Steady and Harmonic Predictions for a Single-Stage Fully Cooled Turbine

A Thesis

Presented in Partial Fulfillment of the Requirements for the Degree Master of Science in the Graduate School of The Ohio State University

By

Mark Brody Wishart, B.S.

Graduate Program in Mechanical Engineering

The Ohio State University

2010

Thesis Committee:

Professor Michael G. Dunn, Advisor

Professor Charles W. Haldeman
ABSTRACT

The research described in this thesis was supported under the NASA/DoD URETI program, and is designed to advance the state-of-the-art in the area of the aerodynamic and heat-transfer performance of a single stage, high-pressure, fully cooled turbine. This particular study is an important computational component of a combined experimental and computational research program conducted at The Ohio State University Gas Turbine Laboratory (OSU GTL). The computational model enhancements included in this particular phase of the overall investigation (Using Numeca’s FINE/Turbo code to generate fully cooled predictions) are a new, highly gridded mesh, the ability of the new code to run isothermal wall solutions, a more correct representation of the experimental geometry, the introduction of film cooling, and finally, generating both steady-state and unsteady harmonic solutions. The code FINE/Turbo has been used by the GTL for several years, and has proven to be a useful predictive tool.

Several modeling details are considered when performing these computations; attention to detail is shown to be especially important when attempting to acquire heat-transfer predictions. A $y+$ sensitivity study shows that surface static-pressure predictions are relatively insensitive to $y+$, but heat-transfer predictions are very sensitive to changes in the $y+$ range. All computations utilize $y+$ values in the range of
1 to 10, which is required by the Spalart-Allmaras turbulence model. This study uses the distributed source term injection approach to model film cooling. To better understand the codes predictive capabilities in the presence of distributed source term cooling, a flat plate experimental and computational investigation was performed. This investigation determined that the minimum average error between the surface heat-transfer predictions and the experimental results is 6.0%. The results provide a "proof of concept" regarding the implementation of distributed source term injection as a method to model cooling holes, and the codes ability to make surface predictions in the presence of this cooling.

For the configuration - both geometry and instrumentation rake locations - of the turbine stage used for this work, two computational models are described; a short channel that does not model the complete experimental configuration, and a new full channel mesh that does model the complete experimental domain. It is found that incorporating the inlet channel (upstream of the vane) does impact the blade heat-transfer predictions by as much a 45%. This difference between the short and full channel heat-transfer predictions is most noticeable on the blade suction surface, below 20% span and above 70% span. Based on this difference, the full channel mesh was chosen for use in this thesis.

For the cooled computations, a method for placing the holes within the computational model, and allocating mass for each cooling row is developed. For the vane, the cooling-hole areas are used to distribute mass across the surface. For the blade, a maximum mass flow model is developed by using the choked flow rate for each row. A further enhancement to the cooled computations is the use of the FINE/Turbo spreading algorithm. For both cooled and un-cooled cases, steady and unsteady
harmonic computational results are presented. The non-linear harmonic method is implemented using 3 harmonics, and isothermal wall conditions. Finally, by relating the experimental vane inlet turbulence intensity to the Fine/Turbo model inlet kinematic turbulent viscosity, it is now possible to specify more realistic inlet turbulence intensity. For these computations, a turbulence intensity of 3.2% is used.

Results are obtained for five different cases. These cases represent variations of the following parameters; cooled vs. un-cooled; short vs. full channel; steady vs. harmonic. For each case, static pressure and surface heat-transfer predictions are presented for the vane and the blade. Comparisons between these predictions are also presented. In general, it is found that both the un-cooled and cooled static-pressure predictions demonstrated good agreement with the data. It is now considered possible to make comparisons between fully cooled computations and fully cooled experimental results. For the heat-transfer, these new predictions represent an improvement over previous predictions; however agreement with the experimental results is still not considered to be good. The most noticeable deviation between the heat-transfer predictions and the data occurs on the blade pressure surface, at approximately 50% wetted distance.
DEDICATION

To My Family
ACKNOWLEDGEMENTS

First, I would like to say that it has been a privilege to work at The Ohio State University Gas Turbine Laboratory. I can confidently say that my time at the Gas Turbine Laboratory, and The Ohio State University, has been a positive experience, and I am grateful to everyone who made this experience possible.

I would like to thank my advisor, Dr. Michael Dunn, for making this opportunity possible, but I am also grateful for his support, guidance, and encouragement during this research. I would also like to thank Dr. Charles Haldeman, for answering countless questions, and providing me with both intellectual guidance, and support, during the development of this work. I would also like to thank Dr. Randall Mathison for his support, especially in the areas of computational modeling and computing. Thank you to Mr. Issam Kheniser, who performed the experimental work for the cooled flat plate. Mr. Kheniser was gracious enough to provide information regarding the computational boundary conditions, as well as his experimental results.

Finally, thank you to Mr. Jeff Barton, Dr. Corso Padova, Dr. Igor Ilyin, Mr. Ken Copley, Mr. Ken Fout, Mrs. Cathy Mitchell, and fellow Graduate Research Assistants, for your support and friendship.
Vita

August 31, 1983 ........ Born - Pittsburgh, Pennsylvania

2006 ................ B.S. Mechanical Engineering, Lafayette College


2008 - 2010 ........ Graduate Research Assistant, The Ohio State University Gas Turbine Laboratory

Publications


Field of Study

Major Field: Mechanical Engineering
CONTENTS

Abstract ................................................................. ii
Dedication ..................................................................... v
Acknowledgements ......................................................... vi
Vita .............................................................................. vii
Table of Contents ............................................................ viii
List of Figures ................................................................. x
List of Tables ................................................................. xiii

1 Introduction ................................................................. 1
  1.1 Background .......................................................... 4
  1.2 Scope of the Current Study ......................................... 8

2 Computational Investigations Using FINE/Turbo at The OSU GTL 11
  2.1 Modeling Film Cooling ............................................. 13
  2.2 Mapping Computational Surface Predictions on Airfoil Geometry .... 15
  2.3 Mesh Cell Wall Width and $y+$ ..................................... 23
  2.4 Inlet Turbulence Intensity ........................................... 28
  2.5 Unsteady Computations using the Non Linear-Harmonic Method .... 30

3 Modeling Cooling Holes with Distributed Source Term Injection 35
  3.1 Flat-Plate Experimental Investigation ............................ 35
  3.2 Flat Plate Computational Investigation ........................... 38
  3.3 Boundary Conditions ............................................... 46
  3.4 Fully Cooled Predictions ............................................ 47

4 The Fully Cooled URETI Turbine Stage ............................... 53
  4.1 URETI Short Channel Mesh ........................................ 55
  4.2 URETI Full Channel Mesh .......................................... 56
<table>
<thead>
<tr>
<th>Chapter</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.3</td>
<td>Implementation of Film Cooling</td>
<td>58</td>
</tr>
<tr>
<td>4.4</td>
<td>Boundary Conditions</td>
<td>67</td>
</tr>
<tr>
<td>5</td>
<td><strong>Computational Results for the URETI Turbine Stage</strong></td>
<td>75</td>
</tr>
<tr>
<td>5.1</td>
<td>Comparison between the Short and Full Channel Mesh</td>
<td>76</td>
</tr>
<tr>
<td>5.2</td>
<td>Comparison between Un-cooled Steady-State and Un-cooled Harmonic</td>
<td>86</td>
</tr>
<tr>
<td>5.3</td>
<td>Comparison between Harmonic Predictions and Experimental Results</td>
<td>96</td>
</tr>
<tr>
<td>6</td>
<td><strong>Conclusion</strong></td>
<td>108</td>
</tr>
<tr>
<td></td>
<td><strong>Bibliography</strong></td>
<td>116</td>
</tr>
</tbody>
</table>
List of Figures

1.1 Image of a cooled single stage turbine section (vane and blade). Taken from Rolls Royce 1992 ................................................. 3
1.2 Computational Work at the OSU GTL. Taken from Mathison et al. 2010 6

2.1 Comparison of "50% span lines" for the FINE/Turbo surface geometry and the corrected surface geometry - URETI blade pressure surface (side view) ......................................................... 17
2.2 Comparison of "50% span lines" for the FINE/Turbo surface geometry and the corrected surface geometry - URETI blade pressure surface (Top view) ......................................................... 17
2.3 Comparison between the FINE/Turbo and the new surface mesh . . . 20
2.4 Area averaging analysis for the new surface data ......................... 21
2.5 Comparison of FINE/Turbo steady-state solution for surface and corrected surface predictions at 50% constant span on the pressure surface of the URETI blade ................................................. 23
2.6 Effects of $y+$ values on the URETI blade, steady, heat-transfer predictions, at 50% span ......................................................... 25
2.7 Effects of $y+$ values on the URETI blade, steady, static-pressure predictions, at 50% span ......................................................... 25
2.8 Vane suction surface and blade pressure surface $y+$ values for the new URETI model ................................................................. 27
2.9 Comparison of Spalding’s inner-law expression with the pipe-flow data of Lindgren (1965) ......................................................... 28
2.10 Comparison between the two kinematic turbulent viscosity settings and run 13 (un-cooled) experimental results .......................... 30

3.1 The Gas Turbine Laboratory Small Calibration Facility (SCF) ........ 36
3.2 Flat plate experimental apparatus ............................................. 38
3.3 Flat plate mesh block diagram ................................................. 39
3.4 Butterfly topology with seven blocks. Image taken from Numeca IGG User Manual ................................................................. 40
3.5 Individual mesh blocks for the flat plate computational domain ........ 41
3.6 Cell distribution at the buffer block ......................................... 42
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.7</td>
<td>Flat plate surface mesh</td>
<td>42</td>
</tr>
<tr>
<td>3.8</td>
<td>Side view of the computational domain as it relates to the physical</td>
<td>43</td>
</tr>
<tr>
<td></td>
<td>geometry</td>
<td></td>
</tr>
<tr>
<td>3.9</td>
<td>Coarse grid levels. Image taken from Numeca IGG User Manual</td>
<td>44</td>
</tr>
<tr>
<td>3.10</td>
<td>Distribution of the plate cooling holes</td>
<td>46</td>
</tr>
<tr>
<td>3.11</td>
<td>$y+$ values at the surface of the un-cooled flat plate</td>
<td>49</td>
</tr>
<tr>
<td>3.12</td>
<td>$y+$ values at the surface of the cooled flat plate</td>
<td>49</td>
</tr>
<tr>
<td>3.13</td>
<td>Flat plate CFD predictions and experimental results</td>
<td>51</td>
</tr>
<tr>
<td>4.1</td>
<td>The Gas Turbine Laboratory Turbine Test Facility (TTF)</td>
<td>54</td>
</tr>
<tr>
<td>4.2</td>
<td>Blade to Blade view of the entire URET I short channel mesh (not to</td>
<td>56</td>
</tr>
<tr>
<td></td>
<td>scale)</td>
<td></td>
</tr>
<tr>
<td>4.3</td>
<td>Blade to Blade view of the entire URET I mesh &quot;rake to rake&quot; model</td>
<td>57</td>
</tr>
<tr>
<td></td>
<td>(not to scale)</td>
<td></td>
</tr>
<tr>
<td>4.4</td>
<td>Turbine stage cooling paths</td>
<td>58</td>
</tr>
<tr>
<td>4.5</td>
<td>Location of the cooling holes for the vane and blade (not to scale)</td>
<td>60</td>
</tr>
<tr>
<td>4.6</td>
<td>URET I blade surface cooling hole locations</td>
<td>61</td>
</tr>
<tr>
<td>4.7</td>
<td>URET I vane surface cooling hole locations</td>
<td>62</td>
</tr>
<tr>
<td>4.8</td>
<td>URET I blade coolant mass flow distribution using the percent area</td>
<td>65</td>
</tr>
<tr>
<td></td>
<td>method</td>
<td></td>
</tr>
<tr>
<td>4.9</td>
<td>URET I blade coolant mass flow distribution using the maximum coolant</td>
<td>65</td>
</tr>
<tr>
<td></td>
<td>method</td>
<td></td>
</tr>
<tr>
<td>4.10</td>
<td>URET I vane coolant mass flow distribution using the percent area</td>
<td>68</td>
</tr>
<tr>
<td></td>
<td>method (TE cooling is row 13)</td>
<td></td>
</tr>
<tr>
<td>4.11</td>
<td>Vane surface mesh with coolant injector spreading turned on. L to R:</td>
<td>68</td>
</tr>
<tr>
<td></td>
<td>Row TE, 12 to 1</td>
<td></td>
</tr>
<tr>
<td>4.12</td>
<td>Blade pressure surface mesh with coolant injector spreading turned</td>
<td>69</td>
</tr>
<tr>
<td></td>
<td>on. L to R: Row 4, 3, 2, 1, 8</td>
<td></td>
</tr>
<tr>
<td>4.13</td>
<td>Blade suction surface mesh with coolant injector spreading turned on.</td>
<td>69</td>
</tr>
<tr>
<td></td>
<td>L to R: Row 7, 6, 5, 4</td>
<td></td>
</tr>
<tr>
<td>4.14</td>
<td>Steady-state absolute total temperature profiles at the R/S interface</td>
<td>71</td>
</tr>
<tr>
<td>4.15</td>
<td>Harmonic time-averaged absolute total temperature profiles at the R/S</td>
<td>72</td>
</tr>
<tr>
<td></td>
<td>interface</td>
<td></td>
</tr>
<tr>
<td>4.16</td>
<td>Inlet Total Temperature Profile</td>
<td>74</td>
</tr>
<tr>
<td>5.1</td>
<td>Case 1: Heat-transfer prediction - un-cooled, steady, short channel</td>
<td>78</td>
</tr>
<tr>
<td></td>
<td>mesh with average inlet temperature</td>
<td></td>
</tr>
<tr>
<td>5.2</td>
<td>Case 1: Static-pressure prediction - un-cooled, steady, short channel</td>
<td>79</td>
</tr>
<tr>
<td></td>
<td>mesh with average inlet temperature</td>
<td></td>
</tr>
<tr>
<td>5.3</td>
<td>Case 2: Heat-transfer prediction - un-cooled, steady, full channel</td>
<td>80</td>
</tr>
<tr>
<td></td>
<td>mesh with average inlet temperature</td>
<td></td>
</tr>
<tr>
<td>5.4</td>
<td>Case 2: Static-pressure prediction - un-cooled, steady, full channel</td>
<td>81</td>
</tr>
<tr>
<td></td>
<td>mesh with average inlet temperature</td>
<td></td>
</tr>
</tbody>
</table>
List of Tables

