Time-Averaged and Time-Accurate Aerodynamic Effects of Rotor Purge Flow for a Modern, Rotating, High-Pressure Turbine Stage and Low-Pressure Turbine Vane

Dissertation

Presented in Partial Fulfillment of the Requirements for the Degree Doctor of Philosophy in the Graduate School of The Ohio State University

By

Brian Richard Green, M.S.

Graduate Program in Mechanical Engineering

The Ohio State University

2011

Doctoral Advisory Committee:

Dr. Michael G. Dunn, Adviser

Dr. Jen Ping Chen

Dr. Sandip Mazumder

Dr. Mohammad Samimy
ABSTRACT

Rotor purge flow cavity seals are used in gas turbine engines to prevent ingestion of the mainstream gas flow into the purge cavity. Ingestion into this cavity leads to an increase in the cavity air temperature and subsequently to the rotor disk and stator metal temperatures leading to higher thermal stresses and reduced disk and stator fatigue life. An over designed cavity seal with an excess amount of purge flow has the downside of increasing engine and aircraft fuel consumption through reduced turbine efficiency. The opposite approach of strengthening the hardware to withstand the higher stress and temperatures would increase the weight of the overall propulsion system. Understanding how the purge flow cavity and cavity seals interact with the mainstream gas is important to producing a balanced design among weight, fuel consumption, efficiency, and fatigue life of surrounding hardware.

The main objective of this research was to address the need for purge flow cavity and rim seal data on engine representative hardware at the proper design corrected operating conditions. This was accomplished through an experimental and computational study of a one and one half stage high-pressure turbine installed at The Ohio State University Gas Turbine Laboratory Turbine Test Facility. The rig housing the turbine stage incorporated many features found in a typical commercial high-pressure turbine such as a cooled high-pressure vane row with hub and shroud cooling, a downstream blade row followed by a downstream vane row, the ability to created elevated radial inlet temperature profiles using a combustor emulator, and a cooling supply line to the purge cavity capable of introducing metered cooling air into that cavity. Multiple runs were performed to study the effects of cooling flows from both an aerodynamic and heat transfer perspective and incorporated instrumentation throughout the rig in order to capture time-
accurate temperature, pressure, and heat flux measurements on the high-pressure vane, high-pressure rotor and the cavity rim seal area. The run matrix included cold rig configurations with no cooling flow, high-temperature uniform inlet profiles at the vane inlet for cases with and without cooling flows, and high-temperature radial inlet profiles with and without cooling flows. The computational study was performed using the Numeca FINE/Turbo code utilizing a multiple blade row model with both steady and harmonic unsteady technique in order to simulate the physics of the experiment including the high-pressure vane, hub and shroud film cooling via a source term injection model, matching inlet conditions to either cold profiles, elevated flat and elevated radial inlet temperature profiles, and isothermal wall boundary conditions in order to accurately model the thermal boundary layers.

Comparisons between the data and the computational results were performed for five different operational conditions: cold inlet and no cooling flow run, an elevated radial inlet temperature profile with purge and without purge cooling, and an elevated flat inlet temperature profile with purge and without purge cooling flow. The solutions were found to match very well to the exit rake measurements from the experiments for both total pressure and total temperature, which builds good confidence in FINE/Turbo’s ability to capture losses accurately through three blade rows. In addition, the leading edge blade profile in the rotating frame of reference also showed excellent agreement with the measured profile with the exception of a local dip in temperature seen only in the data for all runs analyzed. Additionally, time-averaged and time-accurate comparisons of static pressure on the vane hub, stationary cavity wall, and rotating cavity wall were performed for the computational and experimental results and showed very good agreement at all gage locations and operating conditions. Issues were noted at the stationary shroud location above the rotating blade and for the flat inlet profile cases in the cavity region but overall a good comparison was achieved. Time-average comparisons were shown for the thermocouples located on the blade platform and in the stationary and rotating side of the cavity. For the two radial inlet cases and for the cold inlet case, these comparisons showed very good agreement while the two flat inlet profile cases showed that the computational models general under-predicted the static temperature levels both on the rotor platform and
in the cavity. This has been attributed to the isothermal wall boundary conditions such that the temperatures applied to the model based on RTD measurements from the rig are suspected to be low based on the high-pressure vane double-sided heat-flux gage measurements. Increasing the vane hub temperatures did bring the stationary side cavity temperatures up closer to the measured results but had minimal effect on the rotor platform thermocouples.

The cavity rim seal ingestion/ejection phenomena and the aerodynamic impact to the turbine were studied in detail using the computational model and supporting experimental data where appropriate. The presence of the cavity and purge flow cause additional blockage in the high-pressure vane resulting in increased suction side loading at the hub and reduced exit tangential velocity. And as the flow passes over the cavity and mixes with the lower momentum purge flow, the tangential velocity, total temperature and total pressure are further reduced leading to increased incidence angles on the blade suction surface and a reduction in power extraction in the hub region. Ultimately, this reduces adiabatic efficiencies of the turbine when compared to the computational case with no cavity. The ingestion and ejection zones around the annulus at the purge flow and main gas path interface were found to be due to a combination of the high-pressure vane wake, the rotor leading edge bow wake, and the interaction between these two elements. For the run studied, ingestion in the rim seal was not considered deep as the ingested flow did not reach the main portion of the cavity and general did not penetrate beyond the top of the rotor side angel wing.
DEDICATION

To my daughter, Eleanor Claire … May all of your life’s dreams be easily within your reach.
ACKNOWLEDGEMENTS

First of all, I would like to thank Dr. Michael Dunn of The Ohio State University Gas Turbine Laboratory for giving me the opportunity to be a part of the experimental program that spawned the research within. Without his forethought this research program would not be possible. Also, I would like to thank Dr. Charlie Haldeman for your interest, ideas, and help on trouble shooting and understanding all of the various aspects of the experiments. You will be greatly missed at the laboratory Charlie, good luck in your new adventure. And to the entire faculty, staff, and students at The Ohio State University Gas Turbine Laboratory who’s hard work and perseverance have made the experimental program a success.

I would like to personally thank Dr. Randall Mathison. You introduced me to FINE/Turbo and everything took off from there. You took great care of me down to the very last run. From the occasional reboot of the machines to late night phone calls trying to solve the day’s problems to coming in on the weekends to shuffle my runs around or making a late night stop on the drive through during Christmas. I can’t thank you enough for all of your help, wisdom, and thoughts on how to overcome the next challenge we faced (or would face). I did my best to find every breaking point of the computer lab and cluster and you continued to find fixes and workarounds. We learned a lot during the few years of tangling with the computer system and my only wish is to leave you with a stronger, more bulletproof system for future work to come. Thank you for everything Randy.
I would also like to thank Roque Lopez, Autumn Fjeld, and Alain Demeulenaere of NUMECA USA Inc for all of the support. From the occasional “panicked in the middle of the afternoon” phone call to the constant barrage of emails, you kept me running solid the entire time while at the same time helping me to learn the ins and outs of FINE/Turbo. Thank you all and keep the wonderful support going.

Outside of this research, I would like to thank my entire family for all of your support while I trekked through this endeavor along with several other side adventures along the way. I knew this day would come; I was just never sure what date to give you.

To my wife Lori, your endless support throughout both my academic and professional career has been nothing short of amazing. You kept me going when I needed it most and pushed me through to the very end. Many nights were spent looking at each other from across the room while I worked and I knew I had your support during every moment. You took such great care of our daughter Eleanor each night when it was time for her to sleep and for me to work; I cannot thank you enough for the endless support you have always provided. And to both my wife and daughter, thank you for putting up with my mess at the end of the dining room table. It should be gone soon!
VITA

April 12, 1977 ................................................. Born – Lansing, Michigan

1999 ............................................................ B.S. Mechanical Engineering,

Michigan State University

2000 – present ................................................. Mechanical Engineer, GE Aviation

2004 ............................................................ M.S. Mechanical Engineering,

The Ohio State University

2005 – present ................................................. Graduate Research Associate,

The Ohio State University

PUBLICATIONS


FIELDS OF STUDY

Major Field: Mechanical Engineering
TABLE OF CONTENTS

Abstract .................................................................................................................................................. ii

Dedication ............................................................................................................................................... v

Acknowledgements .............................................................................................................................. vi

Vita .......................................................................................................................................................... viii

Publications ............................................................................................................................................ viii

Fields of Study ....................................................................................................................................... ix

List of Tables .......................................................................................................................................... xiv

List of Figures ......................................................................................................................................... xv

List of Symbols ....................................................................................................................................... xxvii

Chapter 1 Introduction .......................................................................................................................... 1

1.1 Review of Purge Flow Cavity Studies ............................................................................................... 11

1.2 Scope of the Current Study ................................................................................................................ 29

Chapter 2 Governing Equations and Solution Techniques .................................................................... 32

2.1 Equations of Motion for Fluid Flow .................................................................................................. 32

2.1.1 Conservation of Mass & the Continuity Equation ....................................................................... 32

2.1.2 Conservation of Momentum and Navier-Stokes Equations......................................................... 35

2.1.3 Conservation of Energy and the Energy Equation ....................................................................... 41

2.1.4 Additional Equations for Closure ............................................................................................... 45

2.2 Closed Form Solutions for Rotor-Stator Cavities .......................................................................... 48

2.2.1 Forced Vortex ............................................................................................................................... 50

2.2.2 Free Vortex .................................................................................................................................. 53

2.2.3 Concluding remarks on the Free and Forced Vortex Closed Form Solutions ............................ 54
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.3</td>
<td>Computational Domain and Grid</td>
<td>198</td>
</tr>
<tr>
<td>5.4</td>
<td>Flow Conditions</td>
<td>211</td>
</tr>
<tr>
<td>5.5</td>
<td>Boundary Conditions</td>
<td>212</td>
</tr>
<tr>
<td>5.6</td>
<td>Solution Techniques and Convergence Criteria</td>
<td>232</td>
</tr>
<tr>
<td>6.1</td>
<td>Data Reduction and Analysis</td>
<td>239</td>
</tr>
<tr>
<td>6.2</td>
<td>Low-Pressure Turbine Vane Exit Plane</td>
<td>242</td>
</tr>
<tr>
<td>6.3</td>
<td>High-Pressure Turbine Vane</td>
<td>256</td>
</tr>
<tr>
<td>6.3.1</td>
<td>Trailing Edge Static Pressure</td>
<td>256</td>
</tr>
<tr>
<td>6.4</td>
<td>High-Pressure Turbine Rotor</td>
<td>267</td>
</tr>
<tr>
<td>6.4.1</td>
<td>Mid-Span Blade Surface Static Pressure</td>
<td>268</td>
</tr>
<tr>
<td>6.4.2</td>
<td>Stationary Shroud Static Pressure</td>
<td>271</td>
</tr>
<tr>
<td>6.4.3</td>
<td>Blade Leading Edge Total Temperature</td>
<td>277</td>
</tr>
<tr>
<td>6.4.4</td>
<td>Blade Platform Static Temperature</td>
<td>286</td>
</tr>
<tr>
<td>6.5</td>
<td>High-Pressure Turbine Vane/Blade Purge Cavity</td>
<td>293</td>
</tr>
<tr>
<td>6.5.1</td>
<td>Stationary Static Pressure</td>
<td>295</td>
</tr>
<tr>
<td>6.5.2</td>
<td>Stationary Static Temperature</td>
<td>305</td>
</tr>
<tr>
<td>6.5.3</td>
<td>Rotating Static Pressure</td>
<td>311</td>
</tr>
<tr>
<td>6.5.4</td>
<td>Rotating Static Temperature</td>
<td>326</td>
</tr>
<tr>
<td>6.6</td>
<td>Concluding Remarks on the Experimental and Computational Comparisons</td>
<td>336</td>
</tr>
<tr>
<td>7.1</td>
<td>One-Dimensional Closed-Form Analytical Model</td>
<td>341</td>
</tr>
<tr>
<td>7.2</td>
<td>Rim Seal Ingress / Egress and Comparison to Published Correlations</td>
<td>345</td>
</tr>
<tr>
<td>7.3</td>
<td>Cavity and Rim Seal Flow Field</td>
<td>349</td>
</tr>
<tr>
<td>7.4</td>
<td>Efficiency and Aerodynamic Impact of Purge Flow</td>
<td>382</td>
</tr>
<tr>
<td>7.5</td>
<td>Concluding Remarks about the In-Depth Analysis</td>
<td>392</td>
</tr>
<tr>
<td>8.1</td>
<td>Conclusions</td>
<td>394</td>
</tr>
</tbody>
</table>
8.1 Conclusions from the Research .................................................................................. 394
8.2 Reccomendations for Future Research .................................................................... 399
8.3 Final Remarks ............................................................................................................. 401
References ...................................................................................................................... 403

Appendix A: Convergence Plots for the NACA0012 Airfoil ............................................ 408
Appendix B: NASA Rotor 37 and NASA Stage 35 Convergence Plots ........................... 417
Appendix C: Flow path and Strut Analysis for the Uncooled One and One-Half High-Pressure Turbine Rig Boundary Conditions................................................................................................. 421
Appendix D: Uncooled One and One-Half Stage High Pressure Turbine Rig Convergence Plots ................................................................. 429
Appendix E: Convergence Plots for All Run 21, Run 22, Run 28, Run 33, and Run 43 Numerical Models ........................................................................................................................................ 435

xiii
LIST OF TABLES

Table 1. Default positioning of the inlet and exit rakes.................................................................69
Table 2. Height of thermocouple junction above the surface..........................................................76
Table 3. Rig operating conditions for all runs studied........................................................................83
Table 4. NACA0012 Cases and Conditions......................................................................................109
Table 5. Rig operating conditions .....................................................................................................156
Table 6. Rig conditions analyzed using Fine/Turbo........................................................................212
Table 7. Purge flow cavity inlet boundary conditions normalized to inlet-averaged values..............223
Table 8. HR142 gage statistics over each time window normalized to inlet-averaged total temperature..228
Table 9. Isothermal surface temperatures normalized to the inlet-averaged total temperature as applied to
       the computational model..............................................................................................................229
Table 10. Comparison the measured rig and the FINE/Turbo analytically predicted operating conditions for
         all runs analyzed........................................................................................................................243
Table 11. Maximum-to-average comparison between the inlet and exit total temperature for the analytical
         results ........................................................................................................................................249
Table 12. Non-dimensional cavity parameters for run 28..................................................................346
Table 13. Efficiency and flow parameter impact for the run 28 purge, no purge, and no cavity cases.....383
LIST OF FIGURES

Figure 1. T-s Diagram with both the Ideal and Real Brayton Cycles.........................................................1
Figure 2. Cross-section of a modern, civil aviation, gas turbine engine.........................................................3
Figure 3. (a) Axial, (b) radial, (c) double lip radial stationary-rotating air seal configurations.........................5
Figure 4. Typical Rotor-Stator Purge Flow Cavity Flow Field........................................................................8
Figure 5. Differential element and coordinate system for the development of the conservation of mass....33
Figure 6. Surface force and its components acting on an arbitrary surface..................................................36
Figure 7. x-Direction notation and sign convention for: (a) positive stresses and (b) negative stresses ....37
Figure 8. Differential element for the application of conservation of energy principal..................................42
Figure 9. Generic Cavity Model......................................................................................................................48
Figure 10. Total temperature distribution within the Rotor-Stator Cavity for a free and forced vortex

assumption....................................................................................................................................................55
Figure 11. Static temperature distribution within the Rotor-Stator Cavity for a free and forced vortex

assumption....................................................................................................................................................56
Figure 12. Total pressure distribution within the Rotor-Stator Cavity for a free and forced vortex

assumption....................................................................................................................................................57
Figure 13. Static pressure distribution within the Rotor-Stator Cavity for a free and forced vortex

assumption....................................................................................................................................................57
Figure 14. Tangential velocity distribution within the Rotor-Stator Cavity for a free and forced vortex

assumption....................................................................................................................................................58
Figure 15. Schematic of the Turbine Test Facility.............................................................................................62
Figure 16. Turbine rig cross-section used for the experimental program (Not To Scale)...............................64
Figure 17. Turbine rig assembly prior to installation into the dump tank .............................................65
Figure 18. Combustor emulator honeycomb structure .................................................................66
Figure 19. Inlet to the combustor emulator .......................................................................................67
Figure 20. Rake positions for both the inlet and exit locations, forward looking aft.......................68
Figure 21. High-pressure vane trailing edge Kulites installed in the hub surface ...............................71
Figure 22. High-pressure vane stationary purge flow cavity instrumentation ....................................72
Figure 23. High-pressure rotor leading edge shrouded thermocouples .............................................73
Figure 24. High-pressure rotor platform thermocouple positions .....................................................74
Figure 25. High-pressure rotor platform thermocouple (a) as installed and (b) post-experiemnt ..........75
Figure 26. High-pressure rotor rotating purge flow cavity instrumentation locations .....................77
Figure 27. High-pressure rotor blade surface and leading edge instrumentation ............................78
Figure 28. Typical boundary conditions for a domain scaling method unsteady calculation ...........102
Figure 29. Typical boundary conditions as applied on a phase-lag unsteady calculation ...............103
Figure 30. Typical boundary conditions as applied for the non-linear harmonic unsteady method for the
          (A) time-averaged solution, (B) harmonic solution .....................................................................105
Figure 31. Percent difference between the theoretical and actual experimental airfoil model ..........110
Figure 32. Block structure for the NACA0012 airfoil: (a) far field, (b) near airfoil view ..................111
Figure 33. Grid density for the NACA0012 Airfoil: (a) Far View, (b) Near View, (c) Mach 0.3 y+ values,
          (d) Mach 0.83 Number y+ values ..............................................................................................112
Figure 34. Case number five: (a) RMS Residual history, (b) inlet and exit mass flow rate ...............116
Figure 35. Pressure Coefficient on the airfoil surface for: (a) 0.3 Mach at 0°, (b) 0.3 Mach at 4°, (c) 0.5
          Mach at 0°, (d) 0.5 Mach at 4°, (e) 0.5 Mach at 8°, (f) 0.75 Mach at 0°, (g) 0.75 Mach at 2°, (h) 0.75
          Mach at 4°, (i) 0.83 Mach at 0° ...............................................................................................117
Figure 36. Z-R plane of the Rotor 37 Flow Path .............................................................................125
Figure 37. Rotor 37 (a) grid density and (b) Y+ values at design point conditions .........................126
Figure 38. Z-R plane of the stage 35 flow path .............................................................................127
Figure 39. Stage 35 (a) grid density and (b) Y+ values at design point conditions .........................128
Figure 40. Rotor 37 convergence history for the near peak efficiency solution for (a) the RMS and maximum residuals and (b) the inlet and exit mass flow rates. ................................................................. 130

Figure 41. Rotor 37 100% speed line comparisons for mass flow rate versus (a) total pressure ratio, (b) total temperature ratio, and (c) adiabatic efficiency .............................................................................. 132

Figure 42. Rotor 37 Peak efficiency solution contours at 50% span of (a) relative total pressure and (b) relative Mach number. .................................................................................................................. 134

Figure 43. Rotor 37 peak efficiency exit radial profile comparisons for (a) total pressure ratio, (b) total temperature ratio, (c) adiabatic efficiency, and (d) tangential flow angle ................................................................. 136

Figure 44. Rotor 37 near stall exit radial profile comparisons for (a) total pressure ratio, (b) total temperature ratio, (c) adiabatic efficiency, and (d) tangential flow angle ................................................................. 139

Figure 45. Stage 35 100% speed line comparisons for mass flow rate versus (a) total pressure ratio, (b) total temperature ratio, and (c) adiabatic efficiency ................................................................. 141

Figure 46. Stage 35 Peak efficiency solution contours at 50% span of (a) relative total pressure and (b) relative Mach number .................................................................................................................. 143

Figure 47. Stage 35 point 3978 (peak efficiency) rotor exit radial profile comparisons for (a) total pressure ratio, (b) total temperature ratio, and (c) adiabatic efficiency ................................................................. 145

Figure 48. Stage 35 point 3978 (peak efficiency) stator exit radial profile comparisons for (a) total pressure ratio, (b) total temperature ratio, (c) adiabatic efficiency, and (d) tangential flow angle ................. 147

Figure 49. Z-R cross-section of the turbine rig flow path used for the grid geometry (Not To Scale). ..... 153

Figure 50. Uncooled turbine rig grid density and Y+ values for the three blade row steady calculation (a) high-pressure vane, (b) high-pressure rotor, and (c) low-pressure vane ........................................................................ 154

Figure 51. Convergence history for the three blade row steady calculation for the (a) RMS and maximum residuals and (b) inlet and exit mass flow rates .............................................................................. 158

Figure 52. Uncooled, three blade row steady analytical contour plots at 50% span for (a) normalized total pressure, (b) normalized total temperature, and (c) Mach number ................................................................. 160

Figure 53. Comparison of analytical and measured time-averaged static pressure on the high-pressure vane surface at (a) 15% span, (b) 50% span, and (c) 90% span ........................................................................................................... 163
Figure 54. Comparison of analytical and measured first harmonic of unsteady pressure on the high-pressure vane surface at (a) 15% span, (b) 50% span, and (c) 90% span .................................................. 165

Figure 55. Comparison of analytical and measured pressure on the high-pressure vane hub surface for (a) time-averaged static pressure and (b) the first harmonic of unsteady pressure ......................... 168

Figure 56. Comparison of analytical and measured pressure on the high-pressure vane shroud surface for (a) time-averaged static pressure and (b) the first harmonic of unsteady pressure ......................... 169

Figure 57. Comparison of analytical and measured time-averaged static pressure on the high-pressure rotor airfoil surface at (a) 15% span, (b) 50% span, and (c) 90% span .................................................. 171

Figure 58. Comparison of analytical and measured first harmonic of unsteady pressure on the high-pressure rotor airfoil surface at (a) 15% span, (b) 50% span, and (c) 90% span ......................... 173

Figure 59. Comparison of analytical and measured unsteady pressure on the high-pressure rotor airfoil surface at (a) 20% wetted distance and (b) -20% wetted distance .................................................. 175

Figure 60. Comparisons of analytical and measured static pressure on the high-pressure rotor platform surface for (a) average pressure and (b) first harmonic of unsteady pressure ........................................... 176

Figure 61. Comparison of analytical and measured unsteady pressure on the high-pressure rotor platform for gage location (a) PRP50, (b) PRP51, (c) PRP52, (d) PRP53, and (e) PRP54 ........................................... 178

Figure 62. Comparison of analytical and measured stationary shroud for (a) average pressure and (b) first harmonic of unsteady pressure ............................................................................. 181

Figure 63. Comparison of analytical and measured unsteady pressure for the shroud at (a) -7% wetted distance, (b) 30% wetted distance, (c) 60% wetted distance, and (d) 94.4% wetted distance ......................... 183

Figure 64. Nominal rig geometry for the modern high-pressure turbine and low-pressure vane (Not To Scale) ........................................................................................................... 193

Figure 65. Purge flow cavity geometry (a) simplifications, (b) final geometry (Not To Scale) ................. 194

Figure 66. Inner and outer flow path surfaces: (a) flow path simplifications, (b) final flow path (Not To Scale) ................................................................................................................. 196

Figure 67. Illustration of the purge flow cavity connections with the high-pressure rotor domain: (a) Full matching connection, (b) Full non-matching connection, (c) the axially non-matching connection... 200
Figure 68. 0-Z planar view of default blocking structure in Fine/Turbo........................................203
Figure 69. R-Z planar view of default blocking structure FINE/Turbo........................................204
Figure 70. Surface grid resolution for the high-pressure vane .......................................................205
Figure 71. Surface grid resolution for the high-pressure rotor .......................................................206
Figure 72. Purge flow cavity blocking structure and tangential cut of the grid resolution ...............207
Figure 73. Surface grid resolution for the low-pressure vane .......................................................208
Figure 74. Contour plot of resulting y+ values from Run 33 for the (a) cooled high-pressure vane (b) high-pressure rotor, (c) purge flow cavity, and (d) low-pressure vane ........................................209
Figure 75. Comparison of rake measured and averaged inlet total pressure profiles for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43 normalized to inlet averaged total pressure ..........213
Figure 76. Comparison of rake measured and averaged inlet total temperature profiles for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43 normalized to inlet averaged total temperature ......217
Figure 77. Cross-section of purge cavity including the lower labyrinth seal (Not To Scale) ...............224
Figure 78. Pressure and temperature history within the purge cavity during the experiment time windows for (a) Run 21, (b) Run 28, and (c) Run 33 .........................................................225
Figure 79. Temperature history during rig operations from gage HR142 ........................................228
Figure 80. Film cooling hole patterns highlighted through local flow injection on the high-pressure vane, hub, and shroud surfaces for the Run 28 solution .................................................................230
Figure 81. Summary of stationary and rotating boundary conditions for all three blade rows (Not To Scale) ..................................................................................................................................231
Figure 82. Run 21 steady three blade row calculation for (a) inlet and exit flow rates, and (b) the RMS and maximum residuals.............................................................................................................233
Figure 83. Run 21 convergence for the unsteady, phase-lag solutions for (a) the full exit mass flow rate history, (b) the last two overlapping periods of exit mass flow rate, (c) the full RMS residual history, and (d) the last two overlapping periods of the RMS residuals ..............................................................................................................................................235
Figure 84. Run 21 convergence for the harmonic solution for (a) exit mass flow rates, and (b) RMS and maximum residuals .......................................................................................................................................238
Figure 85. Comparison of time-average total temperature profiles between the rig exit rakes and the
FINE/Turbo steady and unsteady circumferentially averaged results for (a) Run 21, (b) Run 22, (c)
Run 28, (d) Run 33, and (e) Run 43 ........................................................................................................244
Figure 86. Comparison of the rig and analytical predicted cooled and uncooled exit total temperature
profiles for (a) Run 22 and 28, (b) Run 33 and 43 ..................................................................................248
Figure 87. Comparison of the analytical and measured uncooled-to-cooled temperature difference for (a)
run 22 and run 28 and (b) run 33 and run 43 ..........................................................................................250
Figure 88. Comparison of time-average total pressure profiles between the rig exit rakes and the
FINE/Turbo steady and unsteady circumferentially averaged profile for (a) Run 21, (b) Run 22, (c)
Run 28, (d) Run 33, and (e) Run 43 ........................................................................................................251
Figure 89. Comparison of the rig and analytical predicted cooled and uncooled exit total pressure profiles
for (a) Run 22 and Run 28, (b) Run 33 and Run 43, and (c) all runs .........................................................254
Figure 90. Comparison of the trailing edge hub surface time-averaged static pressure for the rig and
FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e)
Run 43 ..................................................................................................................................................258
Figure 91. Comparison of the Run 21 trailing edge hub surface time-accurate static pressure for the rig and
FINE/Turbo unsteady results for (a) PV110/PV112/PV114 gage location and, (b)
PV111/PV113/PV115 gage location ........................................................................................................261
Figure 92. Comparison of the Run 22 trailing edge hub surface time-accurate static pressure for the rig and
FINE/Turbo unsteady results for (a) PV110/PV112/PV114 gage location and, (b)
PV111/PV113/PV115 gage location ........................................................................................................263
Figure 93. Comparison of the Run 28 trailing edge hub surface time-accurate static pressure for the rig and
FINE/Turbo unsteady results for (a) PV110/PV112/PV114 gage location and, (b)
PV111/PV113/PV115 gage location ........................................................................................................264
Figure 94. Comparison of the Run 33 trailing edge hub surface time-accurate static pressure for the rig and
FINE/Turbo unsteady results for (a) PV110/PV112/PV114 gage location and, (b)
PV111/PV113/PV115 gage location ........................................................................................................265

xx
Figure 95. Comparison of the Run 43 tailing edge hub surface time-accurate static pressure for the rig and FINE/Turbo unsteady results for (a) PV110/PV112/PV114 gage location and, (b) PV111/PV113/PV115 gage location.  ..............................................................266

Figure 96. Comparison of time-average rotor surface pressure between the rig and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43 ............................269

Figure 97. Comparison of the time-average static pressure between the measured and predicted results for the PS2 stationary shroud gage location ...........................................................................273

Figure 98. Comparison of the PS2 stationary shroud gage time-accurate static pressure between the measured and predicted results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43. 274

Figure 99. Run 28 ensemble average diagnostic plot for the PS2 stationary shroud gage location .......277

Figure 100. Comparison of the time-average total relative temperature profiles between the rig rotor leading edge thermocouples and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43 .................................................................279

Figure 101. Comparison of the cooled and uncooled leading edge relative total temperature profiles for (a) the radial profile runs 22 and 28 and (b) the flat inlet profile runs 43 and 33 .................................283

Figure 102. Predicted and measured uncooled-to-cooled temperature difference for (a) run 22 to run 28 and (b) run 43 to run 33 .......................................................................................................................285

Figure 103. Platform thermocouple positions with gage labels (not to scale) ........................................287

Figure 104. Comparison of time-average static temperature on the rotor platform surface between the rig thermocouples and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43 .........................................................................................................................288

Figure 105. Comparison of the nominal and elevated high-pressure vane isothermal wall boundary condition effect on the predicted rotor platform temperatures for (a) Run 28 and (b) Run 43 ........292

Figure 106. Stationary and rotating Kulite locations in the purge flow cavity (Not to Scale) ...........294

Figure 107. Stationary and rotating thermocouple locations in the purge flow cavity (Not to Scale) ....295
Figure 108. Comparison of stationary cavity time-average static pressure locations between the rig and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43 .................................................................................................................. 296

Figure 109. Comparison of the Run 21 stationary cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PV104 and PV105 gage location and (b) the PV107 and PV108 gage location ........................................................................................................ 300

Figure 110. Comparison of the Run 22 stationary cavity time-series static pressure between the rig and the FINE/Turbo unsteady results for (a) the PV104 and PV105 gage location and (b) the PV107 and PV108 gage location ........................................................................................................ 301

Figure 111. Comparison of the Run 28 stationary cavity time-series static pressure between the rig and the FINE/Turbo unsteady results for (a) the PV104 and PV105 gage location and (b) the PV107 and PV108 gage location ........................................................................................................ 302

Figure 112. Comparison of the Run 33 stationary cavity time-series static pressure between the rig and the FINE/Turbo unsteady results for (a) the PV104 and PV105 gage location and (b) the PV107 and PV108 gage location ........................................................................................................ 303

Figure 113. Comparison of the Run 43 stationary cavity time-series static pressure between the rig and the FINE/Turbo unsteady results for (a) the PV104 and PV105 gage location and (b) the PV107 and PV108 gage location ........................................................................................................ 304

Figure 114. Comparison of the stationary cavity time-average static temperature between the rig and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43 .................................................................................................................. 306

Figure 115. Temperature application zones on the high-pressure vane hub and stationary cavity for the increased isothermal wall boundary temperature models .................................................................................. 309

Figure 116. Comparison of the original and the elevated high-pressure vane isothermal wall temperature models to the measured data for(a) run 28 and (b) run 43 ........................................................................................................ 310

xxii
Figure 117. Comparison of the rotating cavity time-average static pressure between the rig and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43. 312

Figure 118. Comparison of the Run 21 rotating cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PRW70 gage location, (b) the PRW71 gage location, (c) the PRW72 gage location, and (d) the PRW74 gage location. 316

Figure 119. Comparison of the Run 22 rotating cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PRW70 gage location, (b) the PRW71 gage location, (c) the PRW72 gage location, and (d) the PRW74 gage location. 318

Figure 120. Comparison of the Run 28 rotating cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PRW70 gage location, (b) the PRW71 gage location, (c) the PRW72 gage location, and (d) the PRW74 gage location. 320

Figure 121. Comparison of the Run 33 rotating cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PRW70 gage location, (b) the PRW71 gage location, (c) the PRW72 gage location, and (d) the PRW74 gage location. 322

Figure 122. Comparison of the Run 43 rotating cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PRW70 gage location, (b) the PRW71 gage location, (c) the PRW72 gage location, and (d) the PRW74 gage location. 324

Figure 123. Comparison of the rotating cavity time-average static temperature between the rig and the FINE/Turbo steady and unsteady profile for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43. 328

Figure 124. Cavity interface instantaneous forward progressing streamlines and cavity static temperature for (a) run 21, (b) run 28, and (c) run 43. 331

Figure 125. Comparison of the original and elevated high-pressure vane isothermal wall temperatures to the measured data for (a) run 28 and (b) run 43. 334

Figure 126. Inlet and outlet radial locations for the analytical and computational comparison. 342
Figure 127. Comparison of the closed form analytical model and the time and tangentially averaged FINE/Turbo results forand (a) total pressure, (b) total temperature, and (c) tangential velocity. 343

Figure 128. Run 28 cavity streamlines from the time-average harmonic unsteady solution. 350

Figure 129. Run 28 vector field in the rim seal area for the time-averaged harmonic unsteady solution. 351

Figure 130. Purge flow forward progressing streamlines for the time-averaged harmonic solution for (a) top looking down view, (b) side view, and (c) forward looking aft side view (Not To Scale). 353

Figure 131. Instantaneous Radial velocity contours on the purge flow cavity and main gas path interface for (a) t=0, (b) t=0.055, (c) t=0.11, (d) t=0.167, (e) t=0.22, (f) t=0.278, (g), t=0.33, (h) t=0.389, (i) t=0.44, and (j) t=0.5. 358

Figure 132. Measured and predicted stationary cavity unsteady static pressure for the (a) PV104 and PV105 location and the (b) PV107 and PV108 location. 369

Figure 133. Measured and predicted stationary cavity unsteady static pressure for (a) the PRW70 location, (b) the PRW71 location, and (c) the PRW72. 371

Figure 134. Instantaneous static pressure contours on the radial planes of the upper stationary and rotating Kulite gage locations for (a) t=0, (b) t=0.055, (c) t=0.11, (d) t=0.167, (e) t=0.22, (f) t=0.278, (g), t=0.33, (h) t=0.389, (i) t=0.44, and (j) t=0.5. 373

Figure 135. Vane exit radial profiles just forward of the cavity for (a) total pressure, (b) total temperature, and (c) tangential velocity. 384

Figure 136. Vane hub surface static pressure at 2% span for (a) the full wetted distance and (b) only the last 40% of the wetted distance. 386

Figure 137. Vane trailing edge hub static pressure comparison between run 22 and run 28 for the computational and experimental rig results. 388

Figure 138. Vane exit radial profiles just aft of the cavity for (a) total pressure, (b) total temperature, and (c) tangential velocity. 389

Figure 139. High-pressure rotor surface static pressure at 2% span or (a) the full wetted distance and (b) the first 30% of the wetted distance. 391

Figure 140. Case number one: (a) RMS Residual history, (b) inlet and exit mass flow rate. 409

xxiv
Figure 141. Case number two: (a) RMS Residual history, (b) inlet and exit mass flow rate ..................................................................410
Figure 142. Case number three: (a) RMS Residual history, (b) inlet and exit mass flow rate ..................................................411
Figure 143. Case number four: (a) RMS Residual history, (b) inlet and exit mass flow rate .................................................412
Figure 144. Case number six: (a) RMS Residual history, (b) inlet and exit mass flow rate ..................................................413
Figure 145. Case number seven: (a) RMS Residual history, (b) inlet and exit mass flow rate .................................................414
Figure 146. Case number eight: (a) RMS Residual history, (b) inlet and exit mass flow rate .................................................415
Figure 147. Case number nine: (a) RMS Residual history, (b) inlet and exit mass flow rate ..................................................416
Figure 148. Rotor 37 stall point convergence history for (a) RMS and maximum residuals, and (b) inlet and exit mass flow rates ..................................................................................................................................418
Figure 149. Stage 35 peak efficiency point convergence history for (a) RMS and maximum residuals, and (b) inlet and exit mass flow rates ..................................................................................................................................419
Figure 150. Forward flow path and strut cross-section (Not To Scale) .....................................................................................421
Figure 151. Forward flow path and strut (a) grid density and (b) resulting + values .................................................................423
Figure 152. Convergence history for the forward strut and flow path steady solution for (a) log of residuals and (b) the inlet and exit mass flow rates ..................................................................................................................................425
Figure 153. Resulting radial profiles at the rake plane for (a) total pressure, (b) total temperature, (c) tangential flow angle, and (d) radial flow angle ..................................................................................................................................427
Figure 154. Uncooled turbine rig two blade row steady solution convergence history for (a) RMS and maximum residuals and (b) the inlet and exit mass flow rates ..................................................................................................................................429
Figure 155. Uncooled turbine rig two blade row unsteady harmonic solution convergence history for (a) RMS and maximum residuals and (b) the inlet and exit mass flow rates ..................................................................................................................................431
Figure 156. Uncooled turbine rig two blade row unsteady phase lag solution convergence history for (a) RMS residuals, (b) the 43rd and 44th period of RMS residuals, (c) the inlet and exit mass flow rate, and (d) the 43rd and 44th period for the exit mass flow rates ..................................................................................................................................432
Figure 157. Run 22 three blade row steady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates ..................................................................................................................................436

xxv
Figure 158. Run 22 three blade row harmonic unsteady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates......................................................437

Figure 159. Run 28 three blade row steady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates.................................................................438

Figure 160. Run 28 three blade row harmonic unsteady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates.....................................................440

Figure 161. Run 33 three blade row steady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates.......................................................................441

Figure 162. Run 33 three blade row harmonic unsteady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates.....................................................443

Figure 163. Run 43 three blade row steady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates.......................................................................444

Figure 164. Run 43 three blade row harmonic unsteady convergence history for the (a) RMS residuals, (b) the inlet and exit mass flow rates...............................................................................446
LIST OF SYMBOLS

Arabic Symbols

A  Area
BR Blow Ratio
c Chord Length
CD Discharge Coefficient
CP Constant Pressure Specific Heat
CV Constant Volume Specific Heat
CW Non-dimensional Purge Seal Flow Rate
e energy
F Force
h Enthalpy
ı X-Direction Vector
ı̂ Y-Direction Vector
KE Kinetic Energy
k Fluid Thermal Conductivity
\( \hat{k} \)  Z-Direction Vector

\( \ln \)  Natural Log, \( \log_e \)

\( L \)  Length

\( m \)  Mass Flow Rate

\( M \)  Mach Number

\( \hat{n} \)  Normal Vector

\( P \)  Pressure

\( p \)  Pressure

\( Q \)  Heat transfer

\( R \)  Universal Gas Constant or Radius

\( \text{Re} \)  Reynolds Number

\( t \)  Thickness

\( T \)  Temperature

\( u \)  Velocity in the x-Direction

\( V \)  Velocity

\( v \)  Velocity in the y-Direction

\( w \)  Velocity in the z-Direction

\( \dot{W} \)  Work

Greek Symbols

\( \rho \)  Fluid Density

\( \mu \)  Fluid viscosity

xxviii
$\omega$  Angular velocity

$\gamma$  Ratio of specific heats

$\eta$  Efficiency

$\tau$  Shear Stress

$T$  Torque

$\Delta$  Delta or Difference

$\sigma$  Normal Stress

$\pi$  Constant Equal to the Ratio of a Circles Circumference to its Diameter

Subscripts

1  Station or Location 1

2  Station or Location 2

a/c  Aircraft

c  Compressor

FD  Free Disk Pumping

MIN  Minimum

r  Radial direction

S  Static property

t  Turbine

T  Total property

u  Velocity in the x-Direction

v  Velocity in the y-Direction
w  Velocity in the z-Direction or Rotational Direction
x  x-direction
y  y-direction
z  z-direction
θ  Tangential direction
CHAPTER 1
INTRODUCTION

The aviation jet engine industry continually is looking to improve cycle efficiency, thus enhancing fuel burn and reduce operating costs for airlines. The Brayton Cycle, shown in Figure 1, is characterized by a compression process (state 2 to 3), heat addition or combustion process (state 3 to 4), expansion through the turbine (state 4 to 5), followed by exhaustion of air from the turbine to the atmosphere (state 5 to 2). This figure shows both an ideal Brayton cycle and a real Brayton cycle with losses in both the compressor and turbine.

![Figure 1. T-s Diagram with both the Ideal and Real Brayton Cycles](image)

---

Figure 1. T-s Diagram with both the Ideal and Real Brayton Cycles
For the Brayton cycle, there are three efficiency definitions that are applied for evaluation of a given thermodynamic design: the propulsive efficiency, the thermal efficiency, and the overall efficiency. The propulsive efficiency relates the rate of work done in propelling the aircraft forward to the rate at which kinetic energy is added to the flow through the engine:

\[
\eta_p = \frac{\text{flight speed x Net Thrust}}{\text{Power to Jet}} = \frac{2V_{A/C}}{V_{A/C} + V_{Jet}}
\]  (1.1)

where \( V_{Jet} \) is the jet velocity of the engine. The thermal efficiency for the Brayton cycle looks at the ratio of the rate of kinetic energy added to the air as it passes through the engine to the rate of energy made available by burning the fuel. The thermal efficiency is defined in equation form as:

\[
\eta_{\text{thermal}} = \frac{\Delta KE_{\text{Air}}}{m_{\text{Fuel}} (\text{Lower Caloric Value})_{\text{Fuel}}} = \frac{W_{\text{NET}}}{m_{\text{air}} C_p(T_4 - T_3)}
\]  (1.2)

And the net work across the engine is simply the difference between the work supplied by the compressor to the air and the work extracted from the air in the turbine:

\[
W_{\text{NET}} = W_T - W_C = m_{\text{air}} C_p(T_4 - T_5) - m_{\text{air}} C_p(T_3 - T_2)
\]  (1.3)

Finally the overall cycle efficiency is the product of the thermal and propulsive efficiency.

Improvements in gas turbine efficiency can be quickly achieved through both raising the compressor exit pressure (overall cycle pressure ratio) and the turbine inlet temperature (overall cycle temperature ratio), however metallurgical considerations often limit how much these cycle parameters can truly be increased. The overall pressure ratios in modern engines is often found to be 40:1 for best efficiency in a commercial application and turbine inlet temperatures can often exceed 2000 Kelvin (K) at take-off conditions and is well above the approximate melting point of 1550 K for the nickel-based alloys used to produce the disks and blades. To combat these high gas temperatures, cooling air is redirected from the exit of the compressor to the high-pressure turbine components to reduce metal temperatures in the form of internal
and film cooling on the blades, vanes, and shrouds and impingement into the rotor bores and cavities. A cross-section of a modern gas turbine engine used for aviation applications is shown in Figure 2. This figure shows the approximate locations of all major modules, airflow direction, and the location of the extraction of cooling flow from the compressor for cooling use in the turbine.

![Diagram of a modern, civil aviation, gas turbine engine](image)

**Figure 2. Cross-section of a modern, civil aviation, gas turbine engine**

This cooling air is considered expensive to the engine cycle due to the amount of work put into raising the pressure through the fan, booster, and compressor. Removing this air also has a negative impact on cycle efficiency as the air is no longer available to do useful work upstream of its introduction point and will tend to create pressure loss and temperature reductions at its introduction point. And often as temperatures increase throughout the turbine section of the engine, more cooling flow is required to keep metal
temperatures on moving and stationary parts workable. New engine design trends are ever increasing
turbine inlet temperature and keeping compressor pressure ratios high. The turbine inlet temperature
increase has a direct impact on the cycle efficiency of a gas turbine engine. This has steadily occurred
since the 1960’s partially due to advances in materials used for turbine components and partially due to the
use of cooling air from the back of the compressor for turbine airfoil cooling. And according to Cumpsty 0,
“it is the improvements in the way the cooling air is used that have the biggest gains.”

Use of such expensive cooling must be done with great efficiency in an effort to reduce the amount needed
to achieve proper cooling levels. For high-pressure turbine disks, low cycle fatigue will always be an issue
of concern with gas turbine engines and cooling flow is necessary to keep the disks from failing. These
rotating disks operate at temperatures often exceeding 900 K with rotational speeds in excess of 12,000
revolutions per minute (RPM) at take-off conditions. Under these conditions, the disks carry a large
amount of rotating energy such that when a failure does occur and the disk is released, the engine’s
structure and casings cannot contain the disk and an uncontained failure results. Uncontained failures
result in significant damage to the engine and could result in damage to the aircraft as well. To ensure the
best use of the cooling flow, rotor purge flow cavity seals have been incorporated into gas turbine engines
for the purposes of sealing the rotating discs from hot gas flow ingestion from the air exiting the combustor.
Ingestion of mainstream gas flow into the rotor cavity will lead to increased cavity air and rotor
temperatures and subsequently reduced rotor fatigue life due to the increased thermal stresses and
decreased material properties. Thus, minimizing and/or eliminating mainstream gas ingestion into the rotor
cavity is critical to maintaining safe operation of the engine.

The purge flow cavity seals are located between the stationary nozzles and the rotating blades in a high-
pressure turbine and are composed of a stationary and rotating portion referred to as the lips of the seal.
The two most common types of stationary-rotating air seals (often referred to as purge seals) are the axial
seal shown in Figure 3(a) and the radial seal shown in Figure 3(b) below. The radial seal configuration can and is often configured with a double lip in the stationary part of the seal as seen in Figure 3(c).

![Diagram of Seal Configurations](image)

(a)

(b) Continued

Figure 3. (a) Axial, (b) radial, (c) double lip radial stationary-rotating air seal configurations
The primary objective of the seal is to create a torturous path that the mainstream flow would have to follow in order to fully penetrate the rotor purge cavity. In addition, the thermodynamic properties (pressure, temperature, etc.) of the purge flow introduced into the cavity are set in order to maintain adequate backflow margin to this mainstream gas penetration. Critical dimensions and features of purge flow seals include the disk radius, the cavity aspect ratio (height-to-width), the radial and axial clearances between the rotating and stationary parts of the cavity, the amount of axial overlap between the stationary and rotating seals, the number of overlapping seals on both the rotating and stationary parts, and the symmetric or non-symmetric design of the seals in the tangential direction.

Several different non-dimensional parameters are often used to describe purge flow cavities, the flow rates within, and the external flow fields just beyond the mainstream gas path seals. Two different Reynolds numbers often appear in the discussion of purge flow cavities, the rotational Reynolds number:

$$Re_\theta = \frac{\rho R_i^2 \omega}{\mu}$$  \hspace{1cm} (1.4)
And the external flow Reynolds number:

\[
\text{Re}_w = \frac{\rho V_z L_{V,Z}}{\mu}
\]  \hspace{1cm} (1.5)

Where \(\rho\) is the fluid density, \(R_H\) is the radius at the hub, \(\omega\) is the angular velocity of the disk, \(\mu\) is the dynamic viscosity of the fluid, \(V_z\) is the mean axial velocity of external flow, and \(L_{V,Z}\) is the axial chord of the upstream vane. In addition, a non-dimensional flow rate is used to describe the level of flow coming into the purge cavity, which is defined as:

\[
C_w = \frac{m_{\text{Purge}}}{\mu R_H}
\]  \hspace{1cm} (1.6)

And a minimum non-dimensional free disk pumping rate of the cavity is defined as:

\[
C_{w,FD} = 0.219 \text{Re}^{0.8}
\]  \hspace{1cm} (1.7)

These non-dimensional characteristics help to classify purge flow cavities and their operation.

Purge cavity flows in general contain a flow structure similar to that depicted in Figure 4. These flows are characterized by one large recirculation zone within the main cavity and small recirculation zones throughout the sealing features, which are dependent upon the type of seal, locations of the seal, and the number of seals.
As the inlet purge flow enters the cavity, it begins to rotate tangentially due to the rotor angular velocity. The rotating wall of the disk cavity then pumps the incoming fluid radially outward towards the main-stream flow rim seals. Once the flow encounters the seal, a portion continues outward while the remaining flow recirculates toward the stationary wall, which is subsequently driven radially inward creating the main recirculation zone. The part of the flow that continues towards the exit of the seals into the main-stream flow will also create additional recirculation zones as the flow turns corners through the seal. These additional recirculation zones can both help to prevent ingestion as well as further exacerbate ingestion of the hotter main-stream air depending on their size and location. What is not shown in Figure 4 is the ingestion flow from the main stream into the cavity; this ingestion process is often caused by unsteadiness within all of the flow fields.
Purge flow cavities and the respective seals are subject to unsteadiness from the main gas path due to its location between a stationary vane and a rotating blade, especially for these transonic turbine stages. The unsteadiness is the result of multiple factors including the stationary vane which creates an upstream wake due to frictional losses on the vane thus creating an unsteady pressure field on the rotating wall and seal, the downstream bow wake created by the rotating blade which creates an unsteady pressure field on the stationary portion of the cavity and seal, and interactions of the vane wake hitting the downstream rotating blade and the rotating blade bow wake hitting the upstream stationary airfoil. Additional unsteadiness is introduced when the upstream nozzle is transonic with exit Mach numbers above one, which creates trailing edge shock waves that will travel downstream and bounce off the rotating blade. All of this unsteadiness leads to non-uniform conditions around the perimeter of the seal, which can result in localized mainstream gas ingestion leading to an increase in rotor and turbine blade dovetail temperatures. If these increased temperatures cannot be tolerated, an increase in total purge flow will be required to maintain seal effectiveness thus leading to a decrease in engine performance. Another effect is the interaction of the purge flow air with the mainstream gas with respect to the rotor blade aerodynamics. This cool, low momentum air can disrupt the aerodynamic flow field that will also lead to reduce turbine efficiency and ultimately increased specific fuel consumption by the engine. Understanding the main drivers for a proper rotor purge flow cavity aerodynamic design and the interaction effects with the surrounding turbomachinery is critical to furthering turbine engine performance.

In an effort to understand these complex interactions within turbomachinery, multiple techniques are often employed: full-scale engine measurements, rig measurements on full or partial scale turbines, and computational studies. Full-scale engine testing has the advantage of measurements applicable to the actual engine hardware and at the operating conditions the engine will see when utilized in the field. However, full-scale engine tests become extremely expensive and are often very limited in the quantity of data that it can provide, in part because of instrumentation limitations. Testing multiple configurations within a full-scale engine program is typically not an option due to the teardown and rebuild cycle needed to reach flow path components, thus adding to the overall costs. Utilizing an experimental rig that duplicates the engine
design corrected conditions can significantly reduce costs while providing the necessary measurements to understand the feature being studied. The rig measurements are often performed on representative engine hardware from an engine such as the high-pressure turbine, the high-pressure compressor, or the fan. In the case of the turbine, the operating conditions are established at reduced temperatures and pressures but are designed to maintain the proper operating conditions such as corrected mass flow and corrected speed, and stage pressure ratio in order to duplicate the correct design and/or off-design conditions desired to study. In the case of the compressor and fan, the component operates at engine speed. For the purposes of this thesis effort, the discussion will be confined to the high-pressure turbine, which utilize a large quantity of instrumentation strategically placed throughout the stage and supporting flowpath hardware in an effort to capture as much information as possible in the areas of concern, but are often limited to single measurements of thermodynamic quantities such as pressure and temperature at discrete locations. Experimental rigs do offer some flexibility in the design/configuration to be tested; however this flexibility is general within the feasible design space offered by the specific rig.

