. Fitness values are used as a feedback function in genetic algorithms denoting definitive criteria for good and bad designs. The weighting of the roulette wheel for this study is based on the fitness value of total pressure loss. If a design has a small value for $\gamma_{ps}$, then that design will have a higher probability of being chosen for crossover and mutation. If a design has a high value for $\gamma_{ps}$, then that design has a smaller probability of being chosen for passing on genetic material.

Mutation occurred by randomly switching a binary digit based on the rate supplied. Mutation can occur in different ways. One way is to set a few random digits to switch per generation, thereby having a fixed mutation rate. Another way is to specify a rate which randomly applies a few mutated genes per generation, allowing for slight variations on the true mutation rate used per generation. For this thesis, each binary digit for every member in a generation was given a random number between 0-100. Mutation occurred if the random number was below the specified mutation rate. For example, if the random number

![Figure 3-8 Weighted roulette wheel used in genetic algorithm’s selection function](image-url)
for a binary digit was 5 and the mutation rate is 0.1, mutation would not occur. If the random number was 0.05 with the same mutation rate, mutation would occur. At a mutation rate of 0.1% and 224 binary digits per member, about 0.224 binary digits were changed per member. The result was about one binary digit would be changed per every four members in the generation, or about 22 per generation. As mutation was random at the specified rate, the occurrence of mutations for multiple binary digits per member was possible.

Elitism was used to accelerate optimum convergence by eliminating the weakest members in a generation and having a pool of elite optimization members for crossover and mutation. Restrictions on the Bezier control points were used to reduce the chance of a non-physical design occurring. An example non-physical design would be if a Bezier curve looped back on itself (Figure 3-9). Duplicate members were also excluded from participating in any generation, as they would provide no new information. Any duplicates were rejected, and a new member was created using the same process at the other members of the generation. Checks for duplicates were performed at the time a member was created to reduce any interference with creating a new generation.

The initial population of 100 contour designs was produced using Latin Hypercube Sampling (LHS) for a diverse set of genetic material for the algorithm. The initial generation was used in its entirety for creating the second generation in the optimization process because the most elite 100 members of the entire optimization were used to create the next generation. Generation 2 and on use the cumulative elite (best 100 members) for
crossover and mutation. This can allow optimization results to be skewed but can also accelerate convergence to a particular design. Some negative effects of elitism include the elimination of vital genetic material that results in obtaining only local extrema instead of the global extrema that is desired. This risk was taken into consideration and elitism was used with the purpose of retaining beneficial genes to accelerate convergence upon a design.

Once the optimization was completed, the optimized contour was produced using additive manufacturing and tested experimentally to give confidence that the optimization is producing physical results and can adequately capture the flow phenomena for design.
4  L2F-EWC Test

Prior to the beginning of the genetic algorithm optimization process, a parametric study was performed to test some of the design limitations and find any undesirable characteristics in the design workflow. After dozens of iterations, the best contour was fabricated and tested in the low speed linear cascade. The best case was chosen based on achieving the lowest predicted area averaged total pressure loss, ease of manufacturability, and designer intuition of likelihood of performing as designed. The general shape of the contour was designed based on successful contouring effects on the flow found in literature. Both the CFD and experimental cases are discussed in this section for the test contour shape, the L2F-EWC.

Figure 4-1 Test non-axisymmetric endwall contour with inlet flow direction. A hill is on the pressure side of the endwall with a peak near the leading edge and a valley near the suction side.
4.1 150% $C_x$ Analysis

The spanwise maximum and minimum amplitudes of the final contour for the test case are 10.9% $C_x$ and 13.5% $C_x$, respectively. Figure 4-1 shows the endwall contour and the inlet flow direction. The amplitudes for the peaks and valleys discussed in Knezevici et al. (2009) for the high-lift front-loaded blade were at 10.1% $C_x$ and 7.2% $C_x$, respectively. The magnitude of the test non-axisymmetric endwall contour peaks and valleys are slightly higher than those found in literature, but this is believed to correspond to differences in blade loading (Knezevici, 2009). The Pack DF ($Z_w=1.4$) had larger peaks and valleys when compared to the Pack B ($Z_w=1.1$) (Praisner, 2013). Following the same trend, an increase in the peak and valley magnitudes for the L2F ($Z_w=1.59$) is reasonable.