2.1 Harmonic method $CPU_{ratio}$ with 1 perturbation ............... 34
3.1 Combination of grid points used in each of the 14 blocks (for the finest multigrid) .................................................. 44
3.2 Mass flow for the flat plate cooling rows ........................ 45
3.3 Flat plate boundary conditions ..................................... 47
3.4 Flat plate percent difference between prediction and experimental results 51
4.1 Mass Flow for URETI Vane and Blade ......................... 66
4.2 Boundary conditions for fully cooled computations ............ 74
4.3 Boundary conditions for un-cooled computations ............... 74
5.1 Computational investigation case matrix .......................... 76
Chapter 1

Introduction

As the need for advances in gas turbine engine efficiency continues, there remains a desire to increase combustor temperatures. By increasing the combustor outlet temperature, designers can realize an increase in cycle efficiency; however, this solution does not come without a cost. Elevated combustor outlet temperatures result in an increased turbine inlet temperature, which means that component temperature management must remain a paramount concern to designers. Two possible approaches to reduce this problem are advancements in turbine materials and effective component cooling. While advancements in airfoil materials, coatings, etc, are important, and over the past several decades significant advancements have been made, these advancements have not been sufficient in completely solving this problem. Within modern engines, first stage turbine temperatures are near, or in excess of the airfoil material melting point. Airfoil cooling is therefore an important element in the thermal protection of turbine components. This cooling can provide an increased margin between material temperatures and melting point, which increases the component life and durability.

Turbine blade cooling is effectively described as the bleeding of gas from the engine compressor so that it can be injected at the turbine airfoil surfaces. Cooling is realized
in two ways; surface (film) cooling and internal cooling. Film cooling is achieved by the injection of coolant at the airfoil surface acting as an insulating layer between the mainstream fluid and the airfoil surface. Figure 1.1 provides an illustration of a cooled turbine section. In this image it can be seen that cooling holes are located on both the vane and the blade. What are not shown in this image are the blade and vane internal cooling passages, which act to cool the airfoil from the inside. The coolant runs through internal passages before reaching the cooling holes. As the cool gas travels through these passages, it removes heat from the metal, helping to reduce the overall metal temperature. This cooling strategy works, but only to a certain point. Simply increasing film cooling is not a feasible solution because the coolant gas is taken from the compressor. This loss of mass in the compressor can decrease the overall cycle efficiency of the engine. Therefore, it is important to maximize the effects of cooling, while minimizing the amount of coolant used. This thesis concentrates on the external component of airfoil film cooling and not the internal component. Other ongoing programs at The Ohio State University Gas Turbine Laboratory (OSU GTL) are investigating the possible improvements in internal cooling passage designs and configurations.
One tool that is used in the development of turbine components is computational fluid dynamics (CFD) software. Such software has made great advancements over the past 30 years, both in aerodynamic and thermal predictions. Past CFD investigations have shown that CFD "Turbo" codes are capable of modeling the aerodynamics within various engine sections (fan, compressor, and turbine), and can provide engineers with realistic predictions. By including film cooling within these models an added level of complexity is introduced; however, coolant flows are present in most modern high-pressure turbines. Computational investigations that do not model coolant flows do not model the physical conditions within a turbine section. It is therefore important,
and relevant, to investigate the ability of CFD codes to predict the aero and thermal performance of fully cooled turbine components.

1.1 BACKGROUND

Short duration experimental research facilities, such as the Turbine Test Facility (TTF) at the OSU GTL have been successful in providing both aerodynamic and heat transfer datasets for full scale turbine stages operating at design corrected conditions of corrected speed, flow function, and stage pressure ratio. Such experiments provide a database against which computational fluid dynamics codes have been, and continue to be, tested and validated. The successful development of CFD over the past three decades has been closely tied to these experimental investigations. Full scale, rotating experiments have provided data that captures the inlet flow field conditions as closely as possible for controlled laboratory measurements using a turbine section. These inlet conditions include temperature profiles, free stream turbulence, and unsteady flow features. Dunn [1] provides an in-depth discussion of the relationship between the experimental research and code development. Dunn [1] argues that the development of CFD can be categorized into three stages. The first of these stages is best described as "knob turning" or the varying of the computational parameters to match known data. Early CFD efforts modeled simple geometries, under very ideal conditions. The second stage of development incorporated predictions into the development of experimental investigations. In this manner, CFD was used as a design tool, and then validated against the experimental results. Currently, CFD has reached a point in development where it is able to model complex geometries with realistic flow characteristics. At this stage, the use of CFD is seen as a critical design tool in the development of turbomachinery components.
One of the first successful codes to make time-accurate predictions of turbine aero-dynamics was the Allison Vane/Blade Interaction (VBI) code, developed in the 1980s. The VBI code was experimentally validated by Dunn et al. [2]. This code was able to solve both the Euler (inviscid) and Navier-Stokes (viscous) equations, but what VBI provided to the development of CFD was an understanding of the interactions between the vane and blade rows. In addition, it was a good demonstration of what can be accomplished when the experimental and the CFD teams work together. A second code introduced in 1991 was UNSFLO, developed by Giles [3]. This code also provided reasonable predictions of experimental results. Giles and Haimes [4] compared predictions to unsteady, inviscid flat plate cascade flows, as well as steady and unsteady, viscous flat plate cascade flows. UNSFLO also showed good agreement between the predictions and experimental results for steady turbine heat transfer. Furthermore, Abhari et al. [5] compared time-resolved heat-transfer predictions using UNSFLO to time-resolved experimental results. From this study it was found the computational and experimental results did agree. Mathison [6] provides a more in-depth discussion of CFD development, including greater detail regarding the VBI and UNSFLO codes. From the literature, it is possible to develop an understanding of how these computational advancements were concurrent with experimental findings. In short, experimental datasets were, and still are, critical to making advancements in CFD possible.
More recent computational investigations conducted at the OSU GTL have included work for multiple geometries. Figure 1.2 taken from Mathison et al. [7], provides an overview of these projects, and shows that these investigations have been broken down into cooled and un-cooled studies. This thesis represents project 1 (top right quadrant), which is a continuation the computational work initiated by Southworth [8] in 2006. This work looked at the URETI geometry, and made both steady-state and time-accurate pressure predictions. The investigation looked at two codes, FINE/Turbo and MSU/Turbo. It was found that FINE/Turbo pressure predictions performed well against MSU/Turbo, and also provided reliable pressure predictions when compared to the experimental results. Heat-transfer prediction were however not reliable. This original model was un-cooled, but Southworth et al. [9] did provide surface pressure predictions for the same geometry using the lumped mass coolant model.
In 2008, Crosh [10] performed a computational investigation, again utilizing the FINE/Turbo code to model a different turbine with a much different blade/shroud geometry. This investigation looked at a one and one-half stage turbine with 38 high pressure and low pressure vanes, and 72 high pressure blades. For the first time at the GTL, predictions using the FINE/Turbo unsteady harmonic method were obtained. Also, this study represented advancement in the laboratory’s computational abilities with FINE/Turbo because the GTL was now able to generate grids. Prior to this point, grids had been provided to the GTL, as was the case for Southworth [8]. This grid generation was far from a trivial assignment due to the fact that Crosh [10] was modeling both a smooth blade casing, and an outer blade casing with 60 circumferential perturbations. It was found that FINE/Turbo again proved to be a useful tool in the prediction of unsteady flow field aerodynamics with a single frequency (a basic turbomachinery configuration). As was true in previous investigations, FINE/Turbo predictions provided good agreement with the experimental results. Finally, the work of Mathison [6] represents a continuation of computational success at the GTL. Mathison made temperature predictions at locations off the airfoil surface for the one and one-half stage turbine. These locations included the blade hub, and instrumented locations just off the blade leading edge surface. Mathison et al. [7] provides comparisons between predictions and experimental results using FINE/Turbo and the unsteady harmonic method. This study included several model improvements as well as a better-quality grid. It was found that the use of isothermal boundary conditions at the wall improved the unsteady predictions. Before this time, isothermal boundary conditions had not been implemented with the harmonic method. The ability to perform such computations provided yet another enhancement to the GTL computational abilities.
1.2 Scope of the Current Study

This study represents the continuation of an experimental and computational research program conducted at the OSU GTL. This program was designed to investigate both aerodynamic and heat transfer performance of a single stage, high-pressure, fully cooled turbine. This research was part of the University Research, Engineering and Technology Institute (URETI) program for Aeropropulsion and Power Technology. The overall goal of the OSU portion of this program (other participants in the URETI effort were the prime contractor Georgia Tech and the other subcontractor Florida A&M University) was to provide experimental results for computational fluid dynamics code validation and model development. In previous studies, it was determined that both design codes and research codes would be utilized. The design codes included FINE/Turbo and STAR-CD, while research codes such as MSU/Turbo and NASA/Turbo were also analyzed against the data. For the current study, the goal was to advance earlier computational work performed at the GTL specific to the URETI program. These advancements included a new, highly gridded mesh, the ability to perform calculations for the isothermal wall condition, a more correct representation of the experimental geometry, the introduction of film cooling, and finally, generating both steady-state and harmonic solutions. Numeca FINE/Turbo has been used by the GTL for several years, and has proven itself a useful design tool in the prediction of the aerodynamics within a turbine stage. FINE/Turbo is a commercially available design code, and has remained the code of choice for the GTL because of past computational successes and the ease with which the code can be applied (Numeca provides very good user support when difficulties with the code are encountered).
This investigation intends to answer four basic questions regarding predictions for a fully cooled single-stage turbine. First, what details must be considered when performing a computational investigation, especially when attempting to obtain heat-transfer predictions. In addition to specific modeling questions, there are also general concerns that must be considered. Take for example the question of how to compare computational results to the experimental results when the two coordinate systems are not equivalent. The second question to be answered is, when utilizing the distributed point source cooling method for the modeling of film cooling, is the code able to make reliable surface predictions? To answer this question, a flat plate investigation was performed in order to better understand the codes ability to model cooling, and provide reliable surface predictions in the presence of cooling. A third question is how to apply this cooling model to the current URETI turbine configuration? To answer this question it is important to not only develop a method for placing the holes within the CFD model, but also develop a mass flow model for all cooling rows. Finally, it is important to ask if this new turbine model provides reliable predictions, and understand what can be learned from these new results. The following chapters address each of these questions.

To better develop answers to these questions, multiple simulations were preformed so that several parameter configurations could be modeled. These configurations represented various "improvements" to the computational model as compared to previous investigations. The parameters were as follows; increased grid density; modeling the full experimental geometry; decreased $y+$ values at the airfoil surfaces; the use of an inlet temperature profile; addition of vane and blade film cooling; the use of isothermal wall boundary condition; and finally, both steady-state and harmonic predictions. Several cases were modeled and compared in order to better understand the
effect of each parameter on the overall prediction. For comparisons between the computational predictions and the experimental results, only the harmonic results were compared because these predictions represented the best case scenario for reliable predictions.

The following discussion aims to not only show the continuation of the investigation conducted as part of the URETI research program, but to also develop a more general discussion of best practices regarding the development of computational models. Such a discussion provides insight regarding the current geometry, and is also applicable across various geometries. It is important to note that a development of best practices will translate across various software platforms, thus the results obtained by this investigation are not limited to FINE/Turbo. In short, this discussion represents a systematic evaluation of the parameters and details associated with the development of computational models for fully rotational airfoil geometries.
Chapter 2

Computational Investigations Using FINE/Turbo at The OSU GTL

The computational fluid dynamics package currently being used at the OSU GTL is FINE/Turbo, a commercially available analysis code. This code was selected for use at the GTL because of its utility as a practical "design" code. A detailed description of this code is provided by Aube and Hirsch [11], while Hirsch [12] outlines the general approach for numerical computations. The code is a Reynolds-averaged Navier-Stokes (RANS) based code, which for the purposes of this investigation, was run utilizing its ability to solve viscous (laminar and turbulent Navier-Stokes) simulations. As the name would suggest, this code is specific to turbomachinery, and can solve both a 2-D and 3-D flows with complex geometries. FINE/Turbo is an explicit solver, which uses a cell centered control volume scheme. The results are second order accurate in both the spatial and temporal domain. The solver can utilize a multigrid approach during a computation in order to speed-up the convergence of steady-state predictions. In general, the advantages to using such a code are that FINE/Turbo has been proven to work for aerodynamic predictions, both in industry, and at the GTL. It is a "commercial design" code, which is relatively easy to use and Numeca provides excellent user support. Also, FINE/Turbo combines the pre-processing (gridding), solving, and post-processing software into one platform. The only disadvantage of
using this software is that the GTL does not presently have available a source code.

Recent publications have illustrated that FINE/Turbo is a reliable predictive tool. Southworth [8] found that the predictions made with FINE/Turbo compared well with vane surface pressure measurements for a high-pressure vane of a fully cooled turbine stage. The present predictions obtained using the newer harmonic method demonstrate slightly better agreement with the experimental results for the blade than the initial predictions of Southworth [8]. Crosh [10] found good agreement between pressure predictions and experimental results for a one and one half stage turbine model. This investigation did include unsteady un-cooled predictions. In general FINE/Turbo is a tool that has been used by the GTL for several years now. With each successive computational investigation, the GTL continues to expand both its computational abilities, and its understanding of the code.