Computational studies utilizing computational fluid dynamics (CFD) have the added advantages of being able to quickly and cost effectively analyze multiple design iterations over a large variety of operating conditions (with the codes ability to handle off-design conditions being a limiting factor). The results provide the necessary thermodynamic properties throughout the entire domain of interest with the added visual capabilities often found in CFD post-processing software. However, the required level of model fidelity and the CFD code’s ability to capture the necessary physics is not well understood when the code is applied to more and more challenging problems. This often requires detailed evaluation with experimental results to determine a CFD code’s abilities and limitations as well as the level of fidelity required to capture the necessary physics. Overall, the best approach is often a combination of both experimental and computational studies to provide proper evaluation and in-depth understanding.
1.1 Review of Purge Flow Cavity Studies

Gas ingestion into the rotor cavity of high-pressure turbines has received a fair amount of attention by engineers in an effort to understand the complex interactions between the high-pressure turbine nozzles, the high-pressure turbine blades, and rotor purge cavity. A significant amount of both experimental and numerical work has been completed in an effort to further this level of understanding. Bayley and Owen [2] performed one of the earliest experimental studies dealing with this problem in 1970. A series of experiments were performed using a stationary disc with a shroud and rotating disc used to create a cavity similar in structure to an axial seal configuration with no external flow present. The stator-to-rotor gap and the shroud-to-rotor gaps were varied over a series of Reynolds numbers in the cavity in order to establish the effects of geometry and flow conditions on the system. Bayley and Owen [2] were able to establish minimum required mass flow curves over a series of cavity Reynolds numbers at or above which ingestion would be prevented:

\[ C_{W, Min} = 0.61G_C \text{Re}_\theta \]  \hspace{1cm} (1.8)

Where \( G_C \) is the seal clearance ratio of the seal clearance \( (S_C) \) divided by the hub radius \( (R_{i0}) \). They also found the cavity pressure distributions to be insensitive to the presence of the shroud below eighty-six percent of the maximum radius and used a superposition of the boundary layer equations and the derived pressure drop equation to predict cavity pressure distributions.

Abe et al. [3] in 1979 assembled a rotating disk rig in order to study the effects of mainstream ingress into the forward purge cavity that would typically be located between the upstream vane and downstream rotor. The rig was designed to study six different rim seal geometries with capability to measure static pressure within the cavity via wall pressure taps, velocities measurements within the cavity using a three headed cobra probe, flow visualization using oil paint methods, and incorporated propane as a tracer gas to measure main stream gas ingestion. The cavity, disk, and rim seal geometry was specified as being a
typical design of gas turbine engines and incorporated an upstream vane to turn the flow 50° from axial; however, no rotating blades were present. The disk was also designed to allow for experimenting with axial spacing at the rim seal by moving the disk fore and aft axially. The author’s found for rim seals with overlapping fins, much like those of Figure 3(b) and (c), that as long as the seals remained overlapping the main stream gas ingestion was held to a minimum. However, once the seals no longer overlapped mainstream ingestion increased dramatically. The most effective method of limiting mainstream gas ingestion found by the author’s was the presence of a radial fin on the rotating side of the rim seal. The presence of such a fin induces additional secondary flows that reduce the ingestion into the cavity space. Overall, three main effects were highlighted that affected mainstream ingestion: Rate of cooling air flow, the axial gap at the exit of the wheel space, and the shape of the static and rotating seal at the exit of the wheel space. No correlation to rotational disk speed or with upstream airflow angle once the axial gaps were corrected was found. In the end Abe et al. [3] concluded that the cooling air flow purge into the disk cavity travels up and exits along the rotating side of the seal while the main stream gas is ingested and travels along the static side of the seal.

Phadke and Owen, at the University of Sussex, conducted a series of experiments in 1988 to study the effects of different seal geometries in a quiescent external environment [4], an axisymmetric external flow environment [5], and a non-axisymmetric external flow environment [6] with all experiments setup in order to determine the minimum flow rates necessary to prevent ingestion. In the quiescent environment, Phadke et al. [4] concluded that the minimum flow rate required for the prevention of ingestion increased with increasing rotation speed and with increasing seal clearance for all seal geometries tested. They also found that the radial clearance seals were more effective (lower minimum flow requirements) than axial clearance seals under similar conditions. In the axisymmetric environment, only three of the original seven cavity seals were tested of which two were axial clearance seals and one radial clearance seal. Two distinct regions were noted for all seals tested: a rotation-dominated region for low ratios of axial flow to rotational Reynolds number followed by an external flow dominated region for large ratios.
In the rotation-dominated region, reduced minimum sealing flow rates with increases in the rotational Reynolds number were noted while in the external flow dominated region the minimum sealing flow was found to be proportional to the external flow Reynolds number. For no external flow, the minimum sealing flow was found to be proportional to the rotational Reynolds number as previously demonstrated by Bayley and Owen [2]. The final study looked at the effects of non-axisymmetric flow on the minimum sealing flow rates. The non-axisymmetric flows were created by the use of honeycomb placed appropriately in the inlet screen such as to create low-pressure zones around the circumference. For this study, four of the original seven seal configurations were studied which included three axial clearance seals and a radial clearance seal. Phadke and Owen concluded that for the external flow dominated region, the minimum sealing flow rate increased with increasing the maximum circumferential static pressure difference ($P_{\text{Max}}$).

A simple correlation was provided for the external flow dominated region based on $G_C$ and $P_{\text{Max}}$:

$$C_{W,\text{Min}} = 2\pi \sqrt{2C_D G_C (P_{\text{Max}})^{\frac{1}{2}}}$$

(1.9)

where $C$ is an empirical constant, $C_D$ is the discharge coefficient and $P_{\text{Max}}$ is

$$P_{\text{Max}} = \frac{1}{2} \frac{P_{\text{Max}} - P_{\text{Min}}}{\rho \bar{W}} \text{Re}^2$$

(1.10)

where $W$ is the uniform velocity at the inner part of the annulus and $P_{\text{Max}}$ and $P_{\text{Min}}$ are the maximum and minimum pressure in the annulus. Their work also concluded that the double-shrouded radial clearance seal appeared to have some advantages over the axial clearance seals in terms of minimizing hot gas ingestion into the cavity.

Ko and Rhode [7] performed a computational study using a two-dimensional, axisymmetric flow solver to solve the Navier-Stokes equations for compressible turbulent flow at actual engine conditions to understand the effects of the interaction between the main stream flow and the cavity flow. The rim seal was that of a
simple axial clearance seal with a shroud on the stationary part. The computational domain was truncated axially to aft of the vane trailing edge to just forward of the blade leading edge and thus no effects of the vane or blade were included thus being replaced with radial distributions of the incoming and outgoing flow properties. Purge flow rates were adjusted for the computations in order to provide a range of flow rates. The results suggested that a recirculation zone exists within the cavity and is the primary transportation mechanism of heat from the mainstream gas path to the blade root and retainer area of the disk. Halving the purge flow increased the recirculation zone significantly thus causing more main stream gas ingestion into the cavity and raising the cavity temperatures and doubling of the purge flow was found to have the opposite effect on both the recirculation zone and the cavity temperatures. The recirculation zone was easily seen to be dependent on the rotational Reynolds number, the external flow Reynolds number, and the sealing flow such that higher rotational Reynolds number or higher sealing flow reduces the size of the recirculation zone and higher external flow Reynolds number causes it to increase in size.

Johnson et al. [8] provided a review of the physical mechanisms associated with the rotor purge cavity as identified in studies performed. Johnson mentions the benefits of improving purge seal technology, this being increased power output, reduced specific fuel consumption, and/or reduced engine weight. Johnson summarizes the physical mechanisms that are important to purge seal ingestion to be disk pumping, the periodic vane/blade pressure field, the three-dimensional geometry in the purge seal region, asymmetries in the purge seal geometry, turbulent transport in the platform overlap region, and flow entrainment.

An experimental study on a research turbine utilizing an axial purge seal geometry was performed by Green and Turner [9]. The purpose of the study was to examine the effectiveness of the purge seal over a range of cooling flow rate values. This study was performed for a single stage, rotating turbine with mainstream flow, without mainstream flow, and with mainstream flow and the rotor blades removed. They concluded that the rotor blades tended to make the main stream flow more axisymmetric in the purge seal region leading to reduced mainstream flow ingestion into the rotor cavity such that the purge seal would be
best placed as far away from the nozzle guide vanes as possible. The work performed by Hills et al. [10] provided an experimental and numerical comparison of an axial purge seal. CFD solutions of the inlet nozzle were performed and compared to static pressure measurements from the experimental rig with good agreement. The nozzle solutions and a potential flow approximate treatment of the nozzle solutions were then used as inlet boundary conditions for a series of numerical analyses of the axial purge seal cavity. The cavity solutions did not resolve the nozzle or blade flows in the mainstream passage and only good agreement was found between the numerical and experimental results at lower seal flow rates. It was concluded by this work that circumferential flow variations associated with the potential flow were more important than the wake and secondary flows at low cavity flows whereas at high cavity flows the measurements and CFD were found to diverge.

The first study completed by Bohn et al. [11] reported a series of experiments which looked at several parameter variations in order to better understand two different purge seal geometries: two flat discs and a stationary axial seal with two different gap widths. The study found that the external (mainstream) Reynolds number had a strong influence on the seal performance such that hot-gas ingestion increased with increased Reynolds number. The effects of rotational speed were also studied which found the flat disc configuration seal effectiveness dropped with decreased rotational speed and the stationary axial seal effectiveness tended to decrease with increasing rotational speed. Numerical simulations were also performed using a commercial solver to perform steady, compressible, three-dimensional solutions of the cavity and the inlet nozzle; however, no comparisons were made with the experimental results. Bohn et al. [12] also performed a second set of experiments that investigated the effects of the presence and absence of the rotor blades on the two aforementioned purge seal geometries. A numerical investigation was conducted using a commercial solver to perform unsteady, compressible, three-dimensional, periodic solutions of the cavity, inlet nozzle, and rotor blades. The experiment concluded that with the blades present, a higher circumferential hub pressure asymmetry was measured which caused a drop in seal efficiency on the flat discs and a significant increase in seal efficiency on the axial seal. No direct comparisons of the numerical and experimental results were provided; however, the numerical results
predicted the positive influence of the rotor blades on the axial seal efficiency and also predicted an ingestion zone that traveled at approximately one-half of rotor speed.

Bohn and Wolff [13] conducted multiple experiments on four different configurations of rim seals using a subsonic, ambient temperature and pressure turbine rig. The rig incorporated full length and height airfoils of an upstream vane and the rotating blade. The experiments looked at a range of non-dimensional sealing mass flow rates through each configuration of seal in an effort to find a correlation back to the minimum sealing flow based on the work of Phadke and Owen [6]. An improved correlation was provided such that:

\[ C_{W,MIN} = 2\pi KG_c \sqrt{\frac{1}{2} C_{P,Max} \Re_{c1}} \]  

(1.11)

Where \( \Re_{c1} \) is the External Flow Reynolds number based on the total velocity at the exit of the upstream vane, \( K \) is a constant based on the configuration of the rim seal, and \( C_{P,Max} \) is:

\[ C_{P,Max} = \frac{P_{Hub,Max} - P_{Hub,Min}}{\frac{1}{2} \rho C_1^2} \]  

(1.12)

Bohn and Wolff [13] found that the constant, \( K \), was dependent on the configuration of the seal and ranged from 0.12 to 1.01 whereas Phadke and Owen [6] used a constant value of 0.6 for all seals. They also concluded that the exit flow angle of the upstream vane is of much less importance than the mainstream parameters (density, dynamic viscosity, and total velocity), the seal configuration, and the pressure delta at the hub just upstream of the cavity.

Johnson et al. [14] developed a simple orifice model for rim seals based on two-dimensional, time-dependent calculations of the Turbine Rim Seal Rig at RWTH Aachen University. Details of the simple orifice model were not provided. Two-dimensional calculations were produced using Fluent and calculating the flow at 10% of the annulus height of the turbine. The calculations studied both a close-
spaced and a wide-spaced vane-to-blade spacing while using the experimental rig data for comparison to the results. The simple orifice model was constructed using the circumferential pressure fields at the seal axial location as a predictor for main stream gas ingestion into the cavity and could be based on either the time-averaged pressure field or the multiple instantaneous pressure fields in which the results would be averaged. The orifice model was shown to be capable of predicting the coolant cavity flow rate required to obtain a given disk-cavity cooling effectiveness; however, the model did require experimentally determined discharge coefficients for the cavity seals in conjunction with the numerical results. They also found that for closely spaced vanes and blades, the time-dependent pressure fields were the dominating ingestion mechanism whereas for more widely space vanes and blades pressure fields are less dominant and conditions could occur where the airfoil less disk-stator rim seal mechanism dominates.

Guo and Rhode [15] performed a numerical study of a radial seal geometry looking at the effects of eccentricity in the radial seal height (gap height between the stationary and the rotating lip of the purge seal). The numerical study was performed was a steady, three-dimensional, full-rotor model of the rotor cavity and purge seal with the domain extending part way into the mainstream flow. No stator-rotor interactions were included, only inlet and exit conditions representative of the axial locations. Their results concluded that no mass ingestion occurred for concentric purge seals but only slight turbulent heat diffusion, the purge seal effectiveness was reduced to 96% for a 5% eccentricity and down to 53% for a 50% eccentricity thus requiring an increase in the minimum coolant flow to prevent ingestion. The maximum ingress location was also noted as independent of the eccentricity and coolant flow levels.

A set of experiments performed by McLean et al. [16] and [17] utilized measurements in the stationary and rotating reference frame to study the differences between root injection, radial injection, and impingement injection. The experimental rig incorporated cooled nozzle and uncooled blade in a single stage, transonic turbine. The experiments were conducted in a continuous flow facility and used exit traverses to measure flow properties about one and a half axial chords downstream of the rotor. In the stationary frame, McLean
et al. [16] concluded that root injection showed the strongest effects on the time averaged profiles and was fundamentally different from the radial and impingement injection. In the rotating frame, McLean et al. [17] noted that the three injection types produced significant, three-dimensional changes to the wake width and position with effects being convected to midspan on the rotating blade. The root injection was again cited as being fundamentally different and having opposite effects to those of the radial and impingement injection. Their ultimate conclusion was that the cooling injection cannot be ignored in turbine design.

Girgis et al. [18] performed a numerical and experimental study on the efficiency trends of adding tangential swirl to the shroud cooling flow injection above a turbine blade and the disk cavity cooling flow injection. The design and testing was performed on a single stage, transonic turbine with a cooled nozzle and uncooled rotor. Measurements of the exit conditions were made on the turbine rig one blade chord downstream of the rotor in an effort to assess the efficiency gains or losses associated with adding the tangential component of swirl to the cooling flow injection. The tests confirmed that controlled addition of purge flow could result in improved turbine efficiency. The CFD simulations were conducted to look at the purge flow effects only using a steady, three-dimensional stage analysis of the entire turbine with the rotor purge cavity explicitly modeled. The combined experimental and computational study concluded that the CFD simulations could predict the trends of efficiency well, but over predicted the benefits. Hills et al. [19] also performed a numerical study in which results were compared to experimental results performed by Hills et al. [10]. The numerical study used the commercial CFD code, Fluent, to examine four different three-dimensional models of the turbine and rotor purge cavity: steady stationary frame (nozzle only), steady rotating frame (blade only), steady mixing plane, and an unsteady sliding interface model with periodic boundaries. Reasonable agreement between the CFD and the experimental results were obtained with the unsteady model being noted as the best. The rotor unsteadiness was found to have a disproportionately large effect on ingestion and was concluded that the rotor will usually lead to more ingestion, which opposes the conclusions formed by Green and Turner [9]. Hills et al. [19] concluded with the derivation of a simple model for the flow across the seal. The model was derived from the inviscid momentum equations in the axial and radial direction and found the following:
\[ \Delta p = \rho \left( \frac{\partial}{\partial t} + \frac{\nu}{r_o} \frac{\partial}{\partial \theta} \right) U + \frac{\rho U^2}{2} C_k \text{sgn}(U) \]  

(1.13)

Cao et al. [20] conducted an experimental and numerical study on the second stage (intermediate stage) of a two-stage (high-pressure/intermediate-pressure) axial turbine. The second stage incorporated an axial rotor purge cavity seal between the stationary nozzle and the rotating blade. The purpose of this study was to understand the effect of the axial gap between the rotating and stationary portion of the seal and the unsteadiness around the seal itself. The computations used Fluent to study 90° and 360° axisymmetric models of the rotor cavity with no blades or vanes modeled (only axisymmetric inlet and exit boundary conditions). The steady models did not converge due to the cavity unsteadiness and thus unsteady models were eventually used to predict the cavity behavior. It was noted that good qualitative and some quantitative agreement was found between the experimental and numerical results. Time-averaged results from the 90° models were found to match the 360° models, but the forced periodic boundary conditions lead to differing unsteady behavior. When the axial gap was reduced, it was noted in both simulations and the measurements that mainstream ingress and unsteadiness in the cavity was reduced for the same cooling flow levels. The flow patterns within the cavity were found both computationally and experimentally to rotate at a 90-97% of rotor speed.

A numerical study was conducted by Jackoby et al. [21] and compared to measurements made by Bohn et al. [12] on another axial purge seal configuration in a one and one-half stage turbine rig. In this numerical study, three companies were involved with three different codes and methodology in an effort to find the most efficient numerical approach to modeling purge flow cavities. Volvo utilized the commercial code, Fluent, to solve a periodic model of the inlet nozzle, rotating blade, exit nozzle, and the front and rear rotor purge cavities. MTU used an in-house code named Trace S/U to perform both steady (Trace S) and unsteady (Trace U) computations on a similar model to the Volvo domain but without the aft rotor purge cavity. The third company, ALSTOM, isolated the cavity from the remainder of the flow path and used
Fluent to perform steady and unsteady calculations on periodic sector and $360^\circ$ models. Reasonable agreement was found between the measurements and the numerical models for high cooling air mass flow rates in the cavities. A large scale, rotating structure was detected in the rotor cavity that significantly influenced the hot gas ingestion. For low cooling flow rates, this structure was found to rotate at about 80% of rotor speed in the front cavity which is similar to the findings of Cao et al. [20], which found this structure to rotate at about 90-97% of rotor speed. The work concluded that the steady CFD models lead to the worst agreement with the experimental data with the Volvo and MTU approach being second best and best results were obtained with the $360^\circ$ unsteady models of the rotor cavity.

A series of investigations took place at Arizona State University on a single stage, turbine rig consisting of partial height vanes and partial height/partial length blades operating at ambient conditions. The first of which was reported by Roy et al. [22] in 2001 who completed a combination of experimental and numerical work by using this rig with a radial purge seal configuration. The numerical analysis used Fluent to perform a steady, rotational, symmetric analysis with no upstream nozzle and a steady, periodic solution with the upstream nozzle present. From the experiment, it was found that the unsteady blade-periodic amplitudes were of the same order of magnitude as the time-averaged nozzle-periodic circumferential variation and that only the unsteadiness persisted into the cavity, not the asymmetry. The numerical results compared favorably to measurements of radial and tangential velocity within the cavity; however, they were found to inadequately predict the ingestion measured in the experiments. Roy et al. [22] concluded that proper predictions would most likely require three-dimensional, unsteady mainstream (nozzle and blade) and cavity solutions.

In 2005, an experimental study using a Pratt & Whitney single-stage turbine with a radial seal configuration was performed by Roy et al. [23] and compared to previous results obtained on a Honeywell single-stage turbine. The rig consisted of partial height nozzles and vanes in both models with airfoil count and nozzle turning angle being the differentiator between the two designs. Time-averaged and time-accurate
measurements were made in and around the rotor purge cavity while a tracer gas was used to determine the level of mainstream flow ingress into the cavity. It was found that mainstream gas flow rate, purge flow rate, and rotor speed had an influence on the flow field that is three-dimensional and unsteady. The measurements also suggested that another key to the mainstream ingestion is the instantaneous pressure field that contains the unsteady blade-passing period as well as the circumferential variation due to the nozzles. Roy et al. [23] concluded that three-dimensional, unsteady CFD may hold the most promise to isolating the many mechanisms associated with ingestion into the purge flow cavity, similar to his conclusions in 2001.

In 2007, Roy et al. [24] utilized the same Pratt & Whitney partial blade height rig to run additional experiments using CO2 as a tracer gas in the purge cavity in order to measure ingestion of the mainstream flow into the cavity and egress of the cavity flow in the mainstream. These experiments also incorporated the use of particle image velocimetry measurements (PIV) in the cavity to resolve the velocity fields in the tangential and radial directions. The experimental results were accompanied by three-dimensional, unsteady numerical analysis using FLUENT to study a periodic sector of the rig. The numerical model utilized full-length blades and modeled similar blade and nozzle counts in order to achieve periodicity unlike the actual turbine rig. The numerical results showed that the mainstream ingestion carried high tangential velocity into the cavity whereas the egress of cavity flow out into the main stream had significantly lower tangential velocity. The velocity fields within the rim seal showed multiple recirculation zones and suggest that the mainstream ingestion process goes from the main gas path to the seal region, through the seal gap and down the stator wall. The PIV measurements confirmed high tangential velocity fields in the background of lower tangential velocity in the cavity near the rim seal. Comparisons of circumferentially averaged, maximum, and minimum tangential and radial velocities were made between the PIV measurements and the numerical simulations, which showed reasonable agreement.
Zhou et al. [25] continued the experiments in 2009 with the Pratt & Whitney partial blade height rig by studying the cavity aspect ratio (height/width) while maintaining rim seal geometry. The study looked at both experimentally and numerically three configurations of cavity aspect ratio with the smallest being similar to those as used in high-pressure turbines of aircraft engines. The largest configuration is that of the nominal rig used by Roy et al. [22], [23], and [24] and the additional smaller configurations were achieved by installing a shelf on the stator of the cavity at the appropriate heights. The numerical simulations were carried out using a similar model to that used by Roy et al. [24] in that it was an unsteady periodic model in which the vane-to-blade ratio was adjusted to maintain periodicity in comparison to the rig. PIV measurements of the flow field throughout the cavity were taken in order to identify the egress and ingestion zones around the circumference of the rotor and were followed up with detailed tangential and radial velocity plots near these two regions. The different cavity aspect ratios were found to have little effect on the tangential and radial velocity fields near the egress locations and the tangential velocity field near the ingestion location. However, it was shown that the radial velocity field near the ingestion location decreased with smaller aspect ratios concluding that ingestion of the mainstream flow into the cavity decreased as the cavity aspect ratio became smaller. The time-averaged numerical results for the tangential and radial velocity were compared to the experimental results for the smallest aspect ratio and show good agreement; however, total ingestion into the cavity was under predicted and was thought to be due to the use of a sector model.

In 2010, Dunn et al. [26] assembled another rig with partial height vanes and partial height/length blades in order to further investigate a single condition both numerically and experimentally in conjunction with Solar Turbines Inc. The numerical model was expanded to include four vanes and blades in an effort to resolve the flow structures not obtainable with previous single blade/vane models and assumes the blade and vane counts to be identical in order to create the periodic model. Multiple grids, steady, unsteady, and the use of species transport to simulate the CO2 gas ingestion were incorporated in order to further understand the flow physics. Comparisons of static pressure at the stator surface and velocity measurements from the PIV were shown with good agreement between the simulations and experiments;
however, no comparisons of the sealing effectiveness could be made due to the time required to converge the species transport in the numerical results. Larger flow structures were seen in the expanded numerical model across the entire sector, but no concrete conclusions with respect to these structures were provided.

Okita et al. [27] performed both an experimental and numerical study on a one and one-half stage turbine rig in an effort to reduce the amount of purge flow needed with a novel design change. This study was aimed at understanding the rotor purge cavity downstream of the rotating blade whereas previous studies have focused on the cavity downstream of the stationary nozzle. Measurements in the stationary frame were taken on the rig and the cooling flow was seeded with a tracer gas to determine mainstream ingestion into the cavity. The numerical modeling was performed using three-dimensional, steady, isolated blade-row models and an unsteady, sliding mesh model of the rotor and downstream strut. The models incorporated steady inlet boundary conditions, an exit static pressure profile, and periodic boundary conditions. The steady numerical models were found to under-predict the cooling effectiveness in the rotor cavity while reasonable agreement between the numerical and experimental results was achieved with the unsteady numerical models. It was concluded that when the blades were incident with the strut, the mainstream ingress was intensified due to the strut bow wave on the stationary side of the purge flow cavity. The redesigned cavity, which incorporated a divider plate on the stationary side, was found to advance the cooling effectiveness that was then verified by the rig experimental results.

Honeywell International reported results from numerical modeling of a sub-sonic high-pressure turbine with a double lip overlap seal forward purge cavity located between the vane and the blade. In 2008, the first part of the study by Mirzamoghadam et al. [28] showed results from numerical analysis using NUMECA International’s FINE/Turbo commercial code with the objective to study a steady mixing plane solution with the mixing plane located aft of the cavity. The first part of the investigation looked at the effects of including the vane trailing edge fillet, which concluded that while the hub Mach numbers were slightly increased leading to less ingestion, the cavity temperatures were reduced less than 3% and thus the fillet
was not a significant influencing factor. Inlet cooling flow was also varied in order to further investigate the ingestion process into the cavity and a correlation between the amount of sealing flow and the resulting ingestion efficiency was provided. The correlation showed that low levels of ingestion were seen even when sealing flows were well above the minimum sealing flow as predicted by the code when purge flow rates were zero. The correlation was compared against findings from Roy et al. [24] and Bohn et al. [13] with reasonable agreement. Mirzamoghadam et al. [29] provided results from the second part of the study in 2009 that included looking at the level of cavity entrained flow (in the absence of purge flow) with decreasing annulus Reynolds number at two constant rotational Reynolds numbers as well as comparing time-averaged unsteady harmonic solution results to a mixing plane located both forward and aft of the cavity. The later study concluded that the mixing plane located aft of the cavity was more representative on the ingestion dynamics as shown by the unsteady analysis implying that the stator wake is more dominant than the blade bow when ingestion is concerned. The cavity entrainment flow in the absence of purge flow was found to increase at a higher rate for the larger rotational Reynolds number as the annulus Reynolds number was increased.

Reid et al. [30] performed an experiment and numerical analysis on a full scale, low speed axial turbine that contained low aspect ratio and low turning vanes and a significant radius change from inlet to exit. The purpose of the combined effort was to characterize the efficiency impacts of purge cavity sealing flow and injection tangential angle, but temperature drops were noted to be small which could have led to large uncertainty. The numerical code used, known as TBLOCK, was used to perform steady multistage mixing plane analysis with the mixing plane located both forward and aft of the cavity and unsteady analysis. The purge flow was found both experimentally and numerically to impact the tangential velocity of the flow field the most and thus attempting to control the tangential velocity of the purge flow could minimize these effects. Direct comparisons of the tangential velocity were not provided between the experimental and computational results, but similar results were provided for each such that one could qualitatively make the comparisons. The results showed that the steady, mixing plane analysis could do a good job of predicting the efficiency impact due to the purge flow introduction with the mixing plane aft of the purge cavity as the
dominant flow feature was the vane wakes for this turbine. Overall, the unsteady calculations were much better as they incorporate both the vane wakes and the rotor bow waves.

Paniagua et al. [31] tested a transonic turbine stage in the von Karman Institute short-duration compression facility and backed the experimental results with three-dimensional, Navier-Stokes simulations (CFX-TASCFLOW) looking at the flows in and around the purge flow cavity for both ingestion and ejection conditions. The computational results were compared to the rig measurements and were found to be in good agreement with time-averaged vane and rotor surface static pressures and radial profiles of flow angles and pressure ratios at the exit of the rotor. The cavity ejection flow was found to cause large blockage at the exit of the transonic vane and for 1.5% of ejection flow the static pressure was increased about 6% with an 11% change in the reaction at mid-span on the blade. The flow at the entrance to the cavity was found to be a competition of the lid driven cavity pattern, the radial pumping of the rotor, and the pressure imposed at the hub of the cavity all of which is dependent about the tangential location of interest. Furthermore, ingestion or ejection of flow was found to be unsteady and also dependent upon the position of the blade relative to the vane.

Pau et al. [32] utilized the CT3 rig at the von Karman Institute compression facility in combination with steady computational results produced by the code named ELSA to study various cooling rates in an effort to isolate the impact of purge flow from platform cooling. The steady analysis utilized a mixing plane forward of the rotor purge cavity such that only the effects of the blade were seen on the cavity flow. Within the wheel space cavity, ejection of the cooling flow was found to reduce the overall pressure fluctuations by up to 50% with a majority of the interaction found to be due to the upstream vane trailing edge shocks. The purge flow ejection was also found to have an impact on the vane shock system over the entire span of the vane. This purge flow was also found to feed the rotor hub passage vortex and enhance the effect of the counter-rotating, pressure side, corner vortex. Rotor leading edge measurements over the entire span showed signs of influence due to the rotor purge flow blockage created in the main gas path
such that rotor inlet Mach numbers were seen to be reduced and along with blade incidence angles at the hub surface. The numerical results were shown to correlate well with rotor exit measurements of temperature and pressure ratios, but largely these results were used to provide qualitative understanding of the flow physics at work.

Pau and Paniagua [33] has further attempted to explain the impact that the rotor purge flow has on the vane aerodynamics as discovered by Pau et al. [32] through the use of calibrated numerical results. The study involved three vane operational conditions namely subsonic flow at Mach 0.73, transonic flow at Mach 1.12, and supersonic flow at Mach 1.33 in which the loss mechanisms were separated into components of boundary layer loss, shock wave loss, and downstream loss due to pressure and suction side mixing. ANSYS-CFX was calibrated to experimental results obtained for hub static pressure measurements taken just downstream of the trailing edge such that inlet conditions were adjusted within the commercial code until a good match was obtained with the experiments. The purge flow interactions were found to be completed dominated by the vane trailing edge hub shock structures, which conversely were found to be affected by the purge flows. The purge flow was found to create a blockage in the passage that reduces Mach numbers downstream and subsequently propagates back upstream causing a thickening of the boundary layer. In the transonic case, the purge flow ejection at a rate of 1% was found to have no effect on the trailing edge shock impingement on the suction side of the airfoil whereas the supersonic case showed that the purge flow forces the shock impingement area upstream. Overall, boundary layer loss was found to increase as ejection rate was increased and reduces shock loss. The downstream mixing of the pressure and suction sides were found to also improve with the purge flow ejection ultimately improving vane efficiency.

A computational only study was completed by Julien et al. [34] in 2010 using ANSYS-CFX and a large scale sector model of approximately 74° in an effort to resolve the large scale flow features of the purge cavity in addition to the high frequency vane-blade interactions. Little detail is provided with respect to the

26
geometry except that it is based on a previous experimental turbine stage rig operating at about 3200 RPM. The computational study involved looking at cases with no purge, low purge, and high purge flow rates in an effort to provide insight into the influence of secondary flows on the ingestion process. Energetic large scale flow features were observed in the computational results and spectral analysis of the lowest purge flow rates at a point within the cavity showed energetic frequencies lower than blade passing and rotating at an angular speed slightly less than rotor speed. According to the results, these large-scale structures can lead to deep ingestion of flow from the main gas path. The high purge flow rate showed two competing frequencies of which one was associated with blade passing; only shallow ingestion was seen with the dominant phenomenon being the vane-blade interaction. At the time of publication, no comparisons to experimental results were provided.

Popovic and Hodson [35] of Whittle Laboratories reported on an experimental and numerical investigation of a highly loaded rotor blade with purge flow injection through an upstream, overlapping seal. The configuration of the seal was similar to a double overlapping radial purge seal. The experiments were performed using a low speed, large scale, linear cascade with the T120 blade (120° of turning) in which static pressure measurements were made along the blade end wall and cooling effectiveness was determined through use of Ethylene as a tracer gas. To complement the experimental investigation, Fluent was used in combination with a single equation turbulence model and a very dense mesh to simulate the experiment and to gain further insight in the purge flow problem. Multiple quantitative and qualitative comparisons were provided between the numerical results and the experimental measurements, which compared favorably. Popovic and Hodson [35] concluded that the leakage flow from the purge cavity provided a limited, concentrated film cooling on the leading edge of the rotor platform and while increasing the purge flow served to improve this film cooling it was with a large increase in aerodynamic losses. They did find that when the mainstream air was ingested into the cavity the mixing between the main stream air and the purge flow air was enhanced resulting in the best aerodynamic efficiency. However, this improved efficiency comes at the risk of increased rim seal and cavity temperatures such that the thermodynamic losses associated with the leakage flow may outweigh the potential aerodynamic benefits.
In 2010, Schuepbach et al. [36] performed several experiments in conjunction with MTU in an effort to look at purge flow performance sensitivities. The experiments looked at two purge flow conditions, one identified as a modest sucking mode in which the flow rate into the cavity is positive and the other is a blowing mode in which the purge flow rate out of the cavity is positive. Detailed time-averaged and time-accurate measurements were shown at a location downstream of the rotor and included flow angles, total pressure, efficiency, and dissipation factor. The results showed that with conventional end walls there was an efficiency drop of 0.6% by going from sucking to blowing mode with the purge cavity flow rate which was attributed to the increase in radial extent of the hub secondary flows. Spectral results also conclude that there was frequency content at around one-half of the blade passing frequency in the hub region, but no further conclusions were provided due to lack of purge cavity instrumentation. In 2011, Schuepbach et al. [37] looked at the influence of the purge flow on the performance of the same turbine rig with end wall contouring on the vane and rotating blade hub surface. This effort, directed at identifying the efficiency impacts, also included a numerical study using ANSYS-CFX to perform unsteady analysis of the rig. Using the base rig from 2010, two designs of end wall contoured vanes and blades were tested that showed efficiencies gains in the rig by as much as 1% when the purge flow was in sucking mode or positive into the cavity. When the purge flow was then turned to blowing mode, it was found that the best end wall contour design had nearly a 2X sensitivity to the purge flow than the base (non-contoured) design with the base design showing the least amount of sensitivity. Based on the numerical analysis, it was found that the best performing end wall contoured vane was characterized by also having the strongest hub perturbation such that when the purge flow is introduced it creates even stronger blowing from the cavity which ultimately introduces more blockage and thus higher losses. Schuepbach et al. [37] concluded that when designing end walls in turbines the purge flow must be accounted for correctly otherwise all improvements could be lost.
The aerodynamic phenomena associated with the interactions between the vane, blade, and the purge flow cavity have been the subject of many experimental and computational studies since the 1970’s. In recent years, there has been a shift from rigs to computational studies although often both are included in an effort to gain as much insight as possible. A majority of the rigs are often ambient temperature and pressure rigs running at low RPM such that conditions are not always representative of engine conditions (pressure ratios, temperature ratios, corrected speeds, and corrected flows) with limited exceptions. Computational results have spanned the map from single blade-row analysis to partial and full-wheel, multi-blade row, unsteady analysis and all have provided some insight into the complex flows associated with purge flow cavity operation in gas turbine engines. While a lot of insight into the aerodynamics surrounding these cavities and cavity seals has been uncovered, a significant amount of information has yet to be discovered as rig testing, instrumentation, and numerical techniques continue to improve. The research described in this thesis intends to add a significant piece of understanding to the overall aerodynamics of high-pressure turbines and rotor purge flow cavities.

1.2 Scope of the Current Study

The current study covers an aerodynamic study of a double-lipped radial purge flow cavity located between an upstream stationary vane and downstream rotating bladed disk, similar to the configuration shown in Figure 3(c). The study incorporates both experimental data obtained from a full scale, rotating rig and a matching set of numerical predictions. The numerical predictions involve the use of unsteady CFD to predict the time-average and time accurate behavior throughout the entire one and a half stage turbine rig which includes the rotor purge cavity, blade tip, and shroud regions. The experimental data is obtained using a full-scale rig that includes the high-pressure turbine (HPT) vane, a rotating HPT rotor, and the low-pressure turbine (LPT) vane as well as a functional purge flow cavity between the HPT vane and blade. The rig is operated at the proper corrected engine design conditions (i.e. corrected speed, design flow
function, pressure ratio, and temperature ratio) to match cruise conditions for the actual engine. The main focus of this study is focused on the rotor purge cavity, but some passage predictions are readily available from both the numerical and experimental investigations and are included for completeness.

The engine hardware used in the experiment is full-scale engine hardware. The engine hardware is representative of a single-stage high-pressure turbine and low-pressure turbine vane that would be found in commercial aviation turbomachinery. Typical commercial high-pressure turbine hardware requires cooling flow for the nozzles, blades and rotor due to high combustor exit temperatures but because this series of measurements represents the base line condition for CFD verification, the cooling capability for cooling has only been introduced on the high-pressure turbine nozzle and purge flow cavity. In this regard, it should be noted that the cooling scheme used for the vane row is very representative of modern HPT vane row cooling practice (for a photograph of this vane row the reader is referred to [71]). Another aspect of high-pressure turbine rotor blades commonly found in commercial engines is a recessed (often termed a ‘squealer’) tip. The rig consists of a majority of blades incorporating the flat tip geometry but several blades with the recessed ‘squealer’ tip geometry of which both will be used for data comparisons. The experiments are performed in a short-duration, blowdown facility that is located at The Ohio State University Gas Turbine Laboratory (OSU-GTL). A full description of the turbine rig, instrumentation, and data acquired will be discussed later.

The numerical simulations use NUMECA International’s FINE/Turbo, a commercial CFD code developed for resolving steady and unsteady flow physics in turbomachinery. The simulations performed using FINE/Turbo includes both steady and unsteady results throughout the entire turbine rig. These results are directly compared to the available data measured at selected locations within the experimental rig to evaluate the CFD code’s ability to capture the unsteady phenomena within and around the purge flow cavity. Once the data comparisons were complete, the simulation results were then used to provide an in-depth look at the time-averaged and time-accurate behaviors.
The results acquired from the combination of the experimental and numerical study are intended to provide a much-needed data set for the CFD community for code evaluation efforts. As well, the investigation will provide an excellent opportunity to evaluate the use of FINE/Turbo for the steady and unsteady calculations for the mainstream gas path as well as the purge flow cavity. The use of everyday computational software and real engine hardware lends itself to make this study as realistic as possible.
CHAPTER 2
GOVERNING EQUATIONS AND SOLUTION TECHNIQUES

The purpose of this chapter is to derive the governing equations for fluid flow. In addition to the derivation of the governing equations, closed form solutions for the complex flow fields within a purge flow cavity will be attempted.

2.1 Equations of Motion for Fluid Flow

In the following sections, the basic equations for fluid flow will be derived in differential form. The derivations will include the Conservation of Mass, Momentum, and Energy equations in Cartesian coordinates.

2.1.1 Conservation of Mass & the Continuity Equation

Conservation of mass principle applied to a system requires that the time rate change of mass within a system remain equal to zero (the mass within a system remains constant):

\[
\frac{DM_{sys}}{Dt} = 0
\]  

(2.1)
Extending equation 2.1 to deforming or non-deforming control volume, the control volume form of the conservation of mass equation is:

\[
\frac{\partial}{\partial t} \int_{CV} \rho \, dV + \int_{CS} \rho \cdot \mathbf{V} \cdot \mathbf{n} \, dA = 0, \quad \mathbf{V} = u \mathbf{i} + v \mathbf{j} + w \mathbf{k}
\]

(2.2)

where \(\mathbf{V}\) is the velocity vector field. Equation 2.2 states that the time rate change of mass over the entire control volume plus the net rate change of mass through the control surface is equal to zero. The control volume over which equation 2.2 will be applied is taken as a small, stationary, cubicle element as shown in Figure 5. At the center of the element is the fluid density and velocity components \(u, v, \) and \(w\) corresponding to the Cartesian coordinates \(x, y,\) and \(z\) respectively.

![Figure 5. Differential element and coordinate system for the development of the conservation of mass](image)

Defining the differential element as small and non-deforming allows for the control volume integral in equation 2.2 to be expressed in the following manner:
\[
\frac{\partial}{\partial t} \int_{cv} \boldsymbol{\rho} \cdot d\omega \approx \frac{\partial \rho}{\partial t} \delta x \delta y \delta z
\] (2.3)

The control surface integral of equation 2.2 can be evaluated by considering the flow in each of the coordinate directions as the element center masses and its gradient in each perspective direction. Starting with the x-direction, the flow through the right face of the element shown in Figure 5 is:

\[
\rho u \bigg|_{\frac{\delta x}{2}} = \rho u + \frac{\partial (\rho u)}{\partial x} \frac{\delta x}{2}
\] (2.4)

And the flow through the left face, in the x-direction, is:

\[
\rho u \bigg|_{-\frac{\delta x}{2}} = \rho u - \frac{\partial (\rho u)}{\partial x} \frac{\delta x}{2}
\] (2.5)

With the net rate of mass outflow in the x-direction:

\[
\left[ \rho u + \frac{\partial (\rho u)}{\partial x} \frac{\delta x}{2} - \rho u + \frac{\partial (\rho u)}{\partial x} \frac{\delta x}{2} \right] \delta y \delta z = \frac{\partial (\rho u)}{\partial x} \delta x \delta y \delta z
\] (2.6)

Performing this balance in the y-direction and z-direction, the net rate of mass outflow is found to be:

\[
\frac{\partial (\rho u)}{\partial x} \delta x \delta y \delta z + \frac{\partial (\rho v)}{\partial y} \delta x \delta y \delta z + \frac{\partial (\rho w)}{\partial z} \delta x \delta y \delta z
\] (2.7)

Combining equations 2.3 and 2.7 and dividing by the elemental volume \(\delta x \delta y \delta z\):

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0
\] (2.8)

Equation 2.8 is the final differential form of the conservation of mass equation for a compressible fluid.
2.1.2 Conservation of Momentum and Navier-Stokes Equations

Newton’s second law of motion for a system states that the time rate change of momentum within a system must be equal to the sum of external forces on that system. In order to develop the differential, linear momentum equation, Newton’s Second law will be applied to a differential element such that:

\[
\delta F = \frac{D(\delta m \cdot V)}{Dt} = \delta x \delta y \delta z \frac{D(\rho \cdot V)}{Dt}
\]

Note that in equation 2.9, D/Dt is the material derivative (applied to the momentum field) that will be discussed later and with the only assumption being that of constant area on each face.

Starting with the left hand side of equation 2.9, there are two types of forces acting on the differential element that must be considered: surface forces and body forces. Surface forces act on the surfaces of the differential element and body forces are distributed throughout the element. The only body force, \(\delta F_B\), which will be considered as the weight of the element expressed as:

\[
\delta F_B = \delta m \cdot g = \rho g \delta x \delta y \delta z, \quad g = g_x \hat{i} + g_y \hat{j} + g_z \hat{k}
\]

The resultant surface forces, \(\delta F_S\), arise due the differential element’s interactions with its surroundings. Each of these surface forces can be resolved into a force that is normal to the surface, \(\delta F_N\), and two forces that are parallel to the surface and orthogonal to each other, \(\delta F_1\) and \(\delta F_2\). These forces are shown for an arbitrary surface in Figure 6 below.
Figure 6. Surface force and its components acting on an arbitrary surface

Each of these surface forces can then be described in terms of a normal stress:

$$\sigma_N = \lim_{\delta A \to 0} \frac{\delta F_n}{\delta A}$$  \hspace{1cm} (2.11)

And shearing stresses:

$$\tau_1 = \lim_{\delta A \to 0} \frac{\delta F_1}{\delta A}$$  \hspace{1cm} (2.12)

$$\tau_2 = \lim_{\delta A \to 0} \frac{\delta F_2}{\delta A}$$  \hspace{1cm} (2.13)

It becomes necessary for the development of a sign convention and reference for these normal and shearing stresses. For simplicity only the x-direction stresses will be discussed, however, these conventions will be applied in the y-direction and z-direction as well. Normal stresses, indicated as $\sigma_{xx}$, are defined as positive if the surface outward normal points in the positive coordinate direction and the shear stresses, indicated as $\tau_{xy}$ (shear stress acting on the x-face in the y-direction) and $\tau_{xz}$ (shear stress acting on the x-face in the z-direction), are also positive if the stress acts in the direction of the outward normal for its direction. Figure
7 shows sign convention and notation for the x-direction shear and normal stresses applied to the x-direction face of the differential element.

Figure 7. x-Direction notation and sign convention for: (a) positive stresses and (b) negative stresses
It is anticipated that these stresses will vary throughout the flow field and thus on each face the stresses are expressed in terms of the stresses at the center of the element and their gradients in each coordinate direction (similar procedure applied in the derivation of the continuity equation) multiplied by the face area to put the stresses in terms of force. After subtracting and adding like terms, the surface force balance in the x-direction thus becomes:

$$\delta F_{sx} = \left( \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \right) \delta x \delta y \delta z$$

(2.14)

Similarly, the y-direction and z-direction are:

$$\delta F_{sy} = \left( \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \sigma_{yx}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \right) \delta x \delta y \delta z$$

(2.15)

$$\delta F_{sz} = \left( \frac{\partial \tau_{zx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z} \right) \delta x \delta y \delta z$$

(2.16)

The right hand side of equation 2.9 will be developed through the material derivative, which is defined in the following manner:

$$\frac{D(\cdot)}{Dt} \equiv \frac{\partial(\cdot)}{\partial t} + u \frac{\partial(\cdot)}{\partial x} + v \frac{\partial(\cdot)}{\partial y} + w \frac{\partial(\cdot)}{\partial z}$$

(2.17)

Applying this definition of the material derivative to momentum in the x, y, and z-direction results in the following:

**x-dir:**

$$\frac{D(\rho u)}{Dt} = \frac{\partial (\rho u)}{\partial t} + u \frac{\partial (\rho u)}{\partial x} + v \frac{\partial (\rho u)}{\partial y} + w \frac{\partial (\rho u)}{\partial z}$$

(2.18)

**y-dir:**

$$\frac{D(\rho v)}{Dt} = \frac{\partial (\rho v)}{\partial t} + u \frac{\partial (\rho v)}{\partial x} + v \frac{\partial (\rho v)}{\partial y} + w \frac{\partial (\rho v)}{\partial z}$$

(2.19)
z-dir: \[ \frac{D(\rho w)}{Dt} = \frac{\partial (\rho w)}{\partial t} + u \frac{\partial (\rho w)}{\partial x} + v \frac{\partial (\rho w)}{\partial y} + w \frac{\partial (\rho w)}{\partial z} \] (2.20)

Combining equations 2.10, 2.14-2.16, and 2.18-2.20 into equation 2.9 and cancelling the elemental volume, \(\delta x \delta y \delta z\) on both sides of the equation, the general differential equations of motion for a fluid can be assembled for each direction:

\[ \rho g_x + \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} = \frac{\partial (\rho u)}{\partial t} + u \frac{\partial (\rho u)}{\partial x} + v \frac{\partial (\rho u)}{\partial y} + w \frac{\partial (\rho u)}{\partial z} \] (2.21)

\[ \rho g_y + \frac{\partial \sigma_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} = \frac{\partial (\rho v)}{\partial t} + u \frac{\partial (\rho v)}{\partial x} + v \frac{\partial (\rho v)}{\partial y} + w \frac{\partial (\rho v)}{\partial z} \] (2.22)

\[ \rho g_z + \frac{\partial \sigma_{zz}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} = \frac{\partial (\rho w)}{\partial t} + u \frac{\partial (\rho w)}{\partial x} + v \frac{\partial (\rho w)}{\partial y} + w \frac{\partial (\rho w)}{\partial z} \] (2.23)

Equations 2.21-2.23 are general equations for all continuum and non-continuum flows. Continuum flows are defined as flows in which the shearing stress is linearly related to the rate of the shearing strain, or in equation form:

\[ \tau = \mu \frac{\partial u}{\partial y} \] (2.24)

where the constant of proportionality is the viscosity of the fluid. These types of flows are also known as Newtonian fluids of which most common fluids, both liquid and gases are Newtonian. Non-continuum or non-Newtonian fluids are fluids in which the shear stress is not linear with the rate of shearing strain. Several types of common non-Newtonian fluids can be classified as shear thinning (such as latex paint), shear thickening (such as quicksand or water-corn starch mixtures) and Bingham plastics that are neither a solid nor a liquid (toothpaste and mayonnaise). Non-Newtonian fluid flows will not be dealt with further as the Newtonian fluid assumption will be directly applied to this research.
For a Newtonian fluid flow, the normal and shear stresses can be written in terms of both pressure and velocity gradients, for the x-direction:

\[ \sigma_{xx} = -p + \mu \left( \frac{4}{3} \frac{\partial u}{\partial x} - \frac{2}{3} \frac{\partial v}{\partial y} - \frac{2}{3} \frac{\partial w}{\partial z} \right) \]  
(2.25)

\[ \tau_{xy} = \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \]  
(2.26)

\[ \tau_{xz} = \mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \]  
(2.27)

The stress terms in equations 2.25-2.27 have similar terms for the y-direction and z-direction. Note that in the normal stress equation, equation 2.25, the pressure has now been included and the viscosity of the fluid, \( \mu \), has been introduced. A further simplification has also been applied to equation 2.25 such that the second coefficient of viscosity, \( \lambda \), has been eliminated through the use of Stoke’s hypothesis that states for air, \( \lambda \approx -2/3\mu \). Substituting these three equations above into equation 2.21 and the similar terms into equations 2.22 and 2.23 results in the Navier-Stokes equations for Cartesian Coordinates:

\[ \frac{\partial}{\partial t}(\rho u) + u \frac{\partial (\rho u)}{\partial x} + v \frac{\partial (\rho u)}{\partial y} + w \frac{\partial (\rho u)}{\partial z} = \rho g_x - \frac{\partial p}{\partial x} + \frac{2}{3} \mu \left( \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{\partial w}{\partial z} \right) \] 

(2.28)

\[ \frac{\partial}{\partial t}(\rho v) + u \frac{\partial (\rho v)}{\partial x} + v \frac{\partial (\rho v)}{\partial y} + w \frac{\partial (\rho v)}{\partial z} = \rho g_y - \frac{\partial p}{\partial y} + \frac{2}{3} \mu \left( \frac{\partial v}{\partial y} - \frac{\partial u}{\partial x} - \frac{\partial w}{\partial z} \right) \] 

(2.29)

\[ \frac{\partial}{\partial t}(\rho w) + u \frac{\partial (\rho w)}{\partial x} + v \frac{\partial (\rho w)}{\partial y} + w \frac{\partial (\rho w)}{\partial z} = \rho g_z - \frac{\partial p}{\partial z} + \frac{2}{3} \mu \left( \frac{\partial w}{\partial z} - \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} \right) \]
\[
+ \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) \right] + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \right] \tag{2.30}
\]

These three equations along with the continuity equation, equation 2.8 above, provide a mathematical description of the flow of Newtonian fluids and apply to both laminar and turbulent flow. For turbulent flow, however, the velocity components each fluctuate randomly with respect to time making analytical solutions of turbulent problems difficult of not impossible. Note that these equations are non-linear, second order, partial differential equations that are not open to exact mathematical solutions with the exception to very few cases that utilize many assumptions. Common assumptions applied to these equations are:

1.) Incompressible flow
2.) Steady flow
3.) One-Dimensional flow
4.) Zero pressure gradient or hydrostatically varying pressure gradient
5.) Free surface or zero shear stress at the surface

Application of some or all of these assumptions results in simplified Navier-Stokes equations that can be solved to provide an exact analytical solution to the problem at hand. Such examples will be explored later.

### 2.1.3 Conservation of Energy and the Energy Equation

While a complete solution to the flow field can be obtained from the use of the Navier-Stokes equations, equations 2.28-2.30, and the continuity equation, equation 2.9, it is desired to also understand the temperature gradients within the complex flow field. To perform such a task, the energy equation will be required in addition to the four aforementioned equations for fluid flow.
The First Law of Thermodynamics states that energy can neither be created nor destroyed; it can only change in form. More fundamentally, energy is conserved. The conservation of energy principal will be applied to a differential volume as shown in Figure 8. An additional variable, $e$, has been introduced at the center of the element that represents the total energy (both internal and kinetic). 