![Figure 4-2. 150% $C_x$ RANS Total Pressure Loss Comparison a.) Planar and b.) Contoured L2F-EWC](image-url)
The area averaged total pressure loss through the passage was calculated in a plane 150% C_x downstream from the leading edge. The predicted reduction in the area averaged passage total pressure loss coefficient was 8.2% for the L2F-EWC when compared with the flat endwall. This reduction in loss in a high AR passage is on the order of other studies (Praisner, 2013; Harvey, 2000). The endwall loss reduction is predicted at 22.9%.

Figure 4-2 shows RANS contours of total pressure loss at the 150% C_x plane. The wake loss is shown to be narrower and closer to the endwall in the L2F-EWC compared with the planar case. The downstream pitchwise averaged total pressure loss is plotted along the span in Figure 4-3. In the L2F-EWC, the loss core is lowered closer to the endwall. This also shows that the 2-D region of the loss wake extends further down the span. The lower loss core is also diminished in strength with an increase in loss adjacent to the endwall with the application of the L2F-EWC. Knezevici et al. (2008) proposed that keeping secondary
flow structures close to the endwall was important for reducing secondary losses and extending the 2D loss region. This contour aimed at directing the flow near the leading edge towards the suction side of the blade in an attempt to reduce the size and strength of the PV to keep most of the endwall flow near the endwall.

The experimental total pressure loss coefficient was measured in a plane 150% $C_x$ downstream of the leading edge. The measurements were made across one pitch and up to 40% span. From 40-50% span the loss was relatively constant. The area averaged passage total pressure loss was calculated and compared to the RANS simulation. The reduction in total pressure loss at 150% $C_x$ is 7.8% and the endwall loss reduction is 19.9%. This reduction in total pressure loss is very close to the CFD predicted benefits. As shown in

![Figure 4-4. 150% $C_x$, Experimental Total Pressure Loss Comparison a.) Planar and b.) Contoured L2F-EWC](image)

Figure 4-4, the experimental total pressure loss contours show the same trends that the CFD predicted in Figure 4-2. The overall loss wake is narrower and located closer to the endwall. The peak loss core in the planar case is lowered from $z/H=0.15$ to $z/H=0.125$. Although,
the passage loss is reduced with the application of non-axisymmetric endwall contouring, a region at \( z/H < 0.05 \) has an increase in loss magnitude. These results are in agreement with the findings of Knezevici, whose total pressure loss contours show a lowering of the higher loss core by a similar amount (2009). The L2F-EWC shows similar contour characteristics to the contour characteristics found in Knezevici et al. and resulted in a similar change to the flow features (2009). Directing incoming flow near the LE towards the SS effectively reduces the size and strength of the PV.

The pitchwise averaged total pressure loss plotted against the span also shows similarities to the CFD (Figure 4-3b). The upper loss core is pulled lower towards the endwall and there is a rise in loss closest to the endwall. The difference between CFD and experimental results is that lowering of the upper loss core is not the main factor contributing to the reduction in loss. The elimination of the lower loss core related to the PV contributes more towards loss reduction in the experimental case.

Further analysis using planes of Q-criterion within the CFD simulation (Figure 4-5), shows that the passage vortex is greatly reduced in strength and size. The PV is caught inside the valley of the endwall contour and does not leave the valley with any substantial size or strength. This keeps the SCSV closer to the endwall, but also increased in strength. This contrasts the observation of Knezevici et al. (2009) that the PV splits into two weaker magnitude vortices with the application of a non-axisymmetric endwall contour. In the present case, the PV does decrease in strength, but it does not split into two and the SCSV increases in strength. SPIV inside the passage (Figure 4-6) shows the movement of the PV from \( y'/C_x = 0.3 \) and \( z/C_x = 0.075 \) (Figure 4-6a) to being inside the contour valley at \( y'/C_x = 0.25 \) (Figure 4-6b). Benton et al. (2014) forced the PV away from the suction surface to
reduce losses. The PV for this case is trapped in the valley near the SS and kept from interacting with the SS.