The GTL has two main computational facilities. The first is an 8 processor LINUX cluster. Each 32 bit processor has a total memory of 2 GB. A computation using a mesh of 4.5 million cells will typically run 5000 iterations in two to three days. The second facility is a 16 processor LINUX cluster. Each 32 bit processor has a total memory of 4 GB. On this system, a computation using a mesh of 4.5 million cells will typically run 5000 iterations in six to eight hours. Both facilities were utilized for this research. Past computational investigations at the GTL used only the 8 processor cluster. Due to its limited number of processors and memory per processor, the grids associated with past studies were kept small to reduce computational time, especially for time-accurate computations. Past grids were on the order of 700,000 cells, as noted by Haldeman et al. [13]. Recently, Mathison [6] and Crosh [10] used the 16 processor cluster with success. This facility has increased the possible size of new
grids for both steady-state and time-accurate computations. The increased memory also makes it possible to compute unsteady computations utilizing the non-linear harmonic method.

Using multiple processors allows for parallel processing of the computation. The basic concept is that the host processor coordinates the computation, while additional processors each handle an independent task (a portion of the total mesh). By distributing the workload among multiple processors, the overall computational time required to reach convergence is accelerated. Also, it becomes possible to run larger computations that would otherwise exhaust the resources of the single processor. The possibility for parallel processing relies on the structure of the mesh, specifically the number of mesh blocks. Blocks represent the smallest possible division of the mesh, thus the number of total processors utilized cannot exceed the number of blocks. FINE/Turbo also attempts to balances the computation evenly across all processors. This balancing can also limit the total number of processors used during a computation. It should be noted that the most efficient parallel processing may not include all available processors.

2.1 Modeling Film Cooling

There are several different methods for the implementation of film cooling within a CFD model. These methods can be broken down into four categories; distributed source term injection, lumped mass, macro model, and micro model. For this study, the distributed source term injection approach was chosen as it is the method implemented by the FINE/Turbo code. This model allows for the injection of coolant through the solid walls into the main flow. This method does not require the meshing of the cooling-hole geometry, and therefore has a low computational cost. In general,
the model does not describe in detail the cooling flow; rather it considers the effects of the coolant flow on the main flow. In FINE/Turbo the cooling hole, and mass injection, is described by a hole diameter, a geometric position on the solid surface, an injection direction and magnitude, a temperature, and turbulent viscosity.

The lumped mass model, used by Southworth et al. [9] introduces the measured coolant mass flow at the inlet/exit of the computational domain (as boundary conditions). The correct mass flow across a stage is modeled; however, the location and direction of the injection is not captured. It was found by Southworth that the performance of the distributed mass, and lumped mass models had good agreement with each other, as well as with the measured surface pressures, across most of the vanes wetted distance (until approximately 50 percent wetted distance). These results were concluded for both steady-state and time-accurate predictions. The macro model represents an enhancement to the CFD code by incorporating an injection model, which approximates the three-dimensional fluid dynamics at the cooling hole. The main hypothesis of such a model, as provided by Burdet et al. [14], is that coolant jet flow behavior near the hole exit is highly three-dimensional and contains flow features that are important when modeling coolant interactions with the mainstream flow. Abhari [15] provides the general structure for this model; however, Tafti and Yavuzkurt [16] developed the groundwork for such a model with work on a two-dimensional discrete hole prediction scheme. In general, the macro model provides a plane jet distribution of the coolant injection into the viscous boundary layer. The model accounts for the jet trajectory, penetration, mixing, and secondary flow development. Isentropic expansion of the coolant through the hole to the surface is assumed. Finally, the computational grid near the hole is packed in the streamwise direction to better resolve the coolant mixing near the injection site.
In contrast to the macro model, the micro model is a detailed gridding approach utilizing millions of cells to model into each hole. These grids model the internal passage, hole passage, and the external airfoil surface. Such models are computationally expensive as they require extensive processing power and time. Work by Leylek and Zerkle [17] have reported on results utilizing this technique. Most recently, Vitt et al. [18] conducted a study to examine the unsteadiness on film cooling effectiveness of a high-pressure turbine airfoil (using the URETI geometry). The numerical model included the gridding of the internal leading edge "impingement chamber", and the full gridding of the leading edge (showerhead) cooling holes. A tetrahedral-prism mesh was implemented to allow for the necessary grid clustering around and within the cooling holes. On the blade alone, a total of 7.7 million cells were used.

2.2 MAPPING COMPUTATIONAL SURFACE PREDICTIONS ON AIRFOIL GEOMETRY

FINE/Turbo has the capability to post-process computational results using the CFView software that is provided by Numeca. In the past, computational investigations conducted at the GTL have relied on this software for mapping of the predicted quantities onto the airfoil surface geometry. FINE/Turbo is able to do this by iterating the surface mesh solution over the known surface geometry. The resolution of the mesh is a function of the number of spanwise grid cuts, and the overall number of cells. It should be noted that this resolution is not constant across the entire surface. In the spanwise direction, cells are pushed towards the hub and shroud. This clustering allows for better resolution of the solution at the solid boundaries, but it results in lower cell density at midspan. This results in a surface solution with cells that are not located across lines of constant span or wetted distance.
For the experimental results, the locations of surface pressure transducers and heat-flux gauges are measured in terms of span (SL) and wetted distance (WD). For the results reported herein, negative WD always represents the airfoils pressure surface and positive WD represents the airfoils suction surface. For a consistent relationship between the experimental results and the predictions, the geometric interpretation must be consistent. Span lines must have a constant percent of total distance between the hub and shroud. Lines of wetted distance must have a constant percent of total distance between the leading and trailing edges. Because FINE/Turbo does not output surface data in constant SL or WD, a modification was required. Figure 2.1 shows the difference between FINE/Turbo spanwise grid lines, and physical lines of constant span for the URETI blade pressure surface. Figure 2.2 shows that although the two blade cuts were taken at approximately the same distance from the hub (the two profiles are almost identical), the spanwise profiles are very different. Using the programming software LabVIEW, a program was created to generate new airfoil surfaces based on percent WD and SL. A program was also created to interpolate between these new surfaces and the FINE/Turbo data files.
Figure 2.1: Comparison of "50% span lines" for the FINE/Turbo surface geometry and the corrected surface geometry - URETI blade pressure surface (side view)

Figure 2.2: Comparison of "50% span lines" for the FINE/Turbo surface geometry and the corrected surface geometry - URETI blade pressure surface (Top view)
To extract surface data from the FINE/Turbo predictions, an airfoil surface model first had to be generated based on surface data provided by the airfoil manufacturer. This geometry was represented by three coordinates \((x, y, z)\), with complete surface perimeter measurements taken at constant vertical \((y)\) positions. In the case of the URET I components, approximately 11 perimeter measurements were provided for each airfoil. To model the surface, this simple geometry was used to extrapolate a much more detailed surface. The blade hub, shroud, and tip gap geometries were also used to generate this new surface, which was in terms of the same three coordinates as before, as well as percent wetted distance and percent span. This new geometry provided the relationship between the basic coordinate geometry \((x, y, z)\), and the WD and SL geometry. This new surface had a constant resolution of approximately 0.1% WD, which allowed for a very precise prediction of the airfoil arch length. By utilizing this surface, it was possible to convert the FINE/Turbo surface predictions into constant wetted distance and span predictions.

To interpolate between the FINE/Turbo data files and the new surface, the program first created a coarse surface based on the 0.1% WD resolution surface. This was done to conserve both computational time, and resources. This new surface had a resolution of 1.0% WD and SL (approximately 10,201 surface points per airfoil side). Figures 2.3a and 2.3b show both the FINE/Turbo surface mesh, and the new surface mesh. For each 1.0% surface point, the three dimensional distance between that point, and all points within the FINE/Turbo data file were calculated. Because the 1.0% surface provided better resolution than the FINE/Turbo surface mesh (particularly in the spanwise direction), a weighted area averaging technique was implemented. The number of points to average could be specified by the user, but it was found that averaging ten points provided a reasonable approximation of the surface predictions.
with a minimum increase in computational time. Figure 2.4 shows the static-pressure prediction for the URETI vane suction surface. The difference between a single point interpolation, 10 point average, and a 20 point average can be seen. The weighting factor was based on the calculated three dimensional distances between the new surface point, and the FINE/Turbo surface grid points. In the case of a ten point average, the ten closest points were used to calculate the new average value at the specific WD and SL. The following equation represents this weighted area averaging technique.

\[ \bar{\chi} = w_1 \chi_1 + w_2 \chi_2 + \ldots + w_n \chi_n \]  \hspace{1cm} (2.1)

In equation 2.1 \( \chi \) represents the surface variable, \( w \) is the weighting factor, and \( n \) is the number of points being averaged. The weighting factor was calculated as follows,

\[ w_i = \frac{w_i}{\sum_{i=1}^{n} w_i} \]  \hspace{1cm} (2.2)
Figure 2.3: Comparison between the FINE/Turbo and the new surface mesh
Once the interpolation was complete, a new data file was generated. This file contained surface predictions at 1.0% WD and SL. These files were then used to plot data at constant span lines, as well as compare entire surface predictions for various computational models. Such analysis allowed for a more accurate understanding of not only the differences between predictions, but also the location of where these differences were concentrated. In general, by using this new (corrected) surface, it was possible to make better comparisons between data and predictions. Figure 2.5 illustrates the difference between the FINE/Turbo surface predictions and the corrected surface predictions for the un-cooled (coolant was turned off) URETI blade at 50% span. This figure also shows that the new surface better predicts the un-cooled data. It should be stated that un-cooled experimental results were taken on blades that had cooling holes, but the coolant flow was turned off. Notice that the greatest different between these two predictions occurs on the pressure surface. The new surface shifts the overall prediction to the left because it precisely estimates
the arch length of the span cut at 50% (unlike the FINE/Turbo surface, as seen in 2.1). Therefore, this shift is most important when considering the agreement between the pressure surface data and predictions. When compared to the data, this difference between the new surface and the FINE/Turbo surface can be a much as 8% of the measurement. Figure 2.5 presents the steady-state solution and as was shown by Southworth [8], the steady-state solution for the blade under predicts the data. When the unsteady solution is utilized, one obtains a much better agreement with the measurements (as was also observed by Southworth [8]). For the purposes of this exercise, the intent was to demonstrate the importance of the geometry correction and not necessarily make a comparison between the predictions and the data. Based on figure 2.5 it is also shown that this shift is important when considering the location of the predicted leading edge (point of maximum static pressure). Overall, this new method for mapping the predictions onto the surface of the airfoil is critical when making comparisons between the predictions and the data.
Figure 2.5: Comparison of FINE/Turbo steady-state solution for surface and corrected surface predictions at 50% constant span on the pressure surface of the URETI blade

2.3 MESH CELL WALL WIDTH AND $y^+$

When performing computations using a RANS solver, it is necessary to place the first grid cell within a specific distance from the solid wall in order to model the boundary layer region. A numerical model must have sufficient grid resolution near the surface (within the boundary layer) to resolve this region of high gradients. Tallman [19] provides a discussion of the wall boundary conditions used during his work with CFD heat-transfer predictions. This computational work was performed for a one and one-half stage turbine (discussed earlier), and used the General Electric proprietary software, TACOMA. Although Tallman used the k-omega turbulence model for his work, the development of the non-dimensional parameters was relevant to this current study. For this study, the one equation Spalart-Allmaras turbulence model was chosen for its ability to simulate the turbulence quantities of low Reynolds num-
ber flows with a faster rate of convergence. This model, developed by Spalart and Allmaras [20], requires that the $y+$ values be between 1 and 10. The parameter $y+$ is defined as,

$$y+ = \frac{u_t y}{\nu}$$

(2.3)

where $y$ is the distance to the nearest wall, $\nu$ is the local kinematic viscosity of the fluid, and $u_t$ is the friction velocity at the nearest wall, and is defined as,

$$u_t = \sqrt{\tau_w / \rho}$$

(2.4)

where $\tau_w$ is the wall shear stress, and $\rho$ is the fluid density at the wall. By changing $y$, it is possible to control the $y+$ range. In FINE/Turbo the variable used to control the $y+$ range is the wall cell width. A study was conducted to show the effects on the steady-state predictions, given several ranges of $y+$ values. The results of four steady-state computations, each with identical boundary conditions and run characteristics, are shown in 2.6 and 2.7.
Figure 2.6: Effects of $y+$ values on the URETI blade, steady, heat-transfer predictions, at 50% span

Figure 2.7: Effects of $y+$ values on the URETI blade, steady, static-pressure predictions, at 50% span
The wall cell width was changed for each prediction in order to control $y+$. Figure 2.6 illustrates the effect of this parameter on heat transfer surface predictions. Notice that the heat-transfer prediction increases (a maximum difference of 86%) as the maximum $y+$ values decrease from 140 to 20. On the pressure surface we see a decrease in the heat transfer as the maximum $y+$ value decreases. On the suction surface we see that values of heat transfer at the leading and trailing edges are most affected by the change in $y+$. Note that suction surface predictions near the leading edge continue to increase, and predictions near the trailing edge decrease. Figure 2.7 illustrates the effect of $y+$ on the blade static pressure surface predictions. These predictions are almost insensitive to the $y+$ range for the given computations. Southworth et al. [9] used $y+$ values in the range of 15 to 100 and had success in making static-pressure predictions when compared to the experimental results.