![Figure 8. Differential element for the application of conservation of energy principal](image)

Performing an energy balance on the differential element in Figure 8, one finds that the rate at which the total energy within the control volume increases is equal to the rate at which total energy enters the control volume, plus the rate at which work is done on the control volume boundary due to surface forces, plus the rate at which work is done on the control volume by body forces, plus the at which heat $s$ added to the control volume by heat conduction, plus the rate at which heat is released in the control volume due to
chemical reactions. For this derivation, the work done by body forces and the heat released due to chemical reactions will be ignored.

For the first term of the energy equation, the rate at which the total energy increases within the control volume, can be defined as the following understanding that the total energy will be a function of both time and space:

\[
\frac{\partial}{\partial t} (\rho e) \delta x \delta y \delta z
\]

(2.31)

The second term, the rate at which energy enters the control volume, is found by first understanding that flux of energy across a boundary is defined as the density multiplied by the velocity, total energy, and the area of the elemental face. Performing a flux balance at each face on the element in all three directions reveals the following:

\[
\left[ \rho u e \delta y \delta z \right]_{x \rightarrow \frac{\delta x}{2}} \left[ \rho u e \delta y \delta z \right]_{x \rightarrow \frac{\delta x}{2}} + \left[ \rho v e \delta x \delta z \right]_{y \rightarrow \frac{\delta y}{2}} \left[ \rho v e \delta x \delta z \right]_{y \rightarrow \frac{\delta y}{2}} + \left[ \rho w e \delta x \delta y \right]_{z \rightarrow \frac{\delta z}{2}} \left[ \rho w e \delta x \delta y \right]_{z \rightarrow \frac{\delta z}{2}}
\]

(2.32)

Dividing equation 2.32 by \( \delta x \delta y \delta z \) and taking the limit as \( \delta x, \delta y, \) and \( \delta z \) approach zero, the final equation for the energy flux balance across the element can be expressed in the following manner:

\[
\frac{\partial}{\partial x} (\rho u e) + \frac{\partial}{\partial y} (\rho v e) + \frac{\partial}{\partial z} (\rho w e)
\]

(2.33)

On each face of the element in Figure 8, the surface forces can be defined by the surface stresses multiplied by the face area. For example, the stress vector on each x-face is defined as the following:

\[
\left( p \hat{i} - \tau_{xx} \hat{i} - \tau_{xy} \hat{j} - \tau_{xz} \hat{k} \right) \delta \delta \delta
\]

(2.34)
The y and z-faces are defined in a similar fashion. Performing the dot product of the stress vector in equation 2.34 with the velocity vector defined in equation 2.2 on each face then dividing by the elemental area and taking the limit as \( \delta x, \delta y, \) and \( \delta z \) approach zero, the equation for the work due to surface forces is obtained:

\[
\frac{\partial}{\partial x} \left( up - u \sigma_{xz} - v \tau_{xy} - w \tau_{xz} \right) + \frac{\partial}{\partial y} \left( vp - v \sigma_{zy} - u \tau_{yz} - w \tau_{zy} \right) + \frac{\partial}{\partial z} \left( wp - w \sigma_{zz} - u \tau_{zx} - v \tau_{zy} \right)
\]

\[(2.35)\]

The final term in the energy equation that will be included for this derivation is the rate at which heat enters the control volume at the surfaces due to conduction, as radiation will be ignored. Using Fourier’s law, the conduction rate equation in three dimensions can be written as the following:

\[
q'' = -k \left( \frac{\partial T}{\partial x} i + \frac{\partial T}{\partial y} j + \frac{\partial T}{\partial z} k \right)
\]

\[(2.36)\]

and performing a conduction rate balance at each face of the element (conduction rate multiplied by the face area) obtains the following equation:

\[
-k \frac{\partial T}{\partial x} \bigg|_{x+\frac{\delta x}{2}}^{x+\frac{\delta x}{2}} + k \frac{\partial T}{\partial y} \bigg|_{y+\frac{\delta y}{2}}^{y+\frac{\delta y}{2}} - k \frac{\partial T}{\partial z} \bigg|_{z+\frac{\delta z}{2}}^{z+\frac{\delta z}{2}} + k \frac{\partial T}{\partial y} \bigg|_{y+\frac{\delta y}{2}}^{y+\frac{\delta y}{2}}}
\]

\[
- k \frac{\partial T}{\partial z} \bigg|_{z+\frac{\delta z}{2}}^{z+\frac{\delta z}{2}} + k \frac{\partial T}{\partial z} \bigg|_{z+\frac{\delta z}{2}}^{z+\frac{\delta z}{2}}}
\]

\[(2.37)\]

Once again, equation 2.37 is divided by the elemental area and the limit as \( \delta x, \delta y, \) and \( \delta z \) goes is taken of the entire equation resulting in the differential form of the conduction rate for the differential element:

\[
\frac{\partial}{\partial x} \left( -k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( -k \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( -k \frac{\partial T}{\partial z} \right) = \nabla \cdot \left( -k \nabla T \right)
\]

\[(2.38)\]
Setting equation 2.31 equal to equations 2.33, 2.35, and 2.38 yields the differential form of the energy equation:

\[
\frac{\partial}{\partial t} (\rho e) = -\frac{\partial}{\partial x} (\rho u e) - \frac{\partial}{\partial y} (\rho v e) - \frac{\partial}{\partial z} (\rho w e) + \frac{\partial}{\partial x} \left( u p - u \sigma_{xx} - v \tau_{xy} - w \tau_{xz} \right) \\
+ \frac{\partial}{\partial y} \left( v p - v \sigma_{yy} - u \tau_{yx} - w \tau_{yz} \right) + \frac{\partial}{\partial z} \left( w p - w \sigma_{zz} - u \tau_{zx} - v \tau_{zy} \right) \\
- \nabla \cdot (-k \nabla T) 
\]  

(2.39)

Subsequent substitution of the normal and shear stresses, equations 2.25-2.27, into equation 2.39 would provide the final differential energy equation.

2.1.4 Additional Equations for Closure

Upon looking at equations 2.8, 2.28, 2.29, 2.30, and 2.39, it can be seen that there are five equations with nine unknowns: density, pressure, x-direction velocity, y-direction velocity, z-direction velocity, temperature, internal energy, viscosity, and thermal conductivity ($\rho$, $p$, $u$, $v$, $w$, $T$, $e$, $\mu$, $k$). Two of these variables, the fluid viscosity and thermal conductivity ($\mu$ and $k$), are transport properties of the fluid. These two properties can be determined as a function of temperature, $T$, using Sutherland’s formulas. Sutherland’s formula for the viscosity, $\mu$, of the fluid is the following:

\[
\mu = C_1 \cdot \frac{T^\frac{3}{2}}{T + C_2} 
\]  

(2.39)
The two constants will be specific to the fluid of use, however for air at moderate temperatures $C_1 = 1.458 \times 10^{-6}$ kg/(m-s-K$^{0.5}$) and $C_2 = 110.4$ K. The thermal conductivity, $k$, can be found through two methods, the first is by Sutherland’s formula, an equation similar to that of equation 2.39:

$$ k = C_3 \cdot \frac{T^2}{T + C_4} $$

(2.40)

where the constants, $C_3$ and $C_4$, are specific to the fluid. For air at moderate temperatures, these constants are found to be $C_3 = 2.495 \times 10^{-3}$ (kg-m)/(s$^3$-K$^{1.5}$) and $C_4 = 194$ K. The second method assumes the ratio of the specific heat at constant pressure, $C_p$, to the Prandtl number, $Pr$, to be constant as is the case for most gases. This method utilizes the definition of the Prandtl number:

$$ Pr = \frac{C_p \cdot \mu}{k} $$

(2.41)

Solving for $k$ in equation 2.41 and providing $C_p/Pr$ ratio for the desired gas allows for quick calculation of the thermal conductivity.

According to the state principal of thermodynamics, the local thermodynamic state is fixed by any two independent variables in the absence of chemical composition change. If these two independent variables were chosen to be $e$ and $\rho$, then equations of the form:

$$ p = p(e, \rho) $$

(2.42)

$$ T = T(e, \rho) $$

(2.43)

are required for full closure. One could further assume the fluid, or gas as in this case, to be a perfect gas such that the intermolecular forces are negligible. For most problems in gas dynamics, this assumption is possible. However, intermolecular forces become important when conditions of high pressure and low temperature exist. A perfect gas obeys the perfect gas equation of state:

$$ P = \rho \cdot R \cdot T $$

(2.44)
where \( R \) is the gas constant \((286.9 \text{ m}^2 / (\text{s}^2 \cdot \text{K})\) for air at standard conditions). Further assuming the gas to be a calorically perfect gas, the specific heat at constant temperature, \( C_v \), the specific heat at constant pressure, \( C_p \), and the ratio of specific heats, \( \gamma \), are assumed to be constant such that the following relationships exist:

\[
\begin{align*}
\epsilon &= C_v \cdot T \\
\eta &= C_v T \\
\gamma &= \frac{C_p}{C_v}
\end{align*}
\]  

(2.45

(2.46

(2.47

and

\[
C_v = \frac{R}{\gamma - 1}
\]  

(2.48

Equations 2.45 and 2.46 are only applicable if \( C_p \) and \( C_v \) are assumed to be independent of temperature. With these relationships, equations 2.42 and 2.43 become the following:

\[
\begin{align*}
p &= (\gamma - 1)\rho \epsilon \\
T &= \frac{(\gamma - 1)\epsilon}{\rho}
\end{align*}
\]  

(2.49

(2.50

With the calorically perfect gas assumption and using Sutherland’s formula for thermal conductivity and viscosity, equations 2.39, 2.40, 2.49, and 2.50 provide the addition four equations and thus closure to the continuity, momentum, and energy equations for fluid flow. The calorically perfect gas assumption is one of several different methods for providing closure to the Navier-Stokes equations. Other such methods include assuming a real gas behavior where the specific heat at constant pressure and the specific heat ratio are a function of temperature or, as in the case of steam, using tables or curve fits to represent the required state relationships.
2.2 Closed Form Solutions for Rotor-Stator Cavities

The Navier-Stokes equations can be applied to cavity flows given the enough information and assumptions in order to predict the velocity profiles within the cavity and properties of the fluid leaving the cavity. For each of the following examples, the generic cavity shown in Figure 9 below will be used.

![Generic Cavity Model](image)

**Figure 9. Generic Cavity Model**

For each example, the flow is assumed to be the following:

1.) Steady, \( \frac{\partial}{\partial t} = 0 \)

2.) Axisymmetric, \( \frac{\partial}{\partial \theta} = 0 \)
3.) Ignore gravitational effects

4.) Ideal gas behavior, \( P = \rho RT \)

5.) Constant \( C_p \) and \( \lambda \)

6.) \( V_R \& V_Z \ll V_0 \)

7.) Isentropic

From these six assumptions, the continuity equation and the Navier-Stokes equations in cylindrical coordinates reduce to the following:

Continuity: \[ \frac{1}{r} \frac{\partial V_\theta}{\partial t} = 0 \] (2.51)

Radial Momentum: \[ \rho \frac{V_\theta^2}{r} = \frac{\partial P}{\partial r} \] (2.52)

Angular Momentum: \[ \frac{V_\theta}{r^2} = \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial V_\theta}{\partial r} \right) \] (2.53)

Axial Momentum: \[ \frac{\partial P}{\partial z} = 0 \] (2.54)

For the energy equation, the one-dimensional equation as applied to a control volume will be utilized:

\[ Q_{1-2} - W_{1-2} = E_2 - E_1 \] (2.55)

Neglecting the kinetic and potential energy changes and applying assumption five to equation 2.55, the Energy equation is obtained:

\[ \dot{Q} - W_S = m \cdot C_p \cdot (T_{T2} - T_{T1}) \] (2.56)

Equations 2.51-2.54 and 2.56 will form the basis for the next few examples.
Three different examples will follow, each of which requires an assumption on the tangential velocity profile within the cavity in order to solve the five equations for the flow within the cavity as shown in Figure 9. The three examples include the assumption of a forced vortex such that the rotating wall applies a moment to the fluid within the cavity and a free vortex in which no moment is applied to the fluid.

2.2.1 Forced Vortex

The first example is that of a forced vortex such that the rotating wall of the cavity applies a moment to the fluid creating a tangential velocity profile in the radial direction:

\[ V_\theta = X \cdot \omega \cdot r \] (2.57)

Where \( \omega \) is the rotation velocity of the wall in radians per second and \( r \) is the radial location within the cavity. For this derivation, it is assumed that the tangential velocity is proportional to the angular momentum of the rotating cavity wall with \( X=1.0 \) however \( X \) could be assumed to be any value between zero and one. It is also assumed that the inlet total temperature \( (T_{T1}) \), inlet total pressure \( (P_{T1}) \), inlet tangential velocity \( (V_{\theta1}) \), and the inlet and exit radii \( (R_1, R_2) \) are known quantities and the static pressure and temperature can be found from the isentropic relationships:

\[ T_{S1} = T_{T1} - \frac{V_{\theta1}^2}{2C_p} = T_{T1} - \frac{\omega^2 R_1^2}{2C_p} \] (2.58)

\[ \frac{P_{T1}}{P_{S1}} = \left( \frac{T_{T1}}{T_{S1}} \right)^{\frac{\lambda - 1}{\lambda}} \] (2.59)
The velocity assumption is shown to be valid for the tangential momentum equation by substitution of equation 2.57 into equation 2.53 which results in the right hand side cancelling with the left hand side. Substituting equation 2.57 and assumption 4 into the radial momentum equation (2.52) and separating variables, the following differential equation is obtained:

\[ \frac{dP_s}{P_s} = \frac{\omega^2 r}{R \cdot T_s} dr \]  \hspace{1cm} (2.60)

At this point, equation 2.60 cannot be integrated as the static temperature, \( T_s \), within the cavity is assumed to vary as a function of the radial location. Thus, an equation for the static temperature as a function of radial location needs to be found. The energy equation can be further reduced by assuming zero heat transfer and knowing the shaft work is simply the torque applied to the fluid by the rotating wall times the angular velocity. Equation 2.56 becomes:

\[ T \cdot \omega = C_p \dot{m}(T_{T2} - T_{T1}) \]  \hspace{1cm} (2.61)

The torque applied to the fluid by the rotating wall is found from applying the conservation of angular momentum to the control volume:

\[ \sum T = \sum (\dot{m} Vr)_{Out} - \sum (\dot{m} Vr)_{In} \rightarrow \tau = \dot{m} (V_{\theta,2} R_2 - V_{\theta,1} R_1) \]  \hspace{1cm} (2.62)

Combining equation 2.62 with the velocity assumption in equation 2.57 and substituting back into equation 2.61 with a little rearrangement, the following relationship between the total temperature at the exit and the total temperature at the inlet is found:

\[ T_{T2} = T_{T1} + \frac{\omega^2 (R_2^2 - R_1^2)}{C_p} \]  \hspace{1cm} (2.63)

Equation 2.63 can be generalized such that this equation can be used to find the total temperature at any location anywhere in the cavity assuming the inlet total temperature and inlet radius is known.
\[ T_T(r) = T_{T1} + \frac{\omega^2(r^2 - R_i^2)}{C_p} \]  

(2.64)

Using a general form of the isentropic relationship which relates the static temperature to the total temperature from equation 2.58:

\[ T_S = T_T - \frac{V_\theta^2}{2C_p} = T_T - \frac{\omega^2 r^2}{2C_p} \]

(2.65)

Equation 2.64 can be re-written in terms of static pressure as a function of radius:

\[ T_S(r) = T_{T1} + \frac{\omega^2}{2C_p} (r^2 - 2R_i^2) \]

(2.66)

And finally, substituting equation 2.66 into equation 2.58 produces a differential equation that can now be integrated:

\[
\frac{dP_S}{P_S} = \frac{\omega^2 r}{R \left( T_{T1} + \frac{\omega^2}{2C_p} (r^2 - 2R_i^2) \right)} dr
\]

(2.67)

Utilizing the integral formula:

\[ \int \frac{x}{ax^2 + b} = \frac{1}{2a} \ln(ax^2 + b) \]

(2.68)

and integrating equation 2.65 from the inlet to the exit of the cavity, the final equation for the static pressure ratio is found:

\[
\frac{P_{S2}}{P_{S1}} = \left[ 1 + \frac{\omega^2}{2C_p T_{S1}} \left( R_2^2 - R_1^2 \right) \right]^{\frac{1}{\lambda-1}}
\]

(2.69)

At this point, all inlet and exit properties can be determined.
2.2.2 Free Vortex

The second example is that of a free vortex assumed within that cavity such that the rotating wall has no effect on the tangential velocity of the incoming fluid. It is also assumed that the inlet total temperature \(T_{11}\), inlet total pressure \(P_{11}\), inlet tangential velocity \(V_{11}\), and the inlet and exit radii \(R_1, R_2\) are known quantities as in the previous example and the static pressure and temperature can be found by using equations 2.58 and 2.59 with the appropriate velocity profiles. The tangential velocity at the exit is found by using the conservation of tangential momentum from equation 2.62 knowing that the torque applied to the fluid by the rotating wall is assumed to be zero:

\[
\sum T = 0 = \sum \left( mV_r \right)_{\text{Out}} - \sum \left( mV_r \right)_{\text{In}} \rightarrow V_{\theta,2} = \frac{V_{\theta,1} \cdot R_1}{R_2} \quad (2.70)
\]

The change in tangential velocity from the inlet to the exit is a simple ratio of radii and can be put into a generic formula that describes the tangential velocity anywhere within the cavity by replacing the radius \(R_2\) with the independent variable \(r\) in equation 2.70. The energy equation in 2.56 is assumed to have zero heat transfer and zero shaft work on the fluid:

\[
W_s = 0 = mC_p (T_{T,2} - T_{T,1}) \rightarrow T_{T,2} = T_{T,1} \quad (2.71)
\]

From these assumptions, it is readily seen that the total temperature throughout the cavity is constant. However, the static temperature still depends on the total temperature and the tangential velocity, which is a function of the radius:

\[
T_s = T_r - \frac{V_{\theta}^2}{2C_p} \quad (2.72)
\]

such that it is not constant throughout the cavity.
Substituting these assumptions into equation 2.52, the radial momentum equation, and separating variables, the following differential equation is found:

$$\frac{dP_s}{P_s} = \frac{V_{\theta,1}^2 R_1^2 dr}{R \frac{2C_p}{r} r(2C_p T_{r,1} - V_{\theta,1}^2 R_1^2)}$$

(2.73)

Utilizing the integral formula:

$$\int \frac{dx}{x(ax^2 - b)} = \frac{\ln(ax^2 - b) - 2 \ln(x)}{2b}$$

(2.74)

And integrating equation 2.73 from the inlet to the exit, the following solution is found for the static pressure ratio between the inlet and the exit:

$$\frac{P_{s2}}{P_{s1}} = \left[ \frac{R_1^2 \left( 2C_p T_{r,1} R_2^2 - V_{\theta,1}^2 R_1^2 \right)}{R_2^2 \left( 2C_p T_{r,1} R_1^2 - V_{\theta,1}^2 R_1^2 \right)} \right]^{\frac{1}{\lambda - 1}} = \left[ \frac{T_{S,2}}{T_{S,1}} \right]^{\frac{1}{\lambda - 1}}$$

(2.75)

All properties can now be calculated at the inlet and the exit of the rotor-stator cavity.

### 2.2.3 Concluding remarks on the Free and Forced Vortex Closed Form Solutions

The free and forced-vortex solutions previously derived do provide a closed form solution of the Navier-Stokes equations for a rotor-stator cavity as shown in Figure 9. Knowing the properties at the total conditions, tangential velocity at the inlet, and the exit radius all total and static properties and velocities can be calculated at the exit. The free and forced-vortex represent the two potential extremes to be found within a cavity and most likely the actual result lies within the bounds of these two assumptions.
To understand how these properties vary, using the solutions derived previously, the properties as a function of radius through the cavity were calculated utilizing air as the working fluid. These calculations assume the inlet to be at a lower radius than the exit, but the equations as previously derived can be used for a cavity with an inlet radius larger than the exit radius as well. Figure 10 shows the distribution of total temperature through a cavity using both the free and forced-vortex assumptions. For the free vortex, the total temperature is constant where the forced-vortex total temperature increases from the inlet to the exit due to work applied to the air from the rotor wall. The static temperature distribution throughout the cavity is shown in Figure 11. Notice that for both solutions the static temperature is rising between the inlet and exit, with the forced vortex having a greater rate of increase.

Figure 10. Total temperature distribution within the Rotor-Stator Cavity for a free and forced vortex assumption
Figure 11. Static temperature distribution within the Rotor-Stator Cavity for a free and forced vortex assumption

Figure 12 shows the total pressure distribution as a function of the cavity radius. The free vortex assumption also produces a constant total pressure as well as constant total temperature throughout the cavity whereas the forced vortex total pressure increase from inlet exit, again due to the work input into the fluid by the rotating wall. The resulting static pressure distribution is provided in Figure 13 where much like the static temperature both the free and forced vortex assumptions produce an increase distribution from inlet to exit. The forced vortex assumption’s rate of increase is greater than that of the free vortex.
Figure 12. Total pressure distribution within the Rotor-Stator Cavity for a free and forced vortex assumption.

Figure 13. Static pressure distribution within the Rotor-Stator Cavity for a free and forced vortex assumption.
Finally, the tangential velocity distribution in the cavity is shown in Figure 14. For the forced vortex assumption, the tangential velocity increases linearly from inlet to exit as anticipated. The free vortex, on the other hand, decreases at a rate slightly less than linear from inlet to exit.

![Graph showing tangential velocity distribution](image)

**Figure 14.** Tangential velocity distribution within the Rotor-Stator Cavity for a free and forced vortex assumption

These two closed form solutions provide a method for calculating the property distributions within a rotor stator cavity assumed to be similar in design to that of Figure 9. However, in reality a majority of the cavities used within turbomachinery are not simple radial cavities but incorporate a complex shape that results from the surrounding hardware. Furthermore, these two solutions ignore the effects of the axial and
radial velocities, unsteadiness, and turbulence that are all inherit within turbomachinery. Once these assumptions are removed from the Navier-Stokes equations, the equations become unsolvable through a closed form solution and thus an analytical approach may provide better insight into the working nature of a rotor-stator cavity.
CHAPTER 3
EXPERIMENTAL MODELING, SETUP, AND INSTRUMENTATION

The purpose of this chapter is to provide an overview of the experimental rig, instrumentation, and the results obtained during the experimental program. The experimental turbine rig design, development, assembly, instrumentation, operation, and data collection was largely performed by the full-time staff of The Ohio State University Gas Turbine Laboratory and the level of involvement for this research was limited to understanding of how the entire operation was performed, observation of several of the test points, and limited data analysis with respect to the instrumentation used primarily for the CFD comparisons. The experimental program described here and the data utilized for the comparison with CFD spanned many years of faculty, technical staff, and students. It would be impossible to incorporate all of the factors that went into making this program successful and the attempt will only be to provide enough of an overview such that the reader can understand the experiments in order to facilitate discussion of the experimental data and analysis as well as the computational work that will be described thoroughly in Chapter 5.

3.1 The OSU-GTL Turbine Test Facility

The experimental portion of this research occurred at the Turbine Test Facility (TTF) now located at The Ohio State University Gas Turbine Laboratory (OSU-GTL). This facility is a short-duration blowdown facility that can be used to provide primary flow and cooling gas flow to a turbine rig housing a full-stage
machine operating at design corrected conditions. Development of the TTF and the associated instrumentation began over 30+ years ago, initially at the Cornell Aeronautical Laboratory, then at CALSPAN Corporation, and now at the OSU-GTL. One the main goals of the TTF is to incorporate the numerous complicating factors found within full-scale turbomachinery into the turbine rig experiments in order to provide realistic data sets to not only facilitate accurate and useful data for CFD prediction comparisons but to also be directly incorporate into the design process.

A general schematic of the TTF is shown in Figure 15. The supply tube, the dump tank, and the Large Cooling Facility (LCF) are responsible for providing the correct flow conditions for the turbine and are reused for each experimental program with only slight modifications. The turbine rig itself sits inside the dump tank and contains the working turbine of interest along with a majority of the instrumentation. Each rig is specifically built for the individual goals of the experimental programs and thus is not considered to be part of the overall TTF. The construction and operation of the TTF was first documented by Dunn [38] but more accessible descriptions of the operation of the facility have been reported in Dunn [39], [40], and [41].
Figure 15. Schematic of the Turbine Test Facility

The facility was originally built as a shock tunnel with a driver section and driven section separated by two diaphragms with the purpose being to produce a high-pressure and high-temperature air source within a reflected-shock reservoir and then the shock processed gas would be expanded through the conical nozzle and provide the heated air for the inlet to the nozzle guide vane row for the turbine stage operation. The shock tube has a 0.47 meter (18.5 inch) diameter by 12.2 meter (40 feet) long driver tube and a 0.47 meter (18.5 inch) diameter by 18.3 meter (60 feet) long driven tube. The driver tube is designed to be sufficiently long that the reflected wave system from the driver endwall would not terminate the test time prematurely. While the shock tube provides the flow conditions desired for running uncooled turbine rigs, the desire to operate with fully-cooled turbine stages and to provide vane inlet temperature profiles has led to discontinued used of the shock tube functionality and the facility is now operated exclusively in blowdown mode. In its current configuration, the diaphragms have been removed and the driver and driven sections are joined in one long (100-feet) high-pressure reservoir controlled by a fast acting valve (FAV). When the FAV is operated, the reservoir flow expands into the evacuated expansion nozzle and dump tank. As the flow passes the turbine rig inlet nozzle, a small amount of the reservoir flow enters the turbine as the main gas path flow. Currently, cooling flows are provided to the turbine rig from a separate facility known as the
LCF that runs in a similar manner acting also as a smaller blowdown facility. A separate FAV triggers the release of the coolant gas flow that passes through the walls of the dump tank via cooling tubes and enters the turbine rig. The sequencing of these two valves is critical and is all done electronically as described below.

A typical experiment for this program utilizing a cooled turbine stage begins when an air turbine is activated to bring the turbine rotor up to a speed slightly below the design speed in the evacuated chamber and an electronic triggering system starts the main data acquisition system followed at a prescribed time by the opening of the FAV of the LCF. The cooling flows are quickly established within the turbine rig immediately after the coolant flow is established the FAV on the main tunnel is triggered releasing the high-pressure air stored in the reservoir into the dump tank and subsequently into the inlet nozzle of the turbine rig. As the air enters the turbine rig inlet nozzle, it passes through a heat exchanger known as the combustor emulator where it is heated up to the desired temperature and the desired temperature profile is imparted to the gas stream. Downstream of the combustor emulator, the flow creates a series of chokes throughout the turbine rig beginning at the throat of the high-pressure turbine nozzle, which is the main flow governing choke point, and at the exit choke of the turbine rig which is used to set the pressure ratio across the turbine rig. As this is happening, the cooling flows are quickly adjusting to the changing pressure field and within approximately 50 milliseconds the flow reaches steady state. After about 750 milliseconds, the FAV on the main tunnel is closed shutting off the main flow to the turbine rig and thus terminating the experiment. It is important to note that the actual data window over which the design point data (100% corrected speed ± 1%) is acquired is only about 15 milliseconds in length depending on the rotation speeds of the rig with the objective to capture about two full rotations. The rotating turbine stage is not constrained so as work is extracted from the flow, the rotor speed increases. This is good in several ways. First, by measuring the moment of inertia of the rotating system and knowing time rate of change of speed one has a direct measure of the torque and thus the aerodynamic performance. Secondly, from the same experiment one can get data at 100% ± 1% of corrected speed and also at other corrected speeds
within a few percent of 100%. In total, a single experiment lasts less than one second thus the term short-duration facility.

3.2 The Turbine Rig

The turbine model or flow path is located in a device commonly referred to as the turbine rig positioned within the expansion nozzle of the shock tunnel facility. A cross-section of the turbine rig used for this experimental program can be seen in Figure 16 below.

![Turbine Rig Diagram](image)

**Figure 16.** Turbine rig cross-section used for the experimental program (Not To Scale)

The turbine rig consists of an inlet duct, combustor emulator, forward inlet flow path, the turbine flow path, an exit nozzle designed to govern the flow through the rig, two slip ring units used to take the rotating
signals to the laboratory stationary frame of reference, the bearing support package, the shaft encoder, an
air motor drive system used to accelerate the turbine disk to the proper design point speeds, and all of the
necessary support structure for the diagnostic and turbine flow path instrumentation. A majority of the
effort for this experimental program is tied to the design, construction, and instrumentation of the turbine
rig. Shown in Figure 17 is a photograph of the actual rig just prior to installation into the dump tank.

![Image](image_url)

**Figure 17.** Turbine rig assembly prior to installation into the dump tank

The turbine flow path for this turbine rig is typical in aerodynamic design to that of a commercial, single
stage, high-pressure ratio, high-pressure turbine and low-pressure vane. The flow path geometry is full
scale of the actual engine hardware and is operated at the proper design point conditions to match cruise

65
conditions of the engine, the parameter matching will be discussed in more detail later. The next few sections will discuss some of the major components and associated instrumentation of the turbine rig.

3.2.1 Combustor Emulator

The combustor emulator is a large, passive heat exchanging device located upstream of the high-pressure vane inlet. It is constructed of thousands of cells called honeycomb made from an Inconel material (nickel based alloy) that run the full 24-inch length of the combustor emulator. A picture of the honeycomb structure can be seen in Figure 18 and a picture of the front of the combustor is shown in Figure 19.

![Combustor emulator honeycomb structure](image)

Figure 18. Combustor emulator honeycomb structure
As the air enters the honeycomb, it is heated to nearly the metal temperature thus transferring the temperature distribution within the combustor emulator to the incoming air stream. Electrical heater rods are placed throughout the combustor emulator such that the temperature distribution within can be carefully managed. The combustor emulator designed for this experiment is capable of producing uniform temperature distributions (circumferentially and radially), one-dimensional radial profiles, or a two-dimensional profile consisting of hot and cold spots around the circumference. This gives the turbine rig the ability to simulate temperature distortions entering the high-pressure vane that would be produced by an upstream combustor. Haldeman et al. [42] provides additional information on the design and construction of the combustor emulator.
3.2.2 Inlet and Exit Rakes

Temperature and pressure rakes are positioned upstream of the inlet to the high-pressure vane and downstream of the low-pressure vane in order to capture both the incoming and outgoing conditions for the turbine stage and one-half machine. The position of the rakes can be seen in Figure 16. These rakes are used to calculate the performance of the rig and also serve as boundary conditions for computational models. At the inlet, there are two total temperature rakes (TU1 and TU2), one total pressure rake (PU2), and three static pressure rakes (PFU1, PFU2, and FFU3). At the exit location, there are also two total temperature rakes (TD1, TD2), one total pressure rake (PD2), and three static pressure rakes (PFD1, PFD2, and PFD3). All of these rakes can be positioned at 45° increments starting from top dead center (TDC) around the circumference at both the inlet at exit. A schematic of the positioning in the turbine rig is shown in Figure 20 and the default position of the rakes for the experimental runs of concern for this research is provided in Table 1.

Figure 20. Rake positions for both the inlet and exit locations, forward looking aft
<table>
<thead>
<tr>
<th>Port Letter</th>
<th>Angular Position</th>
<th>Upstream Rake</th>
<th>Downstream Rake</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>0</td>
<td>Plug</td>
<td>Plug</td>
</tr>
<tr>
<td>B</td>
<td>45</td>
<td>TU1</td>
<td>TD1</td>
</tr>
<tr>
<td>C</td>
<td>90</td>
<td>PFU2</td>
<td>PFD2</td>
</tr>
<tr>
<td>D</td>
<td>135</td>
<td>PFU1</td>
<td>PFD1</td>
</tr>
<tr>
<td>E</td>
<td>180</td>
<td>Plug</td>
<td>Plug</td>
</tr>
<tr>
<td>F</td>
<td>225</td>
<td>PU2</td>
<td>PD2</td>
</tr>
<tr>
<td>G</td>
<td>270</td>
<td>PFU3</td>
<td>PFD3</td>
</tr>
<tr>
<td>H</td>
<td>315</td>
<td>TU2</td>
<td>TD2</td>
</tr>
</tbody>
</table>

Table 1. Default positioning of the inlet and exit rakes

Both the inlet and exit total temperature rakes utilize 0.001-inch diameter wire that is butt welded together to form a thermocouple that has no additional thermal inertia (welding bead is absent) as this allows for high frequency measurements in the range of 1-2kHz. The inlet and exit total pressure and static pressure rakes utilize miniature Kulite pressure transducers that are capable of high frequency measurements in the range of 100 kHz, significantly more than the thermocouples such that the pressure transducers will capture the unsteady behavior quite well.

The inlet total temperature rakes contain nine locations every 10% span from 10% to 90% span from the hub. Nine locations were desired in order to resolve the incoming temperature profiles from the combustor emulator. The inlet total pressure rake only contained five locations every 20% of span from 10% to 90% of span from the hub and the static pressure measurements at the inlet rake location are taken at 100% span on the shroud surface. The inlet total pressure profiles were anticipated to be uniform from tip to hub and thus fewer locations were acceptable. It should be noted that there are four struts in the inlet flow path upstream of the inlet rakes located at positions A, C, E, and G that are thought to have an impact on the measurements taken behind these locations. However, computational analysis of this area shows that the Mach numbers through the upstream inlet flow path area are small, thus losses due to the strut as small and
a negligible difference between static pressure measurements at the strut locations and the non-strut locations should be similar, which is also seen within the rig measurements. The exit total temperature and total pressure rakes have five locations throughout the span at 13.7%, 34.6%, 54.4%, 73.1%, and 90.9% of span from the hub, however, only the top three locations on the total pressure rakes were in full working condition during the experiments.

3.2.3 High Pressure Turbine Vane

The high-pressure turbine vane blade row is made up of 38 equally spaced (circumferentially) vanes that have a strong three-dimensional aerodynamic shape. The vane is transonic with the throat located near the trailing edge on the pressure side of the vane. This throat controls the overall mass flow rate through the turbine during the experimental windows. A film-cooling pattern has been drilled into the surface of the vane along with the film cooling holes in the hub and shroud surfaces. A total of 488 cooling holes and 17 trailing edge slots per vane are incorporated into the film-cooling pattern. The vane surface incorporates 386 holes split among 7 radial rows on the suction surface, 9 radial rows on the pressure surface, and 17 trailing edge slots while the hub surface has 47 holes and the shroud surface has 72 holes.

The high-pressure vane has been instrumented on the vane surface, hub, shroud and purge flow cavity angel wing. The high-pressure vane surface is instrumented with double-sided Kapton heat flux gauges at 140 locations amongst the 38 vanes; this instrumentation will not be utilized for this research and is only mentioned for completeness. Additionally, there are six Kulites measuring static pressure installed on in the hub surface at an axial location just downstream of the vane trailing edge. The Kulites are split up amongst three different vanes (two gauges at each location) with one of the Kulites set in line with the vane
trailing edge and the other at 50% of the circumferential vane passage. A picture of the Kulites as installed within the turbine stage can be seen in Figure 21.

![Kulite at 50% of vane passage and Kulite in-line with vane trailing edge](image)

Figure 21. High-pressure vane trailing edge Kulites installed in the hub surface

The remainder of the instrumentation that is located in the stationary part of the purge cavity angel wing consists of six Kulites, and three 0.0005-inch diameter wire butt-welded thermocouples located at the same positions as the hub surface Kulites. A picture of this instrumentation installed at one of the three locations is shown in Figure 22 below.
There are also nine Pyrex heat flux gauges mounted in these locations, again this instrumentation will not be utilized for this research and is only mentioned for completeness. These installations can also be seen in Figure 21.

3.2.4 High-Pressure Turbine Blade (Rotating)

The high-pressure turbine blade consists of 72 unshrouded, equally spaced (circumferentially) blades that also have a strong three-dimensional aerodynamic shape as dictated by the upstream blade row. The clearance between the tip of the blade and the stationary shroud is ~2% of blade height. A majority of the
blades have flat tips with a few blades incorporating recessed geometry. For this experimental program, the rotating blades are all uncooled and have been rotated closed in order to maintain the proper aerodynamics. The blade is transonic with the passage shock emanating from the pressure side trailing edge over to about 75% wetted distance on the suction surface.

Similar to the high-pressure vane, the high-pressure blade incorporates rotating instrumentation throughout the blade surface, platforms, tip, and purge flow cavity angel wing. The blade leading edge has 36 shrouded thermocouples made from 0.001-inch diameter wire split up amongst four blades around the circumference. The main function of the shroud is to allow the thermocouples to act as total temperature sensors in the rotating frame of reference. A picture of the leading edge thermocouples on one of the blades is shown in Figure 23.

Figure 23. High-pressure rotor leading edge shrouded thermocouples
The shrouded thermocouples are set at 5% span increments starting at 5% of span from the hub surface up to 95% span (19 locations in total). The platform of the high-pressure turbine blade is instrumented with 21 platform thermocouples split up amongst three blades within the rotor. The platform thermocouples are a combination of eighteen 0.001-inch and three 0.0005-inch diameter thermocouples and are placed in the locations shown in Figure 24.

![Diagram]

**Figure 24. High-pressure rotor platform thermocouple positions**

The thermocouples were originally installed into the platform as loop thermocouples protruding 0.025-inch into the flow with some protruding 0.050-inch into the flow above the platform. Due to aerodynamic forces generated during the experiments, the thermocouples did not retain their original shape and upon disassembly it was found that each of thermocouples was at a different height. Figure 25(a) shows a thermocouple as installed prior to the experiments and Figure 25(b) shows a thermocouple after disassembly of the turbine rig.

74
Figure 25. High-pressure rotor platform thermocouple (a) as installed and (b) post-experiment
It is important to consider the thermocouple height above the platform as noted after the experiments as these thermocouples sit within the high thermal gradient boundary layer. Table 2 provides the after-run heights of the thermocouples for the platform (labeled TRP) and the purge cavity angel wing thermocouples (TRW), which will be discussed next. It should be noted that only nine thermocouples survived all of the experimental runs of which only a single thermocouple is the 0.0005-inch diameter wire.

It is important to note that after the initial experiment (the one on which the thermocouples are thought to have bent over) the measured temperature for individual thermocouples at repeat conditions remained very consistent. This result suggested that the position of the thermocouple did not change after the initial bending over of the wire. In addition to the platform thermocouples, there are also 12 Pyrex heat-flux gauges installed on the platform utilizing the same locations as the thermocouples shown in Figure 24. Once again, the heat-flux gauges will not be utilized for this thesis, but are the topic of other thesis works.

<table>
<thead>
<tr>
<th>Gauge Label</th>
<th>Height Above Surface (inch)</th>
<th>Wire Diameter (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRP38</td>
<td>0.022</td>
<td>0.001</td>
</tr>
<tr>
<td>TRP59</td>
<td>0.002</td>
<td>0.001</td>
</tr>
<tr>
<td>TRP39</td>
<td>0.020</td>
<td>0.001</td>
</tr>
<tr>
<td>TRP40</td>
<td>0.035</td>
<td>0.001</td>
</tr>
<tr>
<td>TRP40</td>
<td>0.060</td>
<td>0.001</td>
</tr>
<tr>
<td>TRP46</td>
<td>0.025</td>
<td>0.001</td>
</tr>
<tr>
<td>TRP47</td>
<td>0.020</td>
<td>0.001</td>
</tr>
<tr>
<td>TRP57</td>
<td>0.002</td>
<td>0.0005</td>
</tr>
<tr>
<td>TRP61</td>
<td>0.001</td>
<td>0.001</td>
</tr>
<tr>
<td>TRW62</td>
<td>0.001</td>
<td>0.001</td>
</tr>
<tr>
<td>TRW63</td>
<td>0.002</td>
<td>0.001</td>
</tr>
<tr>
<td>TRW64</td>
<td>0.010</td>
<td>0.001</td>
</tr>
<tr>
<td>TRW65</td>
<td>0.001</td>
<td>0.001</td>
</tr>
</tbody>
</table>

Table 2. Height of thermocouple junction above the surface
The rotating portion of the purge flow cavity angle wing also incorporates thermocouples, Kulites, and heat-flux gauges similar to the high-pressure turbine vane. There are 5 thermocouples and 5 Kulites installed both above and below the angle wing in the locations as shown in Figure 26.

![Diagram of high-pressure rotor rotating purge flow cavity instrumentation locations]

**Figure 26.** High-pressure rotor rotating purge flow cavity instrumentation locations

All 5 Kulite locations are available and from Table 2 it can be seen that 4 out of the 5 thermocouples survived the Build 2 experimental program that consisted of about fifty separate experimental runs. The 10 angel wing heat-flux gauges are located at the same positions as the Kulites and thermocouples shown in Figure 26, however this data will not be utilized within this thesis.

The high-pressure turbine blade surface incorporates additional heat-flux gauges and Kulite pressure transducers throughout the rotor. Figure 27 shows the blade surface instrumentation spread across multiple blades within the rotor along with several of the leading-edge thermocouples. In total, there are four Kulites available on the blade surface in which all four are located at 50% span on the pressure and suction
side of the blade. The blade surface Kulites were kept to a minimum for Build 2 due to previous comparisons completed as part of the Build 1 experimental series between the CFD predictions and the experimental data (many additional Kulite pressure transducers were used in this build), which showed excellent agreement for the uncooled upstream vane. Once such comparison is provided in Green [62] for MSU-TURBO and will be discussed later in section 4.3.3 for FINE/Turbo, which will be used throughout this research. The main purpose behind the Kulites is to serve as a diagnostic measurement only for the CFD predictions. The blade surface is also instrumented with 104 Pyrex heat-flux gauges. The heat-flux gauges are located at multiple spanwise and streamwise locations on the blade; however this instrumentation will not be utilized for this thesis.

Figure 27. High-pressure rotor blade surface and leading edge instrumentation
In addition to all of the aforementioned instrumentation, the rig also incorporate four static pressure Kulites and twelve heat-flux gauges on the non-rotating shroud above the blade and four blade tip heat-flux gauges for the purposes of capturing the physics between the tip cap of the rotating blade and the stationary shroud surface. This instrumentation will not be a part of this thesis and is mentioned only for completeness.

3.2.5 Low-Pressure Turbine Vane

The low-pressure vane row consists of 38 equally spaced (circumferentially) airfoils that are also three-dimensional in aerodynamic shape. The low-pressure vane is also aerodynamically set for counter-rotation of the low-pressure spool relative to the high-pressure spool. For the Build 2 experimental program, this vane did not support any instrumentation.

3.2.6 Other Facility Measurements

There are many additional pieces of instrumentation that are incorporated in the turbine rig to allow for monitoring and quantifying the operation of the rig itself and the facility. These measurements include the TTF dump tank pressure, LCF temperature and pressure, the vibration levels near the bearing package, and temperatures of the bearing package and casings. While these measurements are critical to the successful completion of the overall experimental program, they are less critical to the research being considered within this thesis and thus will not be discussed in any further detail.
3.3 Parameter Matching

As previously mentioned the turbine flow path geometry is full scale actual engine hardware and is operated at the proper design corrected conditions in order to match design point conditions of the engine. In order to do so, the turbine rig must be matched to the engine operating conditions through parameter matching. The primary parameter to match for the turbine is the corrected flow rate, which is often referred to as the flow function through the turbine. The corrected flow rate is:

\[
\dot{m}_{\text{Corr}} = m \sqrt{\frac{T_{\text{Inlet}}}{T_{\text{Ref}}}} \left( \frac{P_{\text{Ref}}}{P_{\text{Inlet}}} \right)
\]  

(3.1)

Where \( P_{\text{Ref}} \) is equal to 101.325 kPa and \( T_{\text{Ref}} \) is equal to 288.15 K. The choke are of the high-pressure vane determines the overall mass flow rate for the turbine rig and the corrected mass flow rate (left hand side of equation 3.1) is set by the turbine designer leaving only the inlet total pressure and total temperature as the two variables that can be used for adjusting the flow rate. The inlet total pressure is typically set at a level that is sufficiently high enough such that flow conditions are representative of the flight envelope but low enough as not to cause significant damage to the turbine rig during operation. The vane inlet total temperature is chosen such that the core-to-wall temperature ratio of the engine can also be matched within the rig:

\[
TR_{C-W} = \frac{T_{\text{Inlet}}}{T_{\text{Metal}}}
\]  

(3.2)

The corrected speed of the rig is also a primary matching parameter between the engine and the turbine rig as this parameter dictates the velocity triangles, incidence angles, etc. throughout the entire turbine. This parameter is defined as:

\[
\omega_{\text{Corrected}} = \omega \sqrt{\frac{T_{\text{Inlet}}}{T_{\text{Ref}}}}
\]  

(3.3)
where $\omega$ is the physical rotating speed of the high-pressure turbine disk and blades in RPM. With the inlet temperature set by the corrected flow rate in equation 3.2, the only variable left to adjust is the physical RPM of the rotor in order to match the corrected speeds between the engine and the turbine rig. In a typical experiment, the rotor RPM can be set to within 1 RPM of the desired value; however, in setting this initial speed prior to activation of the gas flows, one must account for the rotor speed up once the main stream gas and the coolant flow gas begin to flow and the inlet temperature fall-off. Both of these are well understood and hence easily adjusted for during rig operation.

The last primary parameter used for matching turbine design conditions is the turbine stage pressure ratio. The turbine pressure ratio comes in two forms, the total-to-static pressure ratio:

$$\text{PR}_{T-S} = \frac{P_{T,\text{Inlet}}}{P_{S,\text{Exit}}}$$

and the total-to-total pressure ratio:

$$\text{PR}_{T-T} = \frac{P_{T,\text{Inlet}}}{P_{T,\text{Exit}}}$$

For proper design point duplication, it is much better to work with the total-to-total pressure ratio value. The total-to-total pressure ratio is set through use of the adjustable exit choke located downstream of the low-pressure turbine vane while the inlet total pressure can be controlled either by the adjustment of the initial pressure within the supply tube or by repositioning the inlet to the turbine rig further into or away from the inlet throat to the expansion nozzle. The inlet total pressure must be set high enough to maintain the secondary choke at the exit as once both the cooling flow and main stream gas flow are initiated the dump tank pressure begins to rise. The total-to-static pressure ratio carries approximately the same goals as the total-to-total pressure ratio, but provides a less direct method of understanding total work extraction across the turbine. This is a typical measure used throughout the gas turbine industry and thus is carried for consistency. It is also worth mentioning one of the classic parameters for matching, the Reynolds Number:
\[ \text{Re} = \frac{m}{\mu AL} \quad (3.6) \]

In this form of Reynolds Number, \( A \) is the choke area and \( L \) is a length of one meter. This is the typical Reynolds Number form used for turbine applications. While the Reynolds Numbers between the engine and the turbine rig cannot be directly matched due to turbine rig durability with high inlet total pressures, they are typically kept to within 30-40% of each other.

The rest of the turbine parameters used for matching deal with the cooling flows inside the turbine flow path. For each circuit of cooling flow it is important to match the blowing ratio:

\[ BR = \frac{P_x M_x \sqrt{\gamma_x R_x T_x}}{P_{\infty} M_{\infty} \sqrt{\gamma_{\infty} R_x T_x}} \quad (3.7) \]

the coolant-to-metal temperature ratio:

\[ TR_C = \frac{T_{\text{Coolant}}}{T_{\text{Metal}}} \quad (3.8) \]

and the mass flow rate ratio:

\[ MR_C = \frac{\dot{m}_{\text{Coolant}}}{\dot{m}_{\text{Core}}} \quad (3.9) \]

The blowing ratio cannot be set or adjusted for each row of cooling holes within a given circuit and thus is only matched at a single location and the remainder of the locations are allowed to set themselves during rig operation just as they do in the engine environment. The mass flow rate through each circuit is adjusted to meet the correct mass flow rate fractions as would be utilized within the engine and is accomplished through setting choke area within the cooling circuits at the exit of the LCF pressure reservoir.
3.4 Rig Conditions and Run Matrix

The turbine rig was designed to be flexible in that many of the parameters are adjustable throughout the experimental program. The cavity purge flow and the vane cooling flows (both inner and outer circuits) were designed such that a nominal, high, low, and plugged flow (no cooling flow) rate condition could be achieved depending on the individual goals of each experiment. The combustor emulator could also be used to create uniform, radial, radial/circumferential hot streaks, or be turned off to produce a cold inlet temperature profile. The maximum-to-average temperature could also be adjusted from low to high in order to provide different levels of radial and hot streak profiles at the high-pressure vane inlet. The operational characteristics within each run are carefully chosen to provide a balance between repetitive measurements and exploration of a variety of parameters. For this research, five of the many experimental runs were chosen to be analyzed in further detail: Run 21, Run 22, Run 28, Run 33, and Run 43. The operating parameters for each of these runs are provided in Table 3.

<table>
<thead>
<tr>
<th>Run</th>
<th>Max To Avg Inlet TT (K)</th>
<th>TT Ratio</th>
<th>PT-to-PS Ratio</th>
<th>Corrected Flow (kg/sec)</th>
<th>Corrected Speed (RPM)</th>
<th>HPV Film Cooling (% of Inlet)</th>
<th>Purge Cavity Flow (% of Inlet)</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>1</td>
<td>1.40</td>
<td>5.33</td>
<td>4.59</td>
<td>6588.35</td>
<td>0%</td>
<td>0%</td>
</tr>
<tr>
<td>22</td>
<td>1.11</td>
<td>1.47</td>
<td>5.64</td>
<td>4.83</td>
<td>6013.10</td>
<td>0%</td>
<td>0%</td>
</tr>
<tr>
<td>28</td>
<td>1.16</td>
<td>1.57</td>
<td>5.61</td>
<td>4.73</td>
<td>6286.26</td>
<td>12.90%</td>
<td>0.72%</td>
</tr>
<tr>
<td>33</td>
<td>1.02</td>
<td>1.55</td>
<td>5.87</td>
<td>4.69</td>
<td>6275.89</td>
<td>12.90%</td>
<td>0.71%</td>
</tr>
<tr>
<td>43</td>
<td>1.02</td>
<td>1.40</td>
<td>5.94</td>
<td>4.69</td>
<td>6400.51</td>
<td>0%</td>
<td>0%</td>
</tr>
</tbody>
</table>

Table 3. Rig operating conditions for all runs studied

83
Run 21 represents a non-heated inlet (cold temperature profile) experiment with a constant total temperature profile, no high-pressure vane film cooling, and no flow introduction into the purge flow cavity. Run 22 represents a heated inlet experiment with a radial inlet total temperature profile, no high-pressure vane film cooling, and no flow introduction into the purge flow cavity. Run 28 represents a heated inlet experiment with a radial inlet total temperature profile, nominal high-pressure vane film cooling flow, and nominal cooling flow introduction into the purge flow cavity. Run 33 represents a heated inlet experiment with a near constant inlet total temperature profile, nominal high-pressure vane film cooling, and nominal cooling flow introduction into the purge flow cavity. Run 43 represents a heated inlet experiment with near constant inlet total temperature profile, no high-pressure vane film cooling, and no cooling flow introduction into the purge flow cavity. Thus, Run 22 and Run 28 are utilized because they are similar in overall turbine operating conditions with the exception of cooling as is Run 33 and Run 43. These runs will be utilized throughout this research to study a variety of parameters associated with the purge flow cavity including ingestion of main stream gas, unsteady pressure fields within the cavity, interaction of the purge cavity cooling flow with the main stream gas, and overall impact to the turbine efficiency. The results from these five runs of the experiments will be provided later in Chapter 6.
CHAPTER 4
OVERVIEW AND EVALUATION OF THE COMPUTATIONAL FLUID DYNAMICS CODE

The main focus of this research is centered on prediction of the time-averaged and time accurate fluid flow behavior within and around a purge flow cavity positioned between a stationary vane and a rotating turbine disk. For all of the numerical analysis, a commercially available computational fluid dynamics code (CFD) FINE/Turbo of NUMECA International was utilized. FINE/Turbo is multi-purpose, Reynolds Averaged Navier-Stokes, steady and time-accurate, 3-dimensional, viscous CFD code that is specifically adapted to handle turbomachinery applications. The overall package incorporates the pre-processor, solver, and post-processor bundled into a single system that handles the entire analysis from grid-generation to solution visualization.