The PV is closer to the SS inside of the valley of the contour. In the SPIV measurement plane, the SSCSV is relatively small and forms in the corner junction of the
blade endwall. The SSCSV appears to be slightly smaller in the case of the L2F-EWC. The endwall contour reduces the interaction of the PV with the SSCSV by reducing the size/strength of the PV.

Figure 4-7 shows experimental total pressure loss development for the planar and contoured endwall cases measured at the 95%, 105%, 125%, and 150% axial planes. In Figure 4-7b, the higher loss cores near the exit of the passage are increased in magnitude. However, the overall area of these loss regions is smaller and the loss region associated with the PV is weaker with the L2F-EWC. The losses in the SSCSV region are increased in strength with the L2F-EWC. The strengthening of the SSCSV and weakening of the PV has resulted in higher losses along the SS where the endwall flow interacts with the SS flow.

The CV is shown to produce more loss in the L2F-EWC than in the planar case. Knezevici proposed that the increase in loss occurs from a reduction in spanwise flow, pulling loss away from the endwall and up further into the passage (2009). While overall passage loss decreased in the case of the L2F-EWC, there is potential for further reductions in endwall losses by reducing the strength of and losses in the region of the SSCSV and CV.

Figure 4-7. Experimental measurements of total pressure loss coefficient through the exit and downstream of the passage for the a.) planar and b.) contoured endwalls.
An upstream view of Q-criterion isosurfaces in the area of the leading edge is shown in Figure 4-8. The differences in the PS and SS legs of the HV are revealed. In the planar case, the PS leg of the HV has negative vorticity and becomes the PV as it progresses across the passage. The suction side leg of the horseshoe vortex has a positive vorticity in the axial direction but dissipates quickly as it moves into the passage. The SSCSV begins in the region where the SS leg ends. When non-axisymmetric endwall contouring is applied, the SS leg of the HV extends a bit further into the passage. This is credited to more flow going towards the SS rather than the PS. The PV is substantially weaker and moves across the passage further upstream than in the case of the planar endwall. The SSCSV appears larger and stronger in the L2F-EWC.

![Iso surfaces](image)

*Figure 4-8. Leading Edge RANS $Q=5$ Isosurfaces Flooded by axial vorticity comparison*

Loss reductions from implementing non-axisymmetric endwall contouring is dependent on what AR and BL thickness is used. The L2F-EWC effectively reduces the amount of loss at the endwall, but the effects can be confounded with a low AR. At a lower AR the endwall flow has a stronger effect on the midspan region. Figure 4-9 shows a plot of percent changes for area-averaged total pressure loss in CFD by only changing the AR
and maintaining a constant BL thickness. As the AR is decreased, the percent reduction in area-averaged total pressure loss increases.

The L2F-EWC provided insight to the benefits endwall contouring provides to high lift LPT airfoils. An endwall contour was produced with the design goal of directing the flow near the leading edge towards the suction side of the blade. Reductions in area averaged total pressure loss coefficient were achieved both experimentally and computationally. The predictions were not exact matches, but the L2F-EWC gave confidence in the design workflow for use within an optimization routine.

Figure 4-9. CFD Variation of Aspect Ratio Results
5 Optimization of Non-axisymmetric Endwall Contour

5.1 Optimization

Once the computational design tools were verified with the creation of the L2F-EWC, 2500 design cases were ran over the course of 25 generations inside of a genetic algorithm optimization routine. The cost function for the optimization focused on reducing the area averaged total pressure loss coefficient.