From these results, one can generate some conclusions regarding $y+$. First, this parameter is important to the success of the computation, especially when making heat-transfer predictions. Secondly, changing this parameter does not simply "shift" the prediction profile, but it alters the profile. Thirdly, although the 1 to 10 and 1 to 5 ranges are both within the required range, these predictions are not the same (an average difference of 2% for the heat-transfer predictions). Figure 2.9 taken from White [21] illustrates the turbulent boundary layer, and shows that the viscous sub layer only extends to a $y+$ of 5. The Spalart-Allmaras model can resolve this region, but it requires a very fine near wall mesh. When using the Spalart-Allmaras model, a maximum $y+$ of 10 is an acceptable design approximation for the near wall region. Figure 2.8 provides a contour plot of the un-cooled URETI vane and blade $y+$ values. This plot shows that the new values are between 1 and 10, with most values falling below 5. Also note that this plot shows that $y+$ is a quantity that is not completely
controlled by the designer. Although it is possible to design for a $y+$ range using the cell wall width, $y+$ is also a function of the flow field characteristics. This must be taken into consideration when designing the mesh.

Figure 2.8: Vane suction surface and blade pressure surface $y+$ values for the new URETI model
2.4 Inlet Turbulence Intensity

During the experimental investigation, the turbulence intensity was approximately determined. Because the combustor emulator acts and a flow conditioner at the inlet of the experimental test section, it was believed that an appropriate intensity would be in the range of 1% to 5%. Using a general approximation of the experimental geometry and the known run conditions, an intensity of 3.18% was calculated. In the FINE/Turbo interface, the user does not directly set turbulence intensity. Instead, the inlet kinematic turbulent viscosity ($\nu_t$) is specified. For the Spalart-Allmaras turbulence model, the kinematic turbulent viscosity is related to the turbulence intensity using the following equation,
\[ \nu_t = \sqrt{\frac{3}{2}} \bar{u} I l \]

where \( \bar{u} \) is the average main freestream velocity, \( I \) is the turbulence intensity, and \( l \) is the turbulent (eddy) length scale. Initially, \( \nu_t \) was set to 0.0001 \( m^2/s \), which is a general design condition of setting \( \nu_t \) to five times the known viscosity (\( \mu \)); approximately \( \nu_t \) equal to 0.00013 \( m^2/s \) in this case. To have an intensity of 3.2\%, \( \nu_t \) was set to 0.04 \( m^2/s \). Figure 2.10 shows the difference in the predictions for the URETI un-cooled blade at 50% span. Notice that this parameter has its largest influence on the predictions for the pressure surface of the blade, starting at -30% wetted distance. Also notice that using a more correct intensity setting better predicts the experimental results (although the prediction is still well below the data). These results suggest that setting the proper intensity levels at the inlet is very important. These results also suggest that the model is still not adequately predicting the full unsteadiness of the flow field across the pressure side of the blade. It is possible that because the cooling holes represent a discontinuity in the solid surface, they are causing instabilities in the flow. Because the computational model does not recreate the geometry of the holes at the solid surface, the predictions would not account for this flow instability.
Figure 2.10: Comparison between the two kinematic turbulent viscosity settings and run 13 (un-cooled) experimental results

2.5 Unsteady Computations using the Non Linear-Harmonic Method

For this study, both steady-state and time-accurate solutions were obtained. The time-accurate solutions were generated using the non linear-harmonic method. FINE is capable of solving both time marching and harmonic solutions, but the harmonic method was chosen for this work. This choice was made because of the harmonic method’s ability to provide an approximation of the unsteady solution with less computational cost. Most recently, Mathison et al. [7] found that predictions using the harmonic method did agree reasonably well with temperature measurements taken at the leading edge of a blade for a one and one-half stage turbine.
Fransson [22] provides a general description of this method based on the work of He and Ning. Hall et al. [23] developed a method for modeling unsteady flows by decomposing them into a nonlinear mean flow, and a linear harmonically varying unsteady flow. At first this work was performed using a linearized Euler analysis, but He and Ning [24] provided the groundwork for a nonlinear method by introducing the governing equations for a 2D time-averaged and unsteady flow model. At that time, the authors stated that nonlinear time-domain methods were computationally expensive. They concluded that a nonlinear harmonic method, solving the viscous (Navier-Stokes) equations, was both reliable, and computationally efficient. This method was later extended to 3D Navier-Stokes equation by Chen et al. [25] in 2001.

Prior to the harmonic method, Adamczyk et al. [26] presented the average-passage method. This method described the unsteady effects on the time-averaged solution, without generating the full time-accurate solution. This was a continuation of his work on modeling multistage turbomachinery (see Adamczyk [27]), which was first presented in 1984. In short, the method generated the unsteady effects of the time-averaged flow field in order to better simulate the aerodynamics within a multistage turbine. To solve for the unsteady effects, the average-passage method relied on empirical relationships. This method represented a step towards incorporating time-varying effects into the time-averaged equations. In 1999 Adamczyk [28] presented a summary of work regarding 3D multistage turbomachinery CFD models during his International Gas Turbine Institute Scholar Lecture.

The harmonic method utilizes this approach of separating each flow variable into time-averaged and time-varying terms: however, this method generates the time-varying effects of the flow by directly solving for the unsteady perturbations. This
method results in the addition of nonlinear stress terms within the time-averaged equations. These stress terms describe the unsteadiness of the time-averaged solution. A basic assumption of the harmonic method is that within a gas turbine engine, the unsteadiness is periodic in nature (relating to the vane and blade configuration). These unsteady perturbations are calculated through Fourier decomposition in the temporal domain. Solving for each frequency generates the nonlinear stress terms. FINE/Turbo allows the designer to specify the number of harmonic frequencies to solve for, which provides control of the unsteady solution accuracy. Because the time-averaged and unsteady perturbations are interdependent, the coupling between them is an important consideration for this model. He and Ning [24] suggest that a simultaneous time-marching procedure was incorporated because it provided a strong coupling between the two terms, was simple to implement, proved to be highly stable, and provided a fast rate of convergence.

In FINE/Turbo the designer must specify two quantities when utilizing the harmonic method; the number of perturbations, and the number of frequencies per perturbation. First, the number of perturbations, which is the number of blade passing frequencies per blade row, is directly related to the flow conditions generated both upstream and downstream of the rotor. For this study, the blade was only influenced by the effects of wakes from the upstream vane (single stage), and the vane was only influenced by the potential effects from the downstream blade; consequently the number of perturbations was set to 1. The second quantity, number of frequencies per perturbation, is the number of harmonics to solve for during the unsteady calculations. To have a complete recreation of the true time-accurate flow and full continuity across the rotor stator interface, an infinite number of harmonics would need to be solved for in the Fourier series. In practice this is not possible, but FINE/Turbo
does provide qualitative guidelines regarding the number of harmonics to use. It is suggested that for satisfactory solutions, and continuity at an "engineering level", the number of harmonics should be set to no fewer than 3, but possibly as many as 5 in complex configurations. Chen et al. [25] provides a study of the mixing loss across the rotor stator interface for the harmonic methods with 1, 2, and 5 harmonics. All harmonic computations in this study used 3 harmonics.

To calculate the computational time required to run a harmonic simulation, the code provides the following equation for the $CPU_{ratio}$. This ratio relates the amount of time a computation would require using the harmonic method compared to a steady-state simulation.

$$CPU_{ratio} = 1 + 10 \left( \frac{P \times N}{E} \right)$$

(2.6)

Where $E$ is the number of equations used to solve the steady-state turbulent flow using the specified turbulence model. For this study, the Spalart-Allmaras (single equation) turbulence model was used, thus six steady-state equations were used. The terms $P$ and $N$ are the number of perturbations (blade passing frequencies), and the number of frequencies per perturbation (harmonics). It was determined that the harmonic calculations would require 6 times the CPU requirements when compared to the steady-state simulations. Table 2.1 compares the $RAM_{ratio}$ with the $CPU_{ratio}$ for different values of harmonics (i.e. 2, 3, and 4). The $RAM_{ratio}$ is calculated using the same equation above, but this ratio is based on the maximum number of perturbations, and therefore the $RAM_{ratio}$ (memory requirement) for the 3 harmonic computations was also 6 times greater than the steady computations.
Table 2.1: Harmonic method $CPU_{ratio}$ with 1 perturbation

<table>
<thead>
<tr>
<th>Number of Harmonics</th>
<th>$CPU_{ratio}$</th>
<th>$RAM_{ratio}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>4.33</td>
<td>4.33</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>6</td>
</tr>
<tr>
<td>4</td>
<td>7.8</td>
<td>7.8</td>
</tr>
</tbody>
</table>
Chapter 3

Modeling Cooling Holes with Distributed Source Term Injection

For this study, the distributed source term injection approach was chosen as it is the method implemented by the FINE/Turbo code. This model allows for the injection of coolant through the solid walls into the main flow. This method does not require the meshing of the cooling hole geometry, and therefore has a low computational cost. In general, the model does not describe in detail the cooling flow; rather it considers the effects of the coolant flow on the main flow.

3.1 Flat-Plate Experimental Investigation

This experimental investigation was conducted at the OSU GTL Small Calibration Facility (SCF), as seen in figure 3.1. This work was performed and documented by Kheniser [29]. The following is a summary of his discussion. The SCF is best described as a medium duration blowdown facility, where experimental conditions will typically last for 1 to 2 seconds. General information on the development of a blowdown facility of this type is presented by Epstein [30]. The SCF was first established while the GTL group was working at Calspan in the early 1990s, and moved to OSU in 1996 along with the new GTL group. Since that time it has served as the lab's primary
facility for the calibration of total pressure and total temperature rakes. The main blowdown facility consists of a supply tank (main supply), test section, dump tank, and a data acquisition system. For this experimental investigation, the facility was modified with the addition of a cooling supply system. The cooling supply system consists of a second supply tank (cooling supply), and a heat exchanger. Bernasconi [31] outlined the development of the experimental apparatus (test section and flat plate) now in place at the SCF.

Figure 3.1: The Gas Turbine Laboratory Small Calibration Facility (SCF)

During the experimental investigation, the main supply air was provided to the supply tank from the GTL compressed air system. This supply air was pressurized to 75 psi, and heated within the main supply tank to a temperature between 450 $K$ and 550 $K$. The cooling supply tank was a K-bottle pressurized with nitrogen. Prior to the experiment, the test section and dump tank were kept at a near vacuum. This ensured that the flow was choked for the duration of the experiment. This choke
served as the primary method for controlling the test section Mach number during the experiment, and was designed to produce a Mach number of 0.34 within the test section. For the cooling supply system, a choke was also used to control the coolant mass flow from the supply tank, into the heat exchanger. The cooling supply system heat exchanger was contained within an insulated enclosure, which was then filled with dry ice. This system was able to cool the heat exchange to approximately 210 K resulting in a coolant fluid temperature of approximately 300 K, measured within the cooling plenum, downstream of the heat exchanger. In general, it was important that the experiment be able to maintain the correct temperature ratio between the main flow (in excess of 488 K) and coolant flow for an appropriate period of time (approximately 1.5 seconds).

The instrumented flat plate was located within the test section. This plate, as seen in figure 3.2, consisted of four sections. Three of these sections, which represented the base, can be seen in 3.2a. The fourth section was a thin stainless steel insert plate, which rested in the base. The cooling holes were located on this insert plate. The hole pattern represents the pressure side of the URETI turbine blade. The base was designed with a leading edge profile to establish the development of the flow boundary layer across the surface of the plate. The base also served as the mounting surface for the heat-flux gauges. For this experiment, a total of 43 double-sided Kapton heat-flux gauges were used. These gauges were placed at approximately 50% channel width, extending from upstream of the cooling holes, through the cooling region, to the downstream side of the holes. Kheniser [29] provides greater detail regarding both the design of the experimental instrumentation, as well as their location within the test section.
3.2 Flat Plate Computational Investigation

This model was gridded, solved, and post-processed using the FINE/Turbo software package. Although one would generally use the FINE/Turbo code for full-stage applications, it is a versatile code and is able to run stationary simulations. For the gridding, FINE/Turbo Interactive Grid Generator (IGG) was used to generate the mesh. IGG is a structured grid generator which is capable of creating multi-block, structured grids for both 2D and 3D complex geometries. The FINE/Turbo AutoGrid tool is an extension of this software, and is used for the generation of turbomachinery models.
Based on the experimental setup, the computational domain had six faces. Two (the inlet and outlet) were fluid, and the other four, which were the left wall, right wall, top, and bottom walls, were solid. The grid was not extended into the solid surface, thus only heat transfer at the very surface of the solid walls will be discussed. A conjugate heat transfer analysis was not performed. Figure 3.3 illustrates the general layout of the channel. This channel represented the volume directly above the experimental flat plate, starting at the leading edge, and extending downstream, beyond the heat-flux instrumentation. The channel dimensions were modeled as follows; length equal to 0.289 \( m \), width equal to 0.114 \( m \), and height equal to 0.0536 \( m \).

![Figure 3.3: Flat plate mesh block diagram](image)

To allow for parallel processing, which would decrease the total computational time, it was necessary to use more than one mesh block to construct the fluid domain. FINE/Turbo cannot sub-divide mesh blocks, thus in order to spread the computation over multiple processors, multiple grid blocks are required. A "butterfly" mesh was chosen to not only increase the number of blocks, but also allow for increased grid density near the solid surfaces. The butterfly technique is a multi-block approach, which is typically used to mesh curved geometries (e.g. the interior of a pipe). This
method greatly improves the mesh quality of a structured grid. A typical butterfly mesh will consist of seven sub-blocks; one inner block, and six buffer (side) blocks; see figure 3.4.

![Figure 3.4: Butterfly topology with seven blocks. Image taken from Numeca IGG User Manual](image)

The inner block is always present in a butterfly topology, but the buffer blocks can be used as needed to change the shape of the mesh. For the flat plate, four side buffer blocks were used. Buffer blocks were not included at the inlet or outlet faces. The shape of each buffer block can be controlled by changing the "depth" of that block. In general, the number of cells within the buffer block can be changed so that the individual buffer block represent more or less of the overall mesh. A total of 17 radial distributions (cells) were used for each buffer block. The depth for each of these blocks was set to 5% of the total channel cross-sectional area.