Throughout this chapter, FINE/Turbo will be discussed in detail from deriving the RANS equations, to providing details of the numerical methods applied within the code, and finally by applying the code to various test cases to understand the code’s behavior. This paper will not attempt to provide a full validation of FINE/Turbo, but it will include evaluation of the CFD code. These evaluations will include both real world solutions combined with experimental data in an effort to demonstrate creditability of FINE/Turbo’s ability to be used for this research.
4.1 Reynolds Averaged Equations for Fluid Flow

The equations of motion for fluid flow developed in Chapter 2 were derived for the general case of either turbulent or laminar flow. For turbulent flow, the fluid motion is always erratic and unsteady and thus to solve the Navier-Stokes equations in the form previously provided, the velocity and pressure would need to be known at each point of interest for a given instant in time. Since it is a rather unlikely event that such information would be available, the interest is turned to understanding the average quantities about the flow in question. To do so, the Navier-Stokes equations are decomposed into the averaged and fluctuating components for mathematical convenience through a process known as Reynolds Averaging. The Reynolds Averaged equations for fluid flow are derived by decomposing the dependent variables of equations 2.8, 2.28-2.30, and 2.39 into time-mean and fluctuating components followed with a time-average then taken over each equation. For each of the dependent variables, a time-average and fluctuating component in the following manner:

\[
\begin{align*}
  u &= \bar{u} + u' , \quad v = \bar{v} + v' , \quad w = \bar{w} + w' , \quad \rho = \bar{\rho} + \rho' \\
  p &= \bar{p} + p' , \quad T = \bar{T} + T' , \quad e = \bar{e} + e'
\end{align*}
\]

(4.1)

In general, the fluctuations of the fluid properties such as the thermal conductivity, viscosity, and specific heats are considered small in comparison and thus negligible.

Several definitions are used in the Reynolds averaging process, the first being that the time average of a fluctuating variable, by definition, is zero:

\[
\bar{f}' = \frac{1}{\Delta t} \int_{t_0}^{t_0 + \Delta t} f' dt \equiv 0
\]

(4.2)

The time average of the product of an averaged variable and a fluctuating variable is also equal to zero:
\[ \bar{f}g' = 0 \tag{4.3} \]

However, the time average of the product of two fluctuating variables is not equal to zero:

\[ \bar{f}g' \neq 0 \tag{4.4} \]

Also, the time average of the addition of two time average variables is equal to the time average of the variables:

\[ \bar{f} + \bar{g} = f + g \tag{4.5} \]

These properties along with the decomposed variables in equation 4.1 will be used to develop the Reynolds Averaged Continuity, Momentum, and Energy equations.

### 4.1.1 Reynolds Averaged Continuity Equation

To derive the Reynolds Average continuity equation, the appropriate decomposed variables in equation 4.1 are first substituted into equation 2.8, and then a time average of the entire equation is taken:

\[
\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho}'}{\partial t} + \frac{\partial}{\partial x} (\bar{\rho}' u') + \frac{\partial}{\partial y} (\bar{\rho}' u') + \frac{\partial}{\partial z} (\bar{\rho}' u') + \frac{\partial}{\partial x} (\bar{\rho} v') + \frac{\partial}{\partial y} (\bar{\rho} v') + \frac{\partial}{\partial z} (\bar{\rho} v') = 0
\]

Applying the property in equation 4.3 to equation 4.6, it can be reduced to the following, which is the final form of the Reynolds Averaged continuity equation in Cartesian coordinates:

\[
\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x} (\bar{\rho} u') + \frac{\partial}{\partial y} (\bar{\rho} v') + \frac{\partial}{\partial z} (\bar{\rho} w') = 0
\]
### 4.1.2 Reynolds Averaged Navier-Stokes Equation

A similar process as used for the continuity equation is applied to the Navier-Stokes equations in equation 2.28-2.30. For illustration purposes a term-by-term approach will be taken with only the $x$-direction Navier-Stokes equation, but the process would be identical for the $y$-direction and $z$-direction. Applying the variables in equation 4.1 to the first term of the left hand side of equation 2.28 results in the following:

$$
\frac{\partial}{\partial t}[(\overline{\rho} + \rho')u + u'''] = \frac{\partial}{\partial t}\left[\overline{\rho}u + \rho'u' + \overline{\rho}u' + \overline{\rho}u'' + \overline{\rho}u'''ight]
$$

(4.8)

The second term of the left hand side of equation 2.28 is reduced to the following:

$$
\frac{\partial}{\partial x}[(\overline{\rho} + \rho')u + u'''] = \frac{\partial}{\partial x}\left[\overline{\rho}u + \rho'u' + \overline{\rho}u' + \overline{\rho}u'' + \overline{\rho}u'''ight]
$$

(4.9)

The third and fourth terms are processed exactly the same as the second term and result in the following equations:

$$
\frac{\partial}{\partial y}[(\overline{\rho} + \rho')v + v'''] = \frac{\partial}{\partial y}\left[\overline{\rho}v + \rho'v' + \overline{\rho}v' + \overline{\rho}v'' + \overline{\rho}v'''ight]
$$

(4.10)

$$
\frac{\partial}{\partial z}[(\overline{\rho} + \rho')w + w'''] = \frac{\partial}{\partial z}\left[\overline{\rho}w + \rho'w' + \overline{\rho}w' + \overline{\rho}w'' + \overline{\rho}w'''ight]
$$

(4.11)

For the right hand side of equation 2.28, the first and second terms known as the body force and pressure gradient term respectively, are derived as follows:

88
\[(\overline{\rho} + \rho') g_x - \frac{\partial}{\partial x} (\overline{\rho} + p') = \rho g_x - \frac{\partial}{\partial x} (\overline{\rho})\]  

(4.12)

The third term on the right side of equation 2.28 is reduced very similarly to equation 4.12 due to the linear functions of the dependent variables from equation 4.1:

\[\frac{\partial}{\partial x} \left[ \frac{2}{3} \mu \left( 2 \frac{\partial}{\partial x} (\overline{u} + u') - \frac{\partial}{\partial y} (\overline{v} + v') - \frac{\partial}{\partial z} (\overline{w} + w') \right) \right] = \frac{\partial}{\partial x} \left[ \frac{2}{3} \mu \left( 2 \frac{\partial}{\partial x} \overline{u} - \frac{\partial}{\partial y} \overline{v} - \frac{\partial}{\partial z} \overline{w} \right) \right] \]

(4.13)

The same can also be said for the fourth and fifth terms:

\[\frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial}{\partial y} (\overline{u} + u') - \frac{\partial}{\partial x} (\overline{v} + v') \right) \right] = \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial}{\partial y} \overline{u} - \frac{\partial}{\partial x} \overline{v} \right) \right] \]

(4.14)

\[\frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial}{\partial z} (\overline{u} + u') + \frac{\partial}{\partial x} (\overline{w} + w') \right) \right] = \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial}{\partial z} \overline{u} + \frac{\partial}{\partial x} \overline{w} \right) \right] \]

(4.15)

Combining the results of equations 4.8-4.15 into the proper terms in equation 2.28, and with a little rearrangement the x-direction Reynolds Averaged Navier-Stokes equation is found:
\[
\frac{\partial}{\partial t} \left( \bar{\rho} \bar{u} + \rho' u' \right) + \frac{\partial}{\partial x} \left( \bar{\rho} \bar{u} \bar{u} + \bar{u} \rho' u' \right) + \frac{\partial}{\partial y} \left( \bar{\rho} \bar{u} v + \bar{u} \rho' v' \right) + \frac{\partial}{\partial z} \left( \bar{\rho} \bar{u} w + \bar{u} \rho' w' \right) \\
= \bar{\rho} g_x - \frac{\partial}{\partial x} (\bar{p}) + \frac{\partial}{\partial x} \left[ \frac{2}{3} \mu \left( 2 \frac{\partial}{\partial x} (\bar{u}) - \frac{\partial}{\partial y} (\bar{v}) - \frac{\partial}{\partial z} (\bar{w}) \right) - \bar{u} \rho' u' - \bar{p} u' u' - \rho' u' u' \right] \\
+ \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial}{\partial y} (\bar{u}) + \frac{\partial}{\partial x} (\bar{v}) \right) - \bar{v} \rho' u' - \bar{p} u' v' - \rho' u' v' \right] \\
+ \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial}{\partial z} (\bar{u}) + \frac{\partial}{\partial x} (\bar{w}) \right) - \bar{w} \rho' u' - \bar{p} u' w' - \rho' u' w' \right] \\
\tag{4.16}
\]

The y-direction and z-direction Navier-Stokes equations can be derived using the exact same methods and steps as previously shown. Writing all three Reynolds Averaged Navier-Stokes equations in a more compact tensor notation:

\[
\frac{\partial}{\partial t} \left( \bar{\rho} u_j + \rho' u'_j \right) + \frac{\partial}{\partial x_j} \left( \bar{\rho} u_j u_j + \bar{u} \rho' u'_j \right) = \bar{\rho} g_j - \frac{\partial}{\partial x_j} (\bar{p}) \\
+ \frac{\partial}{\partial x_j} \left[ \tau_{ij} - \bar{u}_j \rho' u'_j - \bar{p} u'_j u'_j - \rho' u'_j u'_j \right] \\
\tag{4.17}
\]

\[
\tau_{ij} = \mu \left[ 2 \left( \frac{\partial}{\partial x_j} (\bar{u}_j) + \frac{\partial}{\partial x_i} (\bar{u}_i) \right) - \frac{2}{3} \delta_{ij} \frac{\partial}{\partial x_k} (\bar{u}_k) \right] \\
\tag{4.18}
\]

where the Kronecker delta function, \( \delta_{ij} = 1 \) if \( i= j \), \( \delta_{ij} = 0 \) if \( i \neq j \), has been introduced. Equation 4.17 can be broken down into several components with the entire left hand side being the particle acceleration of the mean flow, and the right hand side is the body force, the mean pressure gradient, laminar-like stress gradients for the mean flow (\( \tau_{ij} \) in equation 4.18), and apparent stress gradients due to the transport of momentum by turbulent fluctuations referred to as the Reynolds stresses. The Reynolds stresses for each of the x,y,z-components of the Reynolds Averaged Navier-Stokes (RANS) equations is shown in the following matrix:
\[
\begin{align*}
-\bar{u} \rho' u' - \bar{p} u' u' - \rho' u' & - \bar{v} \rho' u' - \bar{p} u' v' - \rho' u' v' \\
- \bar{u} \rho' v' - \bar{p} u' v' - \rho' u' v' & - \bar{v} \rho' v' - \bar{p} v' v' - \rho' v' v' \\
- \bar{u} \rho' w' - \bar{p} u' w' - \rho' u' w' & - \bar{v} \rho' w' - \bar{p} v' w' - \rho' v' w' \\
- \bar{w} \rho' u' - \bar{p} w' u' - \rho' w' u' \quad & - \bar{w} \rho' v' - \bar{p} w' v' - \rho' w' v' \\
- \bar{w} \rho' w' - \bar{p} w' w' - \rho' w' w' &
\end{align*}
\] (4.19)

The Reynolds stresses add six additional unknowns to the original Navier-Stokes equations when turbulent flow is assumed.

### 4.1.3 Reynolds Averaged Energy Equation

The Reynolds Averaged energy equation is also derived in a similar manner to the RANS equations shown in the previous section; however, to initiate the process Equation 2.39 is written in a different form for the purposes of this derivation:

\[
\frac{D}{Dt} (\rho e) + p(\nabla \cdot \mathbf{V}) = \frac{\partial Q}{\partial t} + \nabla \cdot (k \nabla T) + \phi
\] (4.20)

where \( \phi \) is the dissipation function defined as the following:

\[
\phi = \mu \left[ 2 \left( \frac{\partial u}{\partial x} \right)^2 + 2 \left( \frac{\partial v}{\partial y} \right)^2 + 2 \left( \frac{\partial w}{\partial z} \right)^2 + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 \right.
\]

\[
\left. + \left( \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right)^2 - 2 \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right)^2 \right] \quad (4.21)
\]

Substituting in the definition of enthalpy:
\[ h = e + \frac{P}{\rho} \]  
(4.22)

converting the enthalpy to static temperature:

\[ dh = C_\rho dT \]  
(4.23)

and finally ignoring the generation term, \( \frac{\partial Q}{\partial t} \), in equation 4.20, the following equation will be the starting point for the Reynolds Averaging process:

\[ \frac{D}{Dt}(\rho C_\rho T) = \frac{D}{Dt}(p) + \nabla \cdot (k \nabla T) + \phi \]  
(4.24)

Starting with the left hand side of equation 4.24, the appropriate mean and fluctuating variables from equation 4.1 are substituted and using the simplifying assumptions in equations 4.2 to 4.5, the left hand side becomes:

\[ \frac{\partial}{\partial t}(\bar{\rho}C_\rho \bar{T}) + \frac{\partial}{\partial x}\left(\bar{\rho}'C_\rho \bar{T}'\right) + \frac{\partial}{\partial y}\left(\bar{\rho}'uC_\rho \bar{T}' + \bar{\rho}'uC_\rho \bar{T} + \bar{\rho}'uC_\rho \bar{T}' + \bar{\rho}'uC_\rho \bar{T}' + \bar{\rho}'uC_\rho \bar{T}'\right) + \]

\[ \frac{\partial}{\partial y}\left(\bar{\rho}'vC_\rho \bar{T}' + \bar{\rho}'vC_\rho \bar{T}' + \bar{\rho}'vC_\rho \bar{T}' + \bar{\rho}'vC_\rho \bar{T}' + \bar{\rho}'vC_\rho \bar{T}'\right) + \]

\[ \frac{\partial}{\partial z}\left(\bar{\rho}'wC_\rho \bar{T}' + \bar{\rho}'wC_\rho \bar{T}' + \bar{\rho}'wC_\rho \bar{T}' + \bar{\rho}'wC_\rho \bar{T}' + \bar{\rho}'wC_\rho \bar{T}'\right) \]  
(4.25)

A similar process is applied to the first term on the right hand side:

\[ \frac{\partial}{\partial t}(\bar{p}) + \frac{\partial}{\partial x}(\bar{\rho}\bar{u}) + \frac{\partial}{\partial x}(\bar{p}'u) + \frac{\partial}{\partial y}(\bar{p}'v) + \frac{\partial}{\partial z}(\bar{p}'w) \]  
(4.26)

and the second term on the right hand side:

\[ \frac{\partial}{\partial x}\left(k \frac{\partial}{\partial x} \bar{T}\right) + \frac{\partial}{\partial y}\left(k \frac{\partial}{\partial y} \bar{T}\right) + \frac{\partial}{\partial z}\left(k \frac{\partial}{\partial z} \bar{T}\right) \]  
(4.27)
The final part of the energy equation requiring averaging is the dissipation function, $\phi$. Reducing equation 4.21 into tensor notation:

$$
\phi = \mu \left[ -\frac{2}{3} \left( \frac{\partial u_i}{\partial x_k} \right)^2 + \frac{1}{2} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)^2 \right] = \tau_{ij} \frac{\partial u_i}{\partial x_j} \quad (4.28)
$$

Then, substituting in the decomposition from equation 4.1, time averaging, and eliminating terms known to be zero results in the final Reynolds Averaged dissipation function:

$$
\bar{\phi} = \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} = \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} + \tau'_{ij} \frac{\partial \bar{u}'_i}{\partial x_j} \quad (4.29)
$$

where $\tau_{ij}$ is evaluated as in equation 4.21. Assembling all of the parts from equations 4.25, 4.26, 4.27, and 4.29, the final Reynolds Averaged energy equation, written more compact in tensor notation, becomes the following:

$$
\frac{\partial}{\partial t} \left( \bar{\rho} C_p \bar{T} + \bar{\rho}' C_p \bar{T}' \right) + \frac{\partial}{\partial x_j} \left( \bar{\rho}' u'_j C_p \bar{T} + \bar{\rho} u_j C_p \bar{T} \right) = \frac{\partial \bar{p}}{\partial t} + u_j \frac{\partial \bar{p}}{\partial x_j}
$$

$$
+ \frac{\partial}{\partial x_j} \left( k \frac{\partial \bar{T}}{\partial x_j} - \bar{p} C_p u'_j T' + \rho u'_j C_p T' - \bar{p} u_j C_p T' - u_j C_p \bar{T}' + \bar{\phi} \right) \quad (4.30)
$$

where the final form of the dissipation function is shown in equation 4.28.

### 4.1.4 Turbulence Modeling for the Reynolds Averaged Navier-Stokes Equations

The equations of motion for fluid flow previously developed in Chapter 2.1 contained six unknowns ($u, v, w, p, \rho, \text{ and } T$) within the six equations, which would normally be considered a well posed problem if the
non-linear, second-order, partial differential characteristics of the equations were ignored. However, Reynolds averaging of the mass, momentum, and energy equations have produced additional unknowns of the form \( \bar{\rho}u'_i u'_j \) for the momentum equations and of the form \( \bar{\rho}C_p u'_i T' \) for the energy equation. Many methods for closure of these terms have been suggested and while many of these do well within a confined set of flow conditions no one solution has been determined to be the ultimate model that provides simplification, accuracy, and generality.

For the purposes of the discussion, the simplified case of the two-dimensional, compressible, thin shear-layer equations in which it is assumed that the boundary layer is thin with respect to a single coordinate. The final set of equations for this type of fluid flow is the following:

\[
\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial (\bar{\rho} \bar{u})}{\partial x} + \frac{\partial (\bar{\rho} \bar{v})}{\partial y} = 0 \tag{4.31}
\]

\[
\frac{\partial (\bar{\rho} \bar{u})}{\partial t} + \bar{\rho} \frac{\partial \bar{u}}{\partial x} + \bar{\rho} \frac{\partial \bar{u}}{\partial y} = -\frac{\partial \bar{\rho}}{\partial x} + \frac{\partial \bar{p}}{\partial y} \left( \mu \frac{\partial \bar{u}}{\partial y} - \bar{\rho} \bar{u}' \bar{v}' \right) \tag{4.32}
\]

\[
\frac{\partial (C_p \bar{T})}{\partial t} + \bar{\rho} \frac{\partial \bar{u}}{\partial x} \frac{C_p \bar{T}}{\partial x} + \bar{\rho} \frac{\partial \bar{v}}{\partial y} \frac{C_p \bar{T}}{\partial y} = \frac{\partial}{\partial y} \left[ \frac{\mu}{P_r} \frac{\partial}{\partial y} (\bar{T}) \right] - \bar{\rho} C_p \bar{v}' \bar{T}' + \bar{u} \left( 1 - \frac{1}{P_r} \right) \left[ \frac{\partial \bar{u}}{\partial y} - \bar{\rho} \bar{u}' \bar{v}' \right] + \frac{\partial \bar{p}}{\partial t} \tag{4.33}
\]

\[
\bar{\rho} = \rho (\bar{p}, \bar{T}) \tag{4.34}
\]

The presence of the \( \bar{u}' \bar{v}' \) and the \( \bar{v}' \bar{T}' \) term in the above equations prevents an analytical solution to equations 4.31 through 4.34 as there are six unknowns \((\bar{u}, \bar{v}, \bar{p}, \bar{T}, \bar{u}' \bar{v}', \bar{v}' \bar{T}')\) and only four equations. Thus two additional equations would be required for closure.
An early assumption was provided by Joseph Boussinesq in 1877 that still forms the basis of many turbulence models used today for solving engineering problems, this assumption stated that a turbulent viscosity, \( \mu_T \), could serve the same purpose as the kinematic viscosity does for laminar flows such that:

\[
-\bar{\rho} \overline{u_i u_j} = 2 \mu_T \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left( \mu_T \frac{\partial \overline{u_k}}{\partial x_k} + \rho \overline{k} \right)
\]  

(4.35)

where \( \mu_T \) is the turbulent viscosity and \( \overline{k} \) is the turbulent kinetic energy. The first term on the right hand side is termed the stress tensor \( S_{ij} \). Evaluating equation 4.35 for the two-dimensional, thin shear layer problem produces the following algebraic equation:

\[
-\bar{\rho} \overline{u_i v_j} = \mu_T \frac{\partial \overline{u_i}}{\partial y}
\]  

(4.36)

where the turbulent viscosity, \( \mu_T \), can be modeled as a function of the turbulent velocity, \( v_T \), and the turbulent mixing length, \( l \):

\[
\mu_T = \rho v_T l
\]  

(4.37)

The problem now reduces to finding a suitable means for evaluating the turbulent velocity and turbulent mixing length, which is accomplished through a variety of methods. Several of these methods utilized for engineering calculations will be briefly discussed.

The simplest of methods is known as the Algebraic or Zerc-equation model in which the Reynolds stresses are replaced by algebraic equations. Prandtl proposed the first of such algebraic turbulence models in 1920’s termed the mixing length model:

\[
\mu_T = \rho l^2 \left| \frac{\partial \overline{u_i}}{\partial y} \right|
\]  

(4.38)

Evaluation of the mixing length in Prandtl’s model is handled differently depending on the type of flow being considered; for flow along a solid boundary the mixing length is evaluated using the following:
$$l_i = \kappa y \left(1 - e^{-y^+ / A^+}\right)$$  \hspace{1cm} (4.39)$$

for the inner region closest to the boundary and where the Von Karmen constant, $\kappa = 0.41$, the damping constant, $A^+ = 26$, and the parameter $y^+$ is a non-dimensional wall distance defined as the following:

$$y^+ = \frac{y \left|u_w / \rho_w\right|}{v_w}$$  \hspace{1cm} (4.40)$$

and for the outer region where $l_i$ exceeds $l_o$:

$$l_o = C_\tau \delta$$  \hspace{1cm} (4.41)$$

where $\delta$ the velocity boundary layer thickness and $C_1$ is a value close to 0.089. Substitution of equations 4.36 and 4.38 back into the original momentum equation in equation 4.32 yields the following:

$$\rho \frac{\partial \vec{u}}{\partial t} + \rho \vec{u} \cdot \frac{\partial \vec{u}}{\partial x} + \rho \vec{v} \cdot \frac{\partial \vec{u}}{\partial y} = - \frac{\partial \rho \vec{v}}{\partial x} + \frac{\partial}{\partial y} \left( \mu \frac{\partial \vec{u}}{\partial y} - \rho l^2 \frac{\partial \vec{v}}{\partial y} \frac{\partial \vec{u}}{\partial y} \right)$$  \hspace{1cm} (4.41)$$

where the unknown Reynolds stress terms have been replaced by one of the existing unknowns and the mixing length, $l$, which utilizes equation 4.38 for the inner region and 4.40 for the outer region of the flow field.

The heat flux terms are modeled based on the similarity between the transport of heat and momentum such that these terms become:

$$-\overline{\rho C_p \nu' T'} = \frac{C_p \mu_T}{Pr_T} \frac{\partial \overline{T}}{\partial y}$$  \hspace{1cm} (4.42)$$

This equation introduces a closure constant, $Pr_T$, which is the turbulent Prandtl number often near 1.0. Through this algebraic example, a system of four equations with four unknowns has been created.
The next class of turbulence modeling is known as the one-equation model. This type of model adds a transport partial differential equation (PDE) for the prediction of the turbulent kinetic energy. An example of such a model assumes the turbulent velocity to be proportional to the square root of the turbulent kinetic energy as such:

$$\mu_T = C_k \bar{p}l(\bar{k})^{\frac{1}{2}}$$

(4.43)

The transport PDE for the turbulent kinetic energy is derived the Navier-Stokes equations and takes on the following form in tensor notation:

$$\rho \frac{\partial \bar{k}}{\partial t} + \rho u_j \frac{\partial \bar{k}}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \bar{k}}{\partial x_j} - \frac{1}{2} \rho \bar{u}_i \bar{u}_j \bar{u}_j - \bar{p}' \bar{u}_j' \right) \frac{\partial \bar{u}_j}{\partial x_j} - \mu \frac{\partial \bar{u}_i'}{\partial x_k} \frac{\partial \bar{u}_j'}{\partial x_k}$$

(4.44)

The first term and second term combined on the left hand side is the rate of increase in the kinetic energy. The second and third term on the right hand side, \(-\frac{1}{2} \rho u_i' u_j' \bar{u}_j' - \bar{p}' \bar{u}_j'\), is typically modeled as a gradient diffusion process:

$$-\frac{1}{2} \rho u_i' u_j' \bar{u}_j' - \bar{p}' \bar{u}_j' = \frac{\mu_T}{\text{Pr}_k} \frac{\partial \bar{k}}{\partial x_j}$$

(4.45)

and when combined with the first term becomes the diffusion rate for the kinetic energy where \(\text{Pr}_k\) is the turbulent Prandtl number which is strictly a constant for closure. The fourth term on the right hand side, the turbulent kinetic energy generation rate, is replaced using the Boussinesq assumption provided in equation 4.35 and the fifth term which is the dissipation rate, \(\varepsilon\), of kinetic energy is given by:

$$\bar{\rho} \varepsilon = \frac{\bar{p} C_D \bar{k}^{\frac{1}{2}}}{l}$$

(4.46)

where \(C_D\) and \(\text{Pr}_k\) are model constants and \(l\) is the ordinary mixing length which needs to be specified algebraically. Substituting equation 4.45, 4.46, and 4.35 into equation 4.44, the final form of the kinetic energy equation is obtained:
\[ \rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\text{Pr}_k} \right) \frac{\partial k}{\partial x_j} \right] + \left( 2 \mu_T S_{ij} \frac{2}{3} \delta_{ij} \rho \frac{\partial k}{\partial x_j} \right) \frac{\partial \tilde{u}_i}{\partial x_j} - \frac{\rho C_{D_k} k^{3/2}}{l} \]  

Using the same algebraic closure model for the heat-flux terms, a system of five equations with five unknowns in which the turbulent mixing length must be specified. This example provided is only one of many different approaches to using a one-equation turbulence model.

The final class of turbulence modeling that will be discussed is the two-equation turbulence model. For these types of models, a second PDE is introduced into the equation set which is utilized to predict the turbulent kinetic energy dissipation rate. The turbulent viscosity is given by:

\[ \mu_T = \left( C_D \right)^{\gamma_3} \rho k^{3/2} \]  

(4.48)

and the mixing length, \( l \), for this is example is the following:

\[ l = \frac{C_{D_k} k^{3/2}}{\varepsilon} \]  

(4.49)

In the turbulent kinetic energy PDE, equation 4.47, the dissipation rate is no longer replaced by the right hand side of equation 4.46 and remains an unknown. The second PDE that is added to the system of equations is the dissipation rate, which follows a very similar format to the turbulent kinetic energy equation:

\[ \rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\text{Pr}_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon_1} \frac{\varepsilon}{k} \left( 2 \mu_T S_{ij} - \frac{2}{3} \delta_{ij} \rho \frac{\partial k}{\partial x_j} \right) \frac{\partial \tilde{u}_i}{\partial x_j} - \frac{\rho C_{\varepsilon_2} \varepsilon^{3/2}}{k} \]  

(4.50)

where several additional model constants \( C_{\varepsilon_1}, C_{\varepsilon_2}, \) and \( \text{Pr}_\varepsilon \) have been introduced. Typically, the heat-flux terms are modeled using the algebraic closure model in equation 4.42. This produces a final set of six equations with six unknowns.
The derivations provided here are very simple in context and have been provided for insight into how closure in the Reynolds Averaged Navier-Stokes equations is achieved. As can been seen in the three different examples of turbulence models previously discussed, turbulence modeling in simplistic form reduces to replacing the Reynolds stress terms with additional unknowns for which either an appropriate algebraic equation or additional partial differential equations can be derived.

4.2 FINE/Turbo Numerical Methods

FINE/Turbo is a multi-purpose, commercially available, CFD code produced by NUMECA International [38] that is specifically adapted to handle turbomachinery applications. The overall package incorporates the pre-processor, solver, and post-processor bundled into a single system that handles the entire analysis from grid-generation to solution visualization. For all analyses shown, the CFD code FINE/Turbo version 8.8-2 was utilized unless otherwise noted. As previously mentioned FINE/Turbo is a three-dimensional, viscous, compressible or incompressible, RANS, steady and time accurate, code solving the equations in Cartesian coordinates and is capable of handling both two-dimensional and three-dimensional complex geometries. The code has the ability to predict inviscid, viscous-laminar, and viscous turbulent flows.

The flow solver within the FINE/Turbo package, named Euranus, is a parallel, three-dimensional, density-based, structured, multi-block solver using the finite volume method. The solver is capable of resolving inviscid (Euler), laminar viscous, or turbulent viscous (Reynolds Averaged Navier-Stokes) flows and has the capability to incorporate gravitational forces where applicable. Spatial discretization can be either a second order accurate central space scheme or an upwind scheme. In the case of the upwind scheme, the user has the option to choose either a first order accurate or second order accurate scheme. In order to suppress the tendency of odd and even point oscillations and to limit overshoots near shock waves, the
central space scheme utilizes Jameson [44] type dissipation with 2\textsuperscript{nd} and 4\textsuperscript{th} order derivatives of the conservative variables. Viscous fluxes are determined in a purely central way and are evaluated on the cell faces through applying Gauss’ Theorem. An explicit Runge-Kutta scheme is incorporated for temporal discretization and can be specified as either a four stage or five stage scheme. The solver also incorporates multi-grid techniques, local time-stepping, and implicit residual averaging in order to speed convergence of the solution. For turbulent flow analysis, multiple turbulence models are available including:

- Baldwin-Lomax [45] algebraic model
- Spalart-Allmaras [46] one-equation model
- Standard Launder-Spalding [47] two-equation k-ε model
- Hakimi [48] Extended wall function two-equation k-ε model
- Chien [49] two-equation k-ε model
- Launder-Sharma [50] two-equation k-ε model
- Yang-Shih [51] two-equation k-ε model
- Lien-Kalitzan [52] code friendly v\textsuperscript{2}-f four-equation model

Each turbulence model in FINE/Turbo has recommended flow type for its most appropriate use. However, each model can be applied to any flow making it perhaps only less appropriate than another.

The code has the ability to perform both steady analysis and time-accurate analysis. The steady analysis, in the case of turbomachinery, can be performed as either isolated blade-rows utilizing inlet and exit profiles to form boundary conditions, a multiple blade-row frozen rotor (single position of a blade with respect to the up and/or downstream stator), or as a multiple blade-row mixing plane approach. The mixing plane approach involves pitchwise averaging of mass, momentum, and energy fluxes at the rotor-stator interface, an exchange of these averages between adjacent blade-rows depending on the flow direction, and a decoding or mixing of these averages. The mixing plane approach in FINE/Turbo incorporates five
boundary conditions for the rotor-stator interaction planes: Conservative Coupling by Pitchwise Rows, Local Conservative Coupling, Full Non-Matching Mixing plane, Full Non Matching Frozen Rotor and Non Reflecting 1D. The Local Conservative Coupling technique bases the flux decomposition on the local flow direction and utilizes the decoding process with the transferred flux variables. This rotor-stator treatment is only recommended for impeller-volute interfaces where significant flow variations are often observed. The Conservative Coupling by Pitchwise Rows provides an exact conservation of mass, momentum and energy across the rotor-stator interface by transferring fluxes between each blade-row but flux decoding is not performed. The main disadvantage of this rotor-stator treatment is that all radial gridlines must be circular arcs in the tangential direction. The Full Non-Matching Mixing Plane is similar to the Conservative Coupling by Pitchwise Rows and removes the circular arc constraints for the interfaces. The Full Non-Matching Frozen Rotor method considers the rotor-stator interface to be a perfect connection, disregards rotor movement, and requires the rotor and stator periodicities to be equal. The Non-Reflecting 1D approach is based on a characteristic analysis of the linearized Euler equations and imposes the characteristics on both sides of the interface. Such a treatment is best utilized in situations where wave reflection is observed.

For time-accurate analysis, the code implements three methods for multiple blade-rows: phase-lag, domain scaling, and a non-linear harmonic method. The domain scaling method is an unsteady computational technique for rotor/stator calculations in which the periodic boundaries are set to be equal between the rotor and stator. For instance, if a turbine with 24 nozzles and 36 blades were being simulated the domain scaling method would utilize two nozzles and three blades for the full domain. If the particular turbine of interest utilized 22 nozzles and 36 blades, a domain of 11 nozzles and 18 blades could be used or the geometry of the nozzles could be scaled to reduce the total number of passages required in each blade row to keep the total computational size to a minimum. Note, however, that scaling of the geometry to force fit periodicity between the rotor and stator has an effect on the final outcome of the analysis. The position of
the rotor relative to the stator is taken into account such that at each time step, the rotor is set at the correct position relative to the stator and the solution is found. At each time step, the rotor-stator interface is treated as a pure connection. The final unsteady solution becomes a succession of instantaneous solutions for each time step. Figure 28 below illustrates the boundary conditions as applied on both the rotor and stator for the domain scaling method at each time step.

Figure 28. Typical boundary conditions for a domain scaling method unsteady calculation

The second method available in FINE/Turbo, phase-lag method, utilizes only a single passage from each blade-row coupled with phase-lag boundary conditions in place of periodic boundaries. This approach allows for a significant reduction in the computational domain as compared to the domain scaling method while modeling the exact rotor-stator blade counts. This method of simulating unsteady flow assumes the
associated frequencies of unsteadiness are at rotor and/or stator passing and its successive harmonics. For each time step, the rotor is clocked into position and the solution is found for that instant in time. In the case where the phase-lag storage tanks are the same size as the number of time steps taken per period, the solution along each phase-lag boundary is then stored and the solution indexes to the next rotor position based upon the time step chosen. As the solution advances periodically with time, these stored values are used to set the boundary conditions along each phase-lag boundary to create a time periodic boundary condition. After each time step, the stored phase-lag boundary condition values are updated based on the resulting solution. The phase-lag method is also applied at the rotor-stator interface in order to properly set up the upstream/downstream boundary conditions at each time step. Figure 29 below illustrates the typical boundary conditions as applied for a phase-lag unsteady analysis. Additional details on the phase-lag approach to unsteady calculations can be found in Chen et al.[53].

![Diagram of phase-lag boundaries](image)

**Figure 29.** Typical boundary conditions as applied on a phase-lag unsteady calculation.
The third method available in FINE/Turbo is known as the non-linear harmonic method. This method incorporates the computational efficiency of a linearized, harmonic, unsteady approach with the non-linear effects of time marching while using a single passage only for each blade-row. The non-linear harmonic method incorporates additional deterministic stress terms into time-averaged Navier-Stokes equations, which are solved simultaneously with the harmonic perturbation equations in a highly coupled solution. Because this method is solving both the time-averaged flow and the harmonic perturbation flow, two sets of boundary conditions are required. Normal inlet and exit boundary conditions for steady flow are typically applied and are coupled with non-reflecting two-dimensional far-field type boundary conditions for the harmonic perturbation equations. The periodic boundaries are held for the time-averaged flow solution and for the harmonic perturbation equations a phase-shifted periodic condition is applied. At the rotor-stator interface, unsteady disturbances from the upstream and downstream blade-row are calculated from a tangential Fourier transform of the flow field. At the outlet of the upstream blade-row the upstream running potential disturbances from the inlet of the downstream solution are applied and on the downstream inlet the incoming wake perturbations are applied from the outlet of the upstream solution. From a time-averaged perspective, the interface incorporates a flux-based approach similar to the Full Mixing Plane interfaces described earlier with the additional deterministic stresses. The deterministic stresses are updated using the harmonic amplitudes obtained at the rotor-stator interface and are used to defined the flux-averaged flow associated with the unsteady effects incorporated. Any imbalance in the total fluxes will cause a jump in the characteristics and will be corrected to ensure total conservation across the interface. This method is best described by Chen et al. [54].

FINE/Turbo can also be utilized to perform single blade-row, unsteady analysis often referred to as wake-blade analysis where only the adjacent upstream wake or downstream potential field is considered. This type of analysis utilizes two-dimensional profiles of the upstream/downstream source as the inlet/outlet boundary condition and incorporates the phase-lag periodic boundaries in the same fashion as the multiple blade-row, phase-lag unsteady analysis previously described.
Figure 30. Typical boundary conditions as applied for the non-linear harmonic unsteady method for the (A) time-averaged solution, (B) harmonic solution
FINE/Turbo also incorporates multiple methods for treating all boundary conditions including walls, inlets, and outlets. For inlets, the boundary conditions can be specified in total conditions (total temperature, total pressure, inlet flow angles), static conditions (static temperature and velocity components), a mass flow and static temperature, total enthalpy with dryness fraction and velocity angles, distributions of flow variables for the non-linear harmonic method, or as conditions set from a rotor-stator interface. These variables can be specified in Cartesian or cylindrical coordinates as either single values, radial profiles, or as two-dimensional profiles in the absolute frame of reference or in the upstream rotating frame of reference. The outlet boundary conditions available include static pressure, average static pressure, radial equilibrium, mass flow rate, or as an imposed characteristic. For radial equilibrium, a static pressure value and radial location must be supplied. Wall boundaries can be given a rotational or translation (Cartesian only) speed and be specified as either Euler type wall with zero normal velocity or as a Navier-Stokes no-slip wall. Thermal wall boundary conditions include adiabatic walls, constant temperature or isothermal walls, constant heat flux walls of which the last two can be specified as constant or as a function of space. Other options for boundary conditions within the code include far-field boundaries, cooling/bleed flow which utilizes point source terms, fluid-particle interaction for modeling of solid particles within the flow, conjugate heat transfer, and a transition model to model transition from laminar to turbulent flow.

One additional feature that FINE/Turbo has incorporated into the overall CFD package is the ability to handle non-matching interfaces beyond just rotor-stator interfaces. Such an interface may be incorporated by modeling non-ideal flow path surfaces with such features as purge cavities, bleed ports, and hub and tip shroud cavities. The interfaces come in two forms: a non-matching interface in which one direction of gridlines on the interface surface match one-for-one and the second direction does not and a full non-matching interface in which neither of the gridline directions match across the interface. Utilizing these types of interfaces can help improve grid quality when incorporating such features.
Overall, FINE/Turbo has taken the steps to incorporate many different forms of boundary conditions, interfaces, and analyses types in order to offer a very flexible yet focused software package for turbomachinery. In the next section, the abilities of FINE/Turbo will be evaluated against several test cases to show that the code is a legitimate choice for the research performed within.

4.3 Test Cases for FINE/Turbo Evaluation

Three test cases will be utilized in to evaluate FINE/Turbo for the purposes of providing assurance that the CFD code is capable for performing the necessary calculations in order to resolve the physics of interest for this research. These test cases are by no means attempting to show complete validation of the CFD code, that exercise is assumed to have been previously performed by the authors of the CFD code. The methodology for creating credibility will include predictions performed by the CFD code compared directly with experimental data. The test cases studied here are: a two-dimensional NACA0012 airfoil at different Mach numbers and angles of attack, NASA Rotor 37, and a three blade-row, uncooled high-pressure turbine with low-pressure turbine vane. The NACA0012 airfoil and NASA Rotor 37 are two widely used test cases for such purposes whereas the third case provides both time-averaged and time-accurate data for comparison.

4.3.1 NACA0012 Airfoil

The National Advisory Committee for Aeronautics (NACA) was originally founded in 1915 and eventually dissolved in 1958 and transferred to the National Aeronautics and Space Administration or NASA as it is
known today. In the 1930’s, NACA created multiple airfoil families described using a series of digits to accurately capture an airfoil’s properties such as camber, maximum camber location, and maximum thickness. The NACA0012 airfoil is one of the four-digit series of airfoils that has zero camber and a maximum thickness of twelve percent of the chord length and can be described by the following equation:

\[
y = \frac{t}{0.2} c \left[ 0.2969 \sqrt{\frac{x}{c}} - 0.1260 \frac{x}{c} - 0.3516 \left( \frac{x}{c} \right)^2 + 0.2843 \left( \frac{x}{c} \right)^3 - 0.1015 \left( \frac{x}{c} \right)^4 \right]
\] (4.51)

Where \( c \) is the chord length, \( t \) is the maximum thickness as a fraction of chord (or 0.12 for the NACA0012 airfoil), \( x \) is the position along the chord from zero to \( c \), and \( y \) is the half thickness at a given value of \( c \). This equation can be utilized to generate the coordinates of the NACA0012 airfoil cross-section.

Thibert et al. [55] tested a two-dimensional NACA 0012 airfoil in a wind tunnel at various Mach numbers and angles of attack in order to provide an experimental database for computer program assessments. The experimental data includes pressure measurements on both the upper and lower surface and force measurements throughout the range of Mach numbers and angles of attack tested. The surface pressure measurements will be compared to the predictions from FINE/Turbo for nine different cases ranging from Mach 0.3 to Mach 0.83 and for angles of attack between zero and eight degrees. Table 4 shows each of the nine cases used for comparison between the computational and experimental results. Note that the corrected Mach number and angle of attack are the final values used to determine the boundary conditions for the analytical analyses. These corrections are necessary due to the setup and methodology applied to the experiment.
<table>
<thead>
<tr>
<th>Case Number</th>
<th>Mach Number</th>
<th>Angle of Attack</th>
<th>Corrected Mach Number</th>
<th>Corrected Angle of Attack</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.3</td>
<td>-0.02</td>
<td>0.3</td>
<td>-0.05</td>
</tr>
<tr>
<td>2</td>
<td>0.299</td>
<td>4.04</td>
<td>0.3</td>
<td>3.59</td>
</tr>
<tr>
<td>3</td>
<td>0.502</td>
<td>-0.02</td>
<td>0.503</td>
<td>-0.06</td>
</tr>
<tr>
<td>4</td>
<td>0.504</td>
<td>4.06</td>
<td>0.504</td>
<td>3.51</td>
</tr>
<tr>
<td>5</td>
<td>0.503</td>
<td>8.02</td>
<td>0.503</td>
<td>6.95</td>
</tr>
<tr>
<td>6</td>
<td>0.756</td>
<td>-0.01</td>
<td>0.755</td>
<td>-0.05</td>
</tr>
<tr>
<td>7</td>
<td>0.753</td>
<td>1.95</td>
<td>0.752</td>
<td>1.49</td>
</tr>
<tr>
<td>8</td>
<td>0.756</td>
<td>3.99</td>
<td>0.754</td>
<td>3.02</td>
</tr>
<tr>
<td>9</td>
<td>0.829</td>
<td>0.05</td>
<td>0.826</td>
<td>-0.22</td>
</tr>
</tbody>
</table>

Table 4. NACA0012 Cases and Conditions

The predictions were performed using a two-dimensional grid in the X-Y plane with solid geometry created from equation 3.51. The actual geometry as built is defined by Thibert et al. [55]; however, the purpose of this test case was to show how the code predicts the experimental data and thus the theoretical geometry was used. Deviations between the actual and theoretical are noted by Thibert et al. [55] and are shown in Figure 31 as a percentage of the theoretical value along the chord. A large dip can be noticed in the upper airfoil surface just past 60% of chord. This dip does not appear in the plot shown by Thibert et al. [55], but it can be readily seen in the coordinates provided.
Figure 31. Percent difference between the theoretical and actual experimental airfoil model

The grid was generated using Interactive Grid Generator (IGG) within the FINE/Turbo software package, which allows the user to create and place blocks within the solid geometry and to set the grid boundary conditions. The grid was generated using four H blocks throughout the computational domain, two blocks of which were manipulated to essentially form a C block that is split at the leading edge of the airfoil. This strategy was used in order to maximize the number of blocks within the grid and hence maximize the number of processors that could be used in the parallel computation. Figure 32 shows the overall blocking strategy for the grid.
Figure 32. Block structure for the NACA0012 airfoil: (a) far field, (b) near airfoil view
The outer boundaries of the grid where set four hundred chord lengths away from the airfoil in order to minimize the effect of boundary conditions on the calculations. The grid density was established to provide three levels of multigrid for the Euranus solver with sufficient airfoil resolution to capture the leading and trailing edge curvature. The final grid density chosen provided 129 points along the upper surface and 129 points along the lower airfoil surface with an offset of 5e-6 meters in order to provide a $y^+$ value between one and ten as recommended by NUMECA [38] for the Spalart-Allmaras turbulence model. Figure 33 shows the far and near field grid density as well as the low and high Mach number resulting $y^+$ values.

Figure 33. Grid density for the NACA0012 Airfoil: (a) Far View, (b) Near View, (c) Mach 0.3 $y^+$ values, (d) Mach 0.83 Number $y^+$ values
Figure 33 Continued

(b)

Continued
The fluid was assumed to be air using real gas properties (compressible flow). Boundary conditions for the grid were set to specify static conditions at the inlet (which require the static temperature and three velocity components) static pressure at the outlet, and far field boundaries were set on the top and bottom boundary surfaces (which require static pressure, static temperature, and the three velocity components). Values for each of the boundary conditions were calculated using the Mach number for each case, an inlet total temperature to be 273 K, and an exit static pressure of 101325 Pascal (Pa). The inlet static temperature is quickly found using the isentropic relationship assuming a specific heats ratio ($\gamma$) of 1.4:

$$T_{\text{static}} = T_{\text{Total}} \left( \frac{1}{1 + \frac{\gamma - 1}{2} M^2} \right)$$

(4.52)
Once the static temperature is calculated for each case, the sound speed \(a\) and the velocity \(V\) can be found from the following equations using an ideal gas constant for air \(R\) of 286.9 m\(^2\)/s\(^2\)-K:

\[
a = \sqrt{\lambda \cdot R \cdot T_{\text{static}}} \tag{4.53}
\]

\[
V = M \cdot a \tag{4.54}
\]

Velocity components in the \(x\)-direction and \(y\)-direction are quickly decomposed through trigonometry from the velocity determined using equation 4.54. The inlet and far field boundary conditions also required the specification of the turbulent viscosity \(\mu_t\) for use with the single equation Spalart-Allmaras [46] turbulence model. This was found using the static temperature in Sutherland’s Formula in equation 2.39 and using a factor of 2.5 as recommended by NUMECA [38] for internal flows.

Once the boundary conditions were set, the solver was set to run for 100 iterations on the coarse and medium grids, and 2000 iterations on the fine grid to achieve at least four orders of magnitude drop in the residuals. A typical convergence history of the root mean square (RMS) and the maximum residual as well as the inlet and exit mass flow rates for case number five in Table 4 are shown in Figure 34 below. This case was only run for approximately 1000 iterations on the fine grid before the RMS residuals began to flatten out just above -5.0 and the inlet and exit mass flow rates had been stable for 600 iterations. The convergence histories for the other eight cases utilized for this test case in Table 4 are provided in Appendix A.
Figure 34. Case number five: (a) RMS Residual history, (b) inlet and exit mass flow rate
For each computational case, the static pressure on the surface of the airfoil was extracted from the solution and converted to a pressure coefficient using the compressible flow form of the equation:

$$C_p = \frac{2}{\lambda \cdot M^2} \left( \frac{p}{p_\infty} - 1 \right)$$

(4.55)

and compared to each data set from Thibert et al. [55]. Negative pressure coefficients correspond to the suction surface and positive pressure coefficients correspond to the pressure surface in the case where the flow angle is not equal to zero. The results of the pressure coefficient comparison between the experiments and the computations can be seen in Figure 35.

![Graph showing pressure coefficient vs. X/C](image)

**Figure 35.** Pressure Coefficient on the airfoil surface for: (a) 0.3 Mach at 0°, (b) 0.3 Mach at 4°, (c) 0.5 Mach at 0°, (d) 0.5 Mach at 4°, (e) 0.5 Mach at 8°, (f) 0.75 Mach at 0°, (g) 0.75 Mach at 2°, (h) 0.75 Mach at 4°, (i) 0.83 Mach at 0°
Figure 35 Continued

(b)

(c)
Figure 35 Continued

(d)

(e) Continued
Figure 35 Continued

(f)

(g) Continued
Figure 35 Continued

(h)

(i)
From each of the comparisons in Figure 35, FINE/Turbo predicts the surface pressure coefficients very well across the spectrum of Mach numbers and attack angles analyzed. The comparisons at 0.3 Mach and 0.5 Mach have excellent agreement between the experimental and analytical results with very little deviation noted throughout both the upper and lower surfaces. The comparisons at 0.75 Mach and 0.83 Mach show good agreement as well however some differences are seen. At 0.75 Mach number, the two degree and four degree angle of attack comparisons in Figure 35(f) and Figure 35(g), respectively show good agreement on the lower surface up to the shock location (X/C of approximately 0.3 and 0.35 respectively). Just past the shock, the analytical results predict a slightly less rapid decline in the pressure coefficient on the backside of the shock. This can also be seen for 0.83 Mach number case in Figure 35(i) at zero degrees angle of attack at an approximate X/C of 0.5. However, if one considers that the predictions assume the flow to be two-dimensional, do not account for deviations in experimental model geometry, and do not take into account the experimental accuracy of the measurements then differences between the experimental and analytical results would be anticipated to some degree. Assuming the flow to be two-dimensional most likely holds well at low angles of attack and low Mach numbers where the comparisons are shown to be very good. However, at higher Mach numbers and angles of attack, such as the 0.75 Mach number case with four degrees of attack angle, the flow might tend to depart from the two-dimensional assumption and thus explain some of the discrepancies. The geometry deviations, as shown in Figure 31, are 0.5% to 0.9% up to the maximum thickness location at x/c of 0.3 and significantly greater aft of the maximum thickness location. Another aspect that should also be taken into consideration is the experimental accuracy of the pressure coefficient measurements, which were noted by Thibert et al. [55] to be equal to or less than 1.8% at 0.7 Mach number (a case not studied). These three effects, two-dimensional flow assumption, the non-modeled geometry deviations, and the experimental accuracy could easily make up the difference in each one of the comparisons, but the experimental accuracy is by far the least of the potential causes. Other sources of variation would be typical sources found within CFD codes such as analytical model discretization (both spatial and temporal), turbulence modeling, and the general RANS assumptions.
4.3.2 NASA Rotor 37 and NASA Stage 35

In the late 1970’s, Reid and Moore [56] designed and tested four compressor stages known as NASA stages 35 to 38 in an effort to investigate the effects of blade camber and aspect ratio on compressor performance. The tests were performed at NASA Glenn Research Center (formerly Lewis Research Center). Stages 35 and 37 utilized low aspect ratio blades and vanes and provided the best overall performance of the group. In addition, Reid and Moore [57] also performed aerodynamic surveys downstream of stage 35 in which radial distributions of total pressure, total temperature, and flow angles were measured. In 1994, Suder et al. [58] and [59] retested Stage 37 as a rotor only configuration (referred to as Rotor 37) using both aerodynamic surveys and laser anemometry in order to study the tip flows in more detail. Radial distributions of static and total pressure, total temperature, and flow angles were measured both upstream and downstream of the rotor. Their data was eventually used by ASME and IGTI (unpublished) to conduct a blind test case and then AGARD conducted a more detailed non-blind test case published by Dunham [60] with the goal of both being to assess the ability of CFD codes to predict the experimental data.