In an attempt to reduce unnecessary bias from the design space, caution was used in the initialization of the optimization routine. Figure 5-1 shows the respective mean spanwise displacement for the endwall of generations 1, 8, 17, and 25. Also shown are the

![Figure 5-1 Generational mean and best contour shape for generation 1 (a, e), 8 (b, f), and 25 (d, h). This shows an unbiased first generation with contour convergence over generations](image-url)
best contour shape for the four generations. The mean for generation one is near zero everywhere, which is desirable since it indicates there was no major bias induced in the first generation. The first generation’s best case resulted in an increase in total pressure loss when compared to the baseline case. Additional generations with respective mean and best contours show the optimization progression. Figure 5-2a shows the spanwise height standard deviation of the first generation of the optimization process. The locations of the Bezier curves are prominent, having the highest standard deviation values. The points closest to the blades, which have a direct relationship to the Bezier control points, exhibit the highest standard deviations within the passage. These are also the regions where there is the most control over the contour shape. The inlet and exit to the passage are fixed transition regions allowing the flat endwall outside the passage to smoothly become contoured inside the passage. This region experiences the smallest amount of change and has the smallest standard deviation in the passage throughout the generations. Using additional control curves would result in more control over the contour shape. Based on the calculated standard deviation of the passage, the current number of six control curves (four non-zero curves) inside the passage provides sufficient control of the contour shape.

Figure 5-2 Spanwise height standard deviation for generations 1, 8, 17, and 25 to show regions where the optimization routine is changing the most.
Later generations show the lowering of standard deviation inside the passage as the population narrows into a subset of the design space and the optimal contour is better approximated. The regions where the standard deviation remains high in the later generations highlight which regions that the optimization algorithm is manipulating to improve the cost function further. In generation 17 (Figure 5-2c), the optimization algorithm is attempting to converge upon the height near the pressure side of the blade along with a region slightly off the suction side of the blade 40%-60% axial chord downstream.

Figure 5-3 shows a plot of the loss values for every member of each generation of the optimization process. As the optimization process progressed, the average passage loss of each generation decreased, and the performance of the best member improved. The variance in loss values also decreased as weaker genes are discarded from the population and better fitting genes were retained. Towards the end of the optimization, the weight on
the roulette selection wheel was increased to accelerate convergence on the best fitting design. This is shown by the last generation having very little variation in their loss values.

Figure 5-4 shows the generation averages with bars of one standard deviation. As the generations progress, the standard deviation bar becomes smaller. This figure also shows the average of the elite population and respective standard deviation bar. The bars on the elite become smaller more quickly than the generation average, and the elite average value steadily decreases through the optimization process. The cumulative best member is also plotted along with the baseline planar case for reference.

The decision to increase the weight on the roulette wheel was justified based on the cumulative best members in Figure 5-4. Between generations 14 and 19 there was almost no change to the best member of the optimization. The elite average and generation average were steadily decreasing, but the cumulative best member was not. Increased weight on the

![Figure 5-4 Generation average, elite, and cumulative best member with bars of one standard deviation.](image)
roulette wheel selection allowed the cumulative best to decrease again. It is a common practice to alter some parameters in a genetic algorithm to promote convergence during an optimization. One method is to use an elite population, which is employed in this study. Another is to increase the mutation rate to cause more diversity in the gene pool (Arora, 2004). Lastly, changing the weight that a fitness value has in the selection process is used to choose a higher fitting member more often for reproduction. This method is what is used starting in generation 19 to accelerate convergence since elitism was being used and the mutation rate was causing sufficient design variations at the current level.

Figure 5-5 shows the generational average $\gamma_t$ plotted against the span with error bars to show locations with the highest variance. The locations with the highest variance were locations that the optimization was still refining in that generation. Locations with very small pitchwise averaged standard deviation bars were not affected in that generation by the optimization. This information was beneficial as the 2D region was virtually unaffected during the optimization process. The focus was predominantly in the endwall loss where
the standard deviation magnitude was significant. The lowering of the standard deviation shows regions where the optimization was affecting and not affecting the loss values. This information can be used with the mean shape to give insight on how a particular shape will affect a certain flow feature. As the generations progress, the regions of loss associated with the SSCSV and PV continued to have the highest amount of variance.