Figure 3.6 shows that the butterfly mesh has a vertical cell distribution clustered at the solid surface. The first cell width was set to 0.0025 mm, which was required to allow for $y+$ values to be in the range of 1 to 10 (as required by the turbulence model). Figure 3.5 shows how the main block was distributed between the center block and the four buffer blocks. Also note that the main block was split in half, further increasing the total number of blocks. Finally, notice that at the inlet, four
buffer blocks were used along the solid surfaces. These block were added (without the generation of a center block) to help push cells towards the inlet face, which allowed for better resolution of the boundary layer initialization. Figure 3.8 illustrates the region of the computational domain as it relates to the physical geometry.

Figure 3.5: Individual mesh blocks for the flat plate computational domain
Figure 3.6: Cell distribution at the buffer block

Figure 3.7: Flat plate surface mesh
A total of 14 grid block were used in this model, with a total of 2.5 million cells. The number of grid points in each direction (vertical, channel length, and channel width) had to be designed to allow for the implementation of a full multigrid solution. Figure 3.9 is taken from the Numeca IGG users manual, and illustrates the cell requirements for each number of multigrids. For this simulation, 3 multigrid levels were necessary. To achieve at least three multigrids, and have the correct grid resolution, a specific number of grid points were used in each block. Table 3.1 lists the various grid levels used for the finest mesh in each of the blocks. Arnone and Pacciani [32] outlined a basic multigrid procedure for both steady and time-accurate solvers. The multigrid methods used by FINE/Turbo and outlined by Arnone and Pacciani [32], is based on the Full Approximation Storage (FAS) method. The multigrid method also made it was possible to perform a grid independence study for this mesh. To do this, both the medium and fine mesh from the multigrid were solved. By generating a solution for each mesh, it was found that the solution was independent of grid size.
Table 3.1: Combination of grid points used in each of the 14 blocks (for the finest multigrid)

<table>
<thead>
<tr>
<th>MultiGrid level</th>
<th>Number of grid points per level</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2  4  6  8  10  12  14  16  18</td>
</tr>
<tr>
<td>2</td>
<td>3  7  11 15  19  23  27  31  35</td>
</tr>
<tr>
<td>3</td>
<td>5 13  21 29  37  45  53  61  69</td>
</tr>
<tr>
<td>4</td>
<td>9 25  41 57  73  89 105 121 137</td>
</tr>
<tr>
<td>5</td>
<td>17 49  81 113 145 177 209 241 273</td>
</tr>
<tr>
<td>6</td>
<td>33 97 161 225 289 353</td>
</tr>
<tr>
<td>7</td>
<td>65 193 321</td>
</tr>
<tr>
<td>8</td>
<td>129 385</td>
</tr>
<tr>
<td>9</td>
<td>257</td>
</tr>
</tbody>
</table>

Care was taken to design the mesh so that the plate surface grid resolution was 1 mm squared. This resolution was set by the minimum distance between cooling holes (1 \( mm \)) and the maximum hole diameter (0.5 \( mm \)). With this resolution, each hole fit inside a single cell, thus no holes overlapped. Figure 3.10 shows the location of the cooling holes on the surface mesh. Notice that each hole has an independent cell location. A total of five rows or 113 cooling holes were modeled as distributed
source term injectors. Figure 3.10 shows these rows, labeled 1 to 5 starting from the top (inlet). Rows 1, 2, and 4 were given hole diameters of 0.5 \textit{mm}, and rows 3 and 5 were given hole diameters of 0.254 \textit{mm}. Each row was placed at a correct distance from both the leading edge and the side walls. From the experimental setup, it was known that each hole had an injection angle of approximately 30-degrees, thus the modeled holes also were given a 30-degree injection angle. The mass flow supply to the coolant plenum was measured during the experiment, but the coolant supplied to an individual hole, or to a row of holes was not measured. To assign mass flow in the computational model, a simple area averaging technique was used. It was assumed that the pressure ratio across each of the holes was equal, which allowed for the assumption that the mass flow was simply a function of the hole area. Based on this assumption it was possible to allocate a percentage of the total coolant flow to each row. Table 3.2 provides the row total coolant mass flows.

Table 3.2: Mass flow for the flat plate cooling rows

<table>
<thead>
<tr>
<th>Row</th>
<th>Hole Area [$m^2$]</th>
<th>Count</th>
<th>Area Percentage</th>
<th>Total Row Mass Flow [kg/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.03E-7</td>
<td>16</td>
<td>21.2%</td>
<td>0.00156</td>
</tr>
<tr>
<td>2</td>
<td>2.03E-7</td>
<td>15</td>
<td>19.9%</td>
<td>0.00146</td>
</tr>
<tr>
<td>3</td>
<td>5.07E-8</td>
<td>33</td>
<td>10.9%</td>
<td>0.00080</td>
</tr>
<tr>
<td>4</td>
<td>2.03E-7</td>
<td>32</td>
<td>42.4%</td>
<td>0.00312</td>
</tr>
<tr>
<td>5</td>
<td>5.07E-8</td>
<td>17</td>
<td>5.6%</td>
<td>0.00041</td>
</tr>
</tbody>
</table>
3.3 **Boundary Conditions**

During the experimental investigation, experiments were performed for cases that had the cooling supply turned on (cooled), and cases that turned off the cooling gas supply (un-cooled). Several runs at both of these conditions were performed. For the computational investigation, it was decided to model one cooled run, and one un-cooled run. The selected runs were run 4 (cooling on) and run 5 (cooling off). For a complete listing of the experimental runs, see Kheniser [29]. Table 3.3 provides the boundary conditions for the two simulated conditions. The boundary conditions for the computational model were as follows. Both the main flow and coolant were modeled as air. All computations were run steady-state, using the turbulent Navier-Stokes mathematical model with the Spalart-Allmaras turbulence model. At the inlet, total quantities were imposed (both pressure and temperature), and the Mach number was extrapolated. At the outlet, mass flow was imposed using...
the "pressure adaptation" boundary condition. This is identical to an imposed static pressure condition, but the exit pressure and mass flow are calculated and modified as the solution converges. This allows the solution to converge to the correct mass flow. The initial condition for mass flow was set using the calculated experimental mass flows (the sum of main flow and coolant flow when coolant was turned on). The initial pressure approximation downstream of the test section was 101300 Pa. For the solid boundaries, it was assumed that during the medium duration blowdown, but solid surface temperatures did not change, thus isothermal wall conditions could be approximated. The experimental average surface temperature was measured using the double-sided heat-flux gauges.

Table 3.3: Flat plate boundary conditions

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>Run 4</th>
<th>Run 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Avg. Inlet Total Temperature [K]</td>
<td>477.5</td>
<td>479.9</td>
</tr>
<tr>
<td>Inlet Total Pressure [kPa]</td>
<td>320.3</td>
<td>309.5</td>
</tr>
<tr>
<td>Freestream Mass Flow [kg/s]</td>
<td>1.8348</td>
<td>1.7961</td>
</tr>
<tr>
<td>Surface Temperature [K]</td>
<td>303.7</td>
<td>306.0</td>
</tr>
<tr>
<td>Coolant Temperature [K]</td>
<td>303.5</td>
<td></td>
</tr>
<tr>
<td>Coolant Mass Flow [kg/s]</td>
<td>0.0073638</td>
<td></td>
</tr>
</tbody>
</table>

3.4 Fully Cooled Predictions

Again, both cooled and un-cooled runs were modeled. These simulations were run steady-state, using a full multigrid initialization approach. Both the coarse, and medium grid levels were run for 100 iterations. This multigrid approach helped to
increase the rate of convergence. On average, each computation reached a global residual of -4.85 after 4,200 iterations. In order to judge the validity of the results, each solutions minimum surface $y+$ values were analyzed to ensure that a proper surface resolution was achieved. Figures 3.11 and 3.12 show that for both the cooled and un-cooled cases, the minimum surface $y+$ values were within the desired range of 1 to 10. It should be noted that for the cooled case, the surface region close to the cooling holes was outside the 1 to 10 range. This was expected, and acceptable, as it was most likely caused by a mismatch between the mainstream and the coolant kinematic turbulent viscosities. In general, $y+$ values across the entire plate were very good, and on average, between 2 and 4. The discontinuity at the inlet was caused by a discontinuity in the grid density at that location. Figure 3.7 shows this discontinuity. It was decided that for the purposes of making comparisons between the experimental results and the predictions this region would not be included. This did not present a problem because data were not collected within that region.

For run 5, the predicted outlet Mach number was 0.31690. For run 4 the value of the predicted outlet Mach number value was 0.31483. The error in these values was approximately 7% given that the experiment was designed with a Mach number of 0.34. The predicted outlet mass flows were 1.7961 kg/s for the un-cooled run 5, and 1.8421 kg/s for the cooled run 5. These predicted values agreed with the experimental values. This agreement provided confidence that the computational investigation was modeling the experimental boundary conditions.
Figure 3.11: $y+$ values at the surface of the un-cooled flat plate

Figure 3.12: $y+$ values at the surface of the cooled flat plate
Results of the computational simulations compared to the experiments can be seen in figure 3.13. The prediction results were extracted at approximately 50% channel width, which represents the approximate location of the heat-flux data. Both the cooled and un-cooled heat transfer results (positive heat transfer being in the direction from the fluid to the wall) are plotted as a function of distance from the leading edge. The location of the cooling holes is also noted by the red dashed lines. In general, the results are encouraging. Table 3.4 provided the percent difference between the experimental results and the computational predictions. This percent difference shows to what degree the simulation over predicted, or under predicted the experimental results. For the un-cooled case, the simulation and experimental results differ an average of 16%, but the prediction trend is representative of what is theoretically expected during parallel, unidirectional flow across a flat plate - Blasius boundary layer solution. It is believed that the presence of the physical cooling holes caused disruptions in the formation of the boundary layer during the experiment. Since the hole geometry is not modeled, this simplification could lead to a more uniform prediction. For a true comparison, the experiment would need to use a flat plate with no cooling holes.

The percent difference for the first cooled run is greater than the un-cooled run, but this prediction is greatly improved with the increase in turbulence intensity. Notice that both run 4 simulations predict the drop in heat-flux, upstream of the first cooling row. Although the heat-flux gauges located within the cooling hole region failed, it can be seen that the computational results do agree with the experimental results downstream of the cooling rows. Table 3.4 shows that for the cooled cases, the average percent difference was at best a 6.0% under prediction of the experimental data.
The results of this investigation provide a "proof of concept" regarding the implementation of distributed source term injection as a method to model cooling holes. It is understood that this flat plate represents an idealized case, one that does not
capture the true complexities of a rotating airfoil. But this study does represent the first steps towards utilizing distributed source term injection for computational investigations at the GTL. This work has helped develop a better understanding of how to model airfoil film cooling, and has provided confidence in both the distributed source term method and FINE/Turbo as a design tool.
The URETI turbine stage is composed of 24 vanes and 38 blades. This turbine is designed to have film cooling supplied to both the vane and blade surfaces. The experimental investigation was conducted using the Turbine Test Facility (TTF) located at the OSU GTL. Figure 4.1 shows the TTF facility. The TTF is a short duration blowdown facility capable of accommodating a full scale rotating turbine that can have more than one stage if desired. A combustor emulator is utilized to provide heat addition to the mainstream flow just ahead of the high-pressure turbine vane row. By using this emulator it is possible to control the temperature profile entering the vane row. The GTL supplies coolant to the vanes and blades by means of the Large Cooling Facility (LCF). The LCF is capable of supplying a constant temperature metered flow to the various cooling gas supply legs for a wide range of supply pressures (and hence blowing ratios). A more detailed discussion of the TTF is presented by Haldeman et al. [33] and Crosh [10]. Further information on the LCF is also provided by [33] and Southworth [8]. No further details regarding these facilities will be discussed.
During the development of the URETI computation model, geometric properties of the vane and blade (e.g. hub and shroud fillets) were modeled. It was discovered that these features caused instabilities in the computations, such as an increase in $y^+$. It was decided that such features would not be included in the final mesh. Also note that while the blade tip was modeled using a more realistic non-constant gap, the tip geometry (e.g. tip recesses and coolant bleed holes) was not modeled. Past studies conducted by Southworth et al. [9] modeled this region with success. The blade tip recess has also been modeled for other, similar geometries; see Green et al. [34]. The final mesh generation was performed using the AutoGrid software version 5 provided by Numeca. As a measure of grid quality, Numeca suggests that the following parameter be satisfied. A minimum orthogonality angle no less than 10-degrees; maximum expansion ratio no greater than 3.0 and a maximum aspect ratio of no greater than 5000.
The Numeca AutoGrid software is a structured meshing tool. Each mesh for the URETI geometry was generated using an O4H topology for both the vane and blade. This topology was constructed of five blocks which include an O-block surrounding the airfoil surface (skin mesh), H-blocks both upstream and downstream of the leading and trailing edges (inlet and outlet blocks), and H-blocks on the pressure and suction sides of the airfoil (up and down blocks). The O-block at the surface allows for greater control of the mesh within the boundary layer region. This multiple block method allows for greater grid resolution at the surface, and allows for a more efficient distribution of the cells; greater density in the boundary region close to the surface.

4.1 URETI Short Channel Mesh

The short channel model was used initially in modeling the experimental setup due to its overall smaller size, and fewer number of grid points. A version of this model was used in a previous investigation, see Southworth [8] with good agreement between the blade surface pressure data and predictions. The previous mesh used approximately 700,000 cells, while the new mesh contained 4.6 million cells organized into a total of 16 mesh blocks. Unlike previous studies, in order to achieve the new desired $y+$ values of 1 to 10, the wall cell width was set to 0.000108-in for the vane and 0.000100-in for the blade. The tip gap was modeled with a gap distance between the blade tip and shroud of 0.0243-in at the leading edge and a gap distance of 0.0286-in at the trailing edge. A total of 101 spanwise discretizations were used throughout the entire mesh. In order to model the rotational portion of the hub, the rotor/stator interface between the vane and blade was placed at a defined distance from the leading edge of the blade. A Z constant line was then used downstream of the blade, also located at the correct distance from the trailing edge. Only the blade skin
surface mesh, blade hub, and rotational mesh blocks associated with the blade tip gap were given a rotational velocity. For this mesh, the grid quality parameters were as follows; minimum orthogonality angle was 18.238-degrees; maximum expansion ratio was 2.444; maximum aspect ratio 595.207. In general this mesh was considered to be of good quality. Figure 4.2 shows the vane and blade mesh at 50% span.