Rotor 37 is constructed of 36 blades made up of multiple circular-arc (MCA) airfoils that achieve a design point pressure ratio of 2.106 at a mass flow rate of 20.19 kg/sec. Stage 35 is also made up of 36 blades and 46 vanes made up of multiple circular-arc (MCA) airfoils which achieve a design point pressure ratio of 1.865 for the rotor and 1.82 for the stage at a mass flow rate of 20.19 kg/sec. Both blades are similar in design utilizing the same forward transonic blade geometry with Rotor 35 having less camber in the aft portion of the blade and thus producing less overall pressure ratio. The hub and shroud flow paths for Rotor 37 and Stage 35 are identical and both rotors have a tip clearance of ~0.2% of blade height while vane 35 is cantilevered over a rotating hub with a clearance of ~0.5% of vane height. The 100% speed point of interest is 17188 RPM.
Both Rotor 37 and Stage 35 are of interest as a test case for FINE/Turbo. Rotor 37 has a higher overall pressure ratio and in the blind ASME/IGTI and non-blind AGARD test cases proved difficult for CFD codes to predict. Stage 35 with the lower total pressure ratio should be easier for FINE/Turbo to predict and also allows for examining the steady mixing plane that has been incorporated into the code due to the stage configuration. Multiple comparisons are made for both configurations for the 100% speed line and include mass flow rate vs. pressure ratio/temperature ratio/adiabatic efficiency and circumferentially averaged radial distributions of pressure ratio/temperature ratio/adiabatic efficiency/flow angles in order to assess FINE/Turbo’s prediction capability using steady analysis techniques. The adiabatic efficiency is calculated using the following equation:

$$\eta_{Ad} = \left( \frac{P_{T,Out}}{P_{T,In}} \right)^{\frac{\gamma-1}{\gamma}} - 1$$

(4.56)

The grid constructed for Rotor 37 is a single blade row, single passage, multi-block grid incorporating the nominal blocking structure from FINE/Turbo’s Autogrid5 program. This nominal blocking structure will be described in more detail in Chapter 5. In addition to the nominal blocking structure, two z-constant lines that create additional blocks within the blade passage were added to designate the boundary between the rotating and non-rotating surfaces of the hub flow path. A Z-R plane of the flow path is shown in Figure 36 below which also shows the locations of the two z-constant lines as well as the inlet and exit boundaries for the grid.
The final grid dimensions are 130 points axially along the blade surface, 65 points circumferentially, and 105 points radially with 41 points through the tip clearance region for a total of 1,960,000 points for the entire grid. The first grid line offset from each surface is set at 2.54e-6 meters in order to provide a $y^+$ value between one and ten as recommended by NUMECA [38] for the Spalart-Allmaras turbulence model. Figure 37(a) shows the final grid density and Figure 37(b) shows the resulting $Y^+$ values at design point conditions for Rotor 37.
Figure 37. Rotor 37 (a) grid density and (b) $Y^+$ values at design point conditions
The grid constructed for Stage 35 is a two-blade row, single passage, multi-block grid incorporating the nominal blocking structure from FINE/Turbo’s Autogrid5 program. This nominal blocking structure will be described in more detail in Chapter 5. In addition to the nominal blocking structure, two $z$-constant lines that create additional blocks within the blade passage were added to designate the boundary between the rotating and non-rotating surfaces of the hub flow path. The rotor-stator interface is located at exactly one-half of the axial distance between the rotor and stator. The Z-R plane of the flow path is shown in Figure 38 below which also shows the locations of the two $z$-constant lines as well as the inlet, exit, and rotor-stator boundaries for the grid.

Figure 38. Z-R plane of the stage 35 flow path
The final grid dimensions is 122 points axially along the blade surface, 49 points circumferentially, and 105 points radially with 25 points through the tip clearance region for a total of 1,180,000 points for the rotor and 122 points axially along the blade surface, 49 points circumferentially, and 105 points radially with 17 points through the hub clearance region for a total of 1,150,000 points for the stator. The total grid size for both blade rows is about 2,330,000 points. The first grid line offset from each surface is set at 2.54e-6 meters in order to provide a $y^+$ value between one and ten as recommended by NUMECA [38] for the Spalart-Allmaras turbulence model. Figure 39(a) shows the final grid density for both the rotor and stator and Figure 39(b) shows the results $Y^+$ values on the rotor and stator at design point conditions.

Figure 39. Stage 35 (a) grid density and (b) $Y^+$ values at design point conditions
Inlet boundary conditions for both Rotor 37 and Stage 35 are specified as total conditions with a constant value for Total Temperature (288.15 K), Total Pressure (101325.4 Pa), tangential angle (6°), radial angle (0°) and the turbulent kinematic viscosity. The turbulent kinematic viscosity is found using the total temperature at the inlet, with Sutherland’s formula (equation 2.39), and a factor of 2.5 as recommended by NUMECA Int. [43]. The exit boundary condition is specified as radial equilibrium about the hub surface where the static pressure is specified. In order to compute the speed lines for both cases, the exit static pressure is initially set to a low enough value to cause the rotors to choke (1.1*Inlet Total Pressure for Rotor 37 and 1.2*Inlet Total Pressure for Stage 35) thus pushing the solutions to their maximum flow rates. The exit static pressure is then raised slowly in a series of successive solutions until the exit mass flow rate begins to diverge and hence the solution stalls providing the minimum mass flow rate. All solid walls are assumed to be adiabatic and the hub flow path and rotor airfoils are specified to rotate at the design speed.
of 17188.7 RPM. Both rotor grids are solved in the rotating frame of reference whereas the stator grid is solved in the stationary frame of reference for Stage 35. Because Stage 35 incorporates the two blade rows, a steady mixing plane approach has been applied to the rotor-stator interface.

A turbomachinery initial solution is utilized in order to start the calculations for each of the successive analyses on both configurations. The solutions are set to run using three levels of multigrid of which 100 iterations are performed on the coarse and medium grid levels and the remainder of the 2200 iterations is performed on the fine grid level. The RMS residuals and the inlet and exit mass flow rates were monitored in order to determine convergence on each solution and typically required 800 to 1200 iterations to reach convergence for both configurations. A typical convergence history for Rotor 37 is provided in Figure 40. The convergence history for the other cases for Rotor 37 and Stage 35 can be seen in Appendix B.

![Graph](image)

**Figure 40.** Rotor 37 convergence history for the near peak efficiency solution for (a) the RMS and maximum residuals and (b) the inlet and exit mass flow rates

Continued
The 100% speed line comparisons of the computations and the experimental data for pressure ratio, temperature ratio, and adiabatic efficiency for Rotor 37 are shown in Figure 41. The computed choke and stall flows for Rotor 37 are 20.75 kg/sec and 19.12 kg/sec compared to the measured choke and stall flow rates of 20.88 kg/sec and 19.38 kg/sec in which the computed values are 0.6% less for the choke flow rate and 1.34% less for the stall flow rate. Overall, the computed total pressure ratio in Figure 41(a) matches the measurements very well across the range of mass flow rates. The computed total temperature ratio given in Figure 41(b) is about 0.5% higher than the experimental data across the mass flow rate range. The adiabatic efficiency in Figure 41(c) is about 1% lower than the experimental data, which is due to the miss in total temperature ratio which can be easily seen in equation (4.56). Both the high total temperature predictions and the low adiabatic efficiency predictions are fairly typical result of CFD codes as pointed out by Chima [61] who also achieved similar results.
Figure 41. Rotor 37 100% speed line comparisons for mass flow rate versus (a) total pressure ratio, (b) total temperature ratio, and (c) adiabatic efficiency
A contour plot of relative total pressure and relative Mach number at 50% span for the peak efficiency solution is provided in Figure 42(a) and (b), respectively. The relative total pressure contour shows a suction side loss starting at the shock location seen in the relative Mach number contour plot and continues aft along the aft section of the airfoil. The loss or separation is due to shock induced boundary layer separation on the airfoil surface.
Figure 42. Rotor 37 Peak efficiency solution contours at 50% span of (a) relative total pressure and (b) relative Mach number.
Radial distributions of total pressure ratio, total temperature ratio, adiabatic efficiency, and tangential flow angle were extracted from the peak efficiency solution and the near stall point solution and compared with the experimental data. The peak efficiency solution radial distributions are provided in Figure 43. The total pressure ratio in Figure 43(a) is captured quite well when compared to the data with the exception being below 30% span where the data shows a drop in total pressure ratio that is not seen in the computation. However, it is important to point out that this miss in total pressure ratio is less than 2.5% of the experimental data and still in very good agreement. The total temperature ratio comparison between the computations and the experimental data is excellent in Figure 43(b) with only the upper 90% span and lower 5% span showing less temperature rise in the experimental data versus the computations. The adiabatic efficiency, in Figure 43(c), also shows very good agreement which is anticipated given the total
pressure and total temperature ratios compared favorably with the experimental data. Figure 43(d), the computed tangential angles at the exit of the rotor are about one to two degrees higher than the experimental data suggests through the mid-span region, but again are in very good agreement overall.

Figure 43. Rotor 37 peak efficiency exit radial profile comparisons for (a) total pressure ratio, (b) total temperature ratio, (c) adiabatic efficiency, and (d) tangential flow angle
Figure 43 Continued

(b)

(c)
The near stall radial distribution comparisons are provided in Figure 44. The near stall comparisons for all values show the exact same trends and have very good agreement with exception to the lower 30% span on the total pressure ratio, the upper 90% span on the total temperature ratio and the one to two degree higher computer tangential angles near mid-span.
Figure 44. Rotor 37 near stall exit radial profile comparisons for (a) total pressure ratio, (b) total temperature ratio, (c) adiabatic efficiency, and (d) tangential flow angle.
Figure 44 Continued

(c)

(d)
The 100% speed line comparisons of the computations and the experimental data for pressure ratio, temperature ratio, and adiabatic efficiency for Stage 35 are shown in Figure 45. The computed choke and stall flows for Stage 35 are 20.88 kg/sec and 19.79 kg/sec compared to the measured choke and stall flow rates of 20.95 kg/sec and 19.54 kg/sec in which the computed values are 0.3% less for the choke flow rate and 1.3% more for the stall flow rate. Overall, both the computed total pressure ratio in Figure 45(a) and the computed total temperature ratio in Figure 45(b) match the experimental data almost perfectly at the higher mass flow rates. At the lower mass flow rates, the total temperature ratio is lower than the experimental data by about 0.2%. The adiabatic efficiency in Figure 45(c) matches very well at the peak efficiency point; however as the mass flow rates decrease the computations diverge from the experimental data by as much as 1%. Overall, the trend is captured very well along with the peak efficiency point.

![Graph showing total pressure ratio vs. mass flow rate](image)

(a) Continued

**Figure 45.** Stage 35 100% speed line comparisons for mass flow rate versus (a) total pressure ratio, (b) total temperature ratio, and (c) adiabatic efficiency
Figure 45 Continued

(b)

(c)
A contour plot of relative total pressure and relative Mach number at 50% span for the peak efficiency solution is provided in Figure 46(a) and (b), respectively. The relative total pressure contour shows a similar suction side loss to that seen in the Rotor 37 solution starting at the shock location seen in the relative Mach number contour plot and continues aft along the aft section of the airfoil. This loss or separation on the airfoil surface is again due to shock induced boundary layer separation. Considering the transonic portions of both Rotor 37 and Rotor 35 are identical, this would be expected.

Figure 46. Stage 35 Peak efficiency solution contours at 50% span of (a) relative total pressure and (b) relative Mach number
Figure 46 Continued

For the Stage 35 peak efficiency solution, radial distributions of total pressure ratio, total temperature ratio, and adiabatic efficiency were extracted downstream of the rotor and downstream of the stator in order to compare with the experimental data. In addition, the tangential angle was extracted downstream of the stator only. Comparing these two locations to the experimental data gives insight into how well the mixing plane steady solution works. The first set of comparisons, shown in Figure 47, is for the radial distributions downstream of the rotor. The total pressure ratio in Figure 47(a) compares very well to the experimental data with small differences noted below 15% of span. These differences are on the order of 1.5% or less in this region. The total temperature ratio in Figure 47(b) shows differences between the computed values and experimental data of about 1% or less in the upper 30% of span and captures the overall trend of the data well. Because of the differences in total pressure ratio below 15% span and total temperature ratio above
70% span, the adiabatic efficiency in Figure 47(c) also under predicts the experimental data in these regions by upwards of 2 to 3%. However, the overall trend in the data is captured well.

Figure 47. Stage 35 point 3978 (peak efficiency) rotor exit radial profile comparisons for (a) total pressure ratio, (b) total temperature ratio, and (c) adiabatic efficiency
Figure 47 Continued

(b)

(c)
The next set of comparisons is for the radial distributions downstream of the stator. These comparisons are shown in Figure 48. In Figure 48(a), the computed total temperature ratio matches the data very well above 20% span, but below 20% span the experimental data shows a slight loss that is not captured by FINE/Turbo. This miss by FINE/Turbo, is less than 1%. The computed total temperature ratio, in Figure 48(b), compares very well with the experimental data with all discrepancies being less than 0.7%. The overall trend is captured very well. The computed adiabatic efficiency in Figure 48(c) is less than 1% different from the experimental data and again compares very well. Despite the very good agreement for total temperature and total pressure ratio, the tangential angle radial distribution downstream of the stator is about 3° to 4° less than the experimental data such that the computations predict more turning in the stator than measured.

![Graph showing total pressure ratio vs. % span from hub](image)

(a) Continued

**Figure 48.** Stage 35 point 3978 (peak efficiency) stator exit radial profile comparisons for (a) total pressure ratio, (b) total temperature ratio, (c) adiabatic efficiency, and (d) tangential flow angle
Figure 48 Continued

(b)

(c) Continued

148
The Rotor 37 computations showed good agreement overall with the experimental data for the computed 100% speed lines and the computed choke mass flow rates, but the computed stall mass flow rate was 1.3% lower than measured for the rotor. Overall, the radial distributions at the peak efficiency and near stall point are in very good agreement. There are some slight differences in the upper 10% span for the total temperature ratio and in the lower 30% span for the total pressure ratio; these results are very similar to those shown by Chima [61]. The 100% speed lines computed choke flow rates for Stage 35 compare very well at the higher mass flow rates. As the mass flow rate tends toward the stall point, the computed total temperature and thus adiabatic efficiencies are under predicted for the stage. Both sets of radial distributions downstream of the rotor and downstream of the stator are in excellent agreement with the experimental data, but downstream of the stator the total pressure ratio is over predicted in the lower 30% span and the tangential angle is about 4° less than the data (more turning). Rotor 37 provides a great test case for the steady solution capability of FINE/Turbo and as can be seen the code does a very good job of capturing the high-pressure ratios and temperature ratios that the rotor was designed to achieve. Stage 35
provides a great test case for the multi-blade row, steady, mixing plane capability of FINE/Turbo and again the code does a good job at predicting the overall performance and flow field of the geometry.

4.3.3 Uncooled One and One-half Stage High-Pressure Turbine Rig

The uncooled one and one-half stage turbine rig Build 1 data set obtained at the OSU GTL was originally used by Green [62] for comparison between MSU-TURBO and the measurements with particular focus on the tip and shroud region of the high-pressure rotating turbine blade. The turbine flow path hardware within the rig is identical to the hardware discussed in Chapter 3, but for this Build 1 turbine configuration case the rig does not incorporate cooling flows within the purge cavity or on the high-pressure vane airfoil, shroud, or hub surface. The rig incorporated multiple Kulite pressure transducers throughout the entire turbine stage in order to capture unsteady pressure on the surfaces of the high-pressure vane, high-pressure blade, and the low-pressure vane. This data set makes for an excellent test case to compare FINE/Turbo’s predictions to the experimental data (and to the results obtained using the GEA version of MSU-TURBO) for the purposes of understanding the unsteady capabilities of the CFD code.

As discussed in Chapter 3, the experimental configuration is full scale, rotating, and running at the proper corrected speeds and flow rates that are representative of cruise conditions for a single stage, transonic, high-pressure ratio commercial turbine. The high-pressure vane and blade are uncooled and the blade is rotated closed in order to maintain the design flow function throughout the turbine. The inlet and exit temperature and pressure are measured via inlet and exit rakes similar to those discussed in Chapter 3 and Kulite pressure transducers are positioned throughout the turbine flow path in order to measure unsteady pressure fluctuations during the experiment. The Kulites are installed on the following locations:

1. The high-pressure vane airfoil surface at 15%, 50%, and 90% span at multiple wetted distances
2. The high-pressure vane hub surface at 0% wetted distance and 50% of pitch (PVIN60) and 103% wetted distance at 0% (PVIN51), 25% (PVIN53), 37.5% (PVIN54), 50% (PVIN55), 62.5% (PVIN56), 75% (PVIN57), and 87.5% (PVIN58) of the vane pitch

3. The high-pressure vane shroud surface at 0% wetted distance and 50% pitch (PVOT43), 52% wetted distance and 50% of pitch (PVOT50), and at 103% wetted distance and 50% pitch (PVOT43), 62.5% pitch (PVOT46), and 75% pitch (PVCT45)

4. The high-pressure rotor airfoil surface at 15%, 50%, and 90% span at multiple wetted distance locations

5. The high-pressure rotor platform surface 0% wetted distance and 77% pitch (PRP50), and at 100% wetted distance at 26% pitch (PRP51), 48% pitch (PRP52), 77% pitch (PRP53) and 92% pitch (PRP54)

6. The high-pressure rotor tip recess for the recessed tip configuration only at three wetted distance locations of 16%, 36%, and 54%

7. The stationary shroud above the rotating blade at -7% wetted distance (PS1, PS2, PS3, PS4) forward of the leading edge, 30% wetted distance (PS5, PS6), 60% wetted distance (PS7), and 94% wetted distance (PS8)

8. The low-pressure vane airfoil surface at 10%, 25%, 50%, 75%, and 90% span at multiple wetted distance locations

9. The low-pressure vane hub surface at wetted distance locations of -63%, -43%, -24%, -12% and 10% with negative values being forward of the leading edge

10. The low pressure vane shroud surface at wetted distance locations of -44%, -37, -26%, and -19%, all forward of the airfoil leading edge

For the purposes of this comparison, the data acquired on the low-pressure vane and the high-pressure rotor tip will not be utilized.
The goal of this test case was produce a full comparison of the harmonic and phase-lag unsteady solution techniques back-to-back with the experimental data. The computational domains were selected in order to utilize the data from the experimental rig with little or no extrapolation of boundary conditions. In order to complete the analysis as desired, multiple computational domains and grids were used: a three blade row domain and a two blade row domain that will be referred to as the stage domain. The stage domain include the high-pressure vane and rotor only in order to keep the computational problem as small as possible in order to produce faster solution turn-around time. The three blade-row domain is used to run the steady analysis only and the stage domains are used only for steady, phase-lag unsteady, and the harmonic unsteady analysis for the given operating conditions. The stage domain includes the high-pressure vane and rotor only in order to keep the computational problem as small as possible in order to produce faster solution turn-around time. Exit boundary conditions for the stage domain are extracted from the steady three-blade row solutions, as high-pressure rotor exit conditions were not measured in the rig.

The grids were generated using Autogrid5 in FINE/Turbo using the default blocking structure that will be discussed in further detail in Chapter 5. The hub and shroud flow path surfaces are assumed to be ideal (smooth) and no purge flow cavities are modeled both forward and aft of the high-pressure blade. The tip gap above the high-pressure blade is incorporated into the computational domain and is \( \sim 2\% \) of blade height and the blade tip is modeled as the flat tip configuration only. A cross-section of the flow path geometry used to generate the computational domains is shown in Figure 49 below.
Figure 49. Z-R cross-section of the turbine rig flow path used for the grid geometry (Not To Scale)

The high-pressure vane is constructed with 169 points on the suction surface and 105 points on the pressure surface axially, 49 points blade-to-blade at the inlet, 81 points blade-to-blade at the exit, and 65 points radially along the vane surface, for a total of 660,000 points in a single passage. The high-pressure rotor has 185 points on the suction surface and 105 point on the pressure surface axially, 57 points blade-to-blade at the inlet and exit, and 73 points radially with 20 points through the tip gap, for a total of 900,000 points in a single passage of the domain. The low-pressure vane has 153 points on the suction surface and 105 points on the pressure surface in the axial direction, 49 points blade-to-blade at the inlet and 65 points blade-to-blade at the exit, and 65 points radially along the vane surface for a total of 624,000 points in a single passage. A three-blade row, single passage domain combines these three grids for a total of 218,4000 points. First cell offset from all surfaces was set to achieve a y+ value between one and ten as directed by NUMECA International [38] for use with the Spalart-Allmaras [46] turbulence model. The final surface offset value used for all wall surfaces (hub, shroud, blades, etc) for all blade rows including
the cavity was determined analytically through trial and error with the best overall value found to be 2.54e-6 meters. The grid resolution and resulting $y^+$ values can be seen in Figure 50.

![Diagram](image)

**Figure 50.** Uncooled turbine rig grid density and $Y^+$ values for the three blade row steady calculation (a) high-pressure vane, (b) high-pressure rotor, and (c) low-pressure vane.
The steady and unsteady solutions each required a similar set of boundary conditions for the inlet plane, exit plane, and walls of the domains. For the steady and unsteady solutions, the inlet plane was set to constant values of total pressure, total temperature, tangential angle, and turbulent dynamic viscosity.
(calculated using Sutherland’s formula in equation 3.5 using the inlet total temperature) while the radial angle was set based on the results from the inlet region analysis shown in Appendix C. These results also show very small variations in the radial profiles of exit total pressure and total temperature and thus help to support the use of constant values for this study. The exit boundary condition for the steady three blade row analysis was specified to use the radial equilibrium equation with the static pressure set at the shroud region (100% span) based on the experimental values as measured. For the unsteady analysis, the exit boundary condition was set to an average static pressure extracted from the three blade row solution at the rotor-stator interface between the high-pressure rotor and the low-pressure vane. All of the wall boundary conditions were specified as adiabatic with the no-slip condition applied. Only the high-pressure rotor surface (including the tip surface) and the high-pressure rotor platform were designated as rotating surfaces as was the entire high-pressure rotor domain. All other surfaces were non-rotating with the no-slip condition. The rotor-stator interfaces for the steady analysis were set to the full non-matching interface boundary condition. A summary and comparison of the flow conditions for the rig and the analytical model are provided in Table 5 below. The total temperature for the analytical model is 5.25% higher than measured and the total pressure ratio is 1.64% lower than the measured data. This is most likely due to the fairly limited data points available from the rig being compared to the full span of the analytical model. All other parameters of interest are within 0.2% of the measured data.

<table>
<thead>
<tr>
<th></th>
<th>TT Ratio</th>
<th>PT-to-PS Ratio</th>
<th>PT-Ratio</th>
<th>Corrected Flow (kg/sec)</th>
<th>Corrected Speed (RPM)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Experimental Values</strong></td>
<td>1.368</td>
<td>5.55</td>
<td>4.66</td>
<td>4.59</td>
<td>6114.40</td>
</tr>
<tr>
<td><strong>FINE/Turbo Steady 3-Row</strong></td>
<td>1.440</td>
<td>5.558</td>
<td>4.583</td>
<td>4.589</td>
<td>6112.89</td>
</tr>
<tr>
<td><strong>% Difference</strong></td>
<td>5.25%</td>
<td>0.15%</td>
<td>-1.64%</td>
<td>-0.03%</td>
<td>0.02%</td>
</tr>
</tbody>
</table>

| Table 5. Rig operating conditions |
For all solutions, the second order accurate, central space, spatial discretization and the four stage, explicit, Runge-Kutta scheme is used for temporal discretization. A Courant-Fredricks-Levy (CFL) number of three is specified in order to remain within the stability region of the explicit scheme. The three blade row and two blade row steady solutions were initialized using the ‘turbomachinery’ solution and were set to run 4000 iterations or until an RMS residual of -6.0 was achieved. Each solution was time-stepped using a global CFL, residual averaging, and three levels of multigrid in which 100 iterations were performed on the coarse and medium grids before performing the final 4000 iterations on the fine grid. Both solutions reached the RMS residual limit in less than 1500 iterations with flat inlet and exit mass flow rates. The convergence history for the three blade row solution can be seen in Figure 51.

The unsteady, harmonic solutions were initialized from the converged steady solution and the convergence history was reset at the beginning of the solution and thus the RMS residual is reset to zero. For each harmonic solution, three harmonics per periodic perturbation are specified per blade row such that the upstream and/or downstream blade row is passed the first three harmonics from the adjacent blade row as boundary conditions in the frequency domain portion of the problem. The solutions are then run for 4000 iterations or until the RMS residuals, inlet mass flow rates, and exit mass flow rates were flat. The convergence history for the unsteady harmonic solution can be seen in Appendix D.

The unsteady, phase-lagged solutions were also initialized from the corresponding converged steady solution in order to a good starting point for the calculation. For each calculation, the convergence history was reset and thus the RMS residuals reset to zero. A dual time-stepping algorithm for the phase-lag solutions was instituted using a second order accurate temporal discretization with 2736 time steps per for revolution of the rotor (or 0.00229 radians per iteration) specified which for each period breaks down into 38 time steps per rotor passing and 72 time steps per vane passing. Fifty inner iterations are taken at each outer iteration until an additional -4.0 on the RMS residuals is achieved. However, due to the initial solution used to start each phase lag unsteady case, the additional four orders of magnitude drop in the
residuals is not achieved as the unsteady solutions were initialized using a solution with an RMS residual of -6.0. However, an additional order of magnitude drop in the residuals is seen. For convergence, both the inlet and exit mass flows and the residuals are monitored looking for each of the parameters to set up periodically. The typical solution required approximately one full revolution of the rotor to achieve the desired periodicity. The turbine stage is transonic and the high-pressure vane is choked at the throat, the inlet mass flow rates are constant throughout the calculation while the exit mass flow rates do set up periodically with mass flow rates at each time step within 0.02% of the previous period. The convergence history for the unsteady harmonic solution can be seen in Appendix D.

---

**Figure 51.** Convergence history for the three blade row steady calculation for the (a) RMS and maximum residuals and (b) inlet and exit mass flow rates

Continued
The contours of absolute total pressure, absolute total temperature, and Mach number for the three blade row steady solution are provided in Figure 52. The transonic operating conditions of the turbine can be clearly seen in all three contours. The Mach number contour in Figure 52(c) clearly shows the shock structure at the exit of the high-pressure turbine vane and at the exit of the high-pressure turbine rotor. The interaction of the shock emanating from the trailing edge and the blade wake at exit of the high-pressure turbine rotor can also be seen.
Figure 52. Uncooled, three blade row steady analytical contour plots at 50% span for (a) normalized total pressure, (b) normalized total temperature, and (c) Mach number
Figure 52 Continued

TTAnorm
1.05
1
0.95
0.9
0.85
0.8
0.75
0.7
0.65

(b) Continued
The analytical and experimental comparisons are broken up into three sections: the high-pressure vane, the high-pressure rotor, and the stationary shroud. Within each section, a comparison of the time-averaged, first harmonic of unsteady pressure from all analytical solutions and the experimental data are provided. The high-pressure rotor and stationary shroud incorporate time-series comparisons in addition to the time-averaged and first harmonic data. For reference, on each of the airfoil surfaces the pressure surface is denoted by negative values of wetted distance and the suction surface is denoted by positive values of wetted distance along the x-axis where zero is the leading edge and one or minus one is the trailing edge.
The first comparison for the high-pressure vane is the time-averaged pressure on the surface of the airfoil at 15%, 50%, and 90% that is shown in Figure 53. At all three spans, the both the suction and pressure surface results compare very well between all analytical models and experimental data. There is some difference noted between the steady and unsteady models on the suction surface beyond the passage shock location at about 50% wetted distance. At 50% and 90% span, both unsteady models compare better than the steady models. It should also be noted that the harmonic unsteady and phase-lag unsteady solutions produce over-lapping results at all locations. Also, the three blade row steady solution is only shown for the high-pressure vane and rotor airfoil surface comparisons. This model was removed from the rest of the comparisons to provide simplicity and clarity as it produced identical results to the two blade row steady solution.

Figure 53. Comparison of analytical and measured time-averaged static pressure on the high-pressure vane surface at (a) 15% span, (b) 50% span, and (c) 90% span
Figure 53 Continued

(b)

(c)
Figure 54, provides a comparison of the first harmonic of unsteady pressure on the airfoil surfaces at 15%, 50%, and 90% span. At all three spanwise locations, the analytical models show no unsteadiness on the pressure surface and up to 50% wetted distance on the suction surface. The experimental data does show very small amplitudes of unsteadiness that are less than 0.1% of the inlet total pressure. These small levels of measured unsteadiness could be attributed to downstream unsteadiness leaking forward through the boundary layer, slight operating point variations during the time window and/or from the resulting manufacturing tolerances from passage to passage. For the remainder of the suction surface, both unsteady models show good comparison with the experimental data in that the level of unsteadiness is on the same order of magnitude but the location of maximum and minimum are different. It is also worth noting that the unsteady harmonic and unsteady phase-lag solutions produce slightly different results, but neither one compares better with the data.

![Graph](image)

**Figure 54.** Comparison of analytical and measured first harmonic of unsteady pressure on the high-pressure vane surface at (a) 15% span, (b) 50% span, and (c) 90% span
The third set of comparisons is for the high-pressure vane hub surface. Figure 55 shows the comparison between the experimental data and the analytical models for the time-averaged pressure and the first
harmonic of unsteady pressure on the inner hub surface. The analytical time-average surface pressure on the hub, shown in Figure 55(a), is over-predicted in comparison to the experimental data by all models at PVIN53, PVIN55, and PVIN58 gage locations. PVIN51 through PVIN58 are at the trailing edge location such that the analytical models are over-predicting some of the circumferential variation of static pressure on the hub surface by up to 10% of inlet total pressure as seen at the PVIN58 location. Conversely, in Figure 55(b) for nearly all locations the first harmonic amplitudes are under predicted by the unsteady analytical models in comparison to the data. However, trends are captured very well across the trailing edge pressure transducers. This miss in time-average pressure and first harmonic of unsteady pressure is most likely due to transducer location differences, shock position differences due to grid resolution, and shock structure moving around due to operating point variation during operation of the turbine. From the Mach number contour plot in Figure 52, very slight differences in tangential location will produce large variations in static pressure. Again, the harmonic and phase-lag unsteady models produce similar results in both cases.
Figure 55. Comparison of analytical and measured pressure on the high-pressure vane hub surface for (a) time-averaged static pressure and (b) the first harmonic of unsteady pressure
The final set of comparisons on the high-pressure vane is for the outer shroud surface. Figure 56(a) shows the time-averaged pressure and Figure 56(b) shows the first harmonic of unsteady pressure for the five measurement locations on the outer shroud. The time-averaged pressure at 0% wetted distance (PVOT43) and one of the 50% wetted distance (PVOT50) transducers compare very well where the second 50% wetted distance gage (PVOT47) and the trailing edge locations at 103% wetted distance are about 5-7% off from the experimental data. The trends are captured very well by both unsteady solution techniques although the amplitudes are over and under-predicting at several of the locations. It is also interesting to note that the phase-lag and harmonic unsteady solutions do produce different results when looking at the first harmonic amplitudes in Figure 56(b) while the time-averaged results are identical.

Figure 56. Comparison of analytical and measured pressure on the high-pressure vane shroud surface for (a) time-averaged static pressure and (b) the first harmonic of unsteady pressure
From the comparisons on the high-pressure vane, it can be seen that all analytical models do an excellent job at capturing the time-averaged behavior on the airfoil, hub, and shroud surfaces. Both unsteady models also do a good job at capturing trends and amplitudes.

The next set of comparisons is for high-pressure rotor rotating surfaces that consist of time-averaged pressure, first harmonic of unsteady pressure, and time-series pressure comparisons on the airfoil and platform surfaces. No comparisons will be provided for the tip region as only the flat tip configuration was analyzed. The time-averaged pressure on the airfoil surface is shown in Figure 57 at 15%, 50%, and 90% span. All analytical models provide similar results and comparison to the data with slight under-predictions of the steady pressure loading on the suction surface at 15% span, but at 50% and 90% span the comparisons improve. Overall, there are some differences between the steady and unsteady models across the entire blade surface and is most noticeable beyond 50% wetted distance on the suction surface.
Figure 57. Comparison of analytical and measured time-averaged static pressure on the high-pressure rotor airfoil surface at (a) 15% span, (b) 50% span, and (c) 90% span
Figure 57 Continued

(c)

Figure 58 compares the first harmonic of unsteady pressure for the analytical and experimental results at 15%, 50%, and 90% locations. The harmonic amplitudes and trends are predicted very well by both unsteady models at all locations. At 15% and 50% span, both models produce very similar results and neither provides a better comparison, but at 90% span the phase-lag model does predict slightly less unsteady pressure amplitudes and compares slightly better to the data.
Figure 58. Comparison of analytical and measured first harmonic of unsteady pressure on the high-pressure rotor airfoil surface at (a) 15% span, (b) 50% span, and (c) 90% span
The variation in the unsteady pressure at two transducer locations at 50% span, which are 20% wetted distance on the suction surface and -20% wetted distance on the pressure surface, is shown in Figure 59(a) and (b), respectively. The comparisons are plotted as a function of the vane passage and provide the fluctuation of pressure about the time-averaged value. The similarity is once again seen between the phase-lag and the harmonic unsteady models while the phase-lag model does show slightly more variation as a function of time. Both models are in very good agreement with the data and maximum and minimum locations and amplitudes. Overall, the additional unsteadiness shown by the phase-lag solution does predict the behavior better away from the maximum and minimum locations.
Figure 59. Comparison of analytical and measured unsteady pressure on the high-pressure rotor airfoil surface at (a) 20% wetted distance and (b) -20% wetted distance.
The rotor platform time-averaged comparisons presented in Figure 60(a) show good comparison for most transducer locations while over-predicting the data at PRP53 and PRP54, which are at the trailing edge of the rotor airfoil. These over-predictions are anywhere from 2% to 15% of the inlet total pressure. The first harmonic of unsteady pressure, on the other hand, compares very well at PRP53 and PRP54 as shown in Figure 60(b) for both models. The level of unsteadiness on the rotor platform at all gage locations is very low and the differences between the measured and analytical results could be attributed to slight variations again in the operating point during the rig time window of operation or from the typical computational modeling issues such as turbulence modeling, grid resolution, and the Reynolds-Average assumptions in the governing equations.

![Graph showing comparisons of analytical and measured static pressure on the high-pressure rotor platform surface for average pressure and first harmonic of unsteady pressure.](image)

Figure 60. Comparisons of analytical and measured static pressure on the high-pressure rotor platform surface for (a) average pressure and (b) first harmonic of unsteady pressure
Time-series comparisons are also provided for the rotor platform surface in Figure 61(a) through (e) for each transducer location. Both trend and levels are captured very well at the PRP50 transducer location that is located near the rotor leading edge. At the other gage locations, some of the trends are captured well and the amplitude of fluctuations about the mean values is predicted to be well within the range of the data. For these comparisons, the phase-lag and harmonic solutions produce different results, but again neither solution produces a better answer when compared with the experimental results for the rotor platform surface.
Figure 61. Comparison of analytical and measured unsteady pressure on the high-pressure rotor platform for gage location (a) PRP50, (b) PRP51, (c) PRP52, (d) PRP53, and (e) PRP54
Figure 61 Continued

(c)

(d)
Similar to the high-pressure vane comparisons, the high-pressure rotor comparisons on the airfoil and platform surfaces show that all of the analytical models predict the time-average trends and amplitudes very well. The unsteady models also capture the trend and amplitude of unsteadiness on the airfoil surface well, but the platform surface misses the overall trends while keeping amplitudes within the range of the experimental data.

The last set of comparisons between experimental and computation results is for the stationary shroud surface above the high-pressure rotating blade. The time-average and first harmonic of unsteady pressure for all gage locations is shown in Figure 62(a) and (b), respectively. The time-averaged pressure is predicted very well at all gage locations for both the steady and unsteady models. Similarly, the unsteady models also capture the first harmonic amplitudes quite well with the largest difference being an under-prediction of the amplitudes by about 1% of inlet total pressure at 94% wetted distance. It is important to
note that at -7% and 30% wetted distance, there are multiple transducer locations positioned around the rotor providing some sense of the variability around the annulus.

Figure 62. Comparison of analytical and measured stationary shroud for (a) average pressure and (b) first harmonic of unsteady pressure
Time-series comparisons for both unsteady models at all gage locations are provided in Figure 63(a) through (d). At all four transducer locations, the measured unsteadiness is predicted very well by both the phase-lag and harmonic unsteady models in both amplitude and trend. At -7% wetted distance, the variability around the annulus can be seen and the analytical results miss some transducer location peak values by up to 2% of the inlet total pressure when compared to the PS3 and PS4 gage locations. A similar trend is seen at 30% wetted distance with the PS5 gage location. Another interesting point is that both unsteady models produced very similar results.
Figure 63. Comparison of analytical and measured unsteady pressure for the shroud at (a) -7% wetted distance, (b) 30% wetted distance, (c) 60% wetted distance, and (d) 94.4% wetted distance.
Figure 63 Continued

(c)

(d)
The high-pressure rotor stationary shroud comparisons also show excellent agreement between the measured and predicted results.

Overall, it can be seen from all of the comparisons presented that FINE/Turbo predicts the time-average and time-accurate surface pressure on both the high-pressure vane and blade very well. For the high-pressure vane, the hub and shroud comparisons do show some differences between the measured and predicted surface pressures particularly at the downstream locations. One potential reason behind the miss could be due to the ideal hub and shroud assumption. It will be discussed later that with the addition of the purge flow cavity even without purge flow a blockage is created at the high-pressure vane which increased hub static pressure loading, alters the circumferential loading, and affects the overall vane exit and rotor aerodynamics in the hub.

The high-pressure rotor comparisons compare to the data very well with both unsteady methods. In particular the high-pressure blade shroud and airfoil surface were noted as being as good and in many instances better than those produced by Green [62] in 2004 using MSU-TURBO. This could be for a variety of reasons including grid resolution and the use of single blade row used by Green [62] versus multiple blade row coupled unsteady analysis used in this work. The rotor platform does show some over-predictions of the time-accurate data on multiple gages whereas the non-linear harmonic balance shows better agreement from a magnitude perspective.

The non-linear, harmonic and the phase-lag unsteady solutions are also seen to produce very similar results all throughout the turbine stage with no model predicting the unsteadiness better than the other. A rough estimate of total computational time for these models is approximately 1 day for steady solutions, 5 days for harmonic solutions, and upwards of 20+ days for the phase-lag solutions. It should be noted, however, that there is the possibility of speeding up the phase-lag solutions with slightly different running techniques related to initial solutions. The non-linear, harmonic balance method does require significantly more RAM.
than the phase-lag or steady solutions. For an approximate 4X increase in computational time, very little
difference is noted between the non-linear harmonic balance and the phase-lag unsteady methods. This
provides the user with additional flexibility without any sacrifices in accuracy.

4.4 Other Applications of FINE/Turbo

In addition to the cases presented in section 4.3, NUMECA’s FINE/Turbo CFD code has been utilized in a
number of other research efforts. Janke et al. [63] used FINE/Turbo to investigate several configurations
for stationary passive cooling on the outer spans of a high-pressure, rotating, shrouded, turbine blade. The
investigation focused on using FINE/Turbo to evaluate several methods for injecting cooling flow for the
purposes of keeping the shroud region of the rotating blade cool. The CFD analysis involved looking at the
entire stage of the MT1 turbine in Cambridge (UK) including the cavity above the shrouded blade using a
steady mixing plane approach. Subsequently, the CFD results were compared to both traverse data at the
stator and rotor exit plane in the form of pitch and yaw angles. At mid-span traverse location behind the
stator, both the pitch and yaw angles were noted to be four degrees different from the experimental data
such that the CFD predicted higher turning and less radial deviation, most likely due to the slight over-
prediction of the mass flow rates. In the tip region, the yaw angles were noted as comparing qualitatively
well but in the shroud cavity both the pitch and yaw angles were significantly off. Measured surface
pressures taken on the blade surface at 50% and 90% span compared favorable to the CFD results. The
rotor exit predicted yaw and pitch angles were noted to compare reasonably good at mid span but hub and
tip secondary flows were clearly missed, mostly likely due to the mixing plane approach applied to the
computational model.
Southworth et al. [64] utilized FINE/Turbo to perform steady and unsteady analyses of a Honeywell single stage, fully cooled turbine rig in which both the vane row and the blade row were film cooled. Both film cooled, using source term injection points, and uncooled solutions were completed. For this study, time averaged and time-accurate surface pressure comparisons were made for the vane, blade, and shroud region about the rotating blade. The results from FINE/Turbo were also shown alongside MSU-TURBO mentioned earlier, but the MSU-TURBO results were for an uncooled turbine stage. The authors’ noted good overall comparisons between codes, methods, and the experimental data for all locations of interest.

Tartinville et al. [65] applied a code friendly variant of the v²-f turbulence model along with several additional turbulence models to multiple cases tried. Three cases of particular note were the DLR annular turbine cascade, the low-pressure turbine cascade known as T106D-EIZ, and a case simply known as the gas turbine nozzle. All cases were compared to experimental results and include the Spalart-Allmaras [46] turbulence as part of the comparisons. For the DLR cascade, pitch-averaged radial distributions of total pressure and circumferential flow angle produced from the FINE/Turbo code predicts the loss cores at 15% and 60% well along with the overturn of flow close to the hub and middle of the passage but was overestimated close to the tip. For the low-pressure turbine cascade, predicted axial distributions of isentropic Mach number were provided for the v²-f turbulence model and compare very well with the experimental data. The author did note that the predictions with other turbulence models were nearly identical and thus left out. The gas turbine nozzle case compared heat-transfer coefficients amongst all of the turbulence models and very good agreement between measured and predicted values were again noted for the single equation Spalart-Allmaras [46] turbulence.

Tartinville et al. [66] also used FINE/Turbo to predict the aerodynamics through the AGTB turbine cascade with leading edge cooling injection operated at multiple blowing ratios. The predictions were focused on isentropic Mach numbers on the surface of the cascade vanes using local source terms and fully discretized holes for the film cooling hole model. For all blowing ratios, the CFD results compared very well and the
source term and fully discretized holes produced similar results. Streamline comparisons between the two models to oil flow visualization did show that the fully discretized holes were much better suited at capturing the local flow features around the injection site. Hildebrandt et al. [67] also performed steady and unsteady (periodic, domain scaling method) analyses using FINE/Turbo on the MT-1 single stage turbine rig geometry in order to compare fully discretized cooling holes to the traditional source term approach. The computational geometry included the film cooled nozzle guide vane (NGV) with both source terms and discretized holes and an uncooled, rotating blade. Comparisons of the CFD results and the experimental rig were provided for the NGV at 50% span at are noted to be very good for all methods shown.

4.5 FINE/Turbo Evaluation Conclusions

The surface pressure coefficient predictions for the NACA0012 two-dimensional airfoil discussed in 4.3.1 show very good agreement between the experimental data and the analytical predictions for all Mach number and attack angles analyzed. There are some differences noted at the higher Mach numbers such as differences in the shock location on the airfoil as well as the rate of pressure drop through the shock. This was an excellent case to see how well FINE/Turbo did as a steady CFD code with fairly simple geometry. FINE/Turbo also did very well at capturing the total pressure rise, total temperature rise, and exit flow angles for NASA Rotor 37 and NASA Stage 35 shown in 4.3.2 that is a good test of the ability of the code to predict high pressure rises and of the steady, multistage mixing plane techniques. The exit profiles for NASA Rotor 37 and the rotor exit and stator exit profiles for NASA Stage 35 are well predicted capturing both the trends and the magnitudes within the experimental data. These results are as good as or better than other published results currently available in the public literature. It is also important to note the difficulty that many had had in predicting the exit profiles for these two cases in particular with the total temperature rise in the tip region and the total pressure rise around 30-40% span on NASA Rotor 37 in which
FINE/Turbo did show some discrepancies. However, to achieve such good results without any adjustments to the grid or code settings does provide some insight into the predictive capabilities of FINE/Turbo. The final test case prepared for this research involved a one and one-half stage uncooled turbine that is of the same geometry as the research within. This test case allowed for both steady and time-accurate comparisons of surface pressure for the high-pressure vane and high-pressure blade to understand how both the steady, phase-lag unsteady, and harmonic unsteady would compare match against the data. Overall, the comparisons at all locations are very good with FINE/Turbo capturing both the time-average and time-accurate trends and amplitudes within the measurements. Another interesting outcome is the similarity as noted between the phase-lag and harmonic unsteady techniques. No one method compared more favorably with the data and thus both are an excellent choice for use. Overall, these three test cases provided an opportunity to understand FINE/Turbo’s capabilities for predicting turbomachinery type flows. In addition to the three examples performed, FINE/Turbo has been used extensively throughout the turbomachinery community to perform similar research on many different geometries and flow conditions. In all cases, the code has performed very well when compared to experimental data and computational predictions, thus providing additional credibility to the use of the CFD code for this research.

The three test case results bring forth another discussion with respect to match experimental data with analytical tools that is worthy of having. Often there are discrepancies between the analytical and experimental results as can be seen through the test case comparisons that have been provided in section 4.3. These differences are due to many factors including steady versus unsteady affect, turbulence modeling simplifications, Reynolds-Averaged assumptions within the governing equations, grid resolution, manufacturing differences between theoretical and actual geometry, measurement error, lack of measurements available for best boundary condition selection, instrumentation placement tolerance, and even facility issues such as background noise which might trip shocks or boundary layers sooner than expected. Some of these factors can be controlled and sometimes even eliminated while others are realities in which their potential effects on the answers need to be understood. Often when CFD predictions match experiments exactly it is due to the experiment being very simple in construction with a lot of
instrumentation and geometry understanding that provides all of the information back to the CFD code for accurate boundary condition assessment leaving no pieces of the puzzle for which the engineer must make assumptions. While often useful for studying smaller pieces of a larger problem, these types of experiments are often difficult to transfer to actual products such as turbomachinery used in the aviation, energy, marine, and other industries as in many instances very little information is known. It is best summed up that in reality there are experimental measurements and analytical predictions, neither one is completely correct and the answer lies somewhere in-between.
CHAPTER 5
COMPUTATIONAL MODELING

The purpose of this chapter is to review the computational modeling of the experimental geometry. The computational modeling is the main thrust of this research. As such, the details will include the computational domain chosen for this problem, the boundary conditions applied, the solution techniques applied, and a look at convergence criteria and monitoring.

5.1 Computational Scope

The scope of the computational efforts includes two parts; the first is to assess the predictive capabilities of NUMECA International’s FINE/Turbo commercial CFD code using the data acquired from the experimental rig and the second is to use the computational results to gain further insight into the interactions between the purge cavity flow and the main gas path flow. The former will be discussed throughout Chapter 6 and the later throughout Chapter 7. The computational domain and boundary conditions were approached from the standpoint of duplicating the experimental rig results as much as possible while keeping the computations as reasonable as possible such as to represent the fidelity of analysis that would be used during the design phase of a high-pressure turbine in industry.

The original scope included analyzing a steady three blade-row solution of the experimental rig in order to generate boundary conditions for a smaller and thus faster running two blade-row solution in which both
phase-lag and harmonic unsteady techniques would be used to predict the unsteady behavior. This approach would also provide quantitative and qualitative comparison between the harmonic and phase-lag methods similar to the study shown in section 4.3.3. During post-processing of the analytical results, it was discovered that the total temperature profiles used in the analysis did not match the desired profiles from the experimental data due to user error. This led to a reduced computational scope of using the harmonic unsteady method only for the three blade-row case for assessing the code’s predictive capability for four of the five experimental runs analyzed as one case (Run 21) that is unaffected by the inlet total temperature boundary condition error.

5.2 Geometry Description

The turbine geometry studied within this research is a single-stage high-pressure turbine and low-pressure vane row that is considered typical of today’s aviation gas turbine engines used for propulsion on narrow body (single aisle) commercial aircraft. The high-pressure turbine is classified as a transonic, highly loaded vane and rotor with an operating total pressure ratio in excess of five and the low-pressure vane is aerodynamically shaped for counter-rotation operation of the low-pressure turbine. The actual airfoil geometries are considered to be very strong in three-dimensional aerodynamic shape. The airfoil counts for each stage are 38 high-pressure vanes, 72 high-pressure turbine blades, and 38 low-pressure vanes. The hardware as developed for the experimental rig is full scale rotating and operating at the proper design corrected conditions, which will be discussed later. The main flow path and surrounding hardware does not contain all of the necessary features in order to maintain proper metal temperatures as in an actual flight engine. The high-pressure vane is film cooled on both the surface of the vane, the hub, and shroud and a purge flow cavity with cooling air injection is located between the high-pressure vane and blade. The high-pressure vane film cooling patterns and the purge flow cavity geometry, however, were specifically developed for rig testing and are not considered to be fully representative of actual engine hardware. The
high-pressure rotating blade, blade platform, blade tip, and stationary shroud above the blade do not utilize nor contain features for film cooling. A cross-section of the nominal rig geometry is provided in Figure 64. A more detailed discussion of the experimental rig can be found in Chapter 3.

Figure 64. Nominal rig geometry for the modern high-pressure turbine and low-pressure vane (Not To Scale)

For the creation of the computational domain, as will be discussed in section 5.3, the nominal design geometry of the main gas flow path, airfoil shapes, and purge flow cavity are used; no attempts were made to incorporate the as-made geometry into the analysis. Only minor alterations and simplifications were performed to the purge flow cavity cross-section in order to provide a better shape for the blocking structure of the grid. These simplifications included replacing several corner fillets with square corners and
changing the cylindrical hole inlet spaced 38 per revolution to an axisymmetric inlet of the same area. A comparison of the nominal and simplified purge flow cavity geometry is provided in Figure 65. The other geometry simplification applied was that of an ideal inner and outer flow path surface with the exception of the purge flow cavity.

Figure 65. Purge flow cavity geometry (a) simplifications, (b) final geometry (Not To Scale)
The other geometry simplifications applied within the computational domain is that of an ideal inner flow path or hub surface and an ideal outer flow path surface or shroud surface. The exception to the hub surface is the inclusion of the purge flow cavity located between the high-pressure vane and high-pressure rotor. The purge cavity opens a connection in the hub surface such that the main gas flow path and the purge cavity are directly connected. A more detailed discussion of this connection within the grid will be discussed later in section 5.3. A comparison of the actual and idealized flow path surfaces is provided in Figure 66.
Figure 66. Inner and outer flow path surfaces: (a) flow path simplifications, (b) final flow path (Not To Scale)
Most of the simplifications made to the hub and shroud surfaces included removing the interfaces between blade rows where joints would normally exist in a mechanical structure. These joints that represent leakage paths and potential steps (both forward and backward) in the flow path are considered to be minor.
5.3 Computational Domain and Grid

The computational domains were selected in order to utilize the data from the experimental rig with little or no extrapolation of boundary conditions. In order to complete the original analysis which included steady, harmonic and phase-lag solutions, multiple computational domains and grids were used: a three blade row domain and a two blade row domain which will be referred to as the stage domain. The three blade-row domain is used to run the steady and harmonic unsteady analysis and the stage domains are used only for the phase-lag unsteady analysis for Run 21. The stage domains only include the high-pressure vane and rotor in order to keep the computational problem as small as possible in order to produce faster solution turn-around time. Exit boundary conditions for the stage domain are extracted from the steady three blade-row a solutions; this process is briefly discussed in section 4.3.3 and will be discussed in more detail in section 5.5. As shown in Green [62], the assumption of removing the low-pressure vane in the stage domain is based upon the comparisons between an unsteady, wake blade solution which utilized only the high-pressure vane as the inlet unsteadiness and a coupled, unsteady, three blade-row solution where the first harmonic of unsteady pressures on the suction surface show very little difference at 15%, 50%, and 90% span on the high-pressure rotor. This is due to multiple effects including a large rotor-stator gap between the high-pressure rotor and the low-pressure vane and a passage shock spanning from the trailing edge to the suction surface of the high-pressure rotor. The large rotor stator gap, which is on the order of one high-pressure blade chord length, allows for significant dissipation of the low-pressure vane static pressure bow wave that propagates forward and causes unsteadiness on the rotor as it passes through these standing bow waves. The high-pressure rotor passage shock does not allow unsteadiness coming from a downstream source such as the static pressure bow wave on the low-pressure vane leading edge to propagate forward through the shock structure.