5.2 Computational Results

Figure 5-6 shows the contour shape (L2F-EWC1) the optimizer converged upon. Characteristics of the contour are hills 1 and 2 (H1 and H2) and valleys 1 and 2 (V1 V2). This shape was similar to the non-axisymmetric endwall contour Praisner et al. (2013) produced on the PS, with a hill near the leading edge and a valley near the trailing edge. However, there were significant differences along the SS of the passage giving it a unique shape. Specifically, Praisner et al. (2013) has a valley near the leading edge and then no additional hill near the SS of the passage.

After the optimizer narrowed down the selected contours, the mechanisms of reducing loss were explored. The L2F-EWC1 has V1 following the SS from the LE to TE, but there was a hill that runs next to V1 almost in-line with the exit flow direction. H1 also acts in a fashion similar to the L2F-EF profile contour by locally increasing the pressure on the PS of the blade. V1 is on the SS of the blade and provides a favorable pressure gradient near the endwall on the SS in the passage, thus reducing the penetration height of loss at the exit of the passage. H2 acts through a similar mechanism to what has been described as an endwall fence. H2 inhibits the crossflow inside the passage that feeds the PV and keeps the PV away from the SS. This was similar to what Benton et al. (2014) achieved by blowing along the SS. Separating the PV away from the SSCSV reduces the
amount of shear that the two vortices experience, thereby reducing the amount of loss production. V2 locally reduces the pressure gradient at the trailing edge of the blade and provides a lower pressure which helps pull the PV away from the SS towards the exit of the passage. The minimum and maximum amplitudes for the contour shape are 22.5% and 14.2% axial chord respectively. These values are within the optimization bounds of ±25% axial chord.

![Diagram of optimized non-axisymmetric endwall contour](image)

*Figure 5-6 Optimized non-axisymmetric endwall contour with different topology labels for the pressure side hill and valley (H1 and V2) and suction side hill and valley (H2 and V1).*

The basic analysis of the contour for the optimization looked at area-averaged total pressure loss. The predicted reduction in area-averaged total pressure loss for this contour shape was 10.6%. This was better than the L2F-EWC discussed in Section 4. Figure 5-7a shows the loss wake for the L2F-EWC1. A substantial change in the shape of this loss contour was seen when compared to the planar case and the L2F-EWC. The upper loss core, coinciding with the SSCSV, was much narrower in the EWC1 than the planar case, almost as narrow as the 2D region above it. There was also a circular loss region close to the endwall. There was a significant reduction in size and loss for the upper loss core,
whereas, there was an increase in the loss attributed to the lower loss core. This was easily seen in Figure 5-7b, which compares the optimized contour with the baseline planar loss distribution.

The EWC1 effectively reduced the upper loss core, but strengthened the lower one. This was different than the EWC case because the test case spatially lowered the upper loss core and significantly reduced the magnitude of the lower loss core.

Further investigation of the flow domain resulted in Figure 5-8, which has streamtraces originating upstream at BL heights of 1.8% and 18% BL thickness. The flow inside the BL was used for this analysis because this flow participates in the loss generation after the separation line inside a turbine passage. The contour levels are shown for spatial reference as this was a top view of a three-dimensional flow. The first BL height (1.8% of

Figure 5-7. 150% axial chord a.) total pressure loss contour plot b.) with pitchwise averaged total pressure loss comparing the baseline to the optimized contour.
the BL) plotted shows very similar behavior between the planar and contoured cases. After
the separation line, the flow was trapped inside one location near the SS and was captured
inside the SSCSV.

![Diagram](image.png)

*Figure 5-8. Streamtraces originating at 1.8% a.) contoured b.) planar and 18% c.)
contoured d.) planar, of the inlet BL height.*

The next BL height for streamline injection was 18% boundary layer thickness. This shows the most significant change between the planar and the contoured case. The planar case shows the flow being captured in the PV. The contoured case uses H2 to split
the flow, effectively separating the PV from interacting with the SSCSV. This allows the
SSCSV to be fed some of the flow from the 17.8% boundary layer thickness height while
reducing the amount of flow being fed into the PV. There was also little to no interaction
between the two vortices which contributed a large amount of loss due to shear in the planar
case. The reduction of the vortical interaction has been achieved in a previous study by
Benton et al. (2014) in his SS blowing experiment. This experiment forcibly moved the PV away from the suction surface greatly reducing the amount of loss in the passage.