![Figure 4.2: Blade to Blade view of the entire URETI short channel mesh (not to scale)](image)

4.2 URETI FULL CHANNEL MESH

This new configuration modeled the vane, blade and the entire flow passage of the experiment between the inlet and exit instrumentation rakes. The mesh contained 5.3 million cells organized into a total of 16 mesh blocks. As was done for the previous mesh, wall cell width was set to 0.000108-in for the vane and 0.000100-in for the
blade. The tip gap was also modeled in the same way as before with gap distances of 0.0243-in at the blades leading edge and 0.0286-in at the blades trailing edge. A total of 101 spanwise discretizations were used throughout the entire mesh. The rotor/stator interface and a Z constant line were used as before to model the correct rotational hub geometry at the blade. The grid quality parameters were as follows; minimum orthogonality angle was 26.14-degrees; maximum expansion ratio was 2.49; maximum aspect ratio 5137.76. Although the maximum aspect ratio did exceed 5000 (within the vane row) there were only 171 surface cells between 5000 and 5137.76 in the entire mesh. The maximum aspect ratio occurred in the vane skin mesh block which is understandable when one considers the wall cell width being used, and the fact that this new mesh is modeling a much larger volume (especially upstream of the vane) without drastically increasing the number of total cells. To put it another way, the cells have been stretched in the axial direction to accommodate for the increased volume, while the wall cell width has been held constant, thus the aspect ratio (cell width to cell streamwise length) has increased. In general, the tradeoff between mesh quality and computational efficiency was considered to be acceptable. Figure 4.3 shows the full vane and blade mesh at 50% span.

![Figure 4.3: Blade to Blade view of the entire URETI mesh "rake to rake" model (not to scale)](image)

Figure 4.3: Blade to Blade view of the entire URETI mesh "rake to rake" model (not to scale)
4.3 Implementation of Film Cooling

As was stated before, the URETI turbine stage is capable of supplying film cooling to both the vane and blade rows. There are 9 rows of cooling holes on the blade and 13 rows on the vane. Combined, this single stage contains over 450 cooling holes. The URETI geometry also contains holes located on the vanes hub and shroud, but these holes were not used during the experimental investigation in order to simplify the CFD, and therefore not included in the following CFD model. Figure 4.4 shows the three independently controlled flow paths providing cooling the vane and blade. The vane inner path supplies the showerhead and pressure surface, while the outer path supplies the suction surface cooling holes and trailing edge slots. The blade cooling is supplied by a single path. Upon entering the interior of the blade, coolant flows through the internal passages, moving in both the streamwise and spanwise directions.

![Figure 4.4: Turbine stage cooling paths](image)

Figure 4.4: Turbine stage cooling paths
The approximate location of each hole was determined using drawings provided by the design company. A program was created to map this "manufacturing" geometry onto the airfoil surface mesh. For most holes, the program was able to determine a location, but some holes were not found. For these holes, a visual inspection of the vane or blade was performed to approximate the location of the hole relative to other holes on the surface. The hole was then placed on the mesh manually. The greatest amount of manual approximation was needed for holes located near the leading edge, in what is known as the "showerhead" region. Figures 4.5a, 4.5b, 4.6a and 4.6b show the location of each cooling row located on the vane and blade surfaces. Figures 4.5c and 4.6c show the cooling hole rows located at the leading edge of the blade and vane. Along with calculating a three dimensional location for the hole position, this program was able to calculate an angle of injection for each hole. FINE/Turbo is capable of defining two angles for the injection vector at each hole. The $k$ and $j$ angles are measured from the $k$ and $j$ lines, which are the two tangent directions on the airfoil surface at any given point. These lines are oriented in the streamwise and spanwise directions respectively. From this description we can say that the $k$ angle is the injectors streamwise angle, and the $j$ angle is the injectors spanwise angle. For rows near the leading edge, the angles of injection were set manually based on visual inspection. For the vane, injection angles were set to 45-degrees in the positive (pointing towards the tip) spanwise direction, and 0-degrees (normal to the surface) in the streamwise direction. For the blade, injection angles were set to 60-degrees in the positive spanwise direction, and 0-degrees in the streamwise direction. All other injection angles were calculated based on the "manufacturing" hole coordinates. Figure 4.7 shows the general location of each row and also shows the numeric label for each row. Note that the vane has 12 rows of cooling holes and 18 trailing edge (TE) slots for cooling. The blade only has 8 rows of cooling holes and the trailing
edge cooling slots were not modeled for the blade.

Figure 4.7: Location of the cooling holes for the vane and blade (not to scale)

Once the position and direction of each injection vector was determined, the next task was to assign a magnitude (mass flow rate). Based on figure 4.4 it can be seen that although the total coolant mass flow being supplied to the vane and blade is well known, what is not well known is the exact mass flow rate for each hole. To assign mass flow, the first method used was a simple area averaging technique. Given the area of each hole, and the total number of holes per row, the mass flow rate per row could be determined by the percent of the total cooling area that row represented. Figure 4.10 shows the cooling mass model for the vane. Note that the inner and outer flows both sum to 100% total flow. This method was a rough assumption because it did not account for the bias in mass flow between the pressure and suction surfaces; it simply assumed all holes were equal. This is however not true, and therefore it was determined that a more robust distribution would be necessary for the blade cooling
Figure 4.5: URETI blade surface cooling hole locations

(a) Blade pressure surface

(b) Blade suction surface

(c) Blade leading edge

61
Figure 4.6: URETI vane surface cooling hole locations
model. Figure 4.8 shows how the mass was allocated to each blade cooling row using this method.

The second method used to distribute mass among the rows was based on pressure differential between the internal passage pressure measurements and the external static pressure measurements. In general, this method assumed that for un-choked holes of equal area, the mass flow rate was a function of velocity, which was determined by the pressure differential across the hole. Because of the limited resolution in the experimental internal pressure data (in particular, data in the spanwise direction) the analysis was performed using a spanwise averaging of the internal and external pressures. This method was performed only on the blade because of limited internal pressure data for the vane. The vane cooling distribution was based on area averaging. By using surface static-pressure predictions from un-cooled computations, a spanwise average pressure profile was generated for the blade. From previous attempted cooled models, it was known that saturation (a calculated coolant velocity greater than \(M\) equal to 1) was occurring at rows 4, 5, 6 and 7. These rows are located on the blades leading edge, and suction surfaces. Garg and Abhari [35] also found that coolant holes on the suction surface, near the hub, were prone to becoming saturated. For this study, the row was determined to be choked if the pressure ratio between the external average pressure and internal pressure was less than or equal to 0.528. It was determined that in fact rows 5, 6 and 7 were choked. The choked mass flow was calculated using the following equation,

\[
\dot{m} = \frac{P_T A}{\sqrt{T_T}} \left( \frac{\gamma}{R_s} \left( \frac{2}{\gamma - 1} \right) \right) \frac{\gamma + 1}{\gamma - 1}
\]

(4.1)

where \(\gamma\) is equal to 1.4, \(R_s\) is equal to 287.058 \(J/(kgK)\), \(A\) is the hole area, and \(P_T\)
and $T_T$ are total internal pressure and temperature measurements. Because all variables were considered constant, the choked mass flow rate for holes in rows 5, 6, and 7 were all equal. The final mass flow rate per row was calculated to be approximately 95% less than what was previously calculated using the area averaging method. It was determined that this mass flow allocation for the blade distributed far too much mass to row 8 (over 60 percent of the total coolant mass flow). Therefore, a new method for the allocation of the coolant mass flow across the blade was considered. For this model, the choked mass flow for each row was calculated using the same method as before. It was understood that not all holes were choked, but by implementing this method, a maximum coolant mass flow model could be obtained. With all the holes choked, the new total coolant mass flow for the blade was calculated to be 94% less than the measured mass flow. By using the area averaging technique, it was calculated that the blade trailing edge cooling (which was not modeled) accounted for approximately 90% of the total coolant. This total difference was acceptable given that the trailing edge accounted for 90%, but also considering that blade tip cooling holes and small losses throughout the system were not modeled. Because this new maximum mass flow model also better allocated mass across the blade surface, it was implemented. After implementing this cooling distribution, no saturated holes were calculated within the model. Figure 4.9 shows the new mass distribution.

For the vane, trailing edge cooling was modeled during the cooled computations. When modeled, this cooling was modeled using the same method as for all other holes (point source terms). Because the exact geometry of this region of the vanes was unknown, an approximation of size and location was used to construct this row. Figure 4.6a and 4.6c show the true geometry of this region, but it was not possible to model the cooling channels with this cooling model. Spreading of the holes was
Figure 4.8: URETI blade coolant mass flow distribution using the percent area method

Figure 4.9: URETI blade coolant mass flow distribution using the maximum coolant method
used to approximate this geometry. The outer flow was distributed evenly among these three rows (11, 12, TE) because the vane suction side and the vane trailing edge cooling gas are supplied by the outer supply path. For the blade, the trailing edge cooling was not modeled. Using the method of coolant distribution discussed previously, the blade trailing edge cooling mass flow rate was determined, and then subtracted from the total blade flow rate. The new mass flow was distributed to the rows using the same differential pressure method. Table 4.1 shows the final coolant mass flows per hole for each of the vane and blade cooling rows.

<table>
<thead>
<tr>
<th>Row</th>
<th>Vane [kg/s per hole]</th>
<th>Blade [kg/s per hole]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000969</td>
<td>0.00269</td>
</tr>
<tr>
<td>2</td>
<td>0.000969</td>
<td>0.00298</td>
</tr>
<tr>
<td>3</td>
<td>0.000969</td>
<td>0.00135</td>
</tr>
<tr>
<td>4</td>
<td>0.000969</td>
<td>0.00143</td>
</tr>
<tr>
<td>5</td>
<td>0.000969</td>
<td>0.00143</td>
</tr>
<tr>
<td>6</td>
<td>0.000969</td>
<td>0.00223</td>
</tr>
<tr>
<td>7</td>
<td>0.000969</td>
<td>0.00148</td>
</tr>
<tr>
<td>8</td>
<td>0.000969</td>
<td>0.00138</td>
</tr>
<tr>
<td>9</td>
<td>0.00107</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.000961</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>0.00267</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.00279</td>
<td></td>
</tr>
<tr>
<td>TE</td>
<td>0.00140</td>
<td></td>
</tr>
</tbody>
</table>

FINE/Turbo has two cooling algorithms. The specifics of each are unknown, but it is important to discuss "spreading" of the cooling holes. When spreading is used, FINE/Turbo allocates a portion of the injector’s total mass to not only the cell where the hole has been placed, but to neighboring cells as well. Essentially, the hole is spread out among multiple cells. This treatment of the cooling holes is helpful for
several reasons. First, this method helps to reduce the possibility of instabilities during the computation. Second, if the surface cell size is not large enough to encompass the entire diameter of the cooling hole, the spreading will more precisely model the holes geometry. This is often true for holes with large injection angles, and highly elliptic cross-sections (e.g. holes in the showerhead region). Finally, Numeca has indicated that if the spreading algorithm is not used, the computation will not notify the user of saturated cooling holes. For these reasons it was determined that spreading was required in this model. Figures 4.11, 4.12, and 4.13 show the final vane and blade surface mesh with the cooling configuration.

4.4 Boundary Conditions

The following boundary conditions were applied to all cooled computations. Starting with the fluid model, FINE/Turbo has several different models to choose from. For steady-state computations, "real air" is used to model the main stream flow and coolant. This gas model contains the following properties; \( R \) set to 287 \( J/(kgK) \); heat capacity, heat conductivity, and dynamic viscosity values as a function of temperature. The mathematical model used was turbulent Navier-Stokes with turbulence modeled using the Spalart-Allmaras turbulence model. For all rotational blocks and surfaces, the rotational speed was set to 12,289.2 \( RPM \).

The rotor / stator (R/S) interface was constructed using the full non-matching mixing plane approach. The mixing plane approach provides a circumferentially averaged flow solution. Upstream wakes and separation are circumferentially mixed at spanwise positions across the interface. These averaged values then pass as boundary conditions between the upstream and downstream blocks. Figure 4.14a shows the upstream (vane side) R/S interface, and 4.14b shows the downstream (blade side)
Figure 4.10: URETI vane coolant mass flow distribution using the percent area method (TE cooling is row 13)

Figure 4.11: Vane surface mesh with coolant injector spreading turned on. L to R: Row TE, 12 to 1
Figure 4.12: Blade pressure surface mesh with coolant injector spreading turned on. L to R: Row 4, 3, 2, 1, 8

Figure 4.13: Blade suction surface mesh with coolant injector spreading turned on. L to R: Row 7, 6, 5, 4
R/S interface. Notice the circumferentially averaged values in 4.14b. A full non-matching mixing plane allows for more relaxed geometric constraints because there are no requirements on the distribution of mesh nodes for the upstream or downstream patches. The cell distribution between the upstream and downstream patches does not need to be equal. As the rotor mesh moves across the stator mesh, a one-to-one matching of cells is not required. This full non-matching mixing plane is also a fully conservative treatment of the R/S interface; conservation of mass, momentum and energy fluxes across the interface. Mansour et al. [36] provided a comparison between two codes, one using an averaged passage RANS model and the other using a mixing plane approach for the R/S interface calculations. This work was done for steady predictions of a four stage axial compressor. It was found that predictions utilizing the averaged passage model were five to six percent "closer" to the test data as compared to the mixing plane approach.