The three blade row computational domain, by definition, consists of the high-pressure vane, high-pressure rotor, low-pressure vane, and the purge flow cavity located between the high-pressure vane and rotor. The
axial extent of the domain is carried from the inlet rake plane to the exit rake plane to align inlet and exit boundary conditions with the measurement planes from the experimental rig. For the tangential extent, a single passage is modeled for each blade row (1/38 of a revolution for the high-pressure and low-pressure vane, 1/72 of a revolution for the high-pressure rotor). The cavity domain is attached to the high-pressure rotor and thus has the same tangential width as a single passage of the rotor (1/72 or a revolution). The two blade row grid is identical in structure and extent, but with the low-pressure vane blade row removed. Hence, discussions to follow on grid block structure and cavity connections apply to both grids.

The cavity domain is connected to the high-pressure rotor grid through a non-matching interface in the axial direction with a tangential extent of one rotor passage to allow for a matching grid tangentially. Initial attempts were made to attach the cavity to the high-pressure rotor grid through a full matching (axial and tangential) connection as this would be the desired geometry. An illustration of the issue is shown in Figure 67(a). However, significant difficulty was encountered when attempting to execute fully coupled, unsteady, phase lag calculations. This was found to be due to the intersection of the vertical walls of the cavity with the horizontal wall of the high-pressure rotor hub where the tightly spaced boundary layer grid needed to be maintained throughout the entire rotor passage height. The tightly spaced grid in the main passage would cause significant turbulence viscosity ratio clipping due to the contraction-expansion-contraction-expansion of the grid through the cavity connection such that the solution would immediately diverge. Next, a full non-matching connection both axially and tangentially was created in order to remove the contraction and expansion zones from the main passage grid. The full non-matching connection initially proved to show better convergence and turbulence viscosity ratio behavior, but upon further inspection the solutions were producing extremely high static pressures on the cavity surface near the connection, which were on the order of one and one-half times the inlet total pressure. The cause of the high static pressures was found to be a fictitious overlap of surfaces caused by misalignment of axial grid lines on either side of the cavity connection. An illustration of this type of connection is shown in Figure 67(b). The final solution was to use a non-matching connection in the axial direction only, however axial grid lines were aligned in the main passage to both sides of the vertical walls of the cavity to avoid the
fictitious overlap as seen on the full non-matching connection. An illustration of this type of connection is shown in Figure 67(c). This cavity connection proved to be very successful as will be shown later.

Figure 67. Illustration of the purge flow cavity connections with the high-pressure rotor domain: (a) Full matching connection, (b) Full non-matching connection, (c) the axially non-matching connection
Axial Mis-Alignment of Grid Lines, Leads to Fictitious Surface Overlap

Smooth Axial Main Passage Grid

Axial Mis-Alignment of Grid Lines, Leads to Fictitious Surface Overlap

Cavity Vertical Surfaces
The high-pressure and low-pressure vanes are solid airfoils from hub to shroud while the high-pressure rotor is unshrouded with a tip gap that is 2.15% of the blade height and is held constant for all analyses. The default multi-block grid structure in Fine/Turbo’s Autogrid5 program was chosen for each blade row and consists of five blocks to build the entire passage grid with a solid airfoil from hub to shroud. This default multi-block structure utilizes an ‘O’ style grid block around the blade surface to form the skin and ‘H’ style grid blocks through the remainder of the passage. A schematic of this structure in the θ-Z plane is shown in Figure 68 below and a similar schematic in the R-Z plane is shown in Figure 69.
Figure 68. θ-Z planar view of default blocking structure in Fine/Turbo
The high-pressure rotor grid incorporates a gridded tip gap above the blade that adds an additional two blocks through the tip gap region: an ‘O’ style grid that runs the outside perimeter of the tip gap similar to the ‘O’ style grid used for the skin of the blade and an inner ‘H’ style grid. The rotor tip grid and the main passage grid are connected one-to-one across the interface. The rotor-stator interfaces were defined approximately one-half the distance between blade rows with radial distributions that match one-to-one. The periodic boundaries are also matched one-to-one for each of the blade rows and the rotor purge cavity.

The grid distributions in each of the three directions were set based similar work performed by Green [62] with some adjustments made for the purge flow cavity addition. No grid resolution study was performed as grid dimensions were deemed to be more than adequate for this research. The high-pressure vane and low-pressure vane both used 65 points in the radial direction while the high-pressure rotor used 81 points.
radially with 21 points spanning the rotor tip gap region. For the tangential and axial direction, each blade row used a slightly different number of points between the inlet and exit due to the use of high angle topography in the blade skin block, which allows the user to put more points on the suction surface of the blade. This is needed due to the much longer surface length of the suction surface as compared to the pressure surface when either high inlet or high exit flow angles are encountered. The high-pressure vane took advantage of the high exit angle topography. In the tangential direction, the grid contains a total of 49 points at the inlet and 81 points at the outlet while in the axial direction there are 101 points on the pressure surface with 165 points on the suction surface for a total grid size of approximately 660,000 points. The final grid resolution on the surface of the high-pressure vane can be seen in Figure 70.

Figure 70. Surface grid resolution for the high-pressure vane
The high-pressure rotor grid used a high inlet angle and high exit angle topography, which translated to 57 points tangentially at both locations and 157 points axially on the suction surface and 101 points axially on the pressure surface for a total grid size of approximately 96,000 points. The final surface grid resolution for the high-pressure rotor can be seen in Figure 71.

![Image of grid resolution](image)

**Figure 71. Surface grid resolution for the high-pressure rotor**

The cavity grid was generated using total of 27 ‘H’ style blocks throughout the geometry. The blocks were arranged manually within the simplified geometry and axial / tangential distributions were set within each block to minimize the expansion ratios to be less than 1.4 from cell to cell and across block interfaces. By default, due to the axial only non-matching connection, the purge flow cavity contained the same 57 points.
tangentially as the high-pressure rotor grid with a final total grid size of 1,400,000 points. Figure 72 below shows the blocking structure and tangential cuts of the final grid resolution within the purge flow cavity.

Figure 72. Purge flow cavity blocking structure and tangential cut of the grid resolution

The low-pressure vane grid used a high inlet angle and high exit angle topography, which translated to 49 points tangentially at the inlet and 65 point tangentially at the exit. Axial refinement of the grid used a total of 153 points on the suction surface and 105 points on the pressure surface for a total grid size of approximately 625,000 points. The final surface grid resolution for the high pressure rotor can be seen in Figure 73 below.
First cell offset from all surfaces was set to achieve a $y^+$ value between one and ten as directed by NUMECA International [38] for use with the Spalart-Allmaras [46] turbulence model. This turbulence model was used for this research as the nonlinear harmonic solver in FINE/Turbo currently only functions with the Spalart-Allmaras [46] model. The final surface offset value used for all wall surfaces (hub, shroud, blades, etc) for all blade rows including the cavity was determined analytically through trial and error with the best overall value found to be $2.54\times 10^{-6}$ meters. A contour plot of the results $y^+$ values for one of the cases studied is shown in Figure 74(a) through (d) for each of the blade rows. Note that for the high-pressure vane, the $y^+$ values at locations of the cooling flow source terms and locations on the hub and shroud aft of the throat location are higher than the desired value 10. Since the high-pressure vane is transonic, conditions at the throat are choked and further downstream the local Mach numbers near the hub surface are approaching Mach 1.4 and thus contributing to the over shoot in desired $y^+$ value, however the over shoot is not extreme (values are only approaching 15) and do not cause concern for the resulting
solution. For the cooling hole locations in the cooled high-pressure vane $y^+$ contour in Figure 74(a). This is due to the source term method of cooling flow introduction which injects the cooling flow through the solid wall thus the velocities at the first cell center off the wall in these locations are significantly higher than would normally be anticipated for a boundary layer flow at the main passage Mach numbers. This creates a fictitious $y^+$ value and thus these regions are ignored.

Figure 74. Contour plot of resulting $y^+$ values from Run 33 for the (a) cooled high-pressure vane (b) high-pressure rotor, (c) purge flow cavity, and (d) low-pressure vane.
Figure 74 Continued

(b)

(c)  Continued

210
In the next section, the boundary conditions applied to the previously discussed computational grids will be discussed.

### 5.4 Flow Conditions

The primary objective behind choosing the three runs was to provide a comparison between a non-cooled cold inlet temperature profile, nominally cooled and uncooled elevated flat inlet total temperature profile, and a nominally cooled and uncooled radially varying inlet total temperature profile to understand how the purge cooling flow affects the rotor aerodynamics. Thus, five flow conditions were analyzed based on runs performed during the experimental rig testing: Run 21, Run 22, Run 28, Run 33, and Run 43. The rig conditions for these three runs are summarized in Table 6 below and a more detailed discussion of the experimental rig can be found in Chapter 3.
<table>
<thead>
<tr>
<th>Run</th>
<th>Max To Avg Inlet TT (K)</th>
<th>TT Ratio</th>
<th>PT-to-PS Ratio</th>
<th>Corrected Flow (kg/sec)</th>
<th>Corrected Speed (RPM)</th>
<th>HPV Film Cooling (% of Inlet)</th>
<th>Purge Cavity Flow (% of Inlet)</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>1</td>
<td>1.40</td>
<td>5.33</td>
<td>4.59</td>
<td>6588.35</td>
<td>0%</td>
<td>0%</td>
</tr>
<tr>
<td>22</td>
<td>1.11</td>
<td>1.47</td>
<td>5.64</td>
<td>4.83</td>
<td>6013.10</td>
<td>0%</td>
<td>0%</td>
</tr>
<tr>
<td>28</td>
<td>1.16</td>
<td>1.57</td>
<td>5.61</td>
<td>4.73</td>
<td>6286.26</td>
<td>12.90%</td>
<td>0.72%</td>
</tr>
<tr>
<td>33</td>
<td>1.02</td>
<td>1.55</td>
<td>5.87</td>
<td>4.69</td>
<td>6275.89</td>
<td>12.90%</td>
<td>0.71%</td>
</tr>
<tr>
<td>43</td>
<td>1.02</td>
<td>1.40</td>
<td>5.94</td>
<td>4.69</td>
<td>6400.51</td>
<td>0%</td>
<td>0%</td>
</tr>
</tbody>
</table>

Table 6. Rig conditions analyzed using Fine/Turbo

5.5 Boundary Conditions

The analysis requires boundary conditions to be specified throughout the computational domain in order to properly match the rig flow conditions previously summarized in section 5.4. The required boundary conditions include the inlet conditions at both the main gas path upstream of the high-pressure vane and at the purge flow cavity inlet, the outlet downstream of the low-pressure vane for the three blade row case and the outlet downstream of the high-pressure rotor for the two blade row case, and for the blade, vane, shroud, and hub surfaces for each blade row.

For the inlet to the main gas path, the total conditions were specified which required the knowledge of the total pressure, total temperature, tangential velocity angle relative to the axial direction, the radial velocity angle relative to the axial direction, and the turbulent kinematic viscosity. The inlet total pressure was applied as a constant radial profile from hub to shroud based on the average of the five spanwise
measurements taken form the inlet rakes just upstream of the high-pressure vane. The inlet total pressure measurements varied 0.8% as a maximum from the average values for all five runs. When one considers that the maximum deviation is less than 4 kPa (0.5 psi), utilizing a constant inlet total pressure profile is a reasonable assumption. Figure 75 shows a comparison of the average values to actual measured inlet total pressure.

Figure 75. Comparison of rake measured and averaged inlet total pressure profiles for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43 normalized to inlet averaged total pressure

(a) Continued
Figure 75 Continued

(b)

(c) Continued
Figure 75 Continued

(d)

(e)
Note for Runs 21, 22, 33, and 43 that the average profile values are within the error bars, which are one standard deviation as calculated from the rig data and Run 28 shows significantly less error in the measurements.

The inlet total temperature for each of the different runs is specified as a varying radial profile based on the measurements at the inlet rake planes. Rake measurements were taken at nine different spanwise locations between 10% and 90% span at two circumferential locations in the inlet. The two rake locations, designated at TU1 and TU2, were then averaged at each spanwise location to produce the final total temperature radial profile for each run shown in Table 6. Since no measurements are taken near the hub or shroud surface, the 10% and 90% measurements were repeated at 0% and 100% respectively to provide full coverage over the entire span. Figure 76 shows the measured rakes values and the final inlet profile used for the inlet total temperature. Figure 76 (a), the result profile for Run 21 is essential a constant profile from hub to shroud with about 1.5° K of variation. This result is expected given the desired inlet conditions for the experimental rig for Run 21. Figure 76 (b) is more of a radial profile peaking at 60% span 80° K above the average while Run 33 in Figure 76 (c) shows significantly less peak in the resulting profile. The resulting total temperature profile applied to the computational analysis for Run 21 and Run 28 fall within the error bars of the measurements suggesting that for these runs taking the average of the two rake measurements is a fairly reasonable assumption to make. Run 33 shows about a 30° K difference at 20% span tapering to 5° K difference at 50% span between the two rake locations while above and below these points TU1 and TU2 agree within the error bars of the measurements. This suggests that Run 33 potentially has both a radial and circumferential variation to the inlet total temperature. However, for this analysis the total temperature profile is only radial and is the resulting average of the TU1 and TU2 rake measurements.
Figure 76. Comparison of rake measured and averaged inlet total temperature profiles for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43 normalized to inlet averaged total temperature.
Figure 76 Continued

(c)

(d) Continued
Next, two different approaches were taken for the two required inlet velocity angles: the tangential angle and the radial angle. The inlet tangential velocity angle, often referred to as $\alpha_Z$, is defined as:

$$\alpha_Z = \arctan\left(\frac{V_T}{V_Z}\right)$$  \hspace{1cm} (5.1)

And the radial velocity angle, $\phi$, is defined as:

$$\phi = \arctan\left(\frac{V_R}{V_Z}\right)$$  \hspace{1cm} (5.2)

The tangential angle is specified as a constant equal to zero over the entire face of the inlet. Within the experimental rig, the flow path between exit of the combustor emulator and the inlet to the high-pressure vane is free of any geometry that would impart turning on the flow and thus the tangential velocity is essentially zero. This length of the flow path is common to the test case shown in section 4.3.3 where the flow path and strut analysis in Appendix C shows essentially no tangential velocity component in Figure 219.
153 to the air entering the high-pressure turbine vane. The inlet radial velocity angle, on the other hand, is greater than zero due to the sloping hub surface at the upstream rake plane (also the inlet to the two blade row and three blade row grids). The radial angle at the rake plane is approximately 13.44° (0.23457 radians) radially upward on the hub surface and 0° on the shroud surface. Again from the flow path and strut analysis in Appendix C, the predicted radial angle profile in Figure 153 at the rake plane changes from approximately 13.44° at the hub to 0° at the shroud as expected. This same profile is also utilized for all three flow conditions analyzed again due to the commonality of the geometry.

The last boundary condition required for the inlet is the turbulent kinematic viscosity. The turbulent kinematic viscosity is applied as a radial profile over the inlet region for the main gas path. This profile was generated using the average inlet total temperature profile as shown in Figure 76 and Sutherland’s Law in equation 3.5 for the kinematic viscosity. A factor of 2.5 was applied to the resulting kinematic viscosity as recommended by NUMECA International [38] for internal flows.

The exit boundary condition for the main gas path required specification of either a mass flow rate, or an exit static pressure, or a characteristic be imposed in order to close the boundary conditions for the main gas path. Since the geometry is known and the flow conditions at high-pressure vane exit are choked, the imposed characteristic was not a desired boundary condition of choice. Similarly, imposing the mass flow rate, while known, could cause the calculation to stall as the choked conditions at the high-pressure vane will only allow so much mass flow to pass through and any slight variances in throat areas versus the experimental rig would cause the calculation to stall as well. This leaves only specifying the exit static pressure of which there are three options: static pressure imposed at the exit, averaged static pressure at the exit, or radial equilibrium equation about the hub or shroud surface. The static pressure imposed boundary condition keeps the static pressure constant across the entire exit plane thus potentially setting up a reflecting condition in which pressure waves could reflect off the exit boundary and propagate forward. Naturally, this is an undesired and unrealistic phenomenon and thus a more relaxed boundary condition was
desired in the form of an averaged static pressure applied across the exit boundary. However, only a single static pressure measurement was taken at the shroud surface during each experiment making the radial equilibrium equation the best option at the risk of being more unstable.

In the end, the three blade-row steady solutions were each run using the radial equilibrium exit boundary condition specifying the measured static pressure from the rig at the shroud surface where it was measured. Once the three blade-row steady solutions are complete, the averaged exit static pressure is extracted from the solution and transferred to the three blade-row harmonic solutions and the exit boundary condition is set to run the averaged exit static pressure. This provides a much more stable and less reflective boundary condition for the unsteady analysis. The average static pressure at the interface between the high-pressure rotor and low-pressure vane is also extracted from the steady solutions and applied to the stage domain exit as an average static pressure boundary condition for the phase-lag solutions.

As previously mentioned, it is desired to reduce the computational efforts of the unsteady calculations that will be performed for the phase-lag unsteady calculation perspective. It is also assumed that the low-pressure vane attributes very little with respect to unsteadiness in the cavity located between the high-pressure vane and rotor due to the presence of a passage shock in the rotor in which no unsteadiness will propagate forward. The axial gap between the high-pressure rotor and low-pressure vane is also significant enough (~1 rotor chord length) such that the bow wakes on the low-pressure vane will dissipate and contribute little unsteadiness on the blade. With this in mind, it is desired to remove the low-pressure vane from the unsteady calculations but no inter blade row measurements were taken during the experiments, which could provide the exit boundary conditions for the high-pressure rotor. Thus, the three blade row case is used to calculate the low field throughout the entire turbine in order to find the exit boundary conditions (i.e. average static pressure for the high-pressure rotor domain) from a purely computationally aspect. The three blade row solution uses the exit boundary conditions derived from the rig measurements as previously described and then is iterated until steady convergence is achieved. The average static
pressure at the high-pressure rotor and low-pressure vane interface is then extracted from the three blade row solution and transferred to the two blade row exit plane as an average static pressure boundary condition. This process is used for all cases studied and is the same process utilized for the test case in section 4.3.3.

For the purge flow cavity inlet, a mass flow rate boundary condition was applied. The mass flow rate inlet boundary condition requires knowledge of the mass flow rate, the direction velocity ratios to the total absolute velocity, the static temperature of the incoming air, and the inlet turbulent kinematic viscosity. The Large Cooling Facility (LCF) system at OSU-GTL, which has been described further in Chapter 3, operates at a starting pressure of 3.7 times inlet averaged total pressure and feeds both the purge flow cavity and the vane film cooling circuits while the gas path. The high-pressure vane film cooling circuit feed pressure is approximately 1.02 times the inlet averaged total pressure and the purge flow cavity circuit feed pressure is approximately 0.44 times the inlet averaged total pressure such that the pressure ratios between the tank and the feed are in excess of 1.893 total pressure ratio required to achieve choked flow. Thus the flow is choked within each of these circuits and the mass flow rates are well known based on separate experiments performed at the OSU-GTL. Mathison [68] provides additional details on the testing of the OSU-GTL LCF system and how the cooling flow rates are established for the facility. Thus, the mass flow rate boundary condition was chosen over specifying total conditions and provides a more stable boundary condition for the purge flow cavity inlet versus the total conditions. The static temperature of the incoming flow was determined using isentropic flow relations knowing the total temperature and total pressure of the incoming flow from rig measurements. A simple iteration method using the incoming Mach number was utilized in MS-Excel and the Goal Seek function to find the final static conditions at the inlet. The velocity ratios are based on the geometry of the rig such that the inlet angle in both the radial (0°) and the tangential (30° from axial) directions are known such that:

\[
\frac{V_r}{V} = 0; \quad \frac{V_T}{V} = 0.500; \quad \frac{V_z}{V} = 0.866025
\]  

(5.3)
Finally, the turbulent kinematic viscosity was found in the same fashion as done for the main gas path using the purge cavity inlet total temperature to find the kinematic viscosity based on Sutherland’s Law and applying the 2.5 factor as recommended by NUMECA International [38] for internal flows. The inlet conditions for the purge flow cavity are summarized in Table 7.

<table>
<thead>
<tr>
<th>Run Number</th>
<th>Purge Inlet Total Temperature / TT(Inlet)</th>
<th>Purge Inlet Total Pressure / PT(Inlet)</th>
<th>Purge Cavity Flow (% of Inlet)</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>N/A</td>
<td>N/A</td>
<td>0</td>
</tr>
<tr>
<td>22</td>
<td>N/A</td>
<td>N/A</td>
<td>0</td>
</tr>
<tr>
<td>28</td>
<td>0.5990</td>
<td>0.4599</td>
<td>0.72%</td>
</tr>
<tr>
<td>33</td>
<td>0.6324</td>
<td>0.4637</td>
<td>0.71%</td>
</tr>
<tr>
<td>43</td>
<td>N/A</td>
<td>N/A</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 7. Purge flow cavity inlet boundary conditions normalized to inlet-averaged values

At the lower boundary of the purge flow cavity, a labyrinth seal is incorporated into the rig to minimize leakage flow. A cross-section of the labyrinth seal is shown in Figure 77. The rotating seal cuts into a stationary Teflon block in order to provide a tight clearance. It is assumed in each of the computational models that there is no leakage flow across the labyrinth seal teeth at the bottom of the cavity.
Figure 77. Cross-section of purge cavity including the lower labyrinth seal (Not To Scale)

Total pressure and temperature gages were positioned just before the inlet to the purge cavity to understand the inlet boundary conditions to the cavity as well as below the labyrinth seal in order to monitor leakage rates through the seal teeth. Plots of the temperature and pressure histories as measured on the rig are shown in Figure 78 for Run 21, 28, and 33.
Figure 78. Pressure and temperature history within the purge cavity during the experiment time windows for (a) Run 21, (b) Run 28, and (c) Run 33
The pressure and temperature history during the experimental runs show the total pressure (PM3) and temperature (TM3) below the labyrinth seal to be fairly constant over the time window of the experiment, similar to the total pressure at the inlet to the purge cavity (PM2) whereas the total temperature into the cavity (TM2) is dropping over the time window for Runs 28 and 33. It would be expected that if the higher temperature purge flow were leaking across the labyrinth seal that a significant rise in temperature behind the seal would be seen over the time window, however this is not the case suggesting that the leakage is minimal.

To verify this assumption of no leakage, the leakage flow rate through the seal is estimated using a preliminary design approach from Vermes[69] often called the Vermes Orifice equation:

$$m = \frac{0.79 \cdot C_D \cdot A \cdot P_o \cdot \beta}{\sqrt{T_o} \cdot \sqrt{1 - \alpha}}$$  \hspace{1cm} (5.4)
Where $\alpha$ and $\beta$ are as follows:

$$\alpha = \frac{8.52}{P - L} + 7.23$$

$$\beta = \frac{\sqrt{1 - \left(\frac{P_s}{P_o}\right)^2}}{N - \ln\left(\frac{P_s}{P_o}\right)}$$

In equations 5.4 to 5.6, $C_D$ is the discharge coefficient, $P_o$ is the total pressure upstream, $P_s$ is the sink pressure downstream, $T_o$ is the total temperature upstream, $A$ is the annular leakage area, $CL$ is the clearance between the top of the teeth and the Teflon block, $N$ is the number of teeth in the labyrinth seal, $P$ is the tooth pitch and $L$ is the tooth tip width. Assuming a $C_D$ of 0.8 and a clearance of 0.0127 mm, a total mass flow through the labyrinth seal of approximately 5% of the mass flow entering the purge flow cavity or about 0.04% of the inlet flow is calculated for Run 28 and Run 33 which is well within the expectations for the experimental rig. In reality, the discharge coefficient of 0.8 is loosely based on the configuration as shown in Figure 77, but reality is that the seal teeth will cut into the Teflon block as the rotor grows under increasing RPM creating a much more tortuous path for the airflow to go over a single seal tooth let alone two teeth such that the actually leakage flow is probably 10-20% less than calculated and thus a good assumption that there is no leakage flow across the labyrinth seal.

The hub, shroud and blade surfaces for each of the blade rows are assumed to be isothermal or constant temperature with temperatures applied based on the experimental rig. This assumption is applied due to the use of a short duration facility to run the experiments, which has been discussed in more detail throughout Chapter 3. For each experiment, the temperature of each surface stays constant due to the short data acquisition time window. A temperature plot from a heat-flux gage located on the high-pressure rotor surface, labeled HR142, is shown in Figure 79 with the statistics shown in Table 8 for Runs 21, 28, and 33.
only. It can be seen that the temperature over the 14 millisecond (ms) time window for Runs 28 and 33 and the 17 ms time window from Run 21 the rotor surface temperature measured by the heat-flux gage is approximately constant.

![Temperature History](image)

**Figure 79.** Temperature history during rig operations from gage HR142.

<table>
<thead>
<tr>
<th>Run Number</th>
<th>Average Temperature (K)</th>
<th>Maximum Temperature (K)</th>
<th>Minimum Temperature (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>0.9619</td>
<td>0.9734</td>
<td>0.9585</td>
</tr>
<tr>
<td>28</td>
<td>0.6219</td>
<td>0.6247</td>
<td>0.6198</td>
</tr>
<tr>
<td>33</td>
<td>0.6182</td>
<td>0.6199</td>
<td>0.6159</td>
</tr>
</tbody>
</table>

**Table 8.** HR142 gage statistics over each time window normalized to inlet-averaged total temperature.
For each of the surfaces within the computational domain, a constant temperature is specified. The temperatures are measured on the rig from multiple resistance temperature devices (RTDs) throughout the rig, which were incorporated in order monitor temperatures during each experimental run. Table 9 below provides a summary of the temperatures applied to each of the analytical runs. For Runs 22, 28, 33, and 43 which utilized the combustor emulator to produce elevated temperature profiles, the high-pressure vane surface temperatures are higher than the surrounding shroud and hub temperatures. This is due to the vane surface having direct line of sight to the combustor emulator thus increasing the amount of radiation heat received (more direct view, increases view factor). Downstream of the high-pressure vane, surface temperatures tend to be cooler and more uniform throughout.

<table>
<thead>
<tr>
<th>Run Number</th>
<th>21</th>
<th>22</th>
<th>28</th>
<th>33</th>
<th>43</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>High-pressure Vane</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shroud</td>
<td>0.990</td>
<td>0.581</td>
<td>0.559</td>
<td>0.591</td>
<td>0.562</td>
</tr>
<tr>
<td>Hub</td>
<td>0.997</td>
<td>0.572</td>
<td>0.558</td>
<td>0.616</td>
<td>0.532</td>
</tr>
<tr>
<td>Airfoil</td>
<td>1.063</td>
<td>0.618</td>
<td>0.559</td>
<td>0.633</td>
<td>0.593</td>
</tr>
<tr>
<td><strong>High-pressure Rotor</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shroud</td>
<td>0.985</td>
<td>0.580</td>
<td>0.558</td>
<td>0.586</td>
<td>0.513</td>
</tr>
<tr>
<td>Hub</td>
<td>0.990</td>
<td>0.569</td>
<td>0.559</td>
<td>0.587</td>
<td>0.524</td>
</tr>
<tr>
<td>Airfoil</td>
<td>0.990</td>
<td>0.580</td>
<td>0.559</td>
<td>0.587</td>
<td>0.513</td>
</tr>
<tr>
<td><strong>Purge Flow Cavity</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stationary Wall</td>
<td>0.997</td>
<td>0.572</td>
<td>0.559</td>
<td>0.616</td>
<td>0.532</td>
</tr>
<tr>
<td>Rotating Wall</td>
<td>0.990</td>
<td>0.580</td>
<td>0.559</td>
<td>0.587</td>
<td>0.513</td>
</tr>
<tr>
<td><strong>Low-pressure Vane</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shroud</td>
<td>0.981</td>
<td>0.567</td>
<td>0.557</td>
<td>0.581</td>
<td>0.516</td>
</tr>
<tr>
<td>Hub</td>
<td>0.981</td>
<td>0.567</td>
<td>0.557</td>
<td>0.581</td>
<td>0.516</td>
</tr>
<tr>
<td>Airfoil</td>
<td>0.981</td>
<td>0.567</td>
<td>0.557</td>
<td>0.581</td>
<td>0.516</td>
</tr>
</tbody>
</table>

**Table 9.** Isothermal surface temperatures normalized to the inlet-averaged total temperature as applied to the computational model.
For Runs 28 and 33, high-pressure vane surface film cooling is incorporated into the rig and is also included into the computational model. This was accomplished by specifying source terms on the vane, hub, and shroud surfaces to match the cooling patterns applied to the turbine rig hardware. A total of 488 cooling holes and 17 trailing edge slots per vane segment are incorporated in the rig and computational models. The vane surface incorporates 386 holes split between 7 radial rows on the suction surface, 9 radial rows on the pressure surface, and 17 trailing edge slots while the hub surface has 47 holes and the shroud surface has 72 holes. For each hole or row of holes, the diameter, mass flow, static temperature, turbulent kinematic viscosity, and the injection angle are specified. This information was obtained from previous analysis performed during the original design of the rig and is applied as received. Figure 80 shows the film cooling pattern on the high-pressure vane, hub, and shroud surfaces by plotting a contour of the local flow injection on the surface of the high-pressure vane for the Run 28 solution.

Figure 80. Film cooling hole patterns highlighted through local flow injection on the high-pressure vane, hub, and shroud surfaces for the Run 28 solution.
The last boundary conditions required are the specification of rotating and non-rotating reference frames and surfaces. For the solution frame of reference, the high-pressure vane and low-pressure vane are solved in the stationary frame of reference. The high-pressure rotor domain and the purge flow cavity are solved in the rotational frame of reference. The hub, shroud and airfoil surfaces within the high-pressure and low-pressure vane domains are stationary. The hub and airfoil surfaces within the high-pressure rotor domain and the rotating surface of the purge flow cavity (aft most surface from the seal tooth to the attachment to the high-pressure rotor domain) are given rotational speeds to match the rotational frame of reference where the shroud and stationary surface of the purge flow cavity are stationary (zero RPM applied). A summary of the rotating and stationary boundary conditions applied to all three blade rows is shown in Figure 81. The two blade row domain utilized these same boundary conditions.

Figure 81. Summary of stationary and rotating boundary conditions for all three blade rows (Not To Scale)
5.6 Solution Techniques and Convergence Criteria

The solutions presented within this work were executed using as much of the default parameters within FINE/Turbo as possible. For all solutions, the second order accurate, central space, spatial discretization and the four stage, explicit, Runge-Kutta scheme is used for temporal discretization. A Courant-Fredricks-Levy (CFL) number of three is specified in order to remain within the stability region of the explicit scheme.

For steady analysis, each solution is started off with a ‘turbomachinery’ solution created by the FINE/Turbo interface. This initial solution uses the inlet boundary conditions, the exit boundary conditions, and static pressure values specified at each of the rotor-stator interfaces along with the conservation of mass and rothalpy along streamlines to produce an initial solution. Only the cavity blocks are initialized from constant values specified based on the inlet conditions calculated using the isentropic flow equations knowing the inlet total conditions and the mass flow rate. Each grid was setup to have a minimum of three levels of multi-grid (Coarse, medium, fine grid) of which all three levels are used. For the coarse and medium grid the levels, the minimum convergence level is set to -3.0 for the RMS residuals such that 100 iterations are performed until either the convergence level is reached or the iteration level is reached on the coarse and medium grids. After which, the remaining iterations are completed on the fine grid until convergence is achieved. This is known as a linear progression from the coarse to the fine grid, other options are available but the intent is to keep parameters as default as possible. To speed up convergence on the steady solutions, local time stepping along with implicit residual averaging is utilized. Solutions are iterated on the fine grid until either a minimum RMS residual of -6.0 is achieved or until the inlet flow, exit flow, and RMS residuals are flat. Typically, the inlet and exit flow rates are flat and within 0.02% or better of each other long before the RMS residuals are flat. A typical convergence plot for the Run 21, steady, three blade row calculation is shown in Figure 82.
Figure 82. Run 21 steady three blade row calculation for (a) inlet and exit flow rates, and (b) the RMS and maximum residuals.
For each of the three steady solutions, both stage and three blade row, the minimum residual of -6.0 as not achieved as can be seen by the RMS residual convergence shown in Figure 82(b) above. However, the inlet flow and exit flow are flat (and thus not changing) long before the RMS and maximum residuals reach a minimum. For this particular case, the inlet and exit mass flows are less than 0.006% suggesting that FINE/Turbo does an excellent job of conserving mass throughout the domain.

The unsteady, phase-lagged solutions are initialized from the corresponding converged steady solution in order to a good starting point for the calculation. For each calculation, the convergence history is reset and thus the RMS residuals reset to zero. A dual time-stepping algorithm for the phase-lag solutions is instituted using a second order accurate temporal discretization with 2736 time steps per revolution of the rotor (or 0.00229 radians per iteration) specified which for each period breaks down into 38 time steps per rotor passing and 72 time steps per vane passing. For each outer iteration, a total of 50 inner iterations were taken until an additional -4.0 on the RMS residuals is achieved. Due to the initial solution used to start each phase lag unsteady case, the additional four orders of magnitude drop in the residuals is not achieved as the unsteady solutions were initialized using a solution with an RMS residual of -5.5. However, an additional order of magnitude drop in the residuals is seen. For convergence, both the inlet and exit mass flows and the residuals are monitored looking for each of the parameters to set up periodically. The typical solution required approximately one full revolution of the rotor to achieve the desired periodicity. Since the turbine stage is transonic and the high-pressure vane is choked at the throat, the inlet mass flow rates are constant throughout the calculation while the exit mass flow rates do set up periodically with mass flow rates at each time step within 0.02% of the previous period. Figure 83 shows the run 21 convergence for the unsteady, phase-lag solution which is typical for all phase-lag, unsteady solutions.
Figure 83. Run 21 convergence for the unsteady, phase-lag solutions for (a) the full exit mass flow rate history, (b) the last two overlapping periods of exit mass flow rate, (c) the full RMS residual history, and (d) the last two overlapping periods of the RMS residuals.

Continued
Figure 83 Continued

(c)

(d)
The unsteady, harmonic solutions are also initialized from the converged steady solution similar to the phase-lag, unsteady solutions. The convergence history is also reset at the beginning of the solution and thus the RMS residual is reset to zero. For each harmonic solution, three harmonics per periodic perturbation are specified per blade row such that the upstream and/or downstream blade row is passed the first three harmonics from the adjacent blade row as boundary conditions in the frequency domain portion of the problem. The solutions are then set to run for 4000 iterations or until the RMS residuals, inlet mass flow rates, and exit mass flow rates are flat. Figure 84 shows the convergence for the Run 21 unsteady, harmonic solution, which is typical for all harmonic, unsteady solutions. The harmonic solutions were able to achieve an RMS residual of about -1.5 or better which provided an additional orders of magnitude drop in the RMS residuals over the steady solution and the inlet and exit mass flow rates are matched to within 0.02%.

The convergence for each of the different solutions (steady, unsteady, harmonic) that were performed for this research is provided in Appendix E.
Figure 84. Run 21 convergence for the harmonic solution for (a) in exit mass flow rates, and (b) RMS and maximum residuals.
CHAPTER 6

COMPUTATIONAL AND EXPERIMENTAL COMPARISON

The goal of this chapter is to provide a comparison between the computation and experimental results in an effort to benchmark the computational. Such benchmarking will provide confidence in the CFD codes ability to predict temperatures and pressures within the confines of the available rig data and allow for further in-depth flow field analysis using as the computational results.

6.1 Data Reduction and Analysis

From an experimental standpoint, the data is presented in several forms: time-series, time-average, and harmonic amplitude or FFT data. For time-series data with respect to vane passing period (i.e. blade surface, rotor cavity, etc), each vane passage of data over a single revolution was clocked into a similar location (on top of each other). An average time-series for a single vane passage was then constructed from the thirty-eight sets of data. This average vane passage time-series, calculated for each available transducer, was then used to calculate the time-averages, FFTs, and time-series via common methods of data post-processing. This technique of data reduction is known as ensemble averaging. A similar procedure was used to generate the desired time-average, time-series, and FFT data sets with respect to blade passing period (the stationary shroud).
The computational results were extracted using the post-processing software available as part of the FINE/Turbo package which is called CFView. There are two functions within CFView that are available for extracting results from a converged solution; these functions are ProbeXYZ and ProbeIJK. The ProbeXYZ function extracts the desired thermodynamic quantity at the specified Cartesian coordinate within the computational domain and thus the ‘probe’ location is stationary and does not rotate with the rotating domains from an unsteady perspective. On the other hand, the ProbeIJK function extracts the desired thermodynamic quantity at the specified i,j,k indices of the grid and thus the ‘probe’ location rotates with the rotating domains from an unsteady perspective. The ProbeXYZ function was typically used to extract the data from stationary instrumentation locations on the high-pressure vane, the stationary wall of the rotor purge cavity, and from the stationary shroud above the rotor tip. The ProbeIJK function was used to extract data from the rotating instrumentation on the rotor surface, rotor platform, and the rotating side of the rotor purge cavity. For the steady solutions, data is extracted at each instrumentation location. The unsteady solutions require extraction of the data at each time step and have different time steps associated with vane passing and rotor passing frequency. This required the harmonic solutions to be reconstructed into instantaneous time steps based on the harmonics. The reconstruction into the time domain was accomplished using Harmo2Time, a function available with the FINE/Turbo software package and the harmonic solutions were reconstructed into 2736 time steps per complete revolution of the rotor which breaks down into 38 time steps per blade passing and 72 time steps per vane passing which was established from the phase lag solutions in order to maintain consistency. Once extracted, the results in the rotating frame of reference are time-averaged and Fourier decomposed with respect to vane passing frequency and stationary frame of reference results were time-averaged and Fourier decomposed with respect to the rotor passing frequency.

Range bars, although available, are not shown in any of the following comparisons on the measured values. One could argue that similar set of range bars could be produced from the analytical results, but this would require detailed measurements of not only the thermodynamic properties (which for the most part is available) but also all of the geometry as well. While this may be a feasible approach once all of the
detailed geometric measurements are incorporated into the computer-aided design (CAD) models, the amount of time and effort required to pre-process, run, and post-process the results would be very extensive and not practical during the design process due to both time and budget constraints. For this very reason, the intent is to show average-to-average comparisons between the measurements and the analytical results as this information would be available to the design engineer very readily without further interpretation.

All pressure and temperature values are shown normalized to the rake averaged inlet total pressure and total temperature for the time-average value comparisons. This provides a good avenue for comparing results from each of the different runs. If inlet total pressure and total pressure are 400 kPa and 500° K respectively (which is within reason for all runs shown), 0.01 or 1% of inlet average difference between measured and predicted is 4 kPa (0.58 psi) and 5° K (9° R). Time-series comparisons are shown for the Kulite locations only as the “as installed” frequency response of these transducers is on the order of 125 kHz, which provides great resolution at the speeds and blade/vane passing frequencies within this experiment. The thermocouples will provide a great measure of the time-average behavior, but the frequency response of these butt-welded devices is on the order of 1 kHz, which is too slow to resolve the time-accurate flow field. The Kulites will be shown as a fluctuation about the mean and calculated as such:

\[ P_i' = \frac{P_i - \bar{P}}{\bar{P}} \quad (6.1) \]

where \( P_i' \) is the instantaneous pressure and \( \bar{P} \) is the time-average pressure calculated from the ensemble average. The time-series data is shown best using this method as it collapses everything around a common mean of zero and thus comparisons between measured and computed can be easily made for the fluctuating components. Each set of comparisons show the three blade row steady (labeled “Steady” and harmonic unsteady (labeled as “Harmonic”) solutions along with the measurements (labeled either “Data” or “Rig”). The run 21 comparisons shown in the following sections will also incorporate the phase-lag unsteady solution (labeled as “Phase-lag”) when the predicted result is available.

241
6.2 Low-Pressure Turbine Vane Exit Plane

The measured total temperature and total pressure from the inlet rakes upstream of the high-pressure vane were used as inlet boundary conditions for the FINE/Turbo predictions along with an exit static pressure measured at the exit rake plane downstream of the low-pressure vane. The exit rakes also measure total pressure and total temperature at multiple spanwise locations thus comparisons for the steady and harmonic unsteady three blade row cases to the experimental measurements were possible to see how FINE/Turbo captures the measured losses throughout the turbine. Table 10 below provides a comparison between the experimental rig and the computational results for the important parameters that dictate the operating point for the turbine, namely total temperature ratio, total pressure ratio, total-to-static pressure ratio, correct speed and corrected flow for all cases analyzed. The difference between the rig and the computational results is shown for each case as a percent difference based on the rig measurements. A majority of the parameters for all cases are less than 1% different with the correct speed and total-to-static pressure ratio being less than 0.5% different. Runs 22 and 43 were off the most as compared to the measurements and in particular with the total pressure ratio and correct flow rates which were 4.88% and 2% low, respectively. Many of the differences can be explained by understanding that the computational results are an average of the entire inlet and exit surface while the rig measurements are an average over 3 to 5 points in the spanwise direction at 1 or 2 tangential locations. Some deviation would be anticipated given the methodology behind these comparisons in Table 10. Overall, the comparison does show that FINE/Turbo does a very good job at capturing losses throughout the calculations and confirms that the computational results are on or close to the same operating point as the experiments.
<table>
<thead>
<tr>
<th>Run #</th>
<th>TT Ratio</th>
<th>PT-to-PS Ratio</th>
<th>PT-Ratio</th>
<th>Corrected Flow (kg/sec)</th>
<th>Corrected Speed (RPM)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Experimenta</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Values</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FINE/Turbo Steady</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>3-Row</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>% Difference</td>
</tr>
<tr>
<td>21</td>
<td>1.405</td>
<td>5.33</td>
<td>4.6</td>
<td>4.610</td>
<td>6588.78</td>
</tr>
<tr>
<td>21</td>
<td>1.406</td>
<td>5.331</td>
<td>4.480</td>
<td>4.574</td>
<td>6587.91</td>
</tr>
<tr>
<td>22</td>
<td>1.474</td>
<td>5.636</td>
<td>4.585</td>
<td>4.839</td>
<td>6013.10</td>
</tr>
<tr>
<td>22</td>
<td>1.485</td>
<td>5.631</td>
<td>4.664</td>
<td>4.603</td>
<td>6027.96</td>
</tr>
<tr>
<td>28</td>
<td>1.572</td>
<td>5.613</td>
<td>4.648</td>
<td>4.75</td>
<td>6289.22</td>
</tr>
<tr>
<td>28</td>
<td>1.543</td>
<td>5.587</td>
<td>4.650</td>
<td>4.714</td>
<td>6320.18</td>
</tr>
<tr>
<td>33</td>
<td>1.557</td>
<td>5.877</td>
<td>4.772</td>
<td>4.82</td>
<td>6275.89</td>
</tr>
<tr>
<td>33</td>
<td>1.548</td>
<td>5.849</td>
<td>4.767</td>
<td>4.705</td>
<td>6292.04</td>
</tr>
<tr>
<td>43</td>
<td>1.498</td>
<td>5.938</td>
<td>4.699</td>
<td>4.69</td>
<td>6400.51</td>
</tr>
<tr>
<td>43</td>
<td>1.519</td>
<td>5.935</td>
<td>4.829</td>
<td>4.598</td>
<td>6402.63</td>
</tr>
</tbody>
</table>

Table 10. Comparison the measured rig and the FINE/Turbo analytically predicted operating conditions for all runs analyzed

The exit rakes also provide picture of the spanwise profiles for the total temperature and total pressure. Two locations of total temperature measurements at 5 spanwise positions each were available from the rig and are shown in Figure 85 compared to the tangentially mass-averaged exit total temperature from the
steady and unsteady harmonic solutions. Overall, the comparisons show that the computational results produced by FINE/Turbo capture the spanwise profile of total temperature very well for each case analyzed. The results from Runs 22, 28, and 33 do show an over-prediction of the total temperature in the upper 50% of span was compared to the data with Run 22 having the largest discrepancy which is predicted to be 2% of inlet total temperature higher than the measurements suggest, but the trend is still captured very well. Run 22 is also the only case of the five analyzed that shows some notable difference in the lower 50% of span between the steady and unsteady harmonic solution; however, differences are minimal but this does show that the mean flow of the unsteady harmonic solutions are altered somewhat by the passing of the deterministic stresses between blade rows. Overall, these comparisons provide confidence in the assumptions discussed throughout Chapter 5 and the results within the turbine flow path predicted by FINE/Turbo.

Figure 85. Comparison of time-average total temperature profiles between the rig exit rakes and the FINE/Turbo steady and unsteady circumferentially averaged results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43

244
Figure 85 Continued

(a)

(b)

(c) Continued
Figure 85 Continued

(d)

(e)
As described in 5.4, the cases analyzed were chosen purposely to provide a comparison between cooled and uncooled runs from the experimental matrix of conditions. Figure 86(a) shows the comparison between Run 22 and Run 28 for an uncooled and cooled elevated radially varying inlet total temperature profile. There is a 4.5% of inlet total temperature delta between the two runs from 90% to about 60% span, which then tapers to a 2% of inlet total temperature difference at 13% span while the profiles are nearly identical. This temperature difference represents the impact of the cooling flow addition on the high-pressure turbine vane. The FINE/Turbo predictions also show this same difference between the cooled and uncooled results thus providing additional confidence in the predictions. Figure 86(b) shows the comparison between Run 33 and Run 43, which are the cooled and uncooled cases with a flat, elevated inlet total temperature profile. These comparisons show a much flatter difference between the cooled and uncooled results, which are different by about 3% of inlet total temperature at 90% span and tapers to about 2% of inlet total temperature at 13% span again with very similar profile shapes. Once again, the analytical predictions also show the same trends from cooled to uncooled results with some slight differences noted in the upper 5% and the lower 5% span where Run 43 has more curvature in the profile than Run 33. This is due to the inlet total-to-wall temperature ratio being slightly higher for Run 43 than Run 33, which is shown in Table 9 of Chapter 4. Apart from this difference, the high-pressure vane cooling flow and the purge flow only affect the exit profile by decreasing the total temperature and does not affect the shape of the profile to any great extent. This is noted for both the flat and the radially cases.
Figure 86. Comparison of the rig and analytical predicted cooled and uncooled exit total temperature profiles for (a) Run 22 and 28, (b) Run 33 and 43
Another interesting note about the exit profiles is the overall shape as compared to the temperature profile at the inlet to the high-pressure vane through the use of a maximum to average profile ratio. A comparison of the inlet and analytical exit total maximum-to-average values are shown in Table 11. For the flat profile cases, Run 33 and 43, the profile tends to become slightly more radial noted by the increase in the maximum-to-average values whereas the radial profile cases, Run 22 and 28, tend to become less radial. This is due to the secondary flows through the turbine helping to migrate the high temperature air into the lower temperature areas.

<table>
<thead>
<tr>
<th>Run</th>
<th>Max-to-Avg Inlet TT (K)</th>
<th>Max-to-Avg Exit TT (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>22</td>
<td>1.11</td>
<td>1.083</td>
</tr>
<tr>
<td>28</td>
<td>1.16</td>
<td>1.072</td>
</tr>
<tr>
<td>33</td>
<td>1.02</td>
<td>1.028</td>
</tr>
<tr>
<td>43</td>
<td>1.02</td>
<td>1.035</td>
</tr>
</tbody>
</table>

Table 11. Maximum-to-average comparison between the inlet and exit total temperature for the analytical results

Another important aspect of the cooled versus uncooled is the realized temperature difference and the abilities of FINE/Turbo for predicting the temperature difference. Figure 87 shows a comparison of the measured temperature difference between the uncooled and cooled solutions in comparison with the analytical solution temperature differences. The run 22 to run 28 differences in Figure 87(a) are predicted about 0.5% to 0.8% of inlet total temperature low compared to the measured difference and similarly in Figure 87(b) the difference between run 43 and run 33 are predicted about 0.4% to 1.4% of inlet total temperature low. For both comparisons, the biggest delta between predicted and measured occurs in the upper 30% span near the tip. Overall, FINE/Turbo is capturing the differences very well.
Figure 87. Comparison of the analytical and measured uncooled-to-cooled temperature difference for (a) run 22 and run 28 and (b) run 33 and run 43.
The low-pressure exit vane total pressure profile comparisons between the rig measured and the analytical steady and unsteady time-averaged results is shown in Figure 88 below. Only three of the five spanwise locations on the exit total pressure rake were available for each of the five runs analyzed. This limits the ability to compare the actual profile shape but does allow for a comparison of magnitude. In the case of all the comparisons, Run 21 has the highest discrepancy at about 46% span between measured and predicted values of 1.7% of inlet total pressure. The remaining runs are less than 1% of inlet total pressure different. This provides additional confidence that FINE/Turbo can predict the behavior within the turbine flow path correctly when the proper boundary conditions are applied.

![Graph showing total pressure profile comparison](image)

**Figure 88.** Comparison of time-average total pressure profiles between the rig exit rakes and the FINE/Turbo steady and unsteady circumferentially averaged profile for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43

251
Figure 88 Continued

(b)

(c)  

Continued
From a total temperature exit profile perspective, the impact of the cooling flow could clearly be seen from an overall reduction of the level of the profile from uncooled to cooled. This same reduction can also be
seen in the exit area averaged values shown in Table 10. When the total pressure profiles radial and flat profile cases are plotted together as in Figure 89 there is little difference between the cooled and uncooled cases in overall profile shape, but run 43 is notably down in magnitude from the other three profiles. Both the data and the computational results support this result. However, there is a slight offset between the radial and flat profile cases most notably at about 50% span that carries up to about 80% span. A slight reduction in total pressure can be seen near the hub in Figure 89(a) for Run 28 compared to Run 22 which the data also supports. This difference is most likely due to the purge flow and will be explored in more detail later. In addition, there is a slight offset between the between Run 33 and Run 43 in Figure 89(b).

(a)  
Continued

Figure 89. Comparison of the rig and analytical predicted cooled and uncooled exit total pressure profiles for (a) Run 22 and Run 28, (b) Run 33 and Run 43, and (c) all runs
Overall, the comparisons between the computational and experimental results at the low-pressure vane exit are very good with only slight differences noted with the Run 21 total temperature profiles. Some of the exit area averaged parameters also show some differences between measured and predicted, but this is attributed to the limited data available from the rig for averaging as compared to the computational models which provide full tangential and spanwise distributions of thermodynamic quantities. The comparisons provide confidence in the FINE/Turbo’s abilities to capture both temperature and pressure losses throughout the turbine with the assumptions that have been applied to the models.

6.3 High-Pressure Turbine Vane

The only available data comparisons for the high-pressure vane are the static pressure measurements taken circumferentially downstream of the trailing edge. This region is important from a comparison standpoint as this pressure field is one of the influencing factors on the purge flow cavity aerodynamics.