The two contours attack different loss producing mechanisms. The L2F-EWC case attacked the PV as the major loss production but neglected the SSCSV. The optimized L2F-EWC1 attacked mainly the interaction between the SSCSV and PV. It did not reduce the loss associated with the PV to the same degree as the L2F-EWC, but it significantly reduced the loss from the SSCSV. An additional measure that could be added to the EWC1 case could focus on reducing the size and strength of the remaining PV to reduce the loss further.

Figure 5-9 compares Q-Criterion slices within the passage for the optimized L2F-EWC1 and the baseline planar case. The baseline planar case has the two distinct flow features previously discussed: the PV and SSCSV. The EWC1 case has one distinct region of rotation in a similar location to the planar case PV. For simplicity, it too has been denoted PV in Figure 5-9. The SSCSV region was not apparent in the contoured case. The PV in the EWC1 was shown to climb over H2 as it moves through the passage, but was prevented from interacting with the SS. Chung et al. showed similar interactions with the

![Figure 5-9. Slices of Q-Criterion comparing the baseline planar vortical structures with the optimized non-axisymmetric endwall contour vortical structures. The SSCSV region is reduced while the PV is pushed away from the suction surface.](image-url)
implementation of an endwall fence (1991). The reduced interaction of the PV on the SS causes the SSCSV loss region to be greatly reduced. This was the opposite of the result shown earlier with the EWC. The L2F-EWC eliminated the loss with the PV while neglecting the SSCSV; whereas, the optimized contour eliminated the SSCSV and neglected the PV. The contour shape does reduce the interaction of the PV with the SS of the blade to reduce loss and the height that the PV can climb up the span.

5.3 Experimental Verification

Numerical simulations provide key information used for the generation of new designs. This information about flow structures and regions of loss help a designer make quick and cost efficient decisions on future designs. Verification that the numerical simulations are capturing the necessary fluid phenomena was crucial for future work. Experimental verification for the optimized non-axisymmetric endwall contour is shown and discussed in this section as the planar L2F and L2F-EWC were in previous sections.

Figure 5-10 compares the planar case with the L2F-EWC1. The development of total pressure loss in the EWC1 case was drastically different than the planar case. The loss region associated with the PV was seen clearly in the EWC1 case. This was the circular loss region that slowly merges with the rest of the loss as it exits the passage. Not seen in

![Figure 5-10](image)

*Figure 5-10. Experimental measurements of total pressure loss coefficient through the exit and downstream of the passage for the a.) planar and b.) contoured endwalls.*
the optimized case was the SSCSV loss region that appears in both the planar and EWC cases. This agrees with the numerical simulation. The absence of the SSCSV reduces the PV interaction with both the SS and SSCSV, keeping the PV close to the endwall, which reduces the amount of affected flow in the passage thus reducing loss.

Figure 5-11 shows the pitchwise averaged total pressure loss for the RANS (Figure 5-11a) and experiment (Figure 5-11b), and compares the planar L2F and optimized L2F-EWC1 cases. As stated before (Section 2.2), the RANS planar case does not exactly predict the PV loss value in the experiment, but the RANS solver was shown to predict the trends contouring would induce on the flow (Section 4). The experimental measurements follow the same trends the simulation predicted. There was an increase in loss in the PV region, a drastic decrease in loss in the SSCSV region, and an increase in loss closest to the endwall. Differences occur within the SSCSV loss region. There was an additional core inside this region in the experiment not captured by the simulation.