When utilizing the non-linear harmonic method for unsteady simulations, a full non-matching, non-reflecting treatment of the R/S interface was utilized. This treatment allowed the solution to maintain better continuity between the upstream, and the downstream flows. Because only a finite number of harmonics (3 harmonics) were used during the calculation, the discontinuity across the R/S interface was not completely eliminated, but this discontinuity would only be eliminated with an infinite number of harmonics. Figure 4.15 shows the upstream and downstream R/S interfaces during the harmonic solution. When compared with 4.14 the improvement in continuity across the R/S interface is visible between steady and harmonic computations. By utilizing the non-reflecting treatment the code ensured that non-physical reflections of waves did not occur at the boundary.
Figure 4.14: Steady-state absolute total temperature profiles at the R/S interface

(a) Upstream R/S interface

(b) Downstream R/S interface
Figure 4.15: Harmonic time-averaged absolute total temperature profiles at the R/S interface
At the inlet, total quantities (temperature and pressure) were imposed. The absolute pressure was a constant 396,780 Pa, and the total temperature was modeled as both an average constant value, as well as a profile. The average value of 465.63 K was constructed from the spanwise temperature profile measured at the inlet instrumentation rake. Finally, an inlet kinematic turbulent viscosity of 0.04 m²/sec was used. At the exit, an average static pressure of 109,340 Pa was imposed. Again, this pressure was constructed from the spanwise pressure profile measured at the outlet instrumentation rake. For the solid walls, isothermal wall conditions were used as they have been found to most closely represent the experimental conditions during a short duration experiment. The computations initial conditions were based on estimates of both static pressure at the inlet and static pressure at the R/S interface. Finally, each computation was set to use three multigrid steps (coarse, medium and the final fine grid) each running for 100 iterations. The total maximum number of iterations was set to 5,000 or once the computation reached a global residual of -6.0. The computation was said to be converged once the global residual, pressure ratio, inlet and outlet mass, and the efficiency all stalled. Not all computations reached the global residual convergence limit, but all solutions did have stalled pressure ratio, inlet and outlet mass, and efficiency values. Table 4.2 provides the boundary conditions for the cooled runs. Figure 4.16 shows the inlet temperature profile used.
Table 4.2: Boundary conditions for fully cooled computations

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Avg. Inlet Total Temperature [K]</td>
<td>450.97</td>
</tr>
<tr>
<td>Inlet Total Pressure [Pa]</td>
<td>399,840</td>
</tr>
<tr>
<td>Inlet Static Pressure [Pa]</td>
<td>399,999</td>
</tr>
<tr>
<td>Exit Static Pressure [Pa]</td>
<td>113,360</td>
</tr>
<tr>
<td>Speed [RPM]</td>
<td>12,298.2</td>
</tr>
<tr>
<td>Surface Temperature [K]</td>
<td>306.1</td>
</tr>
</tbody>
</table>

Table 4.3: Boundary conditions for un-cooled computations

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Avg. Inlet Total Temperature [K]</td>
<td>465.63</td>
</tr>
<tr>
<td>Inlet Total Pressure [Pa]</td>
<td>397,780</td>
</tr>
<tr>
<td>Inlet Static Pressure [Pa]</td>
<td>397,600</td>
</tr>
<tr>
<td>Exit Static Pressure [Pa]</td>
<td>109,340</td>
</tr>
<tr>
<td>Speed [RPM]</td>
<td>12,312.5</td>
</tr>
<tr>
<td>Surface Temperature [K]</td>
<td>306</td>
</tr>
</tbody>
</table>

Figure 4.16: Inlet Total Temperature Profile
CHAPTER 5

COMPUTATIONAL RESULTS FOR THE URETI TURBINE STAGE

For this investigation, three parameters were varied in order to generate the computational cases that are discussed in this chapter. The parameters varied were as follows; short or full URETI channel configuration, cooling gas flow on or turned off, and steady-state computation or unsteady harmonic computation. Table 5.1 provides the case matrix for this computational investigation. For each computation, surface predictions of the vane and blade static pressure, and heat transfer were made. Also, the surface Stanton number was calculated in order to compare cases where boundary conditions were changed. The Stanton number was calculated using equation 5.1.

\[ St = \frac{\dot{m}}{A} \left( C_{p,ref} T_{ref} - C_{p,wall} T_{wall} \right) \] (5.1)

where \( q_{wall} \) is the predicted heat-flux, \( \dot{m} \) is the mass flow rate, and \( A \) is the inlet area. The specific heat \( C_{p,ref} \) and absolute total temperature \( T_{ref} \) were taken as average values at the inlet boundary. Finally, \( C_{p,ref} \) and \( T_{wall} \) represented values at the isothermal airfoil surface.

Using the mapping methods previously described, these surface predictions were mapped onto contour plots for each variable. These contour plots provided a view of
the airfoil surface variables as a function of wetted distance and span. Three comparisons were generated for different cases. Each comparison looked at the difference in surface variables between several computational cases. The percent difference was calculated using equation 5.2. In this format, percent difference represents how the first case \((A)\) over / under predicted the surface values as compared to the second case \((B)\). Because each case represents an "improvement" to the previous case, the percent difference can be seen as the "error" in the prediction, had the "improvement" \((B)\) not been implemented.

\[
PD = \frac{A - B}{B}
\]  

(5.2)

Table 5.1: Computational investigation case matrix

<table>
<thead>
<tr>
<th>Case</th>
<th>Computation</th>
<th>Channel</th>
<th>Run</th>
<th>Inlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Steady</td>
<td>Short</td>
<td>Un-cooled</td>
<td>Average</td>
</tr>
<tr>
<td>2</td>
<td>Steady</td>
<td>Full</td>
<td>Un-cooled</td>
<td>Average</td>
</tr>
<tr>
<td>3</td>
<td>Steady</td>
<td>Full</td>
<td>Un-cooled</td>
<td>Profile</td>
</tr>
<tr>
<td>4</td>
<td>Harmonic</td>
<td>Full</td>
<td>Un-cooled</td>
<td>Profile</td>
</tr>
<tr>
<td>5</td>
<td>Harmonic</td>
<td>Full</td>
<td>Cooled</td>
<td>Profile</td>
</tr>
</tbody>
</table>

5.1 Comparison between the Short and Full Channel Mesh

For this comparison, both the short channel and full channel grids were used. Boundary conditions for each computation remained the same so that a direct comparison between the surface predictions could be made. Both computations represent un-cooled conditions using an average inlet temperature. Figures 5.1 and 5.2 show the heat transfer and static-pressure predictions for the vane and the blade for the short channel mesh (case 1), while figures 5.3 and 5.4 show the same surface predictions
for the full channel mesh (case 2). Figures 5.5 and 5.6 provided a percent difference comparison between the surface predictions.

The most noticeable difference between these two computations can be seen in 5.5b. The greatest difference was located towards the trailing edge of the blade suction surface, from 0 to 30% span and 70 to 100% span. Note that in these locations, the short channel seemed to over predict the heat transfer when compared to the full channel. Figure 5.7 provides plots of the heat-transfer predictions at 25% and 75% span, across the entire blade surface. It can be seen that the greatest difference between predictions was between 60 and 95% wetted distance. In some locations, mostly near the hub, this difference was upwards of 45%. Figure 5.5a shows that the vane surface predictions were relatively unchanged except for locations near the shroud on the suction surface. It is believed that the addition of the inlet channel in case 2 allowed for the development of the flow before entering the turbine section. This resulted in a more distributed temperature profile at the vane and blade inlet. This distributed temperature profile accounted for the reduction in heat transfer at the hub and shroud, which again, was most noticeable on the blade suction surface. In general, these results demonstrated that modeling the entire channel not only allowed for the placement of inlet and outlet boundary conditions at physically accurate distances for the turbine section - impacting the heat-transfer predictions across the blade surface - but is considered to be a better model of the freesteam flow before entering the turbine section. Figure 5.8 is a plot of both heat transfer and static-pressure predictions for the URETI vane at 50% span. These two figures illustrate that the vane surface predictions had less variation between the two cases. Also note that from 5.6 it can be seen that the static-pressure predictions were similar between case 1 and 2, not only across the vane, but also the blade, as seen in figure 5.9.
Figure 5.1: Case 1: Heat-transfer prediction - un-cooled, steady, short channel mesh with average inlet temperature
Figure 5.2: Case 1: Static-pressure prediction - un-cooled, steady, short channel mesh with average inlet temperature
Figure 5.3: Case 2: Heat-transfer prediction - un-cooled, steady, full channel mesh with average inlet temperature
Figure 5.4: Case 2: Static-pressure prediction - un-cooled, steady, full channel mesh with average inlet temperature
Figure 5.5: Heat transfer comparison between cases one and two
Figure 5.6: Static pressure comparison between cases one and two
(a) URETI blade at 25% span

(b) URETI blade at 75% span

Figure 5.7: Heat Transfer comparisons between case 1 and 2 for the un-cooled URETI blade
Figure 5.8: Comparisons between case 1 and 2 for the un-cooled URETI vane
5.2 COMPARISON BETWEEN UN-COOLED STEADY-STATE AND UN-COOLED HARMONIC PREDICTIONS

Comparison between the un-cooled steady-state (case 3) and un-cooled harmonic (case 4) predictions illustrated that there was a difference between the steady-state prediction and the time-averaged, harmonic prediction. Figures 5.10 to 5.11 provide contour plots of the vane and blade surface in case 3, while Figures 5.12 to 5.14 provide the same information for case 4. Because these two cases had the same boundary conditions, a direct heat transfer comparison was made. Figure 5.15 shows this comparison (difference in heat transfer) between the steady and harmonic predictions. For the vane, the greatest difference was seen downstream of the sonic line (approximately 50% wetted distance). For the blade, the steady solution over predicted the heat transfer in the leading edge region, on the suction surface. Note that on the
pressure surface the steady solution also over predicted the harmonic heat-transfer prediction; however, at -40% wetted distance, this relationship changed. Figure 5.16a also shows that at -40% wetted distance and 50% span, the harmonic solution predicted a greater rise in the surface heat transfer. From this result it was understood that the pressure surface of the blade was very dependent on the time-varying nature of the system. Therefore, reliable predictions in this region would only come from unsteady computations.

Figure 5.16 shows both the heat transfer and static-pressure predictions for cases 3 and 4, as well as previous blade surface predictions. From this figure, it is possible to see that the new steady predictions agreed well with the previous predictions. This figure also shows that for the heat transfer and static-pressure predictions, the harmonic computation underpredicted the steady results at the leading edge. Figure 5.17 demonstrates that for the vane static-pressure prediction, upstream of the sonic line (50% wetted distance), the two predictions were almost identical. This was expected because upstream of the sonic line, the vane does not "see" the passing blade. This variation in the two cases showed that downstream of the vane sonic line, an unsteady, time-averaged prediction was necessary to adequately capture the time varying perturbations, which were a function of the blade passing frequency. In summary, these results demonstrated that in order to capture the full flow characteristics, particularly on the blade, an unsteady solution would be required. Again, these results identified the difference between the steady-state solution and the time-averaged, harmonic solution. Although differences in these predictions across the vane were small, the reason for this similarity was understood. For predictions across the blade surface, pressure surface results had the greatest variance between the two solutions. This variation required that harmonic solutions be obtained for both the un-cooled and
cooled computations in order to make comparisons between the predictions and the experimental results.
Figure 5.10: Case 3: Heat-transfer prediction - un-cooled, steady, full channel mesh with inlet temperature profile
Figure 5.11: Case 3: Static-pressure prediction - un-cooled, steady, full channel mesh with inlet temperature profile
Figure 5.12: Case 4: Heat-transfer prediction - un-cooled, harmonic, full channel mesh with inlet temperature profile
Figure 5.13: Case 4: Static-pressure prediction - un-cooled, harmonic, full channel mesh with inlet temperature profile
Figure 5.14: Case 4: Stanton number predictions - un-cooled, harmonic, full channel mesh with inlet temperature profile
Figure 5.15: Heat Transfer comparison between case 3 and 4
(a) Blade heat transfer at 50% span

(b) Blade static pressure at 50% span

Figure 5.16: Comparisons between case 3 and 4 for the URETI blade
Figure 5.17: Comparisons between case 3 and 4 for the URETI vane, static pressure at 50% span

5.3 COMPARISON BETWEEN HARMONIC PREDICTIONS AND EXPERIMENTAL RESULTS

For this comparison, the cooled and un-cooled harmonic predictions (cases 4 and 5) were compared to the measurements. These harmonic predictions represent the best case scenario for making reliable comparisons. For the cooled case, data was taken from run 11 of the URETI dataset. For the un-cooled case, data was taken from runs 7 and 13 of the URETI dataset. This data included two variants of the blade. During the experimental investigation, data was collected on blades that had cooling holes, as well as blades that had the cooling holes blocked (filled blades). These "filled blades" best represent the un-cooled computations. Due to the fact that the hole geometry was not modeled, it was expected that the computational model would not capture the full flow characteristics near the airfoil surface, especially when the
cooling mass flow was turned off (in the model, when the coolant is turned off the holes "disappear" from the surface). The flat plate un-cooled data also seemed to support this hypothesis. By looking at data collected on the filled blades, it was possible to make a more realistic comparison between the computational domain and the experiment.

Again, figures 5.12 through 5.14 provide heat transfer, static pressure, and Stanton number surface predictions for case 4 (un-cooled). Figures 5.18 through 5.20 provide heat transfer, static pressure, and Stanton number surface predictions for case 5 (cooled). Figure 5.21 provides a comparison between the un-cooled and the cooled surface Stanton number. For all cooled surface plots, the approximate locations of the cooling holes are shown as white circles. The vane trailing edge cooling is not marked on these plots, but this cooling region can be seen in figure 5.18 at -95% wetted distance.