6.3.1 Trailing Edge Static Pressure

Two Kulites were placed just downstream of the vane trailing edge at the hub in three vane passages around the annulus in an effort to measure the variation of static pressure within a vane passage. Only two gage locations were available, one directly behind the trailing edge at 0% of the vane passage and the other at 50% of the vane passage or half way between two adjacent vanes. These measurements were not meant to establish the entire trialing edge but more as a diagnostic measurement to make sure the computational models were matching closely to the measurements. The comparisons for the vane trialing edge locations
are provided in Figure 90 for all five runs. The computational results are shown over the entire vane passage with the measurements at the proper location. The two unsteady models and the steady model for Run 21 compare very well to the measured data at both locations falling somewhere between the variation from gage location to gage location. Also interesting to note is the variation within the measured data, which is 3-5% of inlet total pressure depending on the location. The two unsteady models are calculating very similar results while the steady solution shows its biggest difference near the two peaks. Very similar results are seen for Run 28 in Figure 90(c) however the other three comparisons show that the computational models are predicting higher static pressure at the hub surface than is measured by about 3-4% of inlet total pressure to the average of the three gages at each circumferential location. It is difficult to draw conclusions as to why the computational models are predicting high in comparison to the measurements given the minimal amount of information provided by the rig. It could be due to several factors such as runs 22, 33, and 43 were low on corrected flow rate by 2-5%. Run 21 and 28 are less than 1% different to the measured values for corrected flow rate and show the best comparison to the measured data such that this is the most likely culprit as the corrected flow would govern exit Mach numbers, shock location, and shock strength on the high-pressure vane. With a choked vane such as this one, only so much flow can pass through for a given temperature, pressure, and area and with the area fixed as is the case with this rig, it narrows down to temperature and pressure differences.
Figure 90. Comparison of the trailing edge hub surface time-averaged static pressure for the rig and FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43
Figure 90 Continued

(c)

(d) Continued
In addition to the time-average comparisons above, the time-series comparisons are shown in Figure 91 through Figure 95 for run 21, 22, 28, 33, and 43, respectively. For each of these comparisons, the PV110/PV112/PV114 location is just downstream of the vane trailing edge and the PV111/PV113/PV115 location is at 50% of the vane passage at the same axial location. Figure 91 shows the time-series comparisons for both transducer locations with three transducers per location. In Figure 91(a), the trailing edge location, the harmonic unsteady solution predicts the trends and the magnitude of the experimental data quite well. The harmonic unsteady solution does show two valleys at about 45% and 90% of blade passing while the measurements only shows a single valley followed by almost a flat section from 70% to 100% of blade passing. At the mid passage location in Figure 91(b), the harmonic unsteady solution predicts the PV113 very well from a trend and magnitude perspective, but it under-predicts the overall levels of the other two gage locations by a factor of 3-4. This difference between transducer locations is most likely due to manufacturing tolerances from passage to passage which will cause each throat to be slightly different and thus cause a variation in passage flow rate, velocities, Mach Numbers, etc around the annulus and thus producing slightly different static pressure fields.
Figure 91. Comparison of the Run 21 trailing edge hub surface time-accurate static pressure for the
rig and FINE/Turbo unsteady results for (a) PV110/PV112/PV114 gage location and, (b)
PV111/PV113/PV115 gage location
Figure 91 Continued

Very similar results were achieved with the comparisons for run 22, 28, 33, and 43 shown in Figure 92 through Figure 95, respectively and will not be discussed individually for all runs. For each of the runs, the computational model predicts magnitudes and, for the most part, trends within the measurements fairly well. The transducer-to-transducer variability and hence passage-to-passage variability is present for both locations behind the trailing edge at all conditions shown here of which the computational model was never intended to predict. The uncooled runs (21, 22 and 43) do show additional unsteadiness as compared to the cooled runs (28 and 33), but the time-accurate behavior at both locations behind the trailing edge is very similar among all five runs.
Figure 92. Comparison of the Run 22 trailing edge hub surface time-accurate static pressure for the rig and FINE/Turbo unsteady results for (a) PV110/PV112/PV114 gage location and, (b) PV111/PV113/PV115 gage location
Figure 93. Comparison of the Run 28 trailing edge hub surface time-accurate static pressure for the rig and FINE/Turbo unsteady results for (a) PV110/PV112/PV114 gage location and, (b) PV111/PV113/PV115 gage location.
Figure 94. Comparison of the Run 33 trailing edge hub surface time-accurate static pressure for the rig and FINE/Turbo unsteady results for (a) PV110/PV112/PV114 gage location and, (b) PV111/PV113/PV115 gage location
Figure 95. Comparison of the Run 43 trailing edge hub surface time-accurate static pressure for the rig and FINE/Turbo unsteady results for (a) PV110/PV112/PV114 gage location and, (b) PV111/PV113/PV115 gage location.
The trailing edge static pressure field at or near the hub has been shown by Bohn and Wolff [13] to be critical to the cavity behavior and thus must be captured correctly to ensure proper resolution of the behavior. These comparisons show that the computational model is predicting not only the time-average, but the time-accurate static pressure field near the hub with fairly good agreement to the measurements providing further confidence in the model and assumptions.

6.4 High-Pressure Turbine Rotor

The high-pressure rotor incorporates three Kulite transducer locations at the mid-span of the blade surface, leading edge thermocouples, which were used to resolve the incoming temperature profile, platform thermocouples, and a single Kulite location on the stationary shroud located above the blade. The shroud and blade surface Kulites are incorporated to provide a diagnostic measure as to determine where the predicted operating point of the turbine is relative to the experimental results, but it is difficult for these few measurements to truly validate the operating point due to the potential variability in the data.

The leading edge thermocouples, on the other hand, will provide an excellent opportunity to see if FINE/Turbo can propagate the inlet total temperature profile correctly through the high-pressure vane passage. In addition, these measurements will also provide the first look at how well the film cooling is modeled on the high-pressure vane for Runs 28 and 33 as the film cooling patterns were provided from an outside source. In addition, the platform thermocouples also provide a fair amount of detail in the thermal boundary layer and propagation of the temperature distributions downstream.
6.4.1 Mid-Span Blade Surface Static Pressure

One of the diagnostic measurements consisted of three Kulite transducer locations at 50% span on the high-pressure rotor surface. Two of these locations are on the pressure side of the blade while the third location is on the suction side of the blade. While these three measurements combined will not provide total reassurance that the blade loading is correct, it will provide an indication if the incidence angles are within reason as compared to the experiments. Only one run, Run 28, had all three transducer locations functioning, but the other four runs had both a suction surface and a pressure surface location operating regardless. Figure 96 shows the comparison of the full blade loading from the computations to the available measurements. Run 21 in Figure 96(a) compares well with the data from a harmonic and phase-lag perspective in which the suction surface gage actually sits between the two predictions despite the harmonic solution showing slightly more negative incidence on the leading edge such that the stagnation point on the leading edge of the blade is biased toward the pressure surface. This is usually evident by the rounded pressure profile near the leading edge (wetted distance of zero) and the quick drop in static pressure on the suction surface in the first 5-10% of the wetted distance along the blade. Similarly in Figure 96(b), for Run 22 the suction surface gage is biased more towards the steady solution, which again the harmonic solution shows more negative incidence on the blade leading edge. Both solutions over-predicted the pressure surface gage by less than 2% of inlet total pressure. For Runs 28 and 33, the pressure surface locations are over-predicted by about 1-2% of inlet total pressure while the suction surface gages tend to fall between the steady and harmonic solutions with both being slightly more biased to the steady solution. Run 43 in Figure 96(e) shows similar trends to Run 22 in that the harmonic solution predicts more negative incidence on the blade leading edge and over-predicts both gage locations.
Figure 96. Comparison of time-average rotor surface pressure between the rig and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43.
Overall, the comparisons are fairly good agreement for Runs 21, 28 and 33 while Runs 22 and 43 tend to over-predict the measurements and suggest that there is more negative incidence on the blade leading edge. However, due to the minimal amount of data available the actual blade loading cannot be confirmed as predicted by the computational models. It can only be assumed that the results are within an acceptable range of the experimental data and that based on the in section 4.3.3 that FINE/Turbo can do an excellent job at predicting the rotor surface pressure when given the appropriate boundary conditions.

6.4.2 Stationary Shroud Static Pressure

The experimental rig originally incorporated 4 stationary mounted Kulites in the shroud above the rotating blade, but due to transducer mortality as the test sequence progressed only a single shroud Kulite remained throughout all of the runs. This single location is about 30% of the axial wetted distance of the high-
pressure blade aft of the leading edge such that the transducer “sees” the blade tips as they pass by. Since there is only a single transducer, the comparisons are more of a diagnostic much like the high-pressure blade surface Kulites to ensure that the main gas path conditions away from the cavity are representative of the rig.

The time-average comparisons for the steady and harmonic unsteady computational models are shown in Figure 97. For runs 21, 28, and 33 the predicted time-average static pressure from the harmonic unsteady solution is well within 1% of inlet total pressure to the measured data. For runs 28 and 33, it is slightly over-predicted while for run 21 it is under-predicted for both models. The steady model does over-predict the time-average static pressure for run 28 and 33 but about 1% and is still in fairly good agreement. This is most likely due to missing the effects of the high-pressure vane wake propagation, which causes a circumferential variation in time-average pressure on the shroud. For the two uncooled, elevated temperature runs (run22 and 43) the time-average static pressure is over-predicted by 3% of inlet total pressure for both the harmonic unsteady and the steady solution. Only one potential issue comes to mind for the run 22 and 43 time-average over-prediction and that is the assumption that the tip clearance remains constant between all runs at 2.15% of blade height. The rotor physical speed only changes about +/− 500 RPM from runs 28 and 33 to runs 22 and 43 and thus this seems unlikely given that for run 22 the physical speed is slower by about 500 RPM and for run 43 the physical speed is higher by about 500 RPM. This delta would suggest tighter tip clearances for run 43, which would tend to decrease the static pressure and more open clearances for run 22, which would tend to increase the static pressure. Applying this logic, the data suggests that both run 22 and run 43 would have tighter tip clearances than used in the computation, which does not necessarily agree with the physical speed trends.
Figure 97. Comparison of the time-average static pressure between the measured and predicted results for the PS2 stationary shroud gage location

The time-accurate comparison for the PS2 shroud gage is shown in Figure 98. The comparisons for all runs looks very similar in that for the most part the trends of the data are captured quite well except the sharpness of the peak near 70% of blade passage. The magnitude and the lack of sharpness shown in the measurements is not predicted by the harmonic unsteady solution unlike the comparisons shown for the uncooled turbine in Chapter 4.3.3 where the peaks are seen in both the data and the unsteady solutions and compare very well in both trend and magnitude.
Figure 98. Comparison of the PS2 stationary shroud gage time-accurate static pressure between the measured and predicted results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43
Figure 98 Continued

(c)

(d)

Continued
In looking at some of the diagnostic plots that are generated during the ensemble averaging process, it was noticed that the variability from peak-to-peak was only about 2.5% of inlet total pressure across all 72 passages stacked on top of each other. This plot can be seen in Figure 99 for run 28 PS2 gage location. While there is some variability from passage to passage in the measured data, it is certainly not enough to cover the entire miss at the peak location. While tip clearance will also play a factor for the time-series plots, this was again ruled out as the physical speed differences were not enough to cause large changes and the time-average static pressure matched very well on runs 21, 28, and 33. One additional thought could be the film cooling flow pattern applied to the high-pressure vane and the ingestion on the vane pressure surface and subsequent ejection of main stream gas through the cooling holes on the vane suction surface. At this time, there is not solid evidence to point towards the film cooling holes and source terms applied in the computational model, and thus is pure speculation.
Overall, the shroud location predictions do match very well with the measured data which suggests that the computational model flow conditions in the main stream gas away from the purge cavity are matching well with the experimental rig as executed.

### 6.4.3 Blade Leading Edge Total Temperature

As discussed in Chapter 3, thermocouples were installed in the leading edge of several blades in order to capture the temperature profile entering the rotor. The thermocouples were of a shrouded configuration and proved to be effective at capturing the relative total temperature profile (rotating frame of reference). Figure 100 shows the comparisons between the measured and predicted relative total temperature profiles. For Run 21 in Figure 100(a), the profile is predicted very well above 40% span however there is a drop in
the temperature at 35% span that the computational results do not predict. This loss was also noted by Mathison [68] and was originally thought to be due to the absence of cooling flow on the high-pressure vane which already contained the cooling hole pattern. The main stream flow was thought to be ingested at the leading edge into the high-pressure vane internal cavity and then ejected out back out of the cooling holes further downstream and thus causing a temperature loss around 40% and below. Run 22 and Run 43 in Figure 100(b) and (e) also show a similar feature although it is much more localized and has shifted up between 40 and 50% of span, also noted by Mathison [68]. Looking back at the high-pressure vane cooling circuit static pressure and static temperature gages installed in the rig, this theory seems to be accurate as the static pressure gages are seeing nearly 99% of the inlet total pressure and about 65% of the inlet total temperature inside the cooling circuit. This definitely points to ingestion into the high-pressure vane cooling cavity. While the original theory seems appropriate and has some evidence of occurring based on the instrumentation, closer inspection of the cooled solutions shows a similar drop in temperature in Figure 100(c) and (d) between 40% and 50% which tends to hint at a different cause.
Figure 100. Comparison of the time-average total relative temperature profiles between the rig rotor leading edge thermocouples and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43
Figure 100 Continued

(c) 

(d) Continued
Based on these new findings, several theories now present themselves as to what may be happening at mid-span for the relative total temperature profiles:

1. Over-sized cooling holes on the high-pressure vane surface at mid-span due to a manufacturing error and thus flowing much higher than holes at the other spans or the holes in the upper and lower spans could be under-sized and thus under flowing

2. Vane surface geometry issue such that the high-pressure vane surface contour is not to print and causing under-turning at the exit of the vane

3. The internal plenums that feed the film cooling holes within the high-pressure vane are not being pressurized as indicated by the analysis that produce the cooling hole flow distributions

Most likely theories one and two above are not the case. First it is highly unlikely that while drilling the film cooling hole pattern the manufacturing only managed to oversize a very select group of holes and did not to affect the others. And from a surface contour standpoint, the anomaly would have to be significant enough to cause such a drop that it would have been visible when the rig was assembled. The most likely
issue is that the internal plenums are not be pressurized as was originally assumed in the analysis that
design the film cooling pattern for the cooled runs in addition to the original assumption of main stream gas
ingestion for the uncooled runs. From an experimental standpoint, verification would be accomplished
through installing additional static pressure and static temperature gages throughout the internal plenums to
capture the distribution of pressure and temperature. Analytically speaking, a vane grid that discretizes
each of the cooling holes and the internal plenums in order to eliminate the source terms and allow the
cooling flow an plenum cavities to interact directly with the main stream gas flow could be produced.
Source terms do not allow for such interaction and are fixed in temperature, flow rate, and direction. The
grid would need to be built such that it takes advantage of the cooling circuit measurements and eliminate
as much uncertainty as possible. This is a daunting task and will not be under-taken within the scope of
this research.

Apart from this drop in temperature that the predictions have missed, the overall leading edge temperature
profiles have been captured very well with the computational predictions. The largest difference between
the predictions and the experimental data actually occurs at the previously mentioned temperature drops
near mid-span which at its peak are different by only 4% of inlet total temperature. The next location of
significance is near the outer shroud where the harmonic solution tends to over-predict the temperature for
Runs 28 and 33 by about 2% of inlet total temperature. This is most likely due to an issue with the applied
cooling hole pattern and cooling flow distribution as both of these cases incorporate the film cooling on the
high-pressure vane however from them experimental data it is difficult to tell what the issue may be.
Apart from these issues noted, the differences for all other locations are less than 1% of inlet total
temperature which is excellent agreement given all of the uncertainty around film cooling flow
distributions and potential ingestion into the high-pressure vane plenums for the uncooled cases.

A comparison of the uncooled (run 22 and run 43) and cooled (run 28 and run 33) profiles for both the
radial and flat inlet temperature profiles is shown in Figure 101 below. There is a noticeable offset between
the cooled runs and uncooled runs with the cooled runs being anywhere from 1% to 9% of inlet total
temperature lower due to the high-pressure vane film cooling introduction. The offset is obviously seen for
both the analytical and the measured data and for both comparisons the difference is the largest at tip
around 95% span and the smallest around 10% span. It is also noted that in both cases the cooled analytical
solution over-predicts temperatures by about 3% of inlet total temperature in the 95% span region where as
the uncooled analytical solutions predict the temperature to within 0.3% of the measured data. This also
suggests an issue with the high-pressure vane cooling hole pattern as applied in the analysis.

Figure 101. Comparison of the cooled and uncooled leading edge relative total temperature profiles
for (a) the radial profile runs 22 and 28 and (b) the flat inlet profile runs 43 and 33
Based on the results shown in Figure 101, it was also desired to look at the temperature differences between the cooled and uncooled solutions as done with the low-reassure vane exit total temperature profiles. The run 22 to run 28 (radial inlet temperature profile) difference is predicted about 1% to 1.5% of inlet total temperature low and the run 43 to run 33 (flat inlet temperature profile) difference being 0.6% to 3.5% of inlet total temperature low with respect to the measured data. The largest difference between predicted and measured for both comparisons is in the tip region (similar to the exit profile comparisons in section 6.2).
Figure 102. Predicted and measured uncooled-to-cooled temperature difference for (a) run 22 to run 28 and (b) run 43 to run 33.
6.4.4 Blade Platform Static Temperature

The platform static temperature measurements incorporate multiple extracts from the computational results due to the uncertainty in the radial positioning as discussed in section 3.2.4. For each solution, the static temperatures are shown for both the pre-test positioning (labeled as “Pre” with dashed lines) and the post-test inspection positioning (labeled as “Post” with solid lines). The radial positioning of the thermocouples, as will be seen, is vitally important to the comparisons as the thermal boundary layer has a very steep gradient over a small radial height.

For reference, Figure 103 provides the platform positions and gage locations as installed in the experimental rig. From a positioning standpoint on the platform, there are five thermocouples inline axially with the leading edge (TRP 38 and TRP59 are in the same position), one thermocouple on the rear angel wing, and three gages near the passage shock on the rotor (TRP46, TRP47 and TRP61 are in the same position).
The comparisons between the computational and experimental data are provided in Figure 104(a) through (e) for Runs 21, 22, 28, 33, and 43 respectively. Run 21 in Figure 104(a) also incorporates the phase-lag solution along with the steady and harmonic unsteady solutions. For Run 21, both the trends and the absolute level of temperature are captured very well by all three models and with the exception of a few gage locations the three solutions produce very similar trends with some difference in absolute levels. At the TRP 41 location, it is interesting to see that the phase-lag solution from a post-inspection positioning standpoint over-predicts the static temperature by about 1% with the steady solution about 6% high while the harmonic solution is off almost 15% of inlet total temperature. At the TRP57 location on the rear angle, the harmonic and steady solution predict a 7% higher temperature while the phase-lag predicts about 5% lower. All other locations are within 5% of inlet total temperature to the measured data.

The comparisons for Run 22 is shown in Figure 104(b) and for the leading edge gages both the steady and harmonic solution capture trends and absolute levels very well within 2% of inlet total temperature. Once past the leading edge gages, the harmonic and steady solutions begin to diverge from the data and each
other under-predicted the temperature by as much as 4-6% of inlet total temperature respectively at gage TRP47. The Run 28 comparison, in Figure 104(c), shows the steady solution capturing the leading edge temperatures much better than the harmonic solution but once past the leading edge the trend is reversed and the harmonic solution clearly matches better to the data. For Runs 33 and 43, leading edge trends from gage to gage are once again captured well while the absolute levels are predicted low 4-20% with the worst location being TRP61 for both runs. One difference between these two cases and Runs 22 and 28 is that the rotor inlet temperature profile, provided in Figure 100, at 5% span is about 80-83% of inlet total temperature while Runs 22 and 28 are 65% and 70% respectively. The isothermal wall boundary conditions are very similar at around 53-58% for these four cases such that the thermal boundary layers for Runs 33 and 43 have a much higher gradient than Runs 22 and 28 and thus gage height positioning becomes very critical to an accurate comparison.

![Graph showing static temperature data](image)

**Figure 104.** Comparison of time-average static temperature on the rotor platform surface between the rig thermocouples and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43

288
Figure 104 Continued

(b)

(c)  

Continued
Figure 104 Continued

(d)

(e)
The other piece of the puzzle to bear in mind is the isothermal wall temperatures applied in the computational models. There are no direct measurements of the rotor platform surface metal temperatures and boundary conditions were set based on internal RTD measurements of the blade. While for the most part this has proven to be an effective method for supplying boundary conditions, there is still some uncertainty. Using the heat flux gage underside temperature measurements (most representative of the metal temperature) on the high-pressure vane hub and airfoil surface, it was found that for runs 22, 28, 33, and 43 the RTD estimated wall temperatures were low in comparison by anywhere from 40-100° K. This was initially being studied to determine the effect of the high-pressure vane hub isothermal wall boundary conditions on the predicted cavity temperatures for both the stationary and rotating walls which will be discussed later. Subsequently, and additional set of steady and harmonic unsteady analyses for runs 28 and 43 were conducted in order to quantify the potential impact for the cavity which also provided an opportunity to quantify the impact for the rotor platform temperatures. Figure 105 shows the difference between the nominal and the elevated high-pressure vane isothermal boundary conditions for which only the post-test inspection locations are plotted. For run 28, the wall temperatures were raised about 8% of inlet total temperature while for run 43 the vane temperatures were raised about 17% of inlet total temperature. There is very little difference in the predicted rotor platform temperatures which is similar to Mathison’s [68] conclusions as well. For run 43, the elevated high-pressure vane hub temperature does have an effect on the rotor platform gages but only at a single gage location. The most likely root cause of the overall temperature under-predictions is the platform isothermal wall temperatures.
Figure 105. Comparison of the nominal and elevated high-pressure vane isothermal wall boundary condition effect on the predicted rotor platform temperatures for (a) Run 28 and (b) Run 43.
Apart from the under-predicted temperatures on Runs 33 and 43, the remainders of the comparison are very good given the level of assumptions placed on the isothermal boundary conditions for the high-pressure blade. The comparisons also look particularly good at the leading edge locations. The under and over prediction of the temperatures, though, is most likely the effect of two things: the actual height of the thermocouple and the temperature of the platform metal surface during the instrumentation window. The height of the thermocouples was related to an epoxy failure used to hold the thermocouples in position with very little that could have been done to prevent such a failure. The platform surface temperatures could be measured during the time window using thermocouples strategically place about the platform such that either an average or a distribution of temperatures could be established just be the cooling flow introduction into the rig. However, given the amount of instrumentation already incorporated into this experiment it would be quite difficult to add more thermocouples to the mix.

6.5 High-Pressure Turbine Vane/Blade Purge Cavity

The high-pressure vane/rotor cavity was instrumented with both Kulites and thermocouples above and below the angel wings on the stationary and rotating sides of the cavity walls. The stationary side of the cavity has two Kulite locations, one above the lower stationary angel wing and one below with two different vane segments instrumented (total of four gages). These gage locations can be seen in Figure 106 below in the R-Z plane of the rig. In addition to the Kulites, a thermocouple is placed above the lower stationary angel wing one-quarter of a vane pitch away from the Kulites with three thermocouples in total on three different vane segments. On the rotating side of the vane cavity, five Kulites and five thermocouples were originally positioned above and below the angel wing however only four of each survived through the test matrix. Figure 107 shows the thermocouple locations in the R-Z plane of the rig. The four surviving Kulites include two above the angel wing, spaced 40% of the rotor pitch apart starting at 10% of the rotor pitch and two Kulite below the angel wing positioned at 30% and 70% of the blade pitch.
The thermocouples were positioned half way between each Kulite both above and below the angel wing with the surviving gages at 30% and 70% of the rotor pitch above the angel wing and 10% and 50% of rotor pitch below the angel wing. The temperature and pressure data acquired from the rig will provide some insight into how well the computational models are capturing both the time-average and time-accurate flow field within angel wing region.

**Figure 106. Stationary and rotating Kulite locations in the purge flow cavity (Not to Scale)**
6.5.1 Stationary Static Pressure

The Kulite locations on the stationary side of the high-pressure vane/rotor cavity are shown in Figure 108 below for the five runs of interest. For each comparison, PV104 and PV105 are located above the lower angel and PV107 and PV108 are located below the lower angel wing. Run 21, shown in Figure 108(a), provides a comparison of the steady, harmonic unsteady, and the phase-lag unsteady models with the data. Above the lower angel wing, the computational models agree very well with the two gage locations falling between the two data points while below the lower angel wing the computational results are over-predicted by about 5% of let total pressure. A similar result is seen for run 22 and 33 while run 43 is over-predicted for both gage locations by about 5% of inlet total pressure. For these four runs, both the upper and lower angel wing positions are predicted to be nearly identical in absolute pressure level while the rig measurements suggest otherwise. For run 28, in Figure 108(c), the rig measurements show both locations
to be nearly similar in absolute pressure level with the computational models predicting similar results all within about 2% of inlet total pressure to the rig.

Figure 108. Comparison of stationary cavity time-average static pressure locations between the rig and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43
Figure 108 Continued

(b)

(c) Continued
The time-series comparisons for all runs is provided in Figure 109 through Figure 113 that shows two gage locations for the stationary part of the cavity: the PV104/PV105 location which is located above the lower angel wing and the PV107/PV108 location which is below the lower angel wing as shown in Figure 106 above. In each of the PV104/PV105 gage location comparisons for all runs shown below, the computational model predicts the magnitudes within less than 1% peak of inlet total pressure and captures the time-accurate trends very well with the exception of run 43 in Figure 113(a) which is off by about 2% of inlet total pressure. The PV107/PV108 gage location shows a very similar story for runs 21, 22, and 28 where peak magnitudes are missed by up to 1% of inlet total pressure and runs 33 and 43 are matching the peak magnitudes of the measured data. The trends within the data are captured pretty well however for all runs the predictions show a double hump in the time-series data which matches the PV107 gage but a single hump is shown in the PV108 gage. In both the computational predictions and the measured data, the PV107/PV108 gage location shows an approximate 50% reduction in the unsteadiness as compared to the PV104/PV105 gage location which supports the use of the stationary double overlap for reducing the unsteadiness in the actual cavity by creating a tortuous path.
Figure 109. Comparison of the Run 21 stationary cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PV104 and PV105 gage location and (b) the PV107 and PV108 gage location
Figure 110. Comparison of the Run 22 stationary cavity time-series static pressure between the rig and the FINE/Turbo unsteady results for (a) the PV104 and PV105 gage location and (b) the PV107 and PV108 gage location
Figure 111. Comparison of the Run 28 stationary cavity time-series static pressure between the rig and the FINE/Turbo unsteady results for (a) the PV104 and PV105 gage location and (b) the PV107 and PV108 gage location
Figure 112. Comparison of the Run 33 stationary cavity time-series static pressure between the rig and the FINE/Turbo unsteady results for (a) the PV104 and PV105 gage location and (b) the PV107 and PV108 gage location.
Figure 113. Comparison of the Run 43 stationary cavity time-series static pressure between the rig and the FINE/Turbo unsteady results for (a) the PV104 and PV105 gage location and (b) the PV107 and PV108 gage location.
6.5.2 Stationary Static Temperature

Three stationary thermocouple locations were available from the rig and all are positioned below the lower angel wing on the static side of the cavity shown in Figure 107. The comparison of the measured and computationally predicted time-average static temperature is shown in Figure 114 for all runs. Run 21, provided in Figure 114(a) shows the predictions matching the thermocouples to within 0.5% of inlet total temperature for the steady and harmonic unsteady solutions and the phase-lag unsteady solution is still within 1% of inlet total temperature. The remaining runs under-predict the measured cavity static temperatures by a much more significant margin with run 28 being the smallest at about 4% of inlet total temperature and run 43 being nearly 15% of inlet total temperature. The under-prediction could be due to a variety of reasons including not having the proper distance from the wall to the thermocouple in the results extracted from the computational results, the cavity isothermal wall specified temperatures not matching the rig, and the ingested main stream gas being too cool due to the high-pressure vane isothermal wall specified temperatures. The height of the thermocouple is of concern and difficult to assess during operation of the rig however post-test inspections were performed and these heights were used when extracting the data. Additionally, the temperature axially across the width of the cavity is fairly constant partially due to the isothermal wall temperatures on the rotating and stationary side being within a few degrees of each and thus vary little variation is experienced in the vicinity of each of the gage locations. The temperatures specified for the isothermal walls of both the cavity and the vane hub will be discussed next as some additional investigation was performed.
Figure 114. Comparison of the stationary cavity time-average static temperature between the rig and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43.
Figure 114 Continued

(c)

(d)
As previously discussed, the effect of the isothermal wall temperatures for the high-pressure vane were varied after seeing the run 43 comparison of the stationary cavity temperatures between the measured and predicted values. The heat flux gage underside temperature measurements on the high-pressure vane hub and airfoil surface were compared to the RTD measured temperatures originally used to set the isothermal wall boundary conditions and it was found that for all runs the RTD estimated wall temperatures were low by anywhere from 40-100° K. Subsequently, additional steady and harmonic unsteady computational models were developed using the heat flux gage underside temperature measurement as the wall temperature in order to see the effect of the higher wall temperatures on the predicted cavity stationary and rotating static temperatures. The stationary cavity wall was originally assumed to be at the same temperature as the high-pressure vane hub surface. For the new model with the increased high-pressure vane, airfoil and shroud surface temperatures, the stationary cavity wall was run using the higher high-pressure vane hub surface for the portion of the wall that is physically attached to the each vane segment while keeping the original temperature for the remainder of the stationary wall. Figure 115 below shows the region where the wall temperatures were increased for the new model. Runs 28 and 43 were initially
analyzed to determine the impact and the wall temperatures were raised about 8% and 17% of inlet total temperature for the two runs respectively.

Figure 115. Temperature application zones on the high-pressure vane hub and stationary cavity for the increased isothermal wall boundary temperature models

A comparison of the measured, original predictions, and the elevated high-pressure vane surface predictions are shown in Figure 116 below. The elevated temperatures for run 28 bring the predictions directly in-line to the experimental data versus the original predictions which were about 4% of inlet total temperature low. For run 43, the new prediction with the elevated hub surface isothermal wall temperature is now only 4% under-predicted in comparison to the data as compared to the original prediction which was 309
about 14% under-predicted. This suggests that the new elevated temperatures taken from the high-pressure vane double-sided heat flux gauges is most likely more in line with the actual experiment. Closing the additional 4% to the experimental data for run 43 could be one or both of two approaches such that the hub surface temperatures might still be higher than indicated or that the rotor side cavity wall surface temperatures may be higher than original thought.

Figure 116. Comparison of the original and the elevated high-pressure vane isothermal wall temperature models to the measured data for (a) run 28 and (b) run 43
6.5.3 Rotating Static Pressure

The rotating instrumentation in the cavity area includes four Kulite locations of which the PRW70, PRW71, and PRW72 locations are above the high-pressure blade angle wing and the PRW74 location is below the angle wing as seen in Figure 106. The time-average comparisons between the steady/unsteady computational models and the measured data are provided in Figure 117 below. The harmonic solution in all of the comparisons predicts the trends in going from gage location to gage location even predicting the magnitude of change to the same degree as the data. The steady solution also predicts the trends correctly however the magnitude of change is significantly less than measured by about 1% of inlet total pressure. For run 21, 22, 28, and 33, all solutions are offset over-predicting the absolute static pressure in the cavity by about 1-2% of inlet total pressure at all gage locations which exception to the run 28 and run 33 PRW72 location which is much less than 1% of inlet total pressure higher than the measured values. Run 43 is once again over-predicted by 4-6% of inlet total pressure which is consistent with many of the other comparisons.
previously shown. In the cavity region, this could be due to many factors of which include the predicted blade incidence angles which would cause a larger or smaller blade bow wake depending on whether they are over or under predicted as compared to the rig. Another factor could be the upstream high-pressure vane wake strength at the hub being missed which was evident from the Kulite comparisons made at the trailing edge hub in section 6.3.1 as the results over-predicted the static pressure.

(a) Continued

Figure 117. Comparison of the rotating cavity time-average static pressure between the rig and the FINE/Turbo steady and unsteady results for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43

312
Figure 117 Continued

(b)

(c) Continued
Figure 117 Continued

(d)

(e)
The time-accurate comparisons for each of the gage locations are shown in Figure 118 through Figure 122 below for runs 21 to 43 respectively. For run 21, the PRW72 gage location unsteadiness is predicted very well by the harmonic unsteady computational model in both trend and magnitude. The data does suggest a slightly more oscillatory trend however this could easily be passage to passage variation. The remaining gage locations all under-predict the magnitudes by 2-7% of inlet total pressure at the maximum and minimum values and the trends are noticeably different with the general shape being consistent with the data. The comparisons for run 22 in Figure 119 are very similar to those of run 21 in that the PRW71 gage location is predicting very well and the other locations tend to miss maximum and minimum values by about 2-7% of inlet total pressure. Overall, the trends are in general in line with the predicted data however there does appear to be a discrepancy between the amount of unsteadiness (in terms of the number of peaks and valleys) between the computational and measured results. All gage location comparisons for run 28 show that the computational model under-predicts the maximum and minimum values 5-10% of inlet total pressure including the PRW71 gage location. Trends across the vane passage for each location, in general, compare well to the data in terms of peak and valley location while the number of peaks and valleys is different between the two results.

With exception to the PRW71 location, run 33 in Figure 121 captures the maximum and minimum locations and magnitudes to within 2-4% of inlet total pressure. Also interesting to note is that the overall magnitude of the data is reduced in comparison to the other runs by about 30-70% across all locations. The time varying trends are predicted pretty well, but once again the computational results are showing more peaks and valleys than what is seen in the measured data. Run 43 magnitudes, in Figure 122, are back in line with run 21, 22, and 28 and are again under-predicted by 5-10% of inlet total pressure. The PRW71 gage is predicted very well for run 43 in both trend and magnitude while the trends for the other three locations are similar to those as noted for the previous four runs (in general look good but number of peaks and valleys are different). One interesting note for all of the runs is the significant reduction in the level of unsteadiness seen for the PRW74 gage location in comparison to the other three locations. This location is
below the angle wing on the blade and the nearly order of magnitude drop in unsteadiness is due to the shielding affect from main stream gas unsteadiness provided by the double overlap rim seal geometry.

![Graph showing static pressure fluctuation about the mean vs gage location]

*(a) Continued*

Figure 118. Comparison of the Run 21 rotating cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PRW70 gage location, (b) the PRW71 gage location, (c) the PRW72 gage location, and (d) the PRW74 gage location.
Figure 118 Continued

(b)

(c)  Continued
Figure 118 Continued

(d)

Figure 119. Comparison of the Run 22 rotating cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PRW70 gage location, (b) the PRW71 gage location, (c) the PRW72 gage location, and (d) the PRW74 gage location
Figure 119 Continued

(b)

(c) Continued
Figure 119 Continued

Figure 120. Comparison of the Run 28 rotating cavity time-series static pressure between the rig and
the FINE/Turbo unsteady profile for (a) the PRW70 gage location, (b) the PRW71 gage location, (c)
the PRW72 gage location, and (d) the PRW74 gage location.
Figure 120 Continued

(b)

(c)
Figure 120 Continued

(d)

Figure 121. Comparison of the Run 33 rotating cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PRW70 gage location, (b) the PRW71 gage location, (c) the PRW72 gage location, and (d) the PRW74 gage location
Figure 121 Continued

(b)

(c)

Continued
Figure 121 Continued

(d)

Figure 122. Comparison of the Run 43 rotating cavity time-series static pressure between the rig and the FINE/Turbo unsteady profile for (a) the PRW70 gage location, (b) the PRW71 gage location, (c) the PRW72 gage location, and (d) the PRW74 gage location.
6.5.4 Rotating Static Temperature

As previously mentioned, four thermocouples were positioned both above and below the high-pressure blade forward angle wing (two positions above and two positions below) in the rotating reference frame of the cavity. The positions of each thermocouple are shown on the R-Z plane of the cavity in Figure 107. The comparison of the measured and computationally predicted time-average static temperature is shown in Figure 123 for all runs. The comparisons for run 21 provided in Figure 123(a) show that all of the computational models over-predict the static temperature at each gage location by as much as 8% of inlet total temperature. The trend between the gages above the angle wing (TRW62 and TRW63) and below the angle wing (TRW64 and TRW65) is captured in all models however to a much lesser magnitude than dictated by the measured data. For run 22 in Figure 123(b), the predicted magnitudes are off above the angle wing by about 8% of inlet total temperature while the gage below the angle wing are nearly matched at the TRW64 location and about 3% of inlet total temperature high at the TRW65 location. The trend
from above-to-below the angel wing is predicted in the wrong direction, opposite to that for run 21. The magnitude of the TRW64 and TRW65 gages are predicted to within 1% of total temperature for run 28 shown in Figure 123(c) while below the angel wing is over-predicted by 5-7% of inlet total temperature. The predictions for run 28 show the static temperature to be flat between all locations whereas the data suggest the static temperatures above the angle wing should be cooler. Run 33 shows the best overall comparison to the data in that the above angel wing locations are less than 2% of inlet total temperature under-predicted to the data and the under angel wing gage locations are 5% of inlet total temperature under-predicted, however the trends are once again missed. Run 43 is once again the worst overall comparison and under-predicts the below angel wing locations by as much as 15% of inlet total temperature whereas the above angel wing locations are under-predicted by about 2%.

In nearly all cases, the thermocouples under the angel wing are reading temperatures of a 1% to 10% higher than the thermocouples above the angel wing whereas the computational models predict the trends in the opposite direction. This can mean one or several of three possible issues exist: the experimental rig is ingesting main stream flow deeper than the computational models predict, the cavity wall temperatures on either the rotor side or the stationary side are much hotter than originally estimated, or the rotor cavity wall is imparting more work into the cavity air than is predicted causing cavity temperatures to rise above local main stream flow near the hub.
Figure 123. Comparison of the rotating cavity time-average static temperature between the rig and the FINE/Turbo steady and unsteady profile for (a) Run 21, (b) Run 22, (c) Run 28, (d) Run 33, and (e) Run 43.
Figure 124 Continued

(c)

(d)  Continued
To study this difference a little further, the run 21, run 28, and run 43 solution instantaneous forward progressing streamlines on the purge cavity and main stream interface and the static temperature in the cavity at several sections were plotted and are shown in Figure 124 below. For run 21, it is seen that the main stream air ingested into the cavity does penetrate below the forward angler on the rotor platform along the stator side of the cavity wall as expected. However, the ingested air is actually cooler than the air in the cavity due to the rotating wall imparting work into the air in the cavity through friction. If it was a deep ingestion issue for run 21, then the rotating thermocouples under the rotor side angler wing would show cooler temperatures rather than hotter for this case based on the temperatures seen at the ingestion zones. For run 28, the main stream air does not penetrate as deep into the cavity which is most likely due to the positive purge cooling air but the other issue is that the high-pressure vane hub flow that is ingested into the cavity is about the same temperature as the positive cavity purge flow as seen in Figure 124(b). The main stream flow ingestion for run 43 in Figure 124(c) is not shown to penetrate into the rim seal further than the open portion of the cavity although from the static pressure contour it is seen that temperatures hotter than the cavity air are seen as deep as the top side forward tip of the rotor angler wing. Also, the cavity
temperatures below the angel wing are much cooler than the main gas path flow suggesting that the isothermal wall temperatures in the cavity may be too low.

Figure 124. Cavity interface instantaneous forward progressing streamlines and cavity static temperature for (a) run 21, (b) run 28, and (c) run 43
The under-prediction of the data in almost all runs tends to direct the issue toward the isothermal wall boundary condition specified temperatures as has been discussed in previous sections. As discussed in section 6.4.4 and 6.5.2, another set of computational predictions were performed in order to understand the effects of elevated high-pressure vane surface temperatures for runs 28 and 43 on the static temperatures within the cavity and the blade platform temperatures. These new models did not change the specified temperatures on the rotor side of the cavity or the platform as no additional data was available to make such adjustments. However, the effect on the rotating side gages due to the increase in temperature on the stationary side of the cavity can be seen in Figure 125 below. For run 28, the underside angel wing temperatures were increased about 1% of the original predictions and are actually more in-line with the data due to the increasing trend that was picked up from TRW64 to TRW65. However, the two thermocouples above the angel wing are now being over-predicted by nearly 10%. The run 43 comparison
in Figure 125(b) shows that the increased hub isothermal temperatures does have a pretty strong effect on
the thermocouples above the rotor side angle wing raising the static temperatures about 8% over the
experimental data versus the original predictions which were 2-3% under-predicted. The underside angle
gages were also increased about 4% but still remain under-predicted to the data.

(a)                                      Continued

Figure 125. Comparison of the original and elevated high-pressure vane isothermal wall
temperatures to the measured data for (a) run 28 and (b) run 43
The increased isothermal wall temperatures for the high-pressure vane hub surface has definitely been shown to have an impact on the rotor side predicted temperatures at each of the gage locations such that now the computational model is over-predicting the temperature above the angel wing and still maintaining the under-prediction beneath the angel wing. For run 43, based on these results it is a likely combination of rotor side and stationary side isothermal wall temperatures that were originally low for comparison to the rig which would be required to raise both temperatures up to be in agreement with the data. However, the level of adjustment applied to the high-pressure vane hub may be too much as indicated by the rotating thermocouples above the angel wing. The run 28 results point toward the rotor side temperatures possibly being too high as the above angel wing temperatures are over-predicted and would need to come down as the stationary side temperatures are increased in order to maintain a good comparison with the data.
6.6 Concluding Remarks on the Experimental and Computational Comparisons

A comparison between the steady, harmonic unsteady, and for run 21 the phase-lag unsteady computational models to the experimental rig data has been shown for multiple locations throughout the high-pressure turbine stage. The comparisons have included both temperature and pressure data throughout the turbine to ensure that both the purge cavity region and the main gas stream flow field predictions are matching well to the measured data. The exit average operating parameters and the exit profile comparisons for total temperature and total pressure showed that the computational models were not only able to reproduce the operating conditions but could also match the magnitude and the shape of the profiles as seen in the rig. This provides fairly substantial evidence that FINE/Turbo captures the overall temperature and pressure losses through the turbine flow path fairly well.

While the purge cavity instrumentation comparisons was of most interest for this research, additional comparisons of temperature and pressure data were shown for the high-pressure vane and the high-pressure blade. The exit static pressure just downstream of the high-pressure turbine vane provided several circumferential locations to compare the measured and predicted static pressure on the hub surface from both a time-average and time-accurate stand point. The time-average comparisons for the run 21 and the two cooled solutions match with the data very well while the uncooled solutions over-predict the hub surface static pressure with run 43 having the most discrepancy with the data. The time-accurate comparisons showed very similar trends between the measured and predicted data however the maximum and minimum were under-predicted across all five runs analyzed.

The high-pressure turbine blade surface static pressure was only captured in the rig at three locations on the airfoil and for most runs only two gages were available but it did provide a point on both the suction and pressure surface of the blade in all instances. These comparisons mainly provided a diagnostic measure to ensure that the blade surface loading was in a reasonable range as compared to the experimental rig. For
runs 21, 28, and 33 the computational results favored well against the measured data suggesting that the blade loading and most likely the incidence angles on the blade leading edge were predicted similarly to the rig during the operational time window. Runs 22 and 43 over-predict the pressure surface loading as best can be derived from the single measurement possibly suggesting more negative incidence in the computational models versus the rig. The shroud comparisons also showed similar results for the time-average comparisons provided such that runs 22 and 43 were over predicted in comparison to the data while the other three runs were in good agreement. Time-accurate comparisons were shown for the single shroud location in addition to the time-average results which should that overall the trends of the data were in very good agreement however the maximum amplitude was over-predicted. Overall, the diagnostic comparisons do point towards the computational models capturing the operational and flow conditions fairly well.

The high-pressure turbine blade comparisons also included several temperature comparisons that included the leading edge relative total temperature and multiple spanwise locations and the platform static temperature at multiple axial and circumferential locations. Overall the blade leading edge relative total temperatures compared very well between the measured and predicted profiles for both the steady and unsteady models. There is a small section between 40% and 50% span where the data shows a local dip in the measured temperature that is not predicted by the computational models which is currently thought to be due to the high-pressure vane film cooling pattern that is present in the actual vane, modeled in only the cooled solutions as source terms and left out of the uncooled solutions. The blade platform temperature comparisons are very good for run 21 and run 22. For run 28, the steady model predicts the temperatures fairly well on the leading edge of the platform whereas the harmonic unsteady predicts the temperatures very good on the aft section of the platform. For run 33 and run 43, the temperature predicts are under-predicted by a fair amount with respect to the data, especially for the gage locations further aft on the platform. From the leading edge comparisons, the incoming temperature profile is predicted very well such that the initial feeling is that the isothermal wall boundary conditions for the blade platform are the most
likely leading cause for the miss on run 33 and 43. These specified temperatures will dictate the initial temperature gradient off the platform wall through the thermal boundary layer.

The final comparisons shown for the stationary and rotating sides of the cavity included both static pressure and static temperature measurements in the cavity rim seal region which helps to establish how well the computational models predict the behavior where the main stream gas begins to interact with the cavity. The time-average static pressure comparisons on both sides of the cavity showed very good between the computational and measured results. Trends from gage location to gage location were captured by the steady and unsteady solutions very well with the harmonic solution being the better of the two comparisons. Time-accurate comparisons across all gage locations and runs showed that in general the trends were predicted well with peaks and valleys lining up with the data however the overall magnitudes were under-predicted to some extent. Additionally, the unsteadiness predicted by the harmonic solution shows much more oscillatory behavior than the measured data at several gage locations. Only time-average comparisons were provided for the thermocouple locations in the cavity. These comparisons showed that the temperatures in the cavity are under-predicted by both the steady and unsteady computational models. Adjusting the high-pressure vane hub temperatures to match the double-sided heat-flux gage lower temperatures brought the stationary side predictions closer in agreement with the experimental data, caused the rotating side gages above the angel wing to be over-predicted, and had little effect on the platform and cavity thermocouples below the angel wing. Thus it is most likely a combination of increased rotor and stator side wall temperatures that would be needed to match the thermocouple data better.

Overall, the comparisons made between the computational and measured results from the turbine rig have shown that very good agreement was achieved with the assumptions applied in each of the solutions across a varied range operating conditions from room temperature to elevated temperature runs with radial profiles and cooling flows. And given that these computational models were predictions of the rig (with exception
to the elevated high-pressure vane hub analyses); this provides additional confidence in the methods applied for capturing the unsteady flow field in and around the purge cavity rim seal. In the next chapter, comparisons will be made to the analytical model developed in Chapter 2 along with some of the correlations discussed in Chapter 1 and the predicted computational predicted will be look at in further depth and supported with experimental data as appropriate.
CHAPTER 7

IN-DEPTH FLOW FIELD ASSESSMENT OF THE PURGE CAVITY

In Chapter 6, the predictive capabilities of FINE/Turbo across a variety of operating conditions was shown through to be very good through a direct comparison to the experimental data taken on engine representative hardware operating at the proper correct design point conditions. Knowing that that the computation models do provide what appears to be reasonable results, the purpose of this chapter is to take an in-depth look at the computationally predicted flow-field for the steady and harmonic unsteady results to understand the physics in and around the purge flow cavity rim seal. This will include revisiting the one-dimensional equations derived in Chapter 2, comparing findings with other results in the current literature present in Chapter 1, and using zero-to-three dimensional data from the computational solutions to shed light on new findings. Where applicable, the measured data from the rig will be presented in order to support or refute the findings for the computational models.

Throughout this chapter, a majority of the in-depth look will be performed using run 28 only as it represents the most similar configuration to full scale engines with respect to the radial inlet total temperature profile, the high-pressure vane film cooling, and the introduction of the purge flow located between the high-pressure vane and blade. There will be several different cases for run 28 presented: the nominal configuration shown throughout Chapter 6, the nominal configuration without purge flow but with the cavity still present, and the nominal configuration with the purge flow cavity removed (and hence no purge flow). Each solution will be labeled accordingly throughout.

340
7.1 One-Dimensional Closed-Form Analytical Model

Two different models for calculating the thermodynamic properties in a cavity were derived from the Navier-Stokes momentum equations in Chapter 2. Both models assumed the flow to be one-dimensional in the tangential direction (the axial and radial velocities are much smaller in magnitude and thus ignored), steady, and axisymmetric. The forced vortex model made a further assumption about the tangential flow such that it was a function of the angular momentum of the rotating wall of the cavity such that the cavity forces the flow to rotate. Thus, the tangential velocity increases as with radius which subsequently causes increases in the total temperature and total pressure due to the work input of the rotating wall. The free vortex model assumed the tangential velocity to be a function of radius only which creates a constant total temperature and total pressure environment from inlet to exit as the tangential velocity decreases with increased radius. These two models essentially make assumptions at both extremes while most likely purge flow cavities function somewhere in-between.

The two models were used to calculate thermodynamic properties in the pure flow cavity and were compared to tangentially averaged properties from the time-averaged harmonic unsteady computational model at multiple planes. The inlet and exit stations used for the calculations are shown in Figure 126. The calculations were stopped short of the overlapping rim seal as the equations were only derived for the cavity region. While this comparison will not provide insight into ingestion phenomena, it will show how useful simple one-dimensional models can be when applied correctly.
Figure 126. Inlet and outlet radial locations for the analytical and computational comparison

Inlet conditions of total temperature, total pressure, and (for the free vortex only model) tangential velocity were provided as boundary conditions. Originally, the forced vortex model was derived assumed that that the fraction of angular momentum transferred from the wall to the fluid was 100% however it could just as easily be assumed that this net angular momentum transfer is 10% or any number between 0% and 100%. For the comparisons show in Figure 127, the forced vortex model used two assumptions for this net transfer, 100% (or X=1.0) and 50% (or X=0.5). Comparisons are made for the total temperature, total pressure, and tangential velocity as a function of radius. All quantities are normalized to the inlet total temperature, total pressure, and total velocity.
Figure 127. Comparison of the closed form analytical model and the time and tangentially averaged FINE/Turbo results for (a) total pressure, (b) total temperature, and (c) tangential velocity
From Figure 127, it can be seen that computational results of the time-averaged harmonic unsteady model truly lie in-between the forced and free vortex assumption. Clearly, this purge flow cavity does not operate under the free vortex assumption as the total pressure, total temperature, and the tangential velocity rise from inlet to exit which suggests that the rotating wall does impart work on the cooling flow. From a total pressure perspective in Figure 127(a), initially the pressure rises similarly to the X=0.5 forced vortex model however once the flow goes through the mid-height expansion you see the pressure rise increases in rate and eventually levels off to the same rate of rise (slope of the line) of the X=0.5 model. The total temperature comparison in Figure 127(b) is similar in comparison to the total pressure however once past the expansion near mid-height, the total temperature rise levels off and nearly matches the X=0.5 model near the exit radius. This is most likely due to the isothermal wall boundary conditions tempering the total temperature rise in the unsteady computational model. The closed-form analytical models due not take into account the wall temperatures and thus this affect would not be seen in the calculated solution. The tangential velocity in Figure 127(c) shows that the velocity rises at a much steeper rate than the X=1.0
forced vortex solution for the lower portion of the cavity and then once through the expansion at mid-height the velocity rate increase is parallel to the X=1.0 forced vortex solution and nearly matches the X=0.5 forced vortex solution at the exit which is about half of the X=1.0 forced vortex velocity magnitude. This suggests that in the tighter clearance areas between the stationary and the rotating walls, the work input into the fluid by the wall is much higher than in the more open areas, again something not taken into account with the closed-form solutions.