![Figure 5-11. Pitchwise Averaged Pressure Loss Distribution](image)
A comparison of Figure 5-12 with Figure 5-7 shows that EWC1 performed as predicted, with the loss regions moved spatially. The SSCSV loss region becomes as narrow as the 2D region above it and the PV almost becomes a separate structure from the rest of the loss wake. The loss wake was a vertical line most of the way to the endwall except for the region closest to the PV, where the loss was pulled slightly towards the PV. The optimized non-axisymmetric endwall contour was extreme in its approach for loss correction and achieves an extreme change in the loss wake.

Quantifying the reduction in loss has been shown in previous sections by looking at the change in area averaged total pressure loss. The reduction achieved by the EWC1 from experiment was 8.6%. This value was not as large as the predicted amount and was thought to be attributed to the extra loss peak in the SSCSV region and the under prediction of the PV loss region for the planar case. Other reasons for the reduction could be the

![Figure 5-12. 150% C, Experimental Pressure Loss Comparison a.) Planar b.) Optimized Contour](image)
relative locations of the different vortical structures in the passage that vary slightly from the experiment to simulation.

Figure 5-13 compares the 85%, 95%, 105%, 125%, and 150% $C_x$ total pressure loss contours for the RANS and experimental optimized EWC1. Similarities are prevalent in the PV loss region. The PV stays relatively close to the endwall and does not climb up while leaving the passage. There was a loss region due to the shed vortex and suction surface BL. The most notable difference between the RANS and experimental data was the “stem” going from the endwall up to the PV region which was significantly thicker and has a relatively higher loss value in the experiment than it has in the simulation.

5.4 Design Variation Experiments

Additional experimental tests were performed replacing H1 with the L2F-EF profile contour to see the resulting difference. Replacing H1 with the L2F-EF caused the non-axisymmetric endwall contour to perform worse than with the originally designed H1. On the other hand, one could view H2 along with V1 and V2 as increasing the effect the L2F-EF had on reducing loss inside the passage. The L2F-EF has a loss reduction of around 5% and when tested with the hills and valleys of the optimized contour, achieved a 6% reduction in total pressure loss. Altering some of the design criteria for the non-
axisymmetric endwall contour to allow profile contouring in the optimization routine could produce substantially different shapes to reduce loss.

Other tests to investigate the sensitivity of the optimized contour shape included slightly altering some of the key characteristic endwall features. All changes were independent from one another for testing of feature sensitivities. One feature was the large valley on the suction side of the passage. This feature was reduced in depth by adding filler material to reduce its depth (0.5 in from lowest point), though there was still a very noticeable valley. The resulting effects on the area-averaged loss was a reduced benefit from the contour. Another alteration examined was extending the hill H2 to keep the PV away from the suction surface longer. This too resulted in a decreased benefit from the endwall contour. The EWC1 shape was extreme when compared with those in literature, but these ad-hoc experiments indicate the contour was nearly optimal as designed.
6 Conclusions

Non-axisymmetric endwall contouring is a very effective method of reducing endwall losses in LPT blades. The design methodology used to develop an optimized non-axisymmetric endwall contour consisted of verifying the tools performed well for a planar case, for the manually designed L2F-EWC, and lastly as part of an optimization routine that produced the L2F-EWC1.

A commercial RANS solver was used to model a planar high-lift LPT research profile, the L2F. The RANS solver captured the flow phenomena acceptably when compared with ILES and experimental data at a manageable computational cost. An over prediction of loss at the 150% Cx plane for total pressure loss was found, but it was concluded that the solver would be sufficient for the design of a non-axisymmetric endwall contour. This conclusion was made because the flow solver captured the major loss producing mechanisms within the passage. When compared with experimental data at the 150% Cx plane, the RANS solver over predicted area-averaged total pressure loss by 13.8%.