From figure 5.24 it is seen that the new harmonic predictions are an improvement over previous heat transfer (figure 5.24a), and static-pressure predictions (figure 5.24b) for the URETI blade. Notice that both the vane and blade static-pressure predictions did agree with the measurements. Blade static-pressure predictions in figure 5.24b under predicted the data on the pressure surface of the blade, at approximately -25% to -45% wetted distance, but still represented an improvement over previous predictions. Static-pressure predictions for the vane, shown in figure 5.26, captured both the trend and magnitude of the data. In short, the static-pressure predictions were considered to represent reliable results when compared to the data. It is important to note that with these results, it is now considered possible to compare fully cooled static pressure data with fully cooled static-pressure predictions.
With regards to the heat-transfer predictions, as can be seen in figure 5.24a, both the un-cooled and cooled predictions under predicted the heat transfer data; most noticeable on the pressure surface of the blade. Although these predictions did represent an improvement when compared to the previous heat-transfer predictions, these new predictions still missed the blade pressure surface heat transfer "bump". The increased inlet turbulence did improve the predictions, especially on the blade pressure surface. It should also be noted that although the cooled predictions under predicted the run 11 data on the pressure surface, the prediction did match the trend in the data. Also notice that before each cooling row, the heat transfer increased. It is possible that the coolant flow acted as a blockage to the free stream flow moving across the surface of the blade, particularly in the viscous boundary layer (very close to the surface). This blockage caused the free stream flow to stall, generating instabilities in the flow field, which allowed the hot gases to migrate towards the surface, thus increasing the thermal potential and heat transfer. These localized increases can be seen in figure 5.18b at 50% span and -60% wetted distance.

For the filled blades, figure 5.25 shows the comparison between data taken from the filled blades at 50% span, and the un-cooled prediction. Because of the limited number of data points, it was difficult to draw any solid conclusions. It was believed that comparing the filled blade data to the unfilled blade data would help determine if the pressure surface heat transfer bump was caused by ingestion at cooling rows 8 and 1, but unfortunately no data was obtained in this region. Notice that for the first time, the prediction over predicted the data at -75% wetted distance for both the run 7 and 13 measurements. This result, along with the suction surface results were encouraging, but the run 7 data point located at -30% wetted distance still suggested that the heat transfer bump was present on the filled blade, and thus not an effect
caused entirely by the presence of cooling holes, or cooling hole ingestion.

Because data was collected along several lines of constant span, figures 5.22 and 5.23 were generated to show both the measurements along with the surface prediction contour plots. Each rectangle represents the approximate location of experimental instrumentation, either a pressure transducer or a heat-flux gauge. These two plots show harmonic predictions for case 4 and 5 blades. Both the heat transfer and static-pressure predictions are shown along with run 11 (cooled) and run 13 (un-cooled and unfilled) measurements.
Figure 5.18: Case 5: Heat-transfer prediction - cooled harmonic, full channel mesh with inlet temperature profile
Figure 5.19: Case 5: Static-pressure prediction - cooled harmonic, full channel mesh with inlet temperature profile
Figure 5.20: Case 5: Stanton number predictions - cooled harmonic, full channel mesh with inlet temperature profile
Figure 5.21: Stanton number comparison between case 4 and 5
Figure 5.22: Case 4: Un-cooled, harmonic, blade surface predictions with run 13 experimental results
Figure 5.23: Case 5: Cooled, harmonic, blade surface predictions with run 11 experimental results
Figure 5.24: Comparisons between case 4 and 5 for the URETI blade

(a) Blade heat transfer at 50% span

(b) Blade static pressure at 50% span
Figure 5.25: Comparisons between case 4 and filled blade heat transfer data at 50% span

Figure 5.26: Comparisons between case 4 and 5 for the URETI vane, static pressure at 50% span
To better develop conclusions from this work, it is important to once again review the goals of this investigation. This study was designed to investigate both aerodynamic and heat-transfer performance of a single stage, high-pressure, fully cooled turbine. The goal was to advance earlier computational work performed at the GTL, specific to the URETI program, and also to answer several basic questions regarding predictions of a fully cooled, single-stage turbine. Firstly, what details should be considered when performing a computational investigation, especially when attempting to obtain heat-transfer predictions? Secondly, when utilizing the distributed point source cooling method, was the code able to make reliable surface predictions? Thirdly, how should one apply this cooling model to the current URETI turbine stage? Finally, did this new turbine CFD model provide reliable predictions, and what could be learned from these new results?

The $y+$ sensitivity study demonstrates that during the mesh development, it is important to place a sufficient number of cells close to the solid surface within the viscous sublayer. Further, having a $y+$ range between 1 and 10 is critical when using the Spalart-Allmaras turbulence model. From this analysis it was shown that beyond a $y+$ of 5, error in the calculations increases rapidly, and although it was seen
that maximum $y+$ values of 10 were a valid design approximation, $y+$ values of 12 introduced significant error (approximately 25%). Previous studies used $y+$ ranges with maximum limits upwards of 100, but these studies focused on making pressure predictions and the importance of small $y+$ surface values was significantly diminished for these cases. It was shown that surface pressure predictions vary only slightly at regions of large gradients with changes in $y+$. Therefore, it is understandable that reliable pressure predictions were possible with $y+$ ranges well outside the required limit. It is important to remember that although it is possible to design for a $y+$ range by setting the minimum distance between the solid surface and the first cell center, $y+$ is also a function of the flow field characteristics. During these computations, the true range of $y+$ was unknown until the computation had converged, thus the design of the grid was an iterative process.

The use of a full-channel model represented a more precise representation of the experimental setup. Not only does the full-channel model allow for the placement of the inlet and outlet conditions at correct distances from the turbine section, it also reflects the development of the flow prior to entering the vane and blade. This flow development was seen to not only impact the heat transfer distribution across the vane, but also the blade. Originally, the magnitude of the difference was unknown, but subsequent comparisons demonstrated that near the hub and shroud on the blade suction surface this difference was large (as much as 45% in some regions). Therefore, a case can be made for modeling the complete geometry of the experiment. With increased computational resources, it is now possible, and necessary, to consider the full geometry.
Another important consideration that was developed during this work was the use of surface geometry corrections. Such corrections were required when making comparisons between measurements and predictions. Because pressure transducers and heat-flux gauges were located on the airfoil surface using percent wetted distance and percent span, it was important to make sure that the computational results agreed with this geometry scheme. It was found that the "raw" FINE/Turbo predictions did not directly correspond to the physical location of the instrumentation, thus the predictions required a correction. For this study, the surface predictions were interpolated onto a new, physically accurate model of the airfoil surfaces. These new surfaces not only preserved the magnitude of the prediction, but also provided a prediction in terms of percent wetted distance and span. With this new surface mapping, it is now possible to make comparisons between surface predictions and surface data.

For the first time at the GTL, a more accurate interpretation of the inlet turbulence intensity was developed, and implemented, for the URETI computational investigation. When using the FINE/Turbo code with the Spalart-Allmaras turbulence model, the magnitude of the inlet turbulence intensity is specified by setting the kinematic turbulent viscosity. Previous models utilized either default settings or best practice approaches when setting this inlet condition, but this investigation utilized a correlation developed for the Spalart-Allmaras model to relate the kinematic turbulent viscosity to the experimental turbulence intensity. A turbulence intensity study showed that increasing the inlet kinematic turbulent viscosity increased the heat-transfer prediction across the blades pressure surface. In conclusion, when designing a computation, it is important to consider all inlet, outlet, and boundary conditions. All approximations and assumptions of these parameters must be considered.
Again, there are several methods for the implementation of film cooling within a CFD model. Each method is fundamentally different, and represents a certain level of added complexity to the overall computational model. For this investigation, the distributed point source cooling method was utilized. Previous URETI investigations used the lumped mass model to compare predictions to cooled experimental data, and the use of distributed point source cooling technique used here represented the next choice in the progression of model complexity. The flat plate experimental investigation provided a dataset to compare with cooled predictions. These initial cooling predictions represented an idealized case, one that did not capture the true complexities of a rotating airfoil. But this study did represent the first steps towards utilizing distributed source term injection for computational investigations at the GTL.

The work described in this thesis helped develop a better understanding of how to model airfoil film cooling, and provided confidence in both the distributed source term method and FINE/Turbo as a design tool. This study did call into question the validity of heat-transfer predictions close to cooling holes, but experimental data in this region was limited. From these results it was concluded that future experimental studies should increase instrumentation density in the immediate vicinity of the cooling holes. It was also believed that because this cooling model did not include the hole geometry, comparisons between un-cooled measurements and un-cooled predictions were inconsistent. The presence of holes at the surface appeared to disturb the flow, which impacted the measured heat transfer, but un-cooled predictions could not capture this. For future studies, comparisons between un-cooled predictions and datasets taken without the presence of hole geometry (filled blade data) should be considered.
Extending the existing Fine/Turbo cooling model to a cooled full-stage turbine presented several challenges. Among these challenges, the development of a cooling scheme for individual cooling rows was particularly important. During the experiment, total coolant mass flow rates were measured at the inlet of the vane and blade cooling plenums. These flow rates were found to be consistent with the pre-experiment flowing of several independent vanes and blades. Individual hole or individual row cooling flow rates for the vane and blade were unknown. It therefore became important to develop a method for determining these flow rates. The initial technique included simply using hole areas, and row hole counts, to determine the percentage of coolant from each row. Although this method seemed to work well for the vane cooling scheme, it was insufficient for the blade. A second method for assigning cooling to blade cooling rows was based on the pressure ratio between the internal plenum and the exterior pressure data. This method accounted for the fact that more cooling would be present on the suction side of the blade, rather than the pressure side (given the same effective flow area). Although this method accounted for the differences between the two surfaces, and also accounted for the fact that certain rows were choked, it resulted in a coolant distribution which assigned too much mass flow to holes that were possibly ingesting hot gas, not injecting coolant (rows 1 and 8 to be specific). Therefore, a third method was proposed. The basis for this scheme was to treat each hole on the blade as if it were choked, thus a maximum coolant model was developed. This method better distributed mass across the surface, and provided a more realistic interpretation of the coolant distribution.

Both the vane and the blade cooling models made a 2D approximation by assuming that each hole in a given row had the same mass flow rate. It was understood that the pressure distributions, both within the cooling chamber and across the surface, were
not two-dimensional; surface pressure predictions support this. Future cooling models should aim to assign each hole an individual mass flow rate based on the pressure ratio. Such a model would require a better understanding of the internal cooling passage pressure distribution, as well as more external pressure data. But it must be considered that some basic shortcomings of the current distributed source term method are that, it cannot account for the coupling between the holes (holes are linked via the internal cooling passage), nor can it account for time-varying perturbations in the coolant flow rates. Blade internal pressure measurements obtained as part of the URETI dataset show a pulsing within the plenum. This pulsing suggests that the coolant mass flow rate is a function of the unsteady flow features. Both the lack of coupling and the time dependent flow rates suggest that the current cooling model requires some improvements.

One way to account for this coupling between holes, as well the relationship between airfoil internal and external conditions would be to extend the grid through each coolant hole into the plenum. Currently, there are investigations being conducted at the GTL that are looking at extending the grid into cooling cavities. Although these investigations are being performed on turbine geometries other than URETI, the results will provide insight into the application of such cooling models, as well as the codes ability to implement these larger and more complex grids. If such methods are to be investigated in the future, the use of an unstructured mesh should be considered. Currently, FINE/Turbo does have an unstructured meshing tool, but all computational investigations at the GTL have utilized structured grids. In order to decrease cell size, while increasing the total number of cells, an unstructured gridding technique could result in a more computationally efficient, and stable, mesh.
From the computational results, it was seen that for static-pressure predictions, the new harmonic predictions represented an improvement over steady predictions. For the vane, almost no difference, across the pressure surface or most of the suction surface (upstream of the sonic line), was seen between the harmonic and steady predictions. This was expected due to the fact that these surfaces were upstream of the sonic line, and therefore could not "see" the blade passing. Because the unsteady nature of such a system is driven by the blade passing, the harmonic method was an efficient choice over a full, time-accurate solution. These results showed that the harmonic time-averaged solution was not the same as the steady solution, thus when making comparisons to the experimental results, an unsteady solution should be used.

Looking at the comparison between the cooled data and the cooled predictions, it is now considered possible to make pressure comparisons between cooled data and cooled predictions. Although the new heat-transfer predictions represented an improvement over previous predictions, these predictions still underpredicted the data, especially across the pressure surface of the blade. This underprediction of the data continued for the cooled cases, but there is similarity between the prediction and data profiles. It was unclear during the investigation if the presence of the un-cooled pressure surface heat transfer "bump" was being caused by the cooling hole geometry. It was considered possible that even without a coolant flow, the holes could cause disturbances in the flow, leading to increased heat transfer. To help answer this question, the un-cooled predictions were compared to data taken from blades that had their coolant holes filled in. It was difficult to make definitive conclusions from this comparison because only a small amount of data was captured on the filled blades. From this limited un-cooled dataset, a better agreement between the un-cooled predictions and the data was seen. The data suggested that even with no
cooling holes, the same pressure surface heat transfer "bump" was present. Although more data should be collected on filled blades, this comparison suggests that the reason(s) for the discrepancy between the un-cooled predictions and data is not simply the presence of cooling holes.

For now, when compared to the data, both cooled and un-cooled heat-transfer predictions across the blade are not considered to be very good. Even though this study did not result in the generation of good agreement between heat-transfer predictions and measurements, it has provided a systematic evaluation of the parameters and details associated with the development of computational models for full-stage high-pressure turbine geometries. These results suggest that there are still more significant modeling parameters to consider when generating reliable heat-transfer predictions. Future computational efforts can use this investigation as a starting point and strive to develop and understanding of the "other significant modeling parameters", as well as develop an understanding of other modeling techniques. It is not possible to completely remove modeling assumptions from the equation, but through understanding these assumptions, their implications, and rigorously examining predictions against measurements, advancements in computational predictive methods will be possible.
Bibliography