Within the computational cavity, the radial and axial velocities were found to be on the same order of magnitude at the inlet region where the tangential velocity is small. Near the exit region, the axial velocity component was about 1% of the tangential velocity and the radial velocity component was about 10% of the tangential velocity. Thus, near the exit region this assumption holds true but at the inlet it is completely dependent on the inlet conditions. For this cavity, inlet conditions contained both an axial and tangential component as discussed in Chapter 5 and indicated in equation 5.3.

Overall, the comparison between the closed-form, one-dimensional solutions and the computational model did provide good insight into some of the over-arching physics within the cavity such as the rotating wall imparting work on the cavity flow and just how much of that work is most likely being converted.

7.2 Rim Seal Ingress / Egress and Comparison to Published Correlations

While it would be possible to produce a minimum sealing gas correlation based on the unsteady computational model throughout this research, it was determined to be inappropriate with the absence of data from the experimental study. In lieu of such a model, several of the correlations discussed in Chapter 1 will be compared to the analytical results achieved within. Three of the correlations presented were
derived from analogous experimental data sets and include the correlations from Bayley and Owen [2], Phadke and Owen [6], and Bohn and Wolff [13]. Each of these correlations will be discussed in some detail in relation to the purge flow cavity for this turbine rig. In Table 12 below, the non-dimensional cavity parameters discussed throughout Chapter 1 are provided below. These parameters will be the basis for comparison.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value based on Run 28 Operating Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>$G_C$</td>
<td>0.0092</td>
</tr>
<tr>
<td>$G$</td>
<td>0.0438</td>
</tr>
<tr>
<td>$Re_0$</td>
<td>6.34E+06</td>
</tr>
<tr>
<td>$Re_W$</td>
<td>9.46E+06</td>
</tr>
<tr>
<td>$C_W$</td>
<td>13267</td>
</tr>
</tbody>
</table>

Table 12. Non-dimensional cavity parameters for run 28

Bayley and Owen [2] derived a rotational Reynolds number correlation for a single stationary lip rim seal using underside edge pressure as an indicator for ingestion into the cavity such that once it becomes sub-atmospheric ingestion was assumed. The correlation, provided originally in equation 1.8, is the following:

$$C_{W,MIN} = 0.61 G_C \ Re_\theta$$  \hspace{1cm} (7.1)

Where $G_C$ is the shroud clearance ratio and the rotational Reynolds number, $Re_\theta$ is:

$$Re_\theta = \frac{\rho \omega R_O^2}{\mu}$$  \hspace{1cm} (7.2)
Based on the non-dimensional parameters in Table 12, equation 7.2 would suggest a minimum non-dimensional sealing flow of 35600 as compared to the actual flow non-dimensional flow of 13267, a 168% increase in the flow rate. From results shown in section 7.3, both experimental and computational, it is unlikely that the cavity is experiencing ingestion below the lower stationary seal. While Reynolds numbers, non-dimensional flow rates, and non-dimensional seal clearances were similar in to this turbine in their work, Bayley and Owen’s [2] correlation is based on a single, stationary lip rim seal with no blading (stationary or rotating). For this turbine the configuration is a double stationary, single rotating lip rim seal and thus the correlation is not appropriate for this turbine and is included for completeness only.

Phadke and Owen [6] tested a two lip stationary, no rotating lip rim seal in the presence of a non-axisymmetric main stream flow with very a very similar non-dimensional seal clearance to the turbine within this research. However, it should be noted that tested rotational and external flow Reynolds numbers along with the achieved pressure asymmetry was much lower than observed in this turbine by about an order of magnitude or more. Ingestion into the cavity was based on flow visualization using smoke such that any appearance of smoke within the main cavity was declared ingestion of the main stream flow. There correlation shown in equation 1.9 and 1.10 is the following:

\[
C_{W,Min} = 2\pi \sqrt{2CC_D G_C \frac{1}{P_{Max}^{\frac{1}{2}}}} \tag{7.2}
\]

\[
P_{Max} = \frac{\frac{1}{2}(P_{Max} - P_{Maxn})}{\frac{1}{2} \rho \bar{W}} Re_{\bar{W}}^{\frac{2}{2}} \tag{7.3}
\]

Using the maximum annulus pressure difference to calculate \(P_{Max}\) from equation 7.3 and applying the experimentally derived factor of 0.6 for \(\sqrt{2CC_D}\), a minimum sealing flow rate of about 136,000. If the maximum pressure difference at the hub surface near the rim seal is used, the minimum sealing flow rate is reduced to 80,300. In both cases the minimum sealing flow rate is well above those experienced for this turbine rig and it is thus concluded that this correlation, again, is not fitting due to the geometry differences.
Bohn and Wolff[13] performed a series of experiments using a turbine rig that incorporated blading (both stationary and rotating), had the same double stationary and single rotating lip seal and reached similar non-dimensional flow rates as the turbine shown throughout this research in an ambient pressure rig. Their goal was to extend the work of Phadke and Owen [6] by examining the individual characteristics of different seal geometries. Bohn and Wolff [13] utilized CO₂ as a tracer gas and declared ingestion when the cooling efficiency gradient was above 2.1x10⁻⁶ for the location directly below the lowest sealing lip. The final correlation shown in equation 1.11 and 1.12, based on the external flow Reynolds number and a pressure gradient is the following:

\[ C_{W,Min} = 2\pi KG_c \sqrt{\frac{1}{2} C_{P,Max} Re_w} \]  

(7.4)

\[ C_{P,Max} = \frac{P_{Hub,Max} - P_{Hub,Min}}{\frac{1}{2} \rho C_i^2} \]  

(7.5)

This correlation uses the hub static pressure difference for the calculation of \( C_{P,Max} \) and the constant, K, for the rim seal geometry of this turbine is 0.12 versus Phadke and Owen’s [6] derived value of 0.6. Applying equations 7.4 and 7.5 to this turbine, a minimum sealing flow rate of 16,000 is calculated which is more in-line with this turbine and suggests that the flow rate utilized in the experiments may be on the edge of ingestion. This would also depend on how ingestion was defined as it is clear that most authors are using different definitions.

These comparisons to the minimum sealing flow rate correlations show that in order for a correlation to be effective it must be based on the geometry, including main stream gas path blading and rim seal configuration, as well as operational conditions. Early studies which did not involve the gas path blading over-predict the minimum sealing flow rates by as much as ten times what has been demonstrated with this research while the results of Bohn and Wolff [13] which includes all of the necessary features predicts a
minimum sealing flow rate only 16% higher than those achieved with the run 28 testing. Thus the correlation could be deemed useful for preliminary design studies and post-test analysis of data.

7.3 Cavity and Rim Seal Flow Field

The flow field both in the cavity and in the rim seal will be analyzed by using the details of the run 28 harmonic unsteady solution. Both time-average and instantaneous information will be used to show what is happening both in and around the cavity. The high-pressure vane and the high-pressure rotor contribute to the unsteadiness in the cavity through not only their own aerodynamic wakes but also through the interaction of these wakes such that the unsteady computational model becomes very important.

The flow field within the cavity contains many recirculation zones both within the main portion of the cavity and within the rim seal itself. Streamlines from the time-average harmonic unsteady solution projected onto an R-Z plane within the cavity are shown in Figure 128 for run 28. It is important to note that for this cavity the flow field is mainly tangential within the cavity and the rim seal until the section of the rim seal open to the main gas flow is encountered. The streamlines are being shown in this plane to provide an overall sense of what the radial and axial velocity fields are doing. The inlet mass flow or-boards the rotating wall after entering the cavity and is pumped up the rotating wall all the way to the top of the rim seal. Near the areas of restrictions, recirculation zones are setup due to the same mass flow being pump back down the stationary wall. Multiple recirculation zones have set up within the cavity due to this phenomenon.
Next, the velocity vector field in the rim seal area of the cavity is shown in Figure 129 for the time-average harmonic unsteady solution. There are, again, multiple recirculation zones within the rim seal itself that help to seal off the ingested main stream flow from penetrating too deep into the cavity itself. In this case, there is a recirculation zone setup up by the ingested main stream flow where it comes in contact with the purge flow attempting to exit the cavity. The other recirculation zones called out in Figure 129 are a result of the flow being pump up or down the walls turning corners as it makes its way up or down the cavity.
Figure 129. Run 28 vector field in the rim seal area for the time-averaged harmonic unsteady solution

Next, the interactions at the purge flow cavity and main gas path interface will be analyzed. In order to visualize what happens at the interface to the purge flow cavity and the main gas path, the forward progressing streamlines from a time-average perspective are plotted in Figure 130 which shows the a top down, side, and forward looking aft picture. From a top looking down view, there is a large area in which the main gas path air is being ingested that starts at the leading edge of the blade and extends circumferentially toward the adjacent blade for a total length of about one half of a rotor pitch. On the suction side of the blade, the purge and ingested main gas path air is exiting the cavity and becomes entrained with the main gas path air as is progresses downstream along the suction surface. There is a very
small amount of purge flow that is able to travel along the platform boundary layer across the leading edge and becomes entrained in the adjacent blade suction side air flow. From the side view there are several additional observations to be made. Looking at the cavity itself, from a time-average perspective the main gas path air ingestion only penetrates the open part of the cavity although it was shown earlier in Chapter 6 that at some instantaneous time steps the ingested flow was seen to penetrate as deeply as the top side forward tip of the rotor angel wing for run 28. The main gas path air that is ingested into the cavity is seen to be very local to the hub section of the high-pressure vane and is comprised of cooler air due to the isothermal wall boundary conditions and the film cooling flow which has been introduced on the hub surface further forward. It does not ingest hotter air from higher and hotter radial locations and the temperature of the ingested air in this case will be a strong function of the isothermal wall temperature as discussed previously in Chapter 6. For the air flow leaving the cavity on the suction side of the airfoil, it is seen that part of it is pulled upward to the 20% span region fairly quickly while another portion travels along the blade fillet until about 20% wetted distance before being pulled upward. The purge flow leaving the cavity that is pulled along the blade fillet comes from the cavity almost immediately to the left of the leading edge while the remainder of the purge flow is pulled upward to about 20% span. The purge flow is not see to travel along much of the rotor platform and has no residual impact to keeping the platform cooler further after. It only locally travels along the platform in the curved section just forward of the leading edge. If using the purge flow to cool the platform was truly desired, the flow would need to be directed more downstream as it comes out the cavity and a lot more flow would be required such that platform cooling is probably best achieved through the use of film cooling holes if it was required.
Figure 130. Purge flow forward progressing streamlines for the time-averaged harmonic solution for
(a) top looking down view, (b) side view, and (c) forward looking aft side view (Not To Scale)
Figure 130 Continued
The next set of contour plots in Figure 131 looks at the radial velocity on the interface plane between the main gas path and the purge flow cavity. These are instantaneous plots that progress forward in time through half of a vane passing period where the blue zones are negative radial velocity and the red zones are positive radial velocity. A black isoline of zero velocity is also included to help visualize the different zones. At t=0, there are two flow ingestion zones, labeled IG1 and IG2, and two flow ejection zones, labeled EG1 and EG2. The IG2 zone is created by the HPV1 wake as it initially passes over the cavity on that stationary side and as it impinges on the rotor side where the rotation of the wall aides in the local ingestion. The IG1 zone is similar to the rotor side IG2 zone and is created the vane wake impinging on the
rotor side wall. The flow ejection zones are typically found on the suction side of the airfoil as is the case for this time step due to the lower static pressure on the suction side. At \( t=0.055 \), the IG2 zone has grown in intensity as the vane wake is now being influenced by rotor bow wave which is also interacting with the trailing edge shock emanating downstream. This interaction causes the rotor bow wave to be greater in strength and will actually bend the vane wake upstream slightly. The IG1 zone is growing slightly in the circumferential direction which is caused by the rotation wall as discussed earlier. The EG1 zone is beginning to shrink as the IG2 zone grows which causes the EG1 zone to strengthen in radial velocity.

At \( t=0.11 \), the IG2 zone on the rotor side is beginning to loose strength as the HPB1 rotates out of the shock emanating downstream from the HPV1 trailing edge which allows the HPV1 wake to return to its normal trajectory causing the IG2 zone on the stationary side to nearly bridge the axial gap of the interface plane. The IG2 zone can also be seen running circumferentially down the stationary wall side to nearly the trialing edge of the HPV2 airfoil. This was cause by the bending of the HPV1 wake discussed earlier such that it runs nearly parallel to the walls. The EG2 zone is increasing in size growing against the rotating while at the same time losing some radial velocity strength and the EG1 zone is doing exactly the opposite. At \( t=0.167 \), a new ejection zone, EG1’ has been formed due to the IG2 zone cutting off part of the original EG1 zone. The IG1 zone continues to diminish in size and strength as the location is now between two upstream vanes, similar to the IG3 zone. By \( t=0.22 \), the EG1’ zone has now been pulled into the EG2 zone which covers the entire HPB3 platform leading edge and about one-third of the HPB2 platform leading edge. The IG2 zone is fairly stationary at this point as not interactions between the vane and blade are occurring for the HPB2. At \( t=0.278 \), the EG2 zone in front of the HP2 blade is growing in strength particularly near the suction side of the blade. The IG1 zone has all but disappeared and joined with the stationary IG1 zone which is beginning to detach from the stationary wall as the HPB1 blade rotates into the HPV1 shock cause the rotor bow wave to strength and bend the HPV1 wake upstream again. By \( t=0.33 \), the strong part of the IG2 zone has detached from the stationary side and is now riding on the rotating wall circumferentially and begins to stretch in the direction of rotation while the beginning of another stationary side ingestion zone begins to form due to the rotor wake. The EG2 ejection zone on the
suction side of the HPB2 airfoil has continued to grow in strength near the rotating wall as it starts to shrink with the IG1 detachment and movement circumferentially. At t=0.389, the IG2 zone is thinning axially and moving downstream as the HPV1 vane wake continues to feed the new stationary side ingestion zone which is weak but still present. At t=0.44, the stationary ingestion zone of IG1 is beginning to strengthen as the blade rotates through the trailing edge shock. The EG2 zone is concentrating every further on the suction side of the HPB1 airfoil which also causes it to growth in strength. The EG1 zone is growing upstream against the direction of rotation and is weakening as the new stationary IG2 location forms. At t=0.5 the IG2 zone is now coming up to full strength on the stationary side and extending across the axial gap and downstream on the rotating wall.

The ingestions zones are largely created by the high-pressure vane wakes and are subsequently influenced by the blade bow wake interaction with the trailing edge shock structure emanating downstream. The blade bow wake cause the vane wake to bend upstream; and when the blade passes out of the shock the vane wake snaps back in the downstream direction releasing the ingestion zone to travel downstream on the rotating wall side of the cavity where it will weaken and eventually disappear as new ingestion zone is created. The process seems very similar to the oscillating Karman street vortex shedding off of a cylinder.
Figure 131. Instantaneous Radial velocity contours on the purge flow cavity and main gas path interface for (a) t=0, (b) t=0.055, (c) t=0.11, (d) t=0.167, (e) t=0.22, (f) t=0.278, (g) t=0.33, (h) t=0.389, (i) t=0.44, and (j) t=0.5
Figure 131 Continued
Figure 131 Continued

(d) Continued
Figure 131 Continued
Figure 131 Continued
The ingestion and ejection zones are a strong function of the relative positions of both the rotor and stator. Both zones appear to have both stationary and rotating components even within a single zone. In order to truly capture these regions using computational techniques, both the high-pressure vane and the high-pressure blade need to be present and in communication through some form of unsteady technique within the calculation. A similarly statement can be posed for the experimental rig as well.
Next, the unsteady pressure measurements on the rig and the computational predictions for both the stationary side, Figure 132, and the rotating side, of the cavity will be analyzed along with contours of static pressure shown on the radial cutting planes through both gage locations. The static pressure contours are shown in Figure 134 along with the positions of each of the gages that will be discussed next. The purpose of discussing these results again is to correlate what is happening on the gage both experimentally and computational to what is seen in the computational results. This will give some insight into what is causing some of the unsteadiness within the cavity.

For the PV104 and PV105 stationary side gage location in Figure 132(a), the peak of the unsteady pressure at about $t=0.22$ is seen to be due to the rotor bow wave which actually forms at around $t=0$ but doesn’t appear down in the cavity. This suggests that there is a time lag between what happens at the interface plane between the main gas path and the cavity and the stationary Kulite location which is near the rotor side angle wing in radial height. The bow wave also appears to spread in circumferential width as well as loose some strength as travels radially inward through the cavity which goes back to support the use of such rim seals to help attenuate unsteadiness from the main gas path. In looking at PV107 and PV108 gage location in Figure 132(b), the peak occurs out past $t=0.44$ and is also due to the rotor bow wave which has long since diminished at this time step. The secondary and smaller peak in Figure 132(a) near $t=0.44$ is due to the onset of another rotor bow wave in the cavity from the adjacent blade just upstream moving closer. This bow wave in the cavity diminishes before passing by the stationary cavity gage location. A similar secondary peak is also seen at the lower PV107/PV108 location in Figure 132(b) and is due to the same feature only lags with respect to the PV104/PV105 location. The time lag appears to be about one-half of a rotor pitch in length from the main gas path to the PV104/PV105 location and an additional one-half of a rotor pitch to the PV107/PV108 location.
Figure 132. Measured and predicted stationary cavity unsteady static pressure for the (a) PV104 and PV105 location and the (b) PV107 and PV108 location
The rotor side cavity gage measured and predicted results are shown in Figure 133 below. The PRW70 and PRW71 gage locations in Figure 134 are seen to be straddling the rotor bow wave with the PRW71 location sitting right in the middle of it at t=0 which corresponds to the peak in Figure 133(b). As time progresses, the rotor bow wave shifts circumferentially against the rotating of the blade as the vane trailing edge shock attempts to keep the bow wave stationary. By t=0.33, the PRW70 gage has moved into the low pressure zone created by the purge flow ejection one noted earlier in Figure 131(g) on the suction side of the HPB2 blade from a computational perspective as the minima on the PRW70 gage do not necessarily line up between the measured and predicted curves. The peak on the PRW70 gage is due to the rotor bow wave forming again when the HPB2 blade begins to pass the next upstream vane trailing edge, this can be seen in Figure 134(j) on the HPB1 blade at t=0.5 which would occur on the HPB3 blade at an earlier time step. On the PRW72 gage, the peak unsteady pressure occurs as the HPB2 blade bow wave is diminishing and the HPB1 blade bow wave is increasing in intensity. The two bow waves appear to combine across the about one and one-half of a rotor pitch. This peak is followed by a sudden drop in the static pressure level due to the detachment of an ingestion zone from the stationary side of the cavity which was shown in Figure 131(g).
Figure 133. Measured and predicted stationary cavity unsteady static pressure for (a) the PRW70 location, (b) the PRW71 location, and (c) the PRW72 location.
Figure 133 Continue

(c)
Figure 134. Instantaneous static pressure contours on the radial planes of the upper stationary and rotating Kulite gage locations for (a) t=0, (b) t=0.055, (c) t=0.11, (d) t=0.167, (e) t=0.22, (f) t=0.278, (g) t=0.33, (h) t=0.389, (i) t=0.44, and (j) t=0.5
Figure 134 Continued
Figure 134 Continued

(e) Continued
Figure 134 Continued
Figure 134 Continued
Figure 134 Continued
Figure 134 Continued
7.4 Efficiency and Aerodynamic Impact of Purge Flow

The vane exit total pressure, total temperature, and mass flow rates for the run28 purge, no purge, and no cavity cases are shown in Table 13 below. The extraction plane for this data is just forward of the start of the cavity. In addition, the overall adiabatic efficiency for the one and one-half stage turbine as calculated form the FINE/Turbo solutions are provided. The high-pressure vane exit mass flow rates for the no cavity
and no purge cases are 0.9% and 1.06% lower than the purge case for run 28. The high-pressure vane is the choke point for the entire turbine rig and thus sets the overall mass flow rate.

<table>
<thead>
<tr>
<th>Purge</th>
<th>HP Vane Exit Mass Flow Rate</th>
<th>HP Vane Total Temperature</th>
<th>HP Vane Exit Total Pressure</th>
<th>Overall Adiabatic Efficiency</th>
</tr>
</thead>
<tbody>
<tr>
<td>No Cavity</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>99.11%</td>
</tr>
<tr>
<td>No Purge</td>
<td>99.11%</td>
<td>101.08%</td>
<td>99.98%</td>
<td>99.41%</td>
</tr>
<tr>
<td>No Purge</td>
<td>98.94%</td>
<td>100.55%</td>
<td>99.67%</td>
<td>99.44%</td>
</tr>
</tbody>
</table>

Table 13. Efficiency and flow parameter impact for the run 28 purge, no purge, and no cavity cases

Once the nozzle is choked, the mass flow rate is no longer a function of the exit static pressure and is governed by the following equation which assumes a Mach number of one at the minimum area:

\[
m = \frac{A_{\text{throat}} P_T}{\sqrt{T_T}} \sqrt{\frac{\gamma}{R} \left(\frac{\gamma + 1}{\gamma} \right)^{\frac{\gamma+1}{2(\gamma-1)}}} \tag{7.6}
\]

In both cases, the total temperature is higher and the total pressure is lower than the purge flow case such that based on equation 7.1 assuming that gamma, \( \gamma \), is constant (which is a good assumption since the temperatures are not very different) one would expect the mass flow rates of these two cases to be lower. The overall adiabatic efficiency, though, is greater for the no purge and no cavity case. From Table 13, the efficiency reduction or adding the purge cavity with no cooling flow is about 0.03% while introducing the purge flow reduces the efficiency another 0.14%
Looking at the high-pressure vane exit profiles for total temperature, total pressure, and tangential velocity just before the cavity in Figure 135 there is a slight reduction in all quantities between 2% and 5% span for the no purge and purge cases for run 28. This reduction is due to additional blockage created by the flow entering and exiting the cavity opening which includes a combination of main stream gas ingestion and purge flow cavity ejection in the open portion of the rim seal. Without the detailed computational findings, Kahveci et al. [71] and [72] found that the suction side high-pressure vane inner end wall heat transfer was reduce when the purge flow was introduced into the rig and hypothesized that blockage created from the purge flow introduction was the root cause. These findings certainly support Kahveci’s hypothesis. Additionally, the purge appears to have a larger decrement in comparison to the no cavity case that the presence of the cavity causes one level of blockage and the purge flow introduction increases that blockage. This is also supported by the adiabatic efficiencies show earlier in Table 13.

![Graph showing total pressure vs. span from the hub for different cases](image)

**Figure 135.** Vane exit radial profiles just forward of the cavity for (a) total pressure, (b) total temperature, and (c) tangential velocity
Figure 135 Continued

(b)

(c)
The static pressure loading at 2% span for the high-pressure vane also shows the effect of the additional blockage and reduction in total pressure, total temperature and tangential velocity. These loadings are compared in Figure 136 for all three cases. The first comparison in Figure 136(a) shows the entire vane surface loading on both the suction and pressure sides of the airfoil. The biggest differences are seen between 60% and 100% wetted distance on the suction surface which is shown in Figure 136(b) where the purge case and the no purge case have higher surface loading and thus lower Mach numbers than the no cavity case which results in reduced tangential velocity and subsequently lower total pressure and total temperature in the hub region. The loading difference is about a 4% of inlet total pressure increase for the no purge case and an additional 2% increase for the purge case.

Figure 136. Vane hub surface static pressure at 2% span for (a) the full wetted distance and (b) only the last 40% of the wetted distance

Continued
The increased aft loading on the vane trailing edge is directly supported by the experimental rig as static pressure measurements were taken just aft of the trailing edge on the hub surface. Figure 137 below shows the comparison of the trailing edge hub static pressure measurements and the computational results for run 22 and run 28. Recall that run 22 does not have purge flow (but the cavity is still present) and run 28 includes the purge flow. A notable difference at both gage locations can be seen such that run 28 with the purge flow has higher static pressure compared to run 22 for both the data and the computational models which supports the static pressure loading differences seen on the vane surface near the hub.
Figure 137. Vane trailing edge hub static pressure comparison between run 22 and run 28 for the computational and experimental rig results

The total pressure, total temperature, and tangential velocity profiles are also shown just aft of the cavity in Figure 138 in the lower 10% span of the passage. The differences between the three cases are largely concentrated to less than 6% span and by 10% span the profiles nearly sit on top of each other. Below 6% span there are significant reductions in all three quantities. The total pressure drops about 5% of inlet total pressure from the no cavity to no purge case and about 17% from the no cavity to purge case. Similar reduction levels are seen for the total temperature and tangential velocity.
Figure 138. Vane exit radial profiles just aft of the cavity for (a) total pressure, (b) total temperature, and (c) tangential velocity
The high-pressure blade static pressure loading at 2% span is shown in Figure 139 for all three cases. The addition of the purge cavity and subsequently the purge flow is shown to increase the static pressure loading at the leading edge of the blade. The increased loading is a result of the loss of tangential velocity which results in the stagnation point moving from the leading edge to the suction side of the blade reducing the flow Mach number and increasing static pressure. Just as in the case of the tangential velocity profile in Figure 138, adding the cavity and the purge flow causes the largest jump in suction side pressure loading where as just adding the cavity has a minimal effect.
Figure 139. High-pressure rotor surface static pressure at 2% span or (a) the full wetted distance and (b) the first 30% of the wetted distance

In effect, the profile losses due to the purge flow blockage cause the high-pressure vane Mach numbers near the hub over the last 20% of the airfoil to decrease (noted by the rise in static pressure in Figure
136(b)) which directly means a decrease in the exit tangential velocity of the air. As the air then passes over the cavity and is ingested into the cavity (the no purge case), is mixed with the lower momentum air leaving the cavity (the purge case) or a combination of both effects reduces the tangential velocity in the hub region further. The reductions in tangential velocity increase the negative incidence on the rotor leading edge pushing the stagnation point over on to the suction side of the airfoil, increases the suction side static pressure loading more than the slight increase in pressure side loading, thus reducing the total momentum change (change in tangential velocity from inlet to exit). The overall net effect is a reduction in power extraction of the turbine which for an actual gas turbine would require additional fuel in the combustor to raise turbine temperatures for a given power setting which in turns increases the fuel consumption of the engine. This not only demonstrates the need to keep purge flow mass flow rates to a minimum ensure no deep ingestion into the cavity but also shows how important it is to minimize steps and cavities in the flow path. And in order to properly capture the aerodynamics of the turbine from a computational perspective, these features need to be incorporated into the computational models properly.

7.5 Concluding Remarks about the In-Depth Analysis

The flow field both in the cavity and around the cavity rim seal has been analyzed using the computational models. This deeper analysis was only possible through obtaining good correlation between the computational and experimental results show throughout Chapter 6. The computational results were compared to a simple one-dimensional closed-form analytical model which did provide some insight into the mechanisms at work in the cavity, namely the effect the rotating cavity wall has on the cooling air inside. Additionally, several correlations for cavity ingestion were tested using the operating conditions for run 28. These correlations showed that the underlying data needs to contain all of the geometrical features to that of the intended engine. A majority of the insight came from looking at the three-dimensional flow field in the computational results which were used to understand how the purge flow affects the
aerodynamics and the efficiency of the turbine. Overall, the purge flow causes an additional blockage aft of the high-pressure vane increasing suction side airfoil loading and reducing tangential velocities exiting the upstream vane row. This has a downstream impact of increasing suction side incidence on the blade which in turn increase suction side loading and decreases the overall power extraction from the turbine. The mechanisms at work over the open portion of the cavity were also looked at and found to be a combination of the upstream vane wake, the downstream rotor bow wave and the interactions of these two features as the blade rotates past each vane creating both ingestion and ejection locations circumferentially.
CHAPTER 8

CONCLUSIONS

8.1 Conclusions from the Research

The research presented within for the first time provides a detailed comparison between experimentally measured and computational predictions for the flow field in and around a purge flow cavity and rim seal within a one and one-half stage, engine representative, high-pressure turbine rig operating at the proper design corrected conditions. It is important to note that the turbine rig also incorporated many of the complicating factors typically found in gas turbine engines such as the high-pressure vane film cooling and elevated, radial inlet temperature profiles created from a combustor emulator just upstream of the rig inlet. All of these features were incorporated into the computational efforts in order to accurately match the turbine rig and produce a high quality set of predictions.

The main part of this research was geared toward the computational study which involved performing a set of predictions for five different operating conditions from the experimental rig: a cold inlet case with no cooling, an elevated flat inlet temperature profile with with/without cooling, and an elevated radial inlet temperature profile case with /without cooling. The predictions were desired to be both steady and unsteady for all three blade rows in the experimental rig and the commercial CFD code produce by NUMECA International called FINE/Turbo was selected to perform the task. The CFD code is a Reynolds Averaged Navier-Stokes solver capable of performing both steady and unsteady analysis on
turbomachinery and was chosen for its capability and flexibility (as well as availability) in performing the computational studies.

Initially FINE/Turbo was put through the paces to evaluate its abilities against experimental using multiple test cases. For the NACA0012 test case, the code was able to predict the 30+ year old surface pressure loading data for multiple angles of attack and inlet Mach numbers with a fairly high degree of accuracy. Then, using both NASA Rotor 37 and Stage 35, the multi-stage steady capabilities of the code were tested in an effort to predict the operational curves and the exit profiles for total pressure, total temperature, and adiabatic efficiency. Due to the operational characteristics of these two rotors and the high degree of difficulty that has been seen in the past for these types of predictions, FINE/Turbo provided a very good overall comparison to the acquired experimental data that was noted to be as good as or better than those results currently published in the open literature. The predictions were not without flaws and the typical misses noted in other predictions were also seen in the FINE/Turbo predictions however the overall provided good confidence in the multiple blade row modeling implementation within the code.

The last test case used to test out FINE/Turbo’s capabilities was a matching one and one-half stage turbine rig to the one used throughout this research as it was a previous build of the same geometry. The previous experimental build also incorporated Kulite measurements throughout the entire turbine in order to capture time-accurate pressure data in all three blade rows which included the hub, shroud, and airfoil surfaces. This case provided an excellent opportunity to test out both methods (phase-lag and harmonic balance) of performing non-linear, unsteady analysis that have been incorporated into the CFD code given the availability of the time-accurate data. A single prediction of one operating point was completed and the steady, phase-lag unsteady, and harmonic unsteady results were compared to the data from both a time-average and time-accurate standpoint. All three models showed excellent agreement with the time-average data in all locations. From a time-accurate perspective, both the phase-lag and the harmonic unsteady methods produced very similar results at all of the locations used for comparison. It was actually difficult
to determine which model truly performed better as in many instances the results were on top of each other and the experimental data. In the end, it was decided to use the harmonic unsteady method due to the quicker turnaround times experience while running this case as it was pretty evident based on this test case that no loss in accuracy would result.

The computational and experimental comparisons were provided for all five run conditions at all of the pressure and temperature gage locations installed in the rig. The comparisons at the exit of the low-pressure vane (the exit of the turbine) rig were in very good agreement with the measured data and matched the profiles shapes very well for both total temperature and total pressure at all conditions. Comparisons between the cooled and uncooled radial and flat inlet profile runs for exit total temperature showed that the profile shapes were found to be very similar between cooled and uncooled runs with merely a downward temperature shift produced by the introduction of cooling. These trends were seen in both the measured and predicted data. Another interesting note was the total pressure exit profiles which were all nearly identical although run 43 (an uncooled, flat, elevated inlet temperature run) did show slightly more total pressure loss when compared to the other three runs. Overall, the exit profile comparisons did provide good confidence in FINE/Turbo’s abilities to capture both temperature and pressure losses through all three blade rows.

The experimental rig also measured the inlet profile to the rotating blade in the rotating frame of reference for all five runs. The blade leading edge profiles were captured very well by FINE/Turbo with exception to a slight drop in the profile temperature at about 35-50% span for all five cases. Initially this was thought to be due to the absence of vane cooling which lead to ingestion and ejection of the main gas flow path air throughout the film cooling pattern drilled into the high-pressure vane, however this hypothesis provided by Mathison [68] in the absence of the cooled runs. When the cooled run experimental data was investigated, a very similar temperature drop was noted as well leading to the possible conclusion that the inner and outer plenums used to feed the cooling air from LCF to the high-pressure vane may not be evenly distributing cooling air to from hub to tip inside the vane.
The remainder of the comparisons was for the instrumentation in and around the cavity and rim seal area. Starting with the high-pressure vane hub and working downstream, the harmonic unsteady results were found to be in good agreement with the experimental data or all run but run 43 which should much lower static pressure loading on the vane hub than predicted by the computational models. The time-accurate comparisons also showed that the overall trends and magnitudes were captured well although do not match identically to the data. When looking at the hub static pressure trends within the computational models, it was noted that the purge flow cavity creates additional blockage at the exit of the vane causing suction side loading to be increased thus dropping Mach numbers, total pressure, total temperature, and the velocity leaving the vane row. And once the vane exit air crossed the cavity, additional velocity, total temperature and total pressure loss was accumulated resulting in increased incidence on the blade suction surface which increased suction surface loading ultimately dropping the hub power extraction across the rotating blade row. Additional, the temperature and pressure losses generated by the cavity reduce the adiabatic efficiency as calculated for this turbine. This leads one conclusion in that enough purge flow air should be provided in order to maintain positive purge in the cavity however should be minimized to prevent downstream losses.

The cavity Kulites and thermocouples also provided some good and some not so good comparisons. Overall the cavity time-average and time-accurate measured pressure were predicted very well by FINE/Turbo in both magnitude and in trend. Using the measured data in conjunction with the computational model results, the cavity ingestion and ejection behavior was studied to an effort to understand how the main drivers behind the ingestion/ejection phenomena. For all of the runs analyzed, it was not found that the main gas path flow was ingested deep within the cavity as only shallow ingestion to the forward upper tip of the rotor angel was noted and the ingested air mainly consisted of the hub flow from the high-pressure vane. It was determined that high-pressure vane, high-pressure blade, and the
associated interactions between these two blade rows play a dominate role in the cavity ingestion and these interactions could be seen deep within the cavity although at much reduce amplitudes.

The static temperature measured by the thermocouples for run 21, the cold inlet case, was predicted very well while the four cases with increased inlet temperatures were found to under predict the data. Run 43 was once again noted has having the highest discrepancies between the data and the predictions. The cavity static temperature sensitivity to the isothermal wall temperature was briefly studied by using the double-sided heat flux gages on the high-pressure vane to determine a new temperature for the vane hub, shroud and airfoil surface as these gages indicated that the original temperatures applied may be too low. The cavity temperatures were found to be very sensitive to the isothermal wall temperature specification and the increased temperature models predicted stationary wall temperatures for the two cases analyzed much closer to the data while the rotating locations were now over-predicted by as much as the original under-predictions suggesting that both the stationary and the rotating wall temperatures were probably not completely reflective of the rig but are in the right regime.

A similar result was obtained for the rotor platform thermocouple comparisons in that the run 21 cold inlet and the run 22 and run 28 radial profile inlet runs matched fairly good to the measured data. However, the two flat inlet profile cases were found to depart with run 43 differences being the largest. For the two flat inlet profile cases, the temperatures near the wall were significantly higher than the specified wall temperatures as compared to the radial runs such that the isothermal wall temperature and the gage height placement within the thermal boundary layer become extremely important to how well or poor the overall comparison will be. The rotor platform temperatures were found to be insensitive to the vane hub isothermal wall temperature such that they are most likely a strong function of the specified platform temperature as anticipated.
The computational models did not produce an exact match to the experimental data at all locations. This was an anticipated result as there are assumptions made throughout the computational and experimental model that may not be entirely accurate or reflective of actual conditions. Examples of such assumptions includes the use of the harmonic unsteady method which only models the unsteady behavior related to the harmonics of the adjacent blade rows and that the inlet pressure and temperature profiles are axisymmetric about the entire inlet annulus. From an experimental standpoint, only the average values were compared to the data which were produced from ensemble averaging all of the passages around a single rotor revolution. Differences are known to exist around the annulus due to inlet temperature and pressure variations circumferentially, manufacturing tolerance on the flow path hardware as not all vanes and blades were manufactured perfectly and the possibility that frequencies other than those related immediate upstream and downstream airfoil passing frequencies can and do exist. In the end, it is of the best interest to understand the limitations of both the experimental and the computational results and how it may or may not apply to an actual gas turbine engine. In the end, given the assumptions that were put into the computational and experimental models the results were considered to be very good for the first attempt at the cavity predictions for this turbine.

8.2 Recommendations for Future Research

Given the opportunity any engineer would want to incorporate more into both the experiment and the computational efforts in hopes to grab that additional piece of data that might shed some light on the unanswered questions. From an experimental perspective it would be important to include thermocouples on all blade rows specifically installed to measure the surface metal temperatures on the airfoil, hub, shroud, and the cavity walls. While the initial estimates used in the computational model for these temperature (which translates to the isothermal wall temperature specification) produced fairly good correlation between the experiments and the computational results, subsequent analysis of the heat flux
gages and eventually the computational vane hub temperature affects showed that some of the assumptions applied to the higher temperature inlet cases may not have been entirely reflective of the rig. Another potential source of improvement might be the inclusion of temperature and pressure gages within the main part of the cavity well below the cavity seals. This data would provide additional insight into the lurking possibility of non-engine order frequencies within the cavity of which have been noted by some of the currently published works on purge flow cavities. Non-uniform inlet conditions are another concern, while not discussed within this work, which often presented itself within the experimental data analysis when performing the ensemble averages for the instrumentation. This could be best addressed by incorporation of additional inlet and exit total pressure and total temperature rakes around the annulus which could given further insight into how uniform the inlet flow conditions are however there is always some risk that the peaks and troughs could still be missed. The main issue with adding more instrumentation is the physical limitations of the rig as well as competing interests of other engineers looking to acquire data from the rig and it may not be feasible in the end.

From a computational modeling perspective, there is a long list of items that could possibly help to answer the additional questions generated throughout this research. The first effort would be to couple the cooling air flow into the high-pressure vane film cooling plenums with the main gas path flow by including the vane internal plenums and cooling holes on the hub, shroud, and airfoil surfaces directly into the model. Source term injection points were used to model the vane film cooling pattern which may or may not be reflective of distribution of cooling air experienced in the experimental rig. This is a daunting task as noted earlier but is one worthy of study as it may reveal some details that were not present in the source term approach such as redistribution of the cooling air both radially and axially. Another point worthy of consideration is the non-engine order unsteadiness which could be present in the cavity. The nonlinear harmonic and phase-lag unsteady analysis techniques, by definition, assume the unsteadiness to be a function of engine order frequencies produced by the upstream and/or downstream blade rows under consideration. This could truly only be addressed by attempting either a full annulus or half annulus model of the turbine to see if any non-engine order frequencies are excited in the cavity itself and how they might
affect the cavity rim seal area. The additional instrumentation within the main portion of the cavity discussed early would help to guide the best choice of model. Such a model could also provide the stepping stone to study the how the non-axisymmetric inlet conditions affect the downstream aerodynamics around the annulus. When tasks of this size are under taken, one often becomes limited by the computer resources and patients available to take on such large calculations. The typical computational rants of grid density, turbulence modeling, time step size in the unsteady calculations, and computation algorithms definitely apply to the list of future considerations. These aspects could be studied with some regridding of the current problem but in the quest for additional information there is not always a clear direction when studying these effects on a solution.

8.3 Final Remarks

In the end, this research has provided a unique look at the behavior of the flow field for a transonic turbine both in and around the purge flow cavity and rim seal located between the high-pressure vane and high-pressure blade from both a computational and experimental perspective. Others have operated ambient pressure rigs with reduced fidelity in order to capture unsteady data in and around the cavity and complex computational models have been executed in an effort to study the cavity from an in-depth perspective for both rigs and actual engine geometries. However, no effort to date has attempted to bring both an experiment representative of actual engine hardware and operating conditions together with a high fidelity matching set of unsteady predictions to perform both a full comparison to the rig data followed by an in-depth study into what is happening in and around the cavity and what the measured data is seeing. While a lot of knowledge and insight has been gained into how purge flow cavities operate using this type of approach, there are probably more questions than answers that have been generating by the work within. And while it was attempted to answer as many of the questions as possible, often the answers bring about
more questions as the engineering mind seeks to understand every aspect of a good, solid technical challenge.
REFERENCES


APPENDIX A: CONVERGENCE PLOTS FOR THE NACA0012 AIRFOIL

This Appendix contains the convergence plots for each of the nine computational cases run for the NACA0012 airfoil geometry. Note that the case numbers correspond to Table 4 in section 4.3.1. The goal for each calculation was to continue to iterate until both the RMS and Max residuals were flat or until an RMS residual of at least -4.0 is achieved. Figure 140 below shows the convergence history for the 0.3 Mach number, 0° angle of attack case.
Figure 140. Case number one: (a) RMS Residual history, (b) inlet and exit mass flow rate
Figure 141. Case number two: (a) RMS Residual history, (b) inlet and exit mass flow rate

Figure 141 shows the convergence history for the 0.3 Mach number, 4° angle of attack case.
Figure 142. Case number three: (a) RMS Residual history, (b) inlet and exit mass flow rate

Figure 142 shows the convergence history for the 0.5 Mach number, 0° angle of attack case.
Figure 143. Case number four: (a) RMS Residual history, (b) inlet and exit mass flow rate

Figure 143 shows the convergence history for the 0.5 Mach number, 4° angle of attack case.
Figure 144. Case number six: (a) RMS Residual history, (b) inlet and exit mass flow rate

Figure 144 shows the convergence history for the 0.5 Mach number, 8° angle of attack case.
Figure 145. Case number seven: (a) RMS Residual history, (b) inlet and exit mass flow rate

Figure 145 shows the convergence history for the 0.75 Mach number, 0° angle of attack case.
Figure 146. Case number eight: (a) RMS Residual history, (b) inlet and exit mass flow rate

Figure 146 shows the convergence history for the 0.75 Mach number, 4° attack angle case.
Figure 147. Case number nine: (a) RMS Residual history, (b) inlet and exit mass flow rate

Figure 147 above shows the convergence history for the 0.8 Mach number, zero attack angle case. Note that the calculation did not achieve -4.0 or less in RMS residuals, however the inlet and exit flows had stabilized and the residuals within several chord lengths of the airfoil were well -6.0.
APPENDIX B: NASA ROTOR 37 AND NASA STAGE 35 CONVERGENCE PLOTS

The purpose of this appendix is to provide the additional convergence histories for the Rotor 37 stall point and the Stage 35 peak efficiency point as shown in section 4.3.2. All points as computed in the 100% speed lines for both configurations will not be provided. Figure 148 shows the convergence history for the Rotor 37 stall point calculation. This calculation took an additional 800 iterations to achieve convergence as compared to the peak efficiency point shown in Figure 40.
Figure 148. Rotor 37 stall point convergence history for (a) RMS and maximum residuals, and (b) inlet and exit mass flow rates
Figure 149 shows the convergence history for the peak efficiency point for Stage 35. This calculation was stopped after 674 iterations as the inlet and exit mass flow rates had been flat for about 250 iterations. The RMS and maximum residuals were still dropping at the time the calculation was stopped, however the RMS residuals were below -4.5.

(a) Continued

Figure 149. Stage 35 peak efficiency point convergence history for (a) RMS and maximum residuals, and (b) inlet and exit mass flow rates
Figure 149 Continued

(b)
APPENDIX C: FLOW PATH AND STRUT ANALYSIS FOR THE UNCOOLED ONE AND ONE-HALF HIGH-PRESSURE TURBINE RIG BOUNDARY CONDITIONS

The purpose of this Appendix is to document the forward flow path and strut region computational analysis performed using FINE/Turbo to extract the velocity angle inlet boundary conditions for the inlet to the high-pressure vane for subsequent steady and unsteady computations performed for the uncooled one and one-half high-pressure turbine rig test case in section 4.3.3. Figure 150 below shows the cross-section of the forward flow path and strut area between the combustor emulator and the high-pressure vane leading edge.
Figure 150. Forward flow path and strut cross-section (Not To Scale)

The inlet plane for the computational grid is the exit plane of the combustor emulator and the exit plane is the high-pressure vane leading edge. The strut airfoil is a two-dimensional, non-turning airfoil of which four are located 90° apart around the circumference. The grid for the forward flow path section was created using FINE/Turbo’s Autogrid5 program using the default geometry shown in Figure 68. The grid contains 57 points in the radial direction, 74 points axially along the strut surface, and 65 points tangentially for a total of 672,000 grid points. The initial surface offset is set at 2.54e-6 m as used for all other grids. Figure 151 below shows the grid resolution and the resulting $y^+$ values on the surfaces.
Figure 151. Forward flow path and strut (a) grid density and (b) resulting $\pm$ values
The resulting $y^+$ values are smaller than desired however it was found that these values are acceptable for this analysis. Inlet boundary conditions are specified using total temperature and total pressure as measured from the rig, the radial velocity angle which is set to zero, the tangential velocity angle also set to zero, and the turbulence viscosity ratio calculated using the inlet total temperature and the 2.5 factor as recommended for internal flows. All inlet boundary conditions are specified as single values over the entire face of the inlet plane. The exit boundary condition is specified to run to mass flow using the velocity scaling method which adjusts the exit plane pressure profile. The mass flow rate was set to the measured value from the rig. All surfaces are adiabatic with the no-slip condition applied and all rotational
velocities are set to zero. The solution is run using three levels of multigrid (coarse, medium, and fine) running 100 iterations or until an RMS residual of -3.0 is achieved on the first two levels. All remaining iterations are completed on the fine grid level until an RMS residual of -6.0 is achieved or the residuals and mass flow rates remain flat. The resulting convergence is show in Figure 152 below.

Figure 152. Convergence history for the forward strut and flow path steady solution for (a) log of residuals and (b) the inlet and exit mass flow rates
Figure 152 Continued

![Graph showing Mass Flow (kg/sec) vs Iteration]

(b)

Figure 153 below shows the total pressure, total temperature, radial angle, and tangential angle radial profiles at the rake plane of the forward flow path and strut solution. Note that the temperature and pressure profiles are fairly flat suggesting that there is little to no loss through this section of the rig when adiabatic walls are assumed. The tangential profile essentially shows no tangential velocity within the flow field; the extremely low angles calculated for this solution are most likely due to round off error. The radial angle profile shows a nearly linear decrease from the hub angle of 13.44° to 0° at the shroud.
Figure 153. Resulting radial profiles at the rake plane for (a) total pressure, (b) total temperature, (c) tangential flow angle, and (d) radial flow angle
Figure 153 continued

(c)

(d)
APPENDIX D: UNCOOLED ONE AND ONE-HALF STAGE HIGH PRESSURE TURBINE RIG CONVERGENCE PLOTS

The purpose of this appendix is to document the convergence history for each of the numerical simulations that were executed as part of the uncooled high-pressure turbine and low-pressure vane rig test case provided in section 4.3.3. The convergence history for the three blade row steady solution is shown in Figure 51. This solution reached the residual convergence level after only 1500 iterations of which included 100 iterations on the coarse grid and 100 iterations on the medium grid. The inlet and exit mass low rates also set up quickly and remain flat after about 500 iterations.

Figure 154. Uncooled turbine rig two blade row steady solution convergence history for (a) RMS and maximum residuals and (b) the inlet and exit mass flow rates
Figure 154 Continued

(b)

The convergence history for the harmonic unsteady solution is provided in Figure 155 below. The solution was run for the full 4000 iterations however it can be seen that the residuals flatten out with little change about 2000 iterations. The mass flow rate history at both the inlet and the exit show a very similar story but take only about 500 iterations before remaining flat. Note that the mass flow rate history in Figure 155 is on a very small y-axis scale.
Figure 155. Uncooled turbine rig two blade row unsteady harmonic solution convergence history for

(a) RMS and maximum residuals and (b) the inlet and exit mass flow rates
Figure 156 shows the convergence history of the two blade row phase-lag unsteady solution. The RMS residuals only reduce about an order of magnitude over the steady solution while the maximum residuals remain close to their starting position. Note that the residual history is reset at the start of this computation. In Figure 156(b), the overlay of the 36th and 37th period shows the repeating nature of the residuals and in Figure 156(d) it can be seen that the mass flow rates from one period to the next have set up with 0.02% of each other.

Figure 156. Uncooled turbine rig two blade row unsteady phase lag solution convergence history for (a) RMS residuals, (b) the 43rd and 44th period of RMS residuals, (c) the inlet and exit mass flow rate, and (d) the 43rd and 44th period for the exit mass flow rates

Continued
Figure 156 Continued

(b) Log(Residual) vs. Iteration for 36th and 37th Periods

(c) Mass Flow Rate (kg/sec) vs. Iteration for Inlet and Exit Mass Flow

Continued
Figure 156 Continued

![Graph of Iteration vs Mass Flow Rate (kg/sec) showing % Difference between 36th and 37th Periods. The graph displays oscillations in Mass Flow Rate with corresponding % Differences, with labels for 36th Period, 37th Period, and % Difference.](image-url)
APPENDIX E: CONVERGENCE PLOTS FOR ALL RUN 21, RUN 22, RUN 28, RUN 33, AND RUN 43 NUMERICAL MODELS

The purpose of this appendix is to document the convergence history for each of the steady, harmonic unsteady, and phase lag unsteady simulations run for this research. The first set of convergence plots is for the simulations associated with Run 22 as all Run 21 plots are provided in section 5.6. Convergence plots for the three blade row steady solution are provided in Figure 157(a) and (b) for the residuals and mass flow rates respectively.
Figure 157. Run 22 three blade row steady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates

The second set of convergence plots for Run 22 is for the three blade row harmonic unsteady solution in Figure 158(a) and (b) for the residuals and mass flow rates respectively.
Figure 158. Run 22 three blade row harmonic unsteady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates.
The next pair of convergence plots is for the Run 28 steady and harmonic unsteady solutions. The steady three blade row solution convergence is shown in Figure 159(a) and (b) for the residuals and mass flow rates respectively.

(a) Continued

Figure 159. Run 28 three blade row steady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates
(b)

The second set of convergence plots is for the Run 28 three blade row harmonic unsteady solution shown in Figure 160(a) and (b) for the residuals and mass flow rate respectively.
Figure 160. Run 28 three blade row harmonic unsteady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates
The next series of convergence plots are for the steady and unsteady solutions performed for Run 33. The steady three blade row solution for the residuals and the mass flow rate are shown in Figure 161(a) and (b) respectively.

Figure 161. Run 33 three blade row steady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates
Figure 161 Continued

The three blade row harmonic unsteady solution is shown in Figure 162(a) and (b) for the residuals and mass flow rates respectively.
Figure 162. Run 33 three blade row harmonic unsteady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates.
The final set of convergence plots are for the steady and harmonic unsteady three blade row solutions for Run 43. Figure 163(a) and (b) show the convergence history for residuals and mass flow rates respectively for the three blade row steady solution.

Figure 163. Run 43 three blade row steady convergence history for the (a) RMS and maximum residuals and the (b) inlet and exit mass flow rates
The Run 43 three blade row harmonic unsteady convergence history is shown in Figure 164(a) for the residuals and Figure 164(b) for the mass flow rates.
Figure 164. Run 43 three blade row harmonic unsteady convergence history for the (a) RMS residuals, (b) the inlet and exit mass flow rates.

446