A design grid with Bezier control curves was used to define endwall shapes with smooth surfaces. The contoured endwalls were applied to a 3-D RANS mesh using an AFRL mesh morphing tool called MORPH. The morphing tool allowed for a high quality deformed mesh to test the performance of non-axisymmetric endwall contours. The best endwall contour was fabricated using additive manufacturing for use in the tunnel without the removal of the linear cascade blades. This provided a reliable measurement of
total pressure loss difference between the flat and the contoured endwalls. The L2F-EWC effectively decreased the strength of the PV and losses generated by it. The L2F-EWC produces this decrease in strength and loss by trapping the PV inside the valley of the contour, greatly diminishing the PV effect on the passage. The upper loss core associated with the SSCSV was brought closer to the endwall due to the reduced interaction with the PV. The SSCSV was also increased in strength for this case.

The L2F-EWC resulted in a 7.8% experimental reduction in passage total pressure losses through a low-speed linear cascade of turbine blades. The experimentally measured loss reduction was in close agreement with a numerical prediction (-8.2%). The endwall loss reductions for the RANS solver and experimental were 22.9% and 19.9% respectively. The maximum and minimum amplitudes of the endwall height variations were on the order of other researchers at 10.9% C\textsubscript{x} and 13.5% C\textsubscript{x}. For the L2F-EWC test case, it was shown that the CFD predicted the loss trends sufficiently and should work well inside of an optimization loop.

The genetic algorithm for the design cases converged after 25 generations with 100 members per generation. The initialization method for the algorithm employed Latin Hypercube Sampling for a diverse first generation. The resulting optimized contour shape (L2F-EWC1) affected the flow inside the passage differently than the L2F-EWC did. The optimized focused on the SSCSV for loss reduction.

The optimized contour shape achieves loss reduction by combining several passive flow control techniques into the non-axisymmetric endwall contour. The flow control techniques that it combines are an effective endwall fence with regards to H2, a profile contour with H1, and non-axisymmetric endwall contour with the relative locations of the
hills and valleys. The loss reduction the optimized shape achieved was 10.6% for area-averaged total pressure loss with a 39.3% endwall loss reduction. The experimental results give a loss reduction value of 8.6% for total pressure loss and drastically changes the loss wake. The endwall loss reduction was 23.5%, which was substantially lower than the predicted 39.3% but was still better than the 19.9% of the EWC. Differences in these loss values can be attributed to certain loss regions such as the PV region, which the solver under predicted the loss values for the planar case. These numbers also reflect performance in a larger AR passage with a thin BL compared to those found in literature.

The EWC1 did affect the flow inside the passage as the solver predicted. There was a reduced interaction between the PV and SSCSV. The loss associated with the SSCSV was substantially reduced in size and magnitude. There was an increase in loss for the PV region and region closest to the endwall.

The EWC1 was sensitive to shape alterations. Substantial shape changes such as replacing the hill on the PS of the blade with the L2F-EF profile contour resulted in a decrease in aerodynamic performance. Altering other features such as the SS valley and hill resulted again in decreased aerodynamic performance when compared with the “as designed” optimized contour.

6.1 Recommendations for Future Work

A recommendation for advancing the knowledge of non-axisymmetric endwall contours for high-lift blades would be to study off-design conditions. The focus for this thesis was centered on optimizing a contour for certain design conditions. Variability in real engines exists and a non-axisymmetric endwall contour needs to perform equally if not better than conventionally lifted blades. Changing the incidence angle and the flow
speeds are common tests that can be performed to investigate how robust an endwall design is. Shocks are another problem that occur within the turbine section and are neglected in the tested low-speed environment used by this study. Improving the capability of the design system to be used at transonic conditions could be beneficial for real world engine designs.

Running the optimization with higher order Bezier curves could result in a different optimized case. Orienting the curves in the streamwise direction instead of the pitchwise directions also could result in different shaping of the endwall. Another advancement of the design system would allow the capability of optimizing a profile contour with the non-axisymmetric endwall contouring.

Improvements to the optimization routine could entail running the grid independent mesh until the design cases are within a subset of the design space, and then increasing the mesh density for better refinement of the endwall shape. Discussed in Section 2.2, the lower loss core could be captured better by the RANS code with higher mesh density at the expense of computational time. Using a coarser mesh for most of the optimization allows for lower computational times, and then transitioning to a finer mesh in later generations improves the final contour’s accuracy.
References